

User Guide

Siemens Solid Edge 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 - Solid Edge integration supports users when working with Siemens Solid Edge data. The integration is directly embedded in *Windows Explorer* as well as Solid Edge.

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. Solid Edge. It offers possibilities to display extended menu options (ribbons, tree view browser) in the application and to simplify the work with the files.

The supported Solid Edge formats are:




- Solid Edge part (.par)
- Solid Edge assembly (.asm)
- Solid Edge drawing (.dft)
- Solid Edge sheet metal (.psm)
- Solid Edge weldment (.pwd)

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
	Check In
	Check Out
	Undo Check Out
	Get Latest Version
	Get Version
	Show Properties

Function	Title
	Change State
	Search
	Select in Windows Explorer



Edge bar - SOLIDWORKS PDM File Properties

The edge bar is available in the Solid Edge pane and consists of a control strip and the data structure (top and first level) including metadata of the loaded object. Metadata is displayed immediately after loading a model or drawing from a vault.

Users can change the position of the edge bar in Solid Edge 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 Solid Edge. The user performs a **Check Out** via *Windows Explorer*. As a result, the metadata is not up to date in Solid Edge. **Refresh** updates items in edge bar and displays correct check out status.

Data structure

The tree list view represents the first level 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 Solid Edge add-in

2.1 Check In / Check Out

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

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.



During check in a bill of material is created in SOLIDWORKS PDM only for assemblies ([Update BOM](#) (p. 12)).

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.

2.2 Load Versions

Use these functions to load desired versions from SOLIDWORKS PDM to Solid Edge.

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 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.

2.3 Properties

A property is a piece of metadata that can be exchanged bi-directionally between SOLIDWORKS PDM and Solid Edge. Properties are usually saved in CAD files and in SOLIDWORKS PDM fields and can be accessed from the Solid Edge'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.

2.4 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.

2.5 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 Solid Edge. This tool can be used to search for files and folders in the vault using various masks, such as the Solid Edge 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 Solid Edge 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.

2.6 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. 13)

[Creating bill of materials for subassemblies](#) (p. 14)

3 Using the integration - best practices

3.1 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 Solid Edge.
3. To check quantity of assembly components, perform a right-click on the top element in the model tree and select **Occurrence Properties**.
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 Solid Edge 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 **Solid Edge 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 Solid Edge and check out all files again.
8. Perform a right-click on the top element in the model tree and select **Occurrence Properties**. Choose one or two files and change the dropdown value of *User Defined Properties* from **No** to **Yes** and enter a value for the quantity. Press **OK**.
→ Quantity changed and saved.
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 **Solid Edge BOM**
→
Version of top element increased by 1. Quantity of subelement has changed correctly.
11. Switch between the different assembly versions to check BOM changes.

Result

BOM creation successfully.

3.2 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 Solid Edge.
3. To check quantity of assembly components, perform a right-click on the top element in the model tree and select **Occurrence Properties**.
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 Solid Edge 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 **Solid Edge 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 Solid Edge and check out the top element.
8. Perform a right-click on the top element in the model tree and select **Occurrence Properties**. Choose one or two files and change the dropdown value of *User Defined Properties* from **No** to **Yes** and enter a value for the quantity of the subassembly. Press **OK**.
→ Quantity changed and saved.
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 **Solid Edge BOM**
→ Quantity of subassembly has changed.
11. Return to Solid Edge, open the subassembly and check it out.
12. Perform a right-click on the subassembly in the model tree and select **Occurrence Properties**. Choose an underlying component of the subassembly and change the dropdown value of *User Defined Properties* from **No** to **Yes** and enter a value for the quantity. Press **OK**.
→ Quantity changed and saved.
13. Close the dialog and check in the subassembly.
→ Assembly is checked in. BOM is created.
14. Return to SOLIDWORKS PDM and refresh view **Solid Edge BOM**
→
BOM is correct.

- 15.** Check the **Solid Edge BOM** for the top assembly.



Changes of quantity for subassembly components are not available.