

User Guide

Autodesk Inventor integration
for
SOLIDWORKS PDM

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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 - Inventor integration supports users when working with Autodesk Inventor data. The integration is directly embedded in *Windows Explorer* as well as Inventor.

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. Inventor. It offers possibilities to display extended menu options (ribbons, edge bars) in the application and to simplify the work with the files.

The supported Inventor formats are:

- Inventor part (.ipt)
- Inventor assembly (.iam)
- Inventor drawing (.idw)

1.2 Add-in

Activating the integration menu




Starting Inventor for the first time after installation, Inventor prompts the user to activate the add-in. Do the following:

1. In the dialog *Add-in Manager Security Alert* press **Launch Add-in Manager**.
2. In the upcoming dialog *Add-in Manager*, disable **Block** and enable **Loaded/Unloaded** and **Load Automatically**.
3. Press **OK**.

The SOLIDWORKS PDM add-in is now automatically loaded every time Inventor is started. This includes the ribbon and the edge bar.

Ribbon bar

The SOLIDWORKS PDM ribbon bar contains all features of SOLIDWORKS PDM. The following functions are available:

Function	Title
	Check In
	Check Out
	Undo Check Out

Function	Title
	Get Latest Version
	Get Version
	Show Properties
	Change State
	Search
	Select in Windows Explorer

The functions are executable for the loaded root element or selected elements. Multiple selection is possible for functions

- Check in
- Check out
- Undo check out and
- Get latest version

For other functions, the command is executed only for the first selected element in case multiple elements are selected.

Function calls from the ribbon bar update the data in the edge bar immediately.



Edge bar - SOLIDWORKS PDM File Properties

The edge bar consists of the control strip and the data structure (list view) of the loaded object. Furthermore it shows the metadata of all structure elements. Metadata is displayed immediately after loading a model or drawing from a vault.

Users can change the position of the edge bar in Inventor as desired via drag and drop.

Control strip


The control strip has the same features as the ribbon bar.

The functions are executable for loaded root element or selected elements. Multiple selection is possible for functions

- Open File
- Check in
- Check out
- Undo check out and
- Get latest version

For other functions, the command is executed only for the first selected element in case multiple elements are selected.

Since it is possible to change metadata in parallel via *Windows Explorer*, the control strip provides the

function  **Refresh**. This function reads current metadata from SOLIDWORKS PDM server and visualizes them in the data structure.

E.g. an assembly, which is checked in, is loaded in Inventor. The user performs a **Check Out** via *Windows Explorer*. As a result, the metadata is not up to date in Inventor. **Refresh** updates items in edge bar and displays correct check out status.

Data structure

The tree list view represents the file structure of the loaded element. For every structural element, all SOLIDWORKS PDM features can be executed via a context menu. In contrast to function calls via ribbon bar and control strip, they are executed for currently selected object.

2 Functional description of the Inventor 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

If a file is currently opened in a Inventor session, the file status is updated. Files that are not checked out are in read-only mode.

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.

As a result of the check in, the content of the edge bar is also updated.



During check in a bill of material is created in SOLIDWORKS PDM only for assemblies ([Update BOM](#) (p. 14)).

Check Out



Checked in models are always write protected. To remove the write protection, the user needs to check out the desired element. With **Check Out**, the write protection is removed locally on the client and the object is locked for other users.

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.

If a user tries to modify an object that is checked in, he is prompted to check out the file. On confirming the request, the object is checked out. If the user does not confirm the request, the changes may be lost or must be saved in a new file.

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.

Selecting the **Undo Check Out** command opens the *Undo Check Out* dialog. The files, whose changes are undone, are highlighted. After confirming the dialog with **Undo Check Out**, the model is reloaded and the edge bar is updated accordingly.



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 Inventor.



Both functions show warnings if user modifications might get overwritten.

If the user ignores the warning and continues anyway, the CAD structure may be corrupted.

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.

After loading a version, the edge bar displays the corresponding metadata.

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, the object is loaded in the latest version and the edge bar is updated.

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 Inventor. Properties are usually saved in CAD files and in SOLIDWORKS PDM fields and can be accessed from the Inventor'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.

2.5 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.

2.6 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 Inventor. This tool can be used to search for files and folders in the vault using various masks, such as the Inventor 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 Inventor is possible from here.

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

Select in Explorer



It is often useful to find out the location of a selected object in the edge bar in the *Windows Explorer*.

Selecting **Select in Explorer** for a desired file, a new window of the *Windows Explorer* opens and the affected file is marked.

Using this function via the ribbon or control strip, the location of the root or selected element is opened.

2.7 Update BOM



The bill of material is created during check in of the loaded assembly. The usage is described in linked use cases.



The read out BOM is linked with root/top element of the assembly structure. It works only with checked in root element because of the last version number.

Related links

[Creating bill of materials](#) (p. 17)

[Creating bill of materials for subassemblies](#) (p. 18)

3 Using the integration - best practices

3.1 Adding a single file to the vault

How can files be added to SOLIDWORKS PDM via Inventor?

Procedure

1. Start Inventor and create a new assembly, drawing or part via Inventor function **New**.
→
If the user is not logged in and multiple vaults exist, dialog *Select a vault* appears. Continue with step 2.
If the user is not logged in, a login dialog appears. Continue with step 3.
2. Select a vault and press **OK**.
→ Login dialog appears.
3. Enter the login credentials and confirm.
→ Selected vault comes to the foreground.
4. Enter a name for the file and press **Save**.
→ Depending on the configuration, the data card of the saved file appears or not. Refer to *Installation and Administration Guide* for more information.
5. Add/edit desired data values and press **OK**.
6. To check the added values, press **Show Properties** from the integration menu.
7. Press **Check In** from the integration menu.
→ *Check in* dialog appears.
8. Confirm the *Check in* dialog by pressing **Check in**.

Result

Check in process starts and finishes without errors. File is added to the vault successfully.

3.2 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.

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.

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 Loading drawings and models

How can the user load files in Inventor?

Models and drawing can be opened via the vault view or via the file dialog of Inventor. In both cases Dassault Systèmes SOLIDWORKS PDM Professional requires the user to log in if not already done.

After logging in and loading the model from a vault, the edge bar is displayed with the metadata of the files.

If integration functions are used without information about a vault, the dialog *Select a vault* appears showing all available vaults on the client. After selecting a vault, the login dialog appears.

Information on the connection status can be obtained via the SOLIDWORKS PDM icon in Windows hidden icons.

3.4 Creating bill of materials

How can the user create bill of materials in SOLIDWORKS PDM?

Procedure

1. Create a new assembly with some underlying parts, add it to SOLIDWORKS PDM vault and open it in Windows Explorer. Select the top element of the assembly, switch to tab *Bill of Materials* and check view *Inventor BOM*.
→ View not available.
2. Open the top element of the assembly in Inventor.
3. To check quantity of assembly components, press **Bill of Materials** in tab *Assemble*.
→ A message appears that displays that the assembly is not checked out and asks if you want to check out first.
4. Confirm the message dialog with **Yes** to open dialog *Check Out*. Select all structure elements and confirm.
→ Assembly is checked out.
5. To create the Inventor BOM, check in the assembly via the integration ribbon.
→ Assembly is checked in. BOM is created.
6. Go to the vault, select the top element of the assembly, switch to tab *Bill of Materials* and check view **Inventor BOM**.
→ A bill of material is available in SOLIDWORKS PDM . Quantities are correct. Version of top element is 2 and version of subelements is 1.
7. Return to Inventor and check out all files again.
8. Press **Bill of Materials** in tab *Assemble* and do the following:
 - Change the quantity of an underlying element
 - Create a virtual component (e.g. oil with quantity 100)
9. Close the dialog and check in all files again.
→ Assembly is checked in. BOM is created.
10. Return to SOLIDWORKS PDM and refresh view **Inventor BOM**
→
Version of top element increased by 1. Quantity of subelement has changed correctly. Virtual component is available with correct quantity in version 0.
11. Return to Inventor and check out all files again.
12. Press **Bill of Materials** in tab *Assemble* and do the following:
 - Change the quantity of an underlying element
 - Create a virtual component (e.g. stuff with quantity 1)
13. Close the dialog and check in all files again.
→ Assembly is checked in. BOM is created.

14. Return to SOLIDWORKS PDM and refresh view **Inventor BOM**
→ Version of top element increased by 1. Quantity of subelement has changed correctly. Virtual component is available with correct quantity in version 0.
15. Switch between the different assembly versions to check BOM changes.

Result

BOM creation successfully.

3.5 Creating bill of materials for subassemblies

How can the user create bill of materials in SOLIDWORKS PDM?

Procedure

1. Create a new assembly with underlying subassemblies, add it to SOLIDWORKS PDM vault and open it in Windows Explorer. Select the top element of the assembly, switch to tab *Bill of Materials* and check view *Inventor BOM*.
→ View not available.
2. Open the top element of the assembly in Inventor.
3. To check quantity of assembly components, press **Bill of Materials** in tab *Assemble*.
→ A message appears that displays that the assembly is not checked out and asks if you want to check out first.
4. Confirm the message dialog with **Yes** to open dialog *Check Out*. Select all structure elements and confirm.
→ Assembly is checked out.
5. To create the Inventor BOM, check in the assembly via the integration ribbon.
→ Assembly is checked in. BOM is created.
6. Go to the vault, select the top element of the assembly, switch to tab *Bill of Materials* and check view **Inventor BOM**.
→ A bill of material is available in SOLIDWORKS PDM . Quantities are correct. Version of top element is 2 and version of subelements is 1.
7. Return to Inventor and check out the top element.
8. Press **Bill of Materials** in tab *Assemble* and do the following:
 - Change the quantity of the first subassembly
9. Close the dialog and check in the top element.
→ Assembly is checked in. BOM is created.
10. Return to SOLIDWORKS PDM and refresh view **Inventor BOM**
→ Quantity of subassembly has changed.
11. Return to Inventor, open the subassembly and check it out.
12. Press **Bill of Materials** in tab *Assemble* and do the following:
 - Create a virtual component with any quantity

13. Close the dialog and check in the subassembly.
→ Assembly is checked in. BOM is created.
14. Return to SOLIDWORKS PDM and refresh view **Inventor BOM**
→
BOM is correct. Created virtual part is available.
15. Check the **Inventor BOM** for the top assembly.
→
Created virtual component is not available in structure of subassembly.

3.6 Renaming, moving and copying

The tree functions renaming, moving and copying run in the context of the *Windows Explorer* and deal with the correction of file references. The affected files must be checked in, otherwise no processing takes place.

Renaming

The function **Rename** is called via the context menu in the *Explorer*.

After selecting, the user can change the name of the selected file. After changing the name, file references of the affected parents are updated (see tab *Where Used*). The new name also affects older versions.

Moving

Move usually changes the file location. The function can be accessed via **Tools > Move Tree....**

All settings for moving files can be set in the standard *Move Tree* dialog of SOLIDWORKS PDM. It is possible to move parts, complete assemblies and drawings.

In the *Move Tree* dialog, the user can define

- a default target path
- a separate target path per file and/or
- specify new file names

Furthermore the user gets error messages and warnings via the dialog.

SOLIDWORKS PDM executes the action according to the defined settings. This means that after moving the files no longer exist in the source path.

Copying

The function **Copy Tree...** is also available via the menu *Tools*.

Unlike **Move Tree...** the copy function preserves the files in the source directory. In the target directory, new instances of the affected files are created. The dialog *Copy Tree* is similar to the dialog *Move Tree* and supports the user with the settings.

3.7 Restrictions

Version files as template files

Inventor integration does not support using version files as template files.

Renaming newly imported models

Loading assemblies fails with newly imported models when a subassembly is renamed.

The problem is not the renamed file itself, but a neighboring file that has not been renamed.

To avoid this behavior, do the following:

- After importing the new model, open it in Inventor.
- Check out the complete structure.
- Save all components locally.
- Check in the structure again.
- Rename the files.