

User Guide

SOLIDWORKS Electrical integration
for
SOLIDWORKS PDM

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1 Introduction

1.1 Purpose and target audience

This document explains the functions and usage of the integration between Dassault Systèmes SOLIDWORKS Electrical and Dassault Systèmes SOLIDWORKS PDM Professional. The document is written for end users who work with the integration daily.

How to read this document

This document is structured chronologically and you should read it in the order of the chapters described. If you skip chapters, you will miss important information.

Where appropriate, cross-references to other chapters are listed. To quickly return to where you came from after clicking such a link, click the **Back** button in your PDF viewer. Try it right now with [this link](#)!

Notes used



This note highlights additional information about the current content.










This note highlights important instructions.






1.2 Functional overview

The integration connects SOLIDWORKS Electrical with SOLIDWORKS PDM and allows managing electrical engineering information in a central place.

Functions in the SOLIDWORKS PDM menu

The installer creates the integration menu **SOLIDWORKS PDM** in the SOLIDWORKS Electrical ribbon bar.

Function	Title	Description
	Open	Opens a SOLIDWORKS Electrical project from the local SOLIDWORKS PDM vault.
	Check Out	Reserves SOLIDWORKS Electrical project file(s) in SOLIDWORKS PDM for the current user.
	Undo Check Out	Removes the reservation for the SOLIDWORKS Electrical project file(s) and the current user in SOLIDWORKS PDM.
	Check In	Stores the SOLIDWORKS Electrical project file(s) in the local SOLIDWORKS PDM vault.
	Get Latest Version	Updates the current SOLIDWORKS Electrical project with the latest version stored in SOLIDWORKS PDM.
	Get Version	Updates the current SOLIDWORKS Electrical project with a specific version stored in SOLIDWORKS PDM.
	Search	Opens a search dialog to search for objects in the local SOLIDWORKS PDM vault.

Function	Title	Description
	Show	Opens the location of the SOLIDWORKS Electrical project in the local SOLIDWORKS PDM vault.
	Show Card	Opens the data card for the SOLIDWORKS Electrical project folder and container.
	Change State	Pushes the SOLIDWORKS Electrical project through a workflow in SOLIDWORKS PDM.
	Synchronize Libraries	Creates new virtual documents for SOLIDWORKS Electrical manufacturer parts and cables in SOLIDWORKS PDM and synchronizes the attributes in both systems.
	Select Vault	Opens a dialog to select a SOLIDWORKS PDM vault to be used in the integration.

1.3 Starting the integration

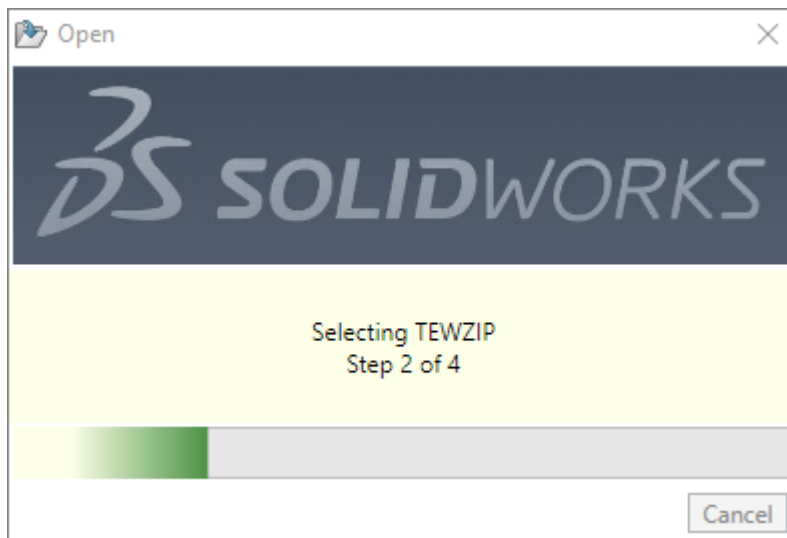
The integration operations can be started from the SOLIDWORKS Electrical ribbon bar. Some operations are also accessible from the pop-up menu in the SOLIDWORKS PDM vault view.

Specify the vault

If you have several SOLIDWORKS PDM vaults configured and want to start working with the integration, select a vault to use first (see [Select Vault](#) (p. 22)).

Start the integration

The progress dialog appears when the integration starts. The dialog shows the name of the integration module, the currently executed action and the progress.



Clicking the button **Cancel** stops the operation. If the current step can not be stopped right now, the button shows **Waiting** instead and the process is stopped when possible.



Integration dialogs and messages are sometimes hidden by a SOLIDWORKS Electrical window. If the integration does not respond, check for open dialogs in the background.

1.4 About data storage in SOLIDWORKS PDM

The integration stores the SOLIDWORKS Electrical project with the data for manufacturing, purchasing and assembly of electrical designs in SOLIDWORKS PDM.

The integration creates a folder for the project in the local SOLIDWORKS PDM vault. The name of the folder can be configured, see **SSA Tool > File Structure > Folders > SOLIDWORKS Electrical Project > Name Rule**. Properties of the project are synchronized with the SOLIDWORKS PDM folder attributes. The project folder contains the project container file (`.tewzip`), and extracted files which are stored in subfolders with configurable names.

All extracted files are linked as SOLIDWORKS PDM custom references to the project container file.

The local SOLIDWORKS PDM vault can store the following information:

Project container files

Project containers contain a complete snapshot of the project files and database. Project containers have the extension `.tewzip`.

Drawing sets in DWG or DXF format

Drawing sheets in an electrical project can be exported to the DWG or DXF format. This format can be used for sharing schematic content with all collaborators who do not have access to SOLIDWORKS Electrical.

Intelligent PDF files

An intelligent PDF file contains all reports and drawings in the project. Additionally there are component and document tree bookmarks that reflect the SOLIDWORKS Electrical data structure. The integration supports two PDF creation modes:

- One PDF for all reports and drawings
- One PDF for each document book

The PDF creation modes can be configured, see **SSA Tool > General Settings > Project export settings > PDF per book**.

Assemblies and BOMs

For each location found in the project, a virtual document with extension `.swe.cvd` is created in the project folder. Assemblies in SOLIDWORKS PDM contain BOMs as set of links to manufacturer parts and cables. Only such manufacturer parts and cables that are synchronized with the function **Synchronize Libraries** in SOLIDWORKS PDM, can be added to the BOM in the SOLIDWORKS PDM assembly.

Other project-related files

An electrical project can contain a variety of supplementary files such as design documents, reports and screenshots. Extracts of such files can also be configured in the **SSA Tool**.

1.5 Manufacturer parts and cable references

Manufacturer parts and cable references are stored in the local SOLIDWORKS PDM vault as virtual documents.

Properties of manufacturer parts and cable references are synchronized with the document attributes in SOLIDWORKS PDM. The mapping of properties and attributes can be configured, see **SSA Tool > File Structure > File > Purchased Part / Purchased Cable**.

The files with the manufacturer parts and cable references are located in the library folder in the SOLIDWORKS PDM vault. The name of the library folder can be configured, see **SSA Tool > General Settings > Library Synchronization > Library Folder**.

1.6 Logs and support

Integration can create log file during execution.

If logging is turned on, during the execution integration produce log file. By default logging is turned off. To change the logging settings, please edit `%XPlmRootDir%\xml\PDMPConnector.xml` file. This file will be created after the installation.

Configuration parameters

Parameter name	Description
EnablePDMPLogging	Turn on/off logging. Possible values: true or false .
PDMPLogFile	Full path to the log file location (for example <code>C:\pdm_integration.log</code>).
PDMPLogLevel	Integer value, specifying how detailed the logging should be. Can be any integer from 1 to 100, where 1 - only critical messages will be logged, 100 - all messages will be logged. Support will always ask for 'Debug' log, please set this value to 100 in this case.

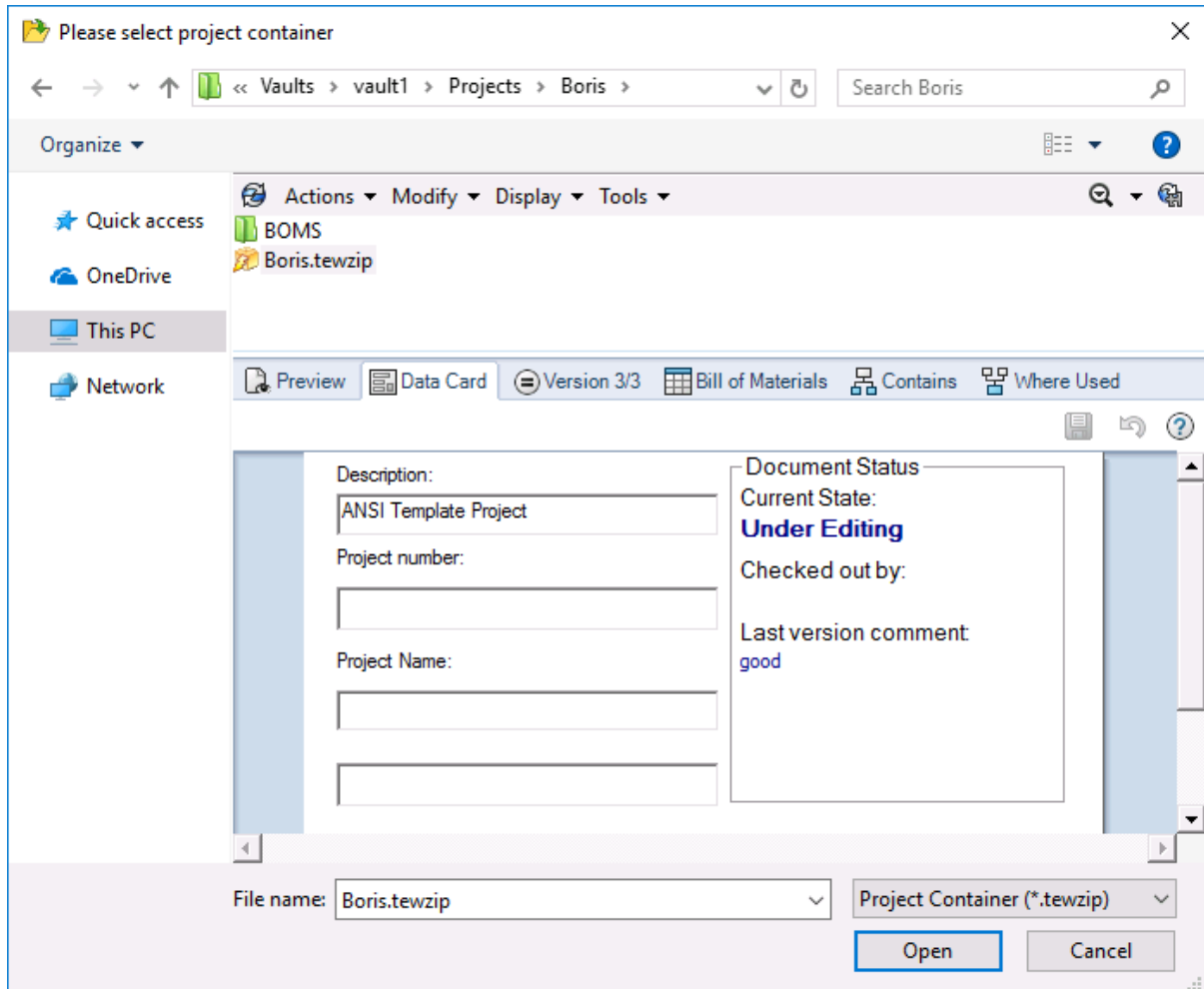


Log file is never deleted or cleaned up by the integration and can grow quite big if logging will be turned on for a long time. Best practice would be to turn it on only for creation of 'Debug' log when contacting support.

2 Usage

2.1 Open

Opens a SOLIDWORKS Electrical project from the local SOLIDWORKS PDM vault.



Navigate to the project folder, select a project container with ending **.tewzip** and click **Open**.

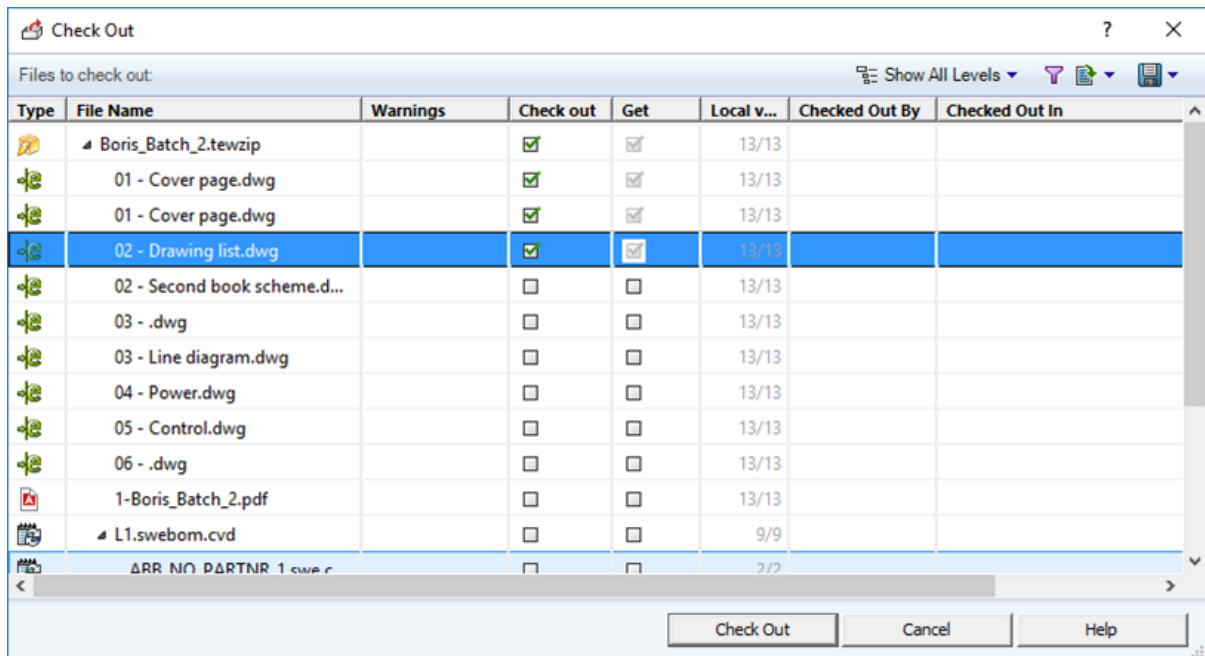
If the project already exists in the project directory, it is updated with the latest version in SOLIDWORKS PDM.



The project is overwritten as a result of this operation. Observe the warning message.

2.2 Check out

Reserves SOLIDWORKS Electrical project file(s) in SOLIDWORKS PDM for the current user.



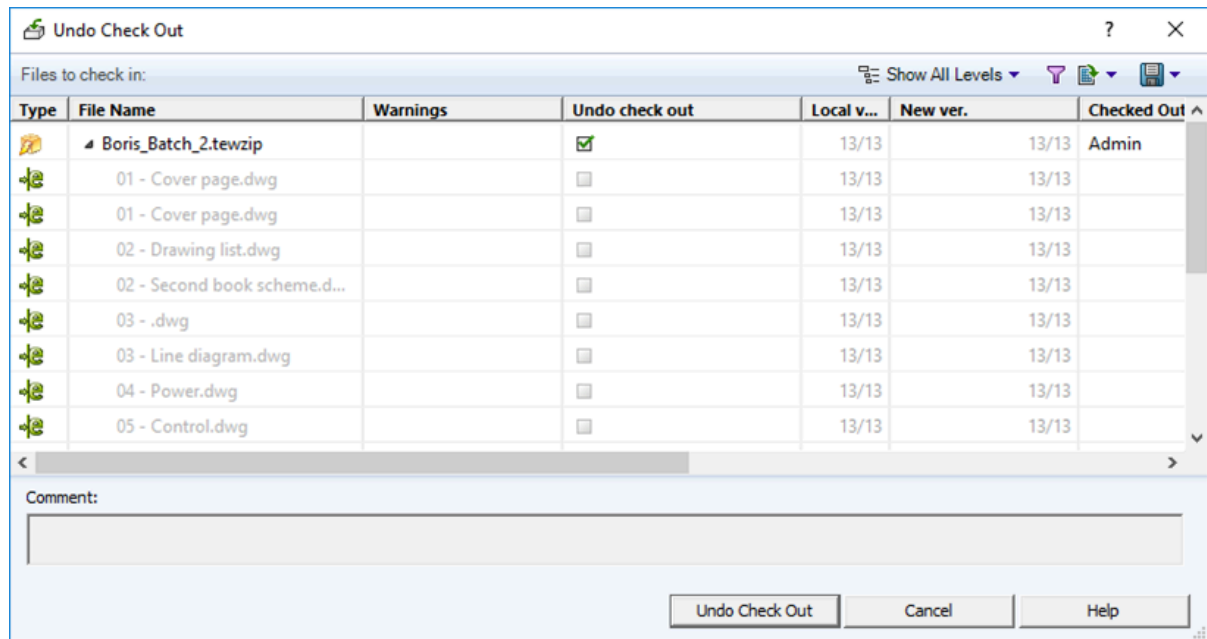
Select the project files to reserve for the current user and click **Check Out**.



The project container must not be checked out in SOLIDWORKS PDM.

2.3 Undo Check Out

Removes the reservation for the SOLIDWORKS Electrical project file(s) and the current user in SOLIDWORKS PDM.



Select the project files from which the reservation for the current user should be removed and click **Undo Check Out**.



The project container must be checked out by the current user in SOLIDWORKS PDM.

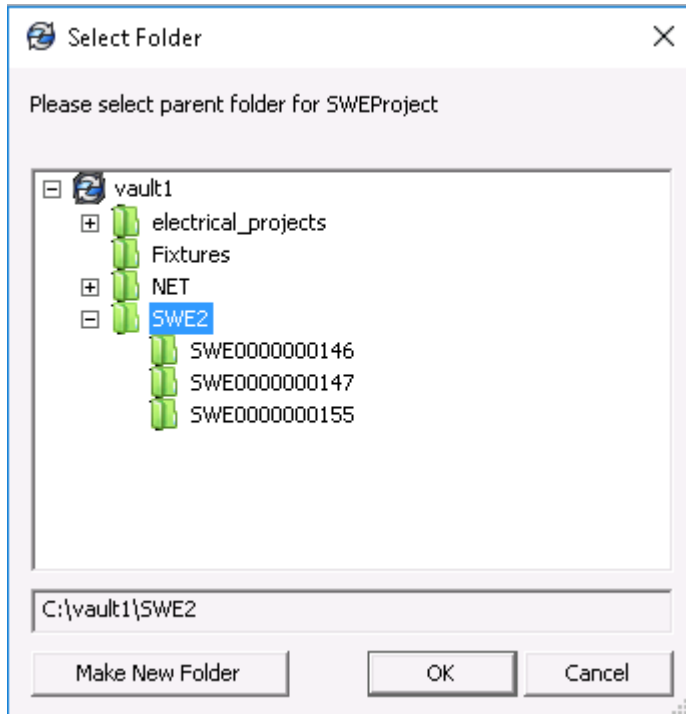
Undo Check Out updates the project with the latest version in SOLIDWORKS PDM.



The project is overwritten as a result of this operation. Observe the warning message.

2.4 Check In

Stores the SOLIDWORKS Electrical project file(s) in the local SOLIDWORKS PDM vault.

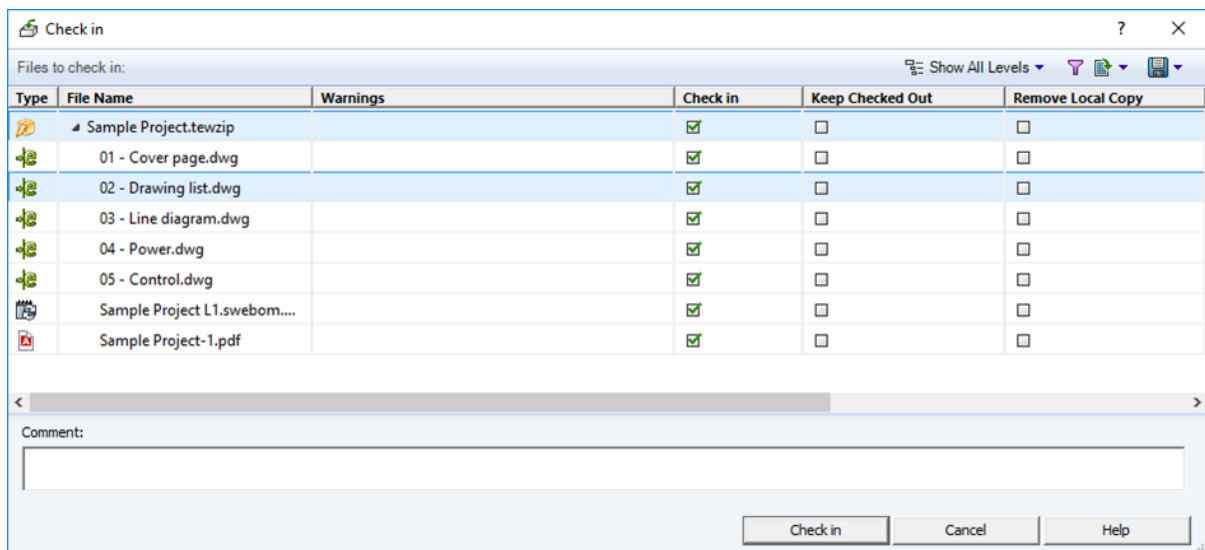


Select the parent folder in which the project data is created and click **OK**.

If the setting **Default Root Folder** is defined in the **SSA Tool**, this folder is pre-selected in this dialog.



The folder selection dialog may not appear if the **Always use default root folder** option in the configuration is set. In this case, the root folder defined by the integration administrator will be used.



The first step of the process opens the dialog **Add Files**. Select the project file(s) to add to the local SOLIDWORKS PDM vault and click **Add**.

The second step opens the dialog **Check In Files**. Select the project file(s) to check in into SOLIDWORKS PDM and click **Check In**.

Enable the option **Remove Local Copy** for the project container file (*.tewzip) to delete the project from SOLIDWORKS Electrical. After that, the project will remain in the vault as an archive only. To start working with the project again, use [Open](#) (p. 10).



In the case when option **Remove project from SW Electrical after check in** is set in the Administration tool, project will be deleted in any case.

Enable the option **Keep Checked Out** to keep the project file(s) checked out by the current user.

If the project contains parts which are not yet synchronized with SOLIDWORKS PDM, a warning message is displayed showing a list of missing parts and their marks.



The list of missing parts is stored in the local folder of the current user and is used during the next execution of **Synchronize Libraries**.

2.4.1 BOM handling

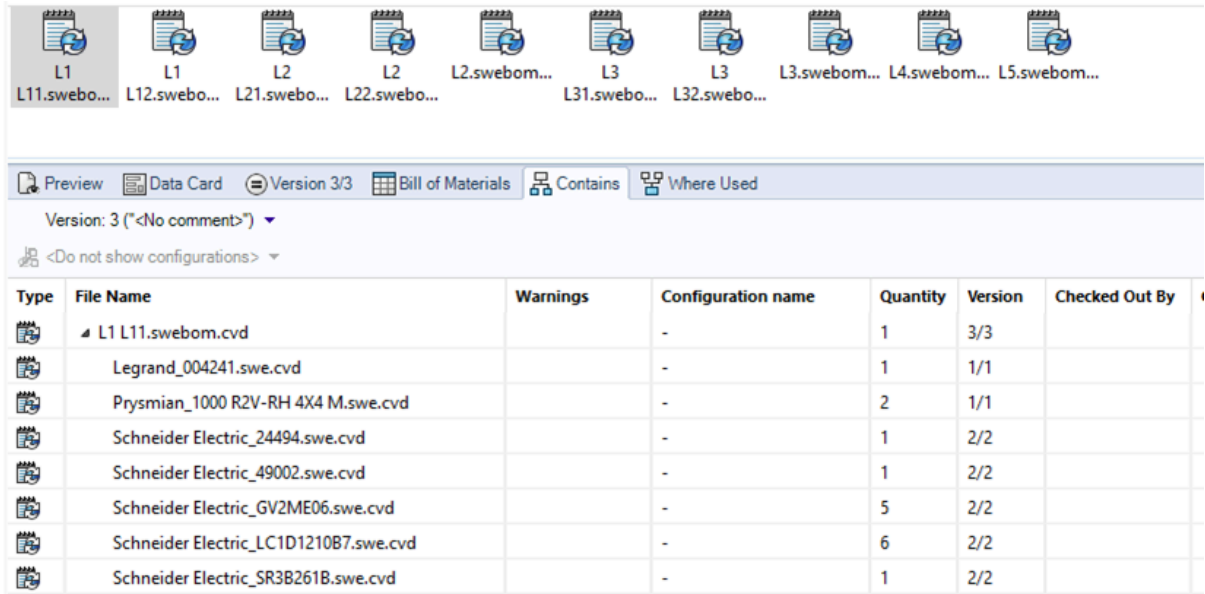
During **Check In**, the integration creates BOM header documents in SOLIDWORKS PDM and links library parts (also represented by virtual documents) to this BOM header document.

There are three possible scenarios of how BOM headers are organized:

- **By Location** - each project location will have a corresponding BOM header with a list of parts and cables used.
- **By Component Type** - three BOM headers will be created: for parts, cables and harnesses.
- **One Global** - one BOM header, containing all parts and cables will be created.


BOM headers are represented in SOLIDWORKS PDM as virtual documents with the extension **.swebom.cvd**.

To find which parts are used in a BOM header, click on the file and select the tab **Contains** or **Bills Of Materials**.



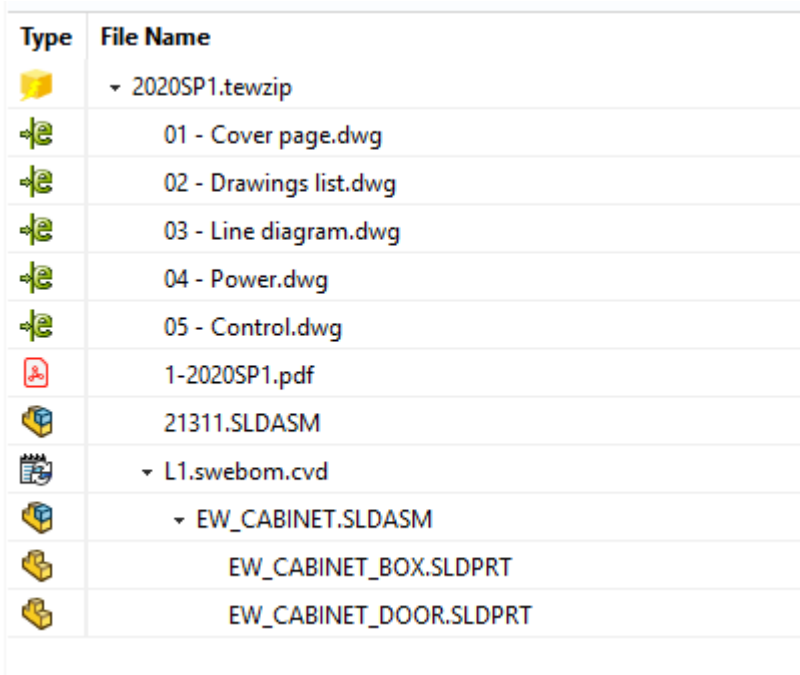
The screenshot shows the SolidWorks PDM interface. At the top, there is a toolbar with icons for L1, L2, L3, and L4 levels, and a search bar. Below the toolbar, there is a table with columns: Type, File Name, Warnings, Configuration name, Quantity, Version, and Checked Out By. The table lists several files, including L1 L11.swebom.cvd, Legrand_004241.swe.cvd, Pysmian_1000 R2V-RH 4X4 M.swe.cvd, and Schneider Electric files. The interface also shows a 'Version: 3' dropdown and a 'Where Used' button.

Type	File Name	Warnings	Configuration name	Quantity	Version	Checked Out By
	L1 L11.swebom.cvd		-	1	3/3	
	Legrand_004241.swe.cvd		-	1	1/1	
	Pysmian_1000 R2V-RH 4X4 M.swe.cvd		-	2	1/1	
	Schneider Electric_24494.swe.cvd		-	1	2/2	
	Schneider Electric_49002.swe.cvd		-	1	2/2	
	Schneider Electric_GV2ME06.swe.cvd		-	5	2/2	
	Schneider Electric_LC1D1210B7.swe.cvd		-	6	2/2	
	Schneider Electric_SR3B261B.swe.cvd		-	1	2/2	

 Before starting **Check In**, it is recommended to run [Synchronize Libraries](#) (p. 19) first. After that select **Process > Management > Update Data** and follow the instructions. This ensures that part properties in the project are updated with the current attribute values from SOLIDWORKS PDM.

2.4.2 3D handling

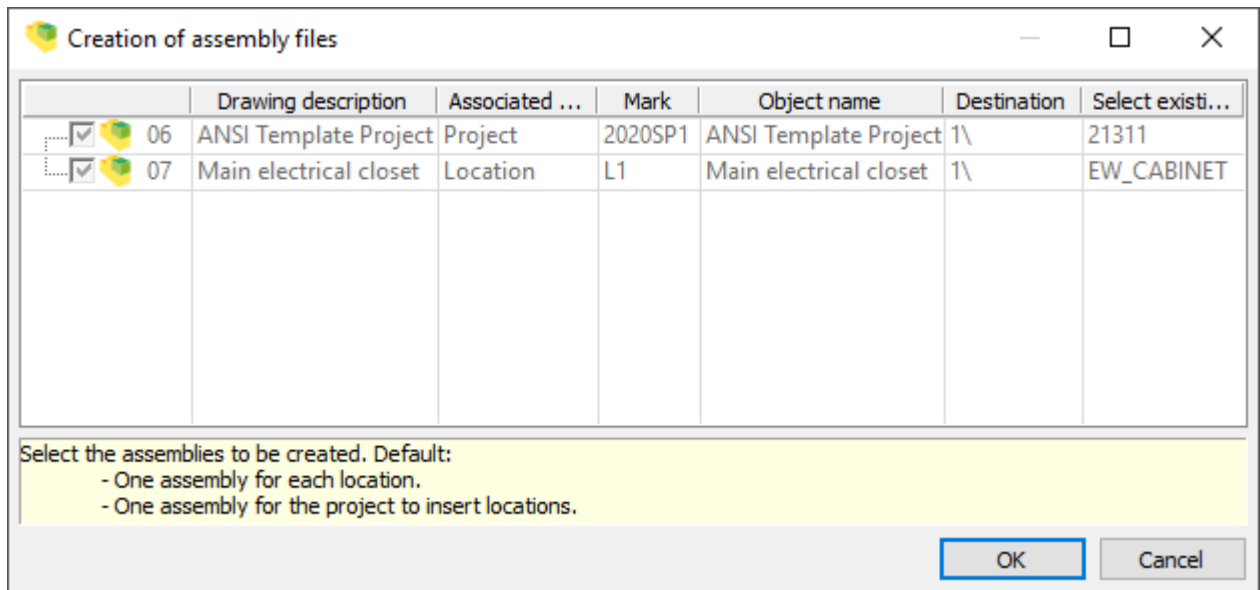
During **Check In**, the integration creates links in SOLIDWORKS PDM between the documents representing the project and locations, and corresponding 3D assemblies.




The screenshot shows the SolidWorks PDM interface. At the top, there is a toolbar with icons for L1, L2, L3, and L4 levels, and a search bar. Below the toolbar, there is a table with columns: Type, File Name, Warnings, Configuration name, Quantity, Version, and Checked Out By. The table lists several files, including 2020SP1.tewzip, 01 - Cover page.dwg, 02 - Drawings list.dwg, 03 - Line diagram.dwg, 04 - Power.dwg, 05 - Control.dwg, 1-2020SP1.pdf, 21311.SLDASM, L1.swebom.cvd, EW_CABINET.SLDASM, EW_CABINET_BOX.SLDPRT, and EW_CABINET_DOOR.SLDPRT. The interface also shows a 'Version: 3' dropdown and a 'Where Used' button.


Type	File Name	Warnings	Configuration name	Quantity	Version	Checked Out By
	2020SP1.tewzip		-	1	3/3	
	01 - Cover page.dwg		-	1	1/1	
	02 - Drawings list.dwg		-	2	1/1	
	03 - Line diagram.dwg		-	1	2/2	
	04 - Power.dwg		-	1	2/2	
	05 - Control.dwg		-	5	2/2	
	1-2020SP1.pdf		-	6	2/2	
	21311.SLDASM		-	1	2/2	
	L1.swebom.cvd		-	1	2/2	
	EW_CABINET.SLDASM		-	1	2/2	
	EW_CABINET_BOX.SLDPRT		-	1	2/2	
	EW_CABINET_DOOR.SLDPRT		-	1	2/2	

Select **Process > SOLIDWORKS assembly > Select existing file** and assign the project and locations in the dialog.



 3D assemblies must reside in the SOLIDWORKS PDM vault. Links to assemblies located outside of the vault are ignored.

During **Check In**, the link between the `tewzip` file and the 3D-assembly file assigned to the project is set. If locations also have assigned assemblies, they are also linked.

 Locations are processed only if the BOM mode **By Location** is used.


Related links

[3D parts handling](#) (p. 20)

2.5 Get Latest Version

Updates the current SOLIDWORKS Electrical project with the latest version stored in SOLIDWORKS PDM.

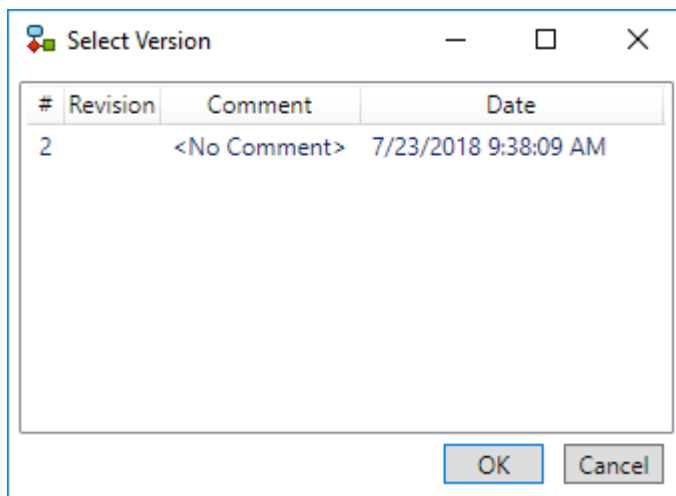
Select the project file before executing the function.


 The project is overwritten as a result of this operation. Observe warning message.

2.6 Get Version

Updates the current SOLIDWORKS Electrical project with a specific version stored in SOLIDWORKS PDM.

Select a specific version to download and open.



 The project is overwritten as a result of this operation. Pay attention to the warning message shown.

2.7 Search

Opens a search dialog to search for objects in the local SOLIDWORKS PDM vault.

Access the SOLIDWORKS PDM search functionality quickly from within SOLIDWORKS Electrical.

2.8 Show

Opens the location of the SOLIDWORKS Electrical project in the local SOLIDWORKS PDM vault.

Make sure the project is open in SOLIDWORKS Electrical, checked in into SOLIDWORKS PDM and set to **Current**.

If the project container file is already available in SOLIDWORKS PDM, **Show** opens the current project folder and selects the project container file.

If the project is stored in SOLIDWORKS PDM but the project container file hasn't been checked in yet, **Show** opens the root folder and selects the current project folder.

2.9 Show Card

Opens the data card for the SOLIDWORKS Electrical project folder and container.

The **Data Card** dialog contains three sections:

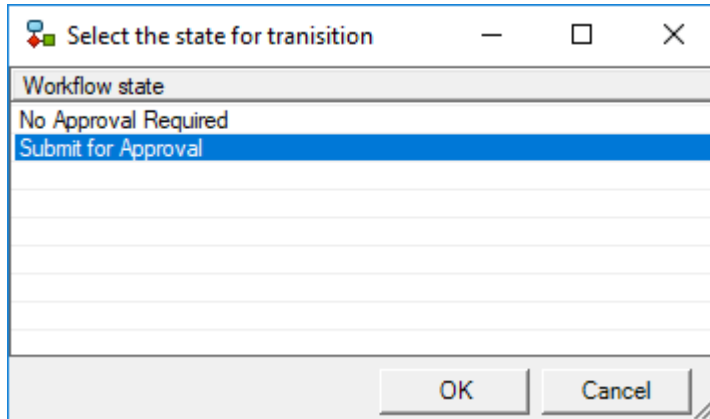
- A header section with information about the project container state
- A tab with the **Project Folder** data card
- A tab with the **Project Container** data card

- **SWE Version:** Shows the SOLIDWORKS PDM version of the project container opened in SOLIDWORKS Electrical. It may differ from the latest version in SOLIDWORKS PDM.
- **Local Vault Version:** Shows the version of the project container in the local SOLIDWORKS PDM vault and the current version in the SOLIDWORKS PDMPDM vault on the server.
- **Project Folder:** The data card can be edited by clicking the tab **Edit Values**.
- **Project Container:** If the project container is checked out by the current user, it is also possible to edit this data card.
- **OK:** Saves changes to SOLIDWORKS PDM.
- **Cancel:** Closes the dialog without saving.

Make sure the project is open in SOLIDWORKS Electrical, checked in into SOLIDWORKS PDM and set to **Current**.

2.10 Change State

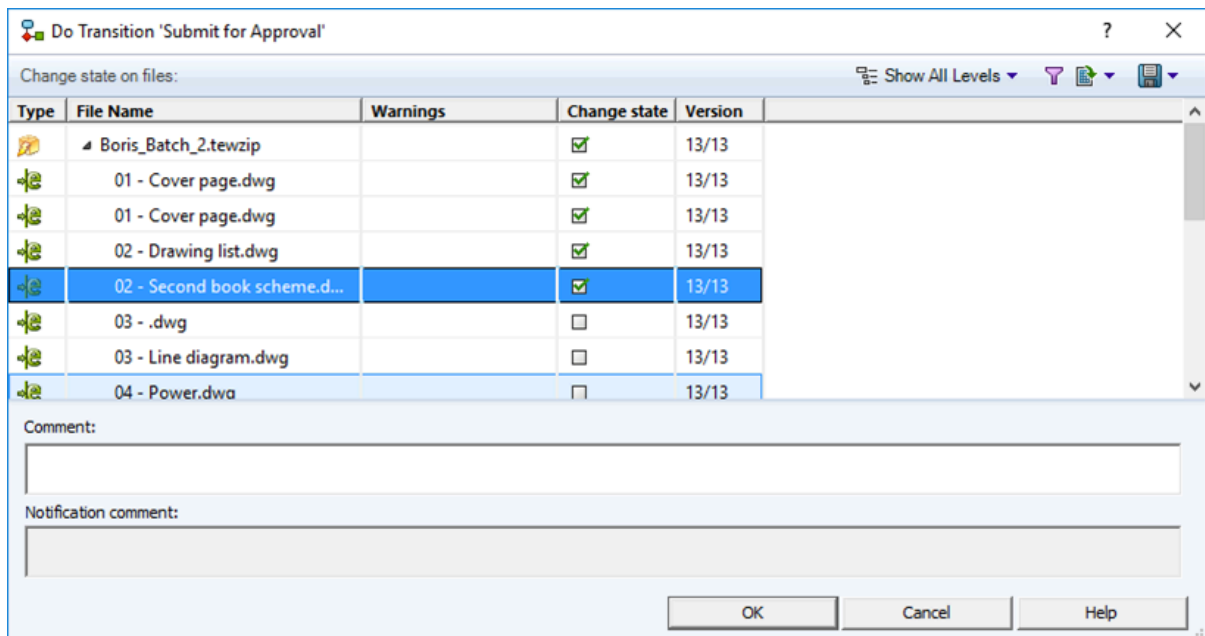
Pushes the SOLIDWORKS Electrical project through a workflow in SOLIDWORKS PDM.



Make sure the project container is checked in.

Select a workflow state first. Only allowed states are shown in the list.

The transition dialog appears in which the project files suitable for transition are shown.



Enable the option **Change State** for the files to be changed.

2.11 Synchronize Libraries

Creates new virtual documents for SOLIDWORKS Electrical manufacturer parts and cables in SOLIDWORKS PDM and synchronizes the attributes in both systems.

Manufacturer parts and cables without a corresponding virtual document in SOLIDWORKS PDM are considered missing during **Check In** and are not included in the BOM.

The library is stored in the configured folder in SOLIDWORKS PDM, see **SSA Tool > General Settings > Library Folder**.

Synchronization is performed in batch mode and can take some time depending on the library size.

Library synchronization can be executed without user interaction, to be run overnight. To do this start `connector_runner.exe` from installation directory.



After synchronization, the changes to the library have to be updated in the project, otherwise they will not appear in the schematics. Select **Process > Management > Update Data** and follow the instructions.

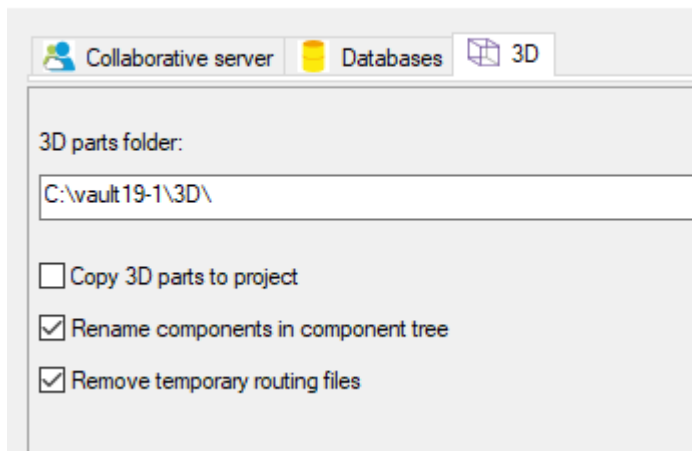
2.11.1 3D parts handling

During **Synchronize Libraries**, the integration creates links between the virtual documents representing the electrical parts and their 3D models.

Prerequisites:

- 3D parts storage in SOLIDWORKS Electrical is pointing to a location inside the vault (**Tools > Application Settings > 3D > 3D parts folder**) and the 3D representations of parts are placed within this location.
- The setting **Copy 3D parts to project** is switched off (**Tools > Application Settings > 3D > Copy 3D parts to project**).

Application settings



As a result of **Synchronize Libraries**, documents representing the electrical parts in the SOLIDWORKS Electrical library which are assigned to 3D models, will be linked in the vault to the corresponding 3D files.

SOLIDWORKS Electrical view:

Manufacturer part properties AMP : 3-350819-2

Properties User data Circuits, terminals

Data sheet:
Mark root:
Description (English): RECEPTACLE ASSY,ULTRA-FAST, .250 SERIES,16-14 AWG
Commercial reference (English):

Supplier
Supplier name:
Stock number:

Information
Illustration
Line diagram: 3-350819-2
Scheme: 3-350819-2
3D part: 3-350819-2.SLDPRT
2D footprint: 3-350819-2
Connection label:
Printed circuit board file:

SOLIDWORKS PDM view:

Version History View File New Folder

Actions Modify Display Tools

Name	Checked Out By	Size	File Type	State
AMP_3-350819-2.swe.cvd		0 bytes	Virtual docum...	Under Editing
Hager_KZ004.swe.cvd		0 bytes	Virtual docum...	Under Editing

Preview Data Card Version 2/2 Bill of Materials Contains Where Used

Version: 2 ("<No comment>")
<Do not show configurations>

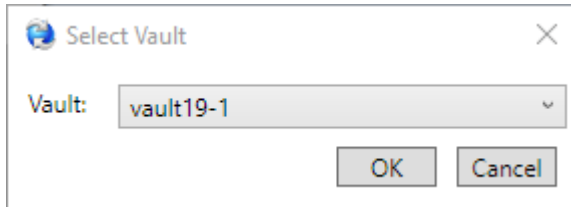
Type	File Name	Found In
▼	AMP_3-350819-2.swe.cvd	C:\vault19-1\SWE_Library\Connectors
3D	3-350819-2.sldprt	C:\vault19-1\3D

Related links

[3D handling](#) (p. 15)

2.12 Select Vault

Opens a dialog to select a SOLIDWORKS PDM vault to be used in the integration



If you have only one SOLIDWORKS PDM vault configured, you can skip the **Select Vault** operation and the vault will be selected automatically.

2.13 Special notes

Additional information when duplicating a project.

Copy project in SOLIDWORKS Electrical

In SOLIDWORKS Electrical, it is possible to copy a project. If the original project was stored in SOLIDWORKS PDM via **Check In**, the copied version will also create a new SOLIDWORKS PDM folder upon **Check In**.

Copy project in SOLIDWORKS PDM

If a project was copied in SOLIDWORKS PDM, the integration will detect this and will unlink it from the old location upon **Open**.

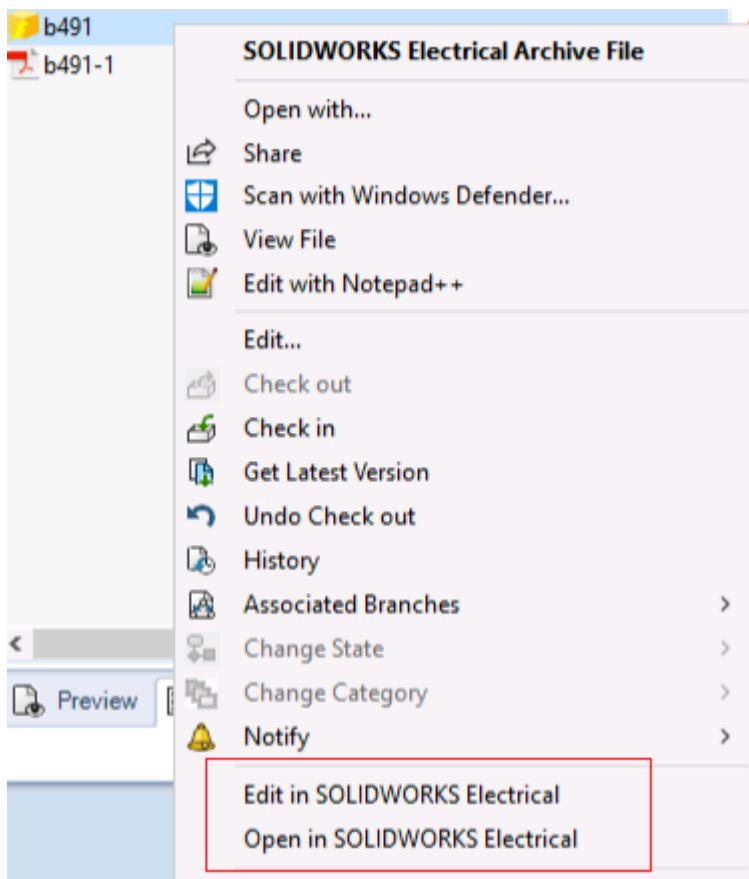


The suggested way to copy a project in SOLIDWORKS PDM is to copy the project container file (**.tewzip**) only and to leave the other project files such as drawings, PDF files, etc. untouched. Having such files in the new, copied project folder will cause naming conflicts upon **Check In**.

2.14 Using vault pop-up menu

You can perform some of the integration operations directly from the SOLIDWORKS PDM view.

In the vault view, right-mouse click on the project container archive (***.tewzip**) and select an operation from the pop-up menu:



Description of operations

Table 1:

Operation name	Description
Edit in SOLIDWORKS Electrical	Opens the selected project archive in SOLIDWORKS Electrical (see Open (p. 10)) and checks out the project container.
Open in SOLIDWORKS Electrical	Works the same as Edit , but doesn't check out the project container.



The integration and SOLIDWORKS Electrical must be installed to perform these operations.



Sometimes the progress dialog of an operation does not appear on top of other windows. Check the taskbar to bring up the window.

3 Troubleshooting

3.1 Integration does not respond

If the integration does not respond, check for open dialogs in the background. Integration dialogs and messages are sometimes hidden by a SOLIDWORKS Electrical window.

3.2 Function is not working

If a function from within the SOLIDWORKS PDM toolbar is not working, restart SOLIDWORKS Electrical and try again.

If this does not help, set the logging level to **All**, retry the action and retrieve the log files from the configured destination directory. Then zip the log files and send them to the XPLM support for analysis.

3.3 Check-in operation fails when exporting data

If the check in process in the step "Exporting Data from SWE" fails, make sure the PDFCreator utility is installed. Without the utility, SOLIDWORKS Electrical cannot export project data correctly, so that the integration cannot add files to the vault.