

Administration Guide

SOLIDWORKS Electrical integration
for
SOLIDWORKS PDM

Valid for product version: 2024 SP2

Published: 21.05.2024 | Build: 546 | Revision: 9fdea1228

Legal information

© 1995-2024, Dassault Systèmes SolidWorks Corporation, a Dassault Systèmes SE company, 175 Wyman Street, Waltham, Mass. 02451 USA. All Rights Reserved.

The information and the software discussed in this document are subject to change without notice and are not commitments by Dassault Systèmes SolidWorks Corporation (DS SolidWorks).

No material may be reproduced or transmitted in any form or by any means, electronically or manually, for any purpose without the express written permission of DS SolidWorks.

The software discussed in this document is furnished under a license and may be used or copied only in accordance with the terms of the license. All warranties given by DS SolidWorks as to the software and documentation are set forth in the license agreement, and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of any terms, including warranties, in the license agreement.

Patent notices

SOLIDWORKS® 3D mechanical CAD and/or Simulation software is protected by U.S. Patents 6,219,049; 6,219,055; 6,611,725; 6,844,877; 6,898,560; 6,906,712; 7,079,990; 7,477,262; 7,558,705; 7,571,079; 7,590,497; 7,643,027; 7,672,822; 7,688,318; 7,694,238; 7,853,940; 8,305,376; 8,581,902; 8,817,028; 8,910,078; 9,129,083; 9,153,072 and foreign patents, (for example, EP 1,116,190B1 and JP 3,517,643).

eDrawings® software is protected by U.S. Patent 7,184,044; U.S. Patent 7,502,027; and Canadian Patent 2,318,706.

U.S. and foreign patents pending.

Trademarks and product names for SOLIDWORKS products and services

SOLIDWORKS, 3D ContentCentral, 3D PartStream.NET, eDrawings, and the eDrawings logo are registered trademarks and FeatureManager is a jointly owned registered trademark of DS SolidWorks.

CircuitWorks, FloXpress, PhotoView360, and TolAnalyst are trademarks of DS SolidWorks.

FeatureWorks is a registered trademark of Geometric Ltd.

SOLIDWORKS 2018, SOLIDWORKS Standard, SOLIDWORKS Professional, SOLIDWORKS Premium, SOLIDWORKS PDM Professional, SOLIDWORKS PDM Standard, SOLIDWORKS Workgroup PDM, SOLIDWORKS Simulation, SOLIDWORKS Flow Simulation, eDrawings, eDrawings Professional, SOLIDWORKS Sustainability, SOLIDWORKS Plastics, SOLIDWORKS Electrical, SOLIDWORKS Composer, and SOLIDWORKS MBD are product names of DS SolidWorks.

Other brand or product names are trademarks or registered trademarks of their respective holders.

COMMERCIAL COMPUTERS SOFTWARE - PROPRIETARY

The Software is a “commercial item” as that term is defined at 48 C.F.R. 2.101 (OCT 1995), consisting of “commercial computer software” and “commercial software documentation” as such terms are used in 48 C.F.R. 12.212 (SEPT 1995) and is provided to the U.S. Government 14 (a) for acquisition by or on behalf of civilian agencies, consistent with the policy set forth in 48 C.F.R. 12.212; or (b) for acquisition by or on behalf of units of the Department of Defense, consistent with the policies set forth in 48 C.F.R. 227.7202-1 (JUN 1995) and 227.7202-4 (JUN 1995).

In the event that you receive a request from any agency of the U.S. Government to provide Software with rights beyond those set forth above, you will notify DS SolidWorks of the scope of the request and DS SolidWorks will have five (5) business days to, in its sole discretion, accept or reject such request. Contractor/Manufacturer: Dassault Systèmes SolidWorks Corporation, 175 Wyman Street, Waltham, Massachusetts 02451 USA.

Copyright notices for SOLIDWORKS PDM Professional product

Outside In ®Viewer Technology, © 1992-2012 Oracle

©2011, Microsoft Corporation. All rights reserved.

Table of Contents

Legal information.....	ii
1 Introduction.....	6
1.1 Purpose and target audience.....	6
2 Installation.....	7
2.1 System requirements.....	7
2.2 Installing integration.....	7
2.3 Importing integration add-in.....	8
2.4 Defining vault and library folder.....	9
3 Configuration.....	10
3.1 Using the SSA Tool.....	10
3.1.1 Overview.....	10
3.1.2 General Settings.....	10
3.1.3 BOM Settings.....	12
3.1.4 File Structure.....	12
3.1.4.1 Add/Remove Electrical Project Data Files.....	15
3.1.4.2 Export of arbitrary files.....	15
3.1.4.3 Rule Expression Editor.....	16
3.1.4.4 Edit Mapping Rule.....	16
3.1.5 Import/Export Configuration.....	17
3.2 Attribute mapping example.....	17
3.3 Special cases of attribute mapping.....	21
4 Update.....	23
4.1 Modifying installation.....	23
4.2 Repairing installation.....	24
4.3 Updating installation.....	25
5 Uninstallation.....	27
5.1 Removing installation.....	27
6 Troubleshooting.....	28
6.1 Logs and error messages.....	28

7	References.....	30
7.1	Silent-mode installation.....	30
7.2	Working with overlay packages.....	34

1 Introduction

1.1 Purpose and target audience

This document explains how to install and configure the integration between Dassault Systèmes SOLIDWORKS Electrical and Dassault Systèmes SOLIDWORKS PDM Professional. The document is written for system administrators who manage the integration.

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.

2 Installation

2.1 System requirements

Before installing the integration, check that the following requirements are met.

Operating system

- Windows 10/11 (x64)
- Local administrator rights

Supported CAD and PDM versions

- Dassault Systèmes SOLIDWORKS Electrical 2024 SP0 or higher
- Dassault Systèmes SOLIDWORKS PDM Professional 2024 SP0 or higher

The Dassault Systèmes SOLIDWORKS PDM Professional vault is connected and a local view has been created.

2.2 Installing integration

Make sure that all installation media and licenses are available, and then start the installation.

About this task

With this unified installer, you can install Dassault products in the same installation directory and use them in parallel. You can update all components individually without affecting the functionality of other installed products.





The following procedure applies to a new installation. To update an existing installation, see [Updating installation](#) (p. 25) for more information.



You can install, modify, repair, or uninstall also in silent-mode. See [Silent-mode installation](#) (p. 30) for more information.

Procedure

1. Copy the installer archive *.7z.exe to the client computer running SOLIDWORKS Electrical.
2. Close all open applications related to the integration.
3. Extract the archive and start Setup-*.exe with administrator rights.
4. Install any Visual C++ runtimes that you are prompted for.
→ The runtimes are installed, and the installation wizard appears.
5. Click **Next** to start the wizard.
→ The step *License agreement for end-users* appears.
6. Accept the license agreement and click **Next**.
→ The step *Installation path* appears.
7. Check the installation directory. It must point to the directory *CAD Integration* in the SOLIDWORKS PDM installation directory, for example *C:\Program Files\SOLIDWORKS Corp\SOLIDWORKS PDM\CAD Integration\CAD Integration*. Click **Next**.

- If another product is already installed, you cannot change the installation directory. The product installed first defines this path for other products.
 - If you have received an overlay package from Dassault, or want to apply your own, enable the option **Apply custom files after installation** and enter the path to the overlay. Overlays contain customized files that overwrite the installed files as the last step. See [Working with overlay packages](#) (p. 34) for more information.
 - To backup the existing installation, enable the option **Backup current installation**. Define the backup scope first. Then click **Browse** and select a backup directory.
-  Always create a backup when you update or change an installation. This makes it easier to compare and merge the files later.
- The step *ECAD components* appears.
8. Select the application(s) to be integrated. If required, select additionally a version or other settings. Click **Next**.
- The step *Ready to install* appears.
9. To start installation, click **Install**.
-  During installation, a progress dialog is shown with detailed information on each MSI currently installed. If there are installation issues, the process stops and you can examine the error in the lower dialog section. If required, click **Save to log** and send the log file to support@xplm.com for further investigation. This new progress dialog is available from installer versions 24.3.3.606 onwards.
- For older installers, a common CMD window with the progress is shown. Do not click into this window or installation cannot proceed. To continue if you clicked in, press **Enter** or **Esc**.
10. To close the wizard after installation, click **Finish**.

Result

Installation is complete. You can find log files for all installed components in the directory `C:\ProgramData\XPLM Solution GmbH\logs`.

2.3 Importing integration add-in

The installer has provided the integration add-in which you must import in SOLIDWORKS PDM.

Procedure

1. Open the SOLIDWORKS PDM administration console.
2. Log in to a vault you want to use.
3. Right-click *Add-ins* and select **New Add-in**.

→ The SOLIDWORKS PDM add-in directory appears. If not, navigate to it, for example `C:\Program Files\SOLIDWORKS Corp\SOLIDWORKS PDM\CAD Integration\SOLIDWORKS PDM Electrical Addin`.
4. Select the file `SOLIDWORKS Electrical Integration Add-In.dll` and click **Open**.

→ The properties window of the add-in appears.

5. Go to **Files** and click **Add Files**.
→ The SOLIDWORKS PDM add-in directory is appears again.
6. Change the file filter to show all files.
7. Select all files and click **Open**.
→ All required files are added.
8. Click **OK**.
9. In the warning dialog, click **OK**.

Result

The add-in **SOLIDWORKS Electrical Integration Add-In** is imported below the *Add-ins* node.

What to do next

To launch the integration administration tool, right-click **Add-Ins > SOLIDWORKS Electrical Integration Add-In** and select **SOLIDWORKS Electrical Integration Administration**.

To verify if the help function works, click **Help**.

2.4 Defining vault and library folder

Procedure

Define vault:

1. In SOLIDWORKS Electrical, select **SOLIDWORKS PDM > Administration > Select Vault** to open the dialog *Select Vault*.
2. Select a local SOLIDWORKS PDM vault from the list.
3. Click **OK** to commit and close the dialog.

Define library folder:

4. Launch the SOLIDWORKS PDM administration tool.
5. Login to a vault.
6. Navigate to **Add-ins > SOLIDWORKS Electrical Integration Add-in**.
7. Right-click and select **SOLIDWORKS Electrical Integration Administration**. Administration Tool will launch.
8. Select desired vault folder in **General Settings > Library synchronization > Library Folder**.

Result

Vault and library folder are configured.

What to do next

Configure naming, sub folder and mapping rules.

3 Configuration

3.1 Using the SSA Tool

3.1.1 Overview

The SSA Tool is an administration tool to modify the SOLIDWORKS Electrical and SOLIDWORKS PDM integration configuration.

The SSA Tool is started from SOLIDWORKS PDM by right-clicking **Vault name** > **Add-ins** > **SOLIDWORKS Electrical Integration Add-in** and selecting **SOLIDWORKS Electrical Integration Administration**.

On start, the SSA Tool automatically loads the configuration and presents it on the screen.



The configuration is stored in the vault database and is active on all computers where the integration with the vault is used.

Description of the user interface

Tabs

- **General Settings:** Defines the configuration for the most important system values.
- **BOM Settings:** Defines the settings for the handling of Bills Of Materials in the integration.
- **File Structure:** Defines the configuration for file and folder names in SOLIDWORKS PDM and the mapping rules for the attributes synchronization.

Buttons

- **OK:** Applies changes to the configuration files and closes the application.
- **Cancel:** Closes the application.
- **Export:** Exports the configuration to the file system
- **Import:** Imports a configuration from a previously exported configuration file (or files in case of migration from previous version of the integration)
- **Help:** Shows the help file.

3.1.2 General Settings

Defines the configuration for the most important system values.

Description of the user interface

Table 1: General

Field name	Description
Always use default root folder	When set, the integration will not ask the user to select a root folder during the Check-In operation. The folder name specified in the setting File Structure > Working Folder > SOLIDWORKS Electrical Project > Default Root Folder will be used. If this value is empty, the root of the vault will be used.

Field name	Description
Remove project from SW Electrical after check in	When set, the integration will always remove the project from SOLIDWORKS Electrical database after check in to PDM. "Remove local file" option on the check in dialog will be ignored in this case.

Table 2: Library synchronization


Field name	Description
Library Folder	Defines the root folder for the library synchronization. All manufacturer part/cable files are created under this root folder. The definition can contain the name of the folder or the full path within a particular vault.
SWE Libraries	<p>Defines names of the SOLIDWORKS Electrical component libraries to be used during the libraries synchronization. A comma separated list of the names is allowed. The "Select Libraries" dialog can be called to simplify the selection process. This dialog is available by pressing the "..." button next to the text field.</p> <p> SOLIDWORKS Electrical must be running to show this dialog.</p>

Table 3: Project Export Settings

Field name	Description
One PDF per book	Defines an option to export one PDF for each project book instead of one PDF for all books.
Use export settings from SOLIDWORKS Electrical	When set, options set by the user in SOLIDWORKS Electrical Project Export PDF, UI will be used when generating the PDF via check-in.



Table 4: Drawing

Field name	Description
Format	Defines the format of the drawing files - DWG or DXF.
Version	Defines the version of DWG or DXF format to be used.

3.1.3 BOM Settings

Defines the configuration for the BOM handling in the integration.

Description of the user interface

Field name	Description
BOM Headers Grouping	<p>Defines how the BOM will be organized in SOLIDWORKS PDM.</p> <p>There are three possible scenarios:</p> <ul style="list-style-type: none"> ■ 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. <p> It is important to specify a naming rule for BOM headers which corresponds to the grouping type. This is required to avoid file naming conflicts in SOLIDWORKS PDM.</p> <p>Example</p> <ul style="list-style-type: none"> ■ If By Location type is used, the naming rule should contain some formula which will produce a unique name for each location. ■ If By Component Type is used, the type-suggested naming rule can use a formula <code>"{COMPONENT.Type}_BOM.swebom.cvd"</code>
Exported BOM objects	<p>Defines which types of components will be included in a BOM</p> <ul style="list-style-type: none"> ■ Parts - include manufactured parts ■ Cables - include cables ■ Harnesses - include parts and cables used in harnesses. <p> Harnesses do not belong to any location, so if you are using 'By Location' BOM grouping, parts and cables which belong only to harness will be ignored.</p>

3.1.4 File Structure

Defines the configuration for folder and file names in the local SOLIDWORKS PDM vault and the mapping rules for the attributes synchronization.

Overview

This dialog is divided in two sections:

- Located to the left, there is the tree panel for **Working Folders** and **Files**. Each node underneath the root node represents one object type involved in the synchronization process.
- Located to the right, there is the property panel with properties corresponding to the selected node. The root nodes for **Working Folder** and **Files** have no properties at all and the property panel remains empty.

Description of the user interface

Tree panel	Property panel	Description
<ul style="list-style-type: none"> ■ Working Folder <ul style="list-style-type: none"> □ SOLIDWORKS Electrical Project 	Name Rule	<p>Defines the folder naming rule. This field is mandatory.</p> <p>Clicking {} opens the Rule Expression Editor (p. 16) that helps to fill the property value.</p> <p>If left blank, root folder of the vault is used.</p>
	Default Root Folder	<p>Defines the default root folder when the project is saved for the first time on Check In.</p> <p>The path is relative to the vault root, for example <code>Projects\SWE</code>.</p>
	Mapping rules	<p>Defines mapping rules consisting of three values:</p> <ul style="list-style-type: none"> ■ Name of the SOLIDWORKS PDM attribute ■ Mapping direction ■ Name of the SOLIDWORKS Electrical attribute <p>Double-clicking the rule, as well as clicking Add or Edit, opens the Edit Mapping Rule (p. 16) dialog where the rule is edited.</p>
<ul style="list-style-type: none"> ■ Files <ul style="list-style-type: none"> □ BOM □ Purchased Part □ Purchased Cable □ Drawing □ TEWZIP □ PDF □ Other Object n 	Name Rule	<p>Defines the folder naming rule. This field is mandatory.</p> <p>Clicking {} opens the Rule Expression Editor (p. 16) that helps to fill the property value.</p>
	Sub Folder	<p>Defines naming rules for file(s) stored in the vault's subfolders.</p> <p>The path is relative to the root folder, for example <code>DWG\{BOOK.Tag-{BOOK.Description}</code>.</p> <p>Clicking {} opens the Rule Expression Editor (p. 16) that helps to fill the property value.</p>
	Mapping rules	<p>Defines mapping rules consisting of three values:</p> <ul style="list-style-type: none"> ■ Name of the SOLIDWORKS PDM attribute ■ Mapping direction ■ Name of the SOLIDWORKS Electrical attribute <p>Double-clicking the rule, as well as clicking Add or Edit, opens the Edit Mapping Rule (p. 16) dialog where the rule is edited.</p>

About name rules

Name rule settings are applied by check-in to SOLIDWORKS PDM objects during the first creation. If the settings are changed later, only newer objects are affected, and existing objects are not changed.

About subfolder rules

The same behavior applies to the subfolder settings except for Purchased Parts and Purchased Cables. For these objects, the settings are applied each time. If the subfolder rule has changed, existing objects in SOLIDWORKS PDM are moved to the new location during **Synchronize Libraries**.

About mapping rules

The integration is shipped with a default attribute mapping configuration based on SOLIDWORKS Electrical and SOLIDWORKS PDM default installations. The configuration can be adapted to customer specific requirements.

The following object attributes can be configured:

Tree panel name	SOLIDWORKS Electrical	SOLIDWORKS PDM	Used by integration function
Working Folder > SW Electrical Project	Project	Project folder	Open, Check In
Files > BOM	Location	BOM Header - Virtual document .swebom	Check In
Files > Purchased Part	Manufacturer Part	Part - Virtual document .swe	Synchronize Libraries
Files > Purchased Cable	Cable	Cable - Virtual document .swe	Synchronize Libraries
Files > Drawing	Drawing export	File .dwg or .dxf	Check In
Files > TEWZIP	Project Archive export	File .tewzip	Check In
Files > PDF	PDF export	File .pdf	Check In

A new mapping rule is added by clicking **From SW PDM to SW Electrical** or **From SW Electrical to SW PDM** located in the column headers.

The mapping direction is changed by clicking **Mapping Direction**. This can be useful when there are many attribute mapping rules configured and the master-side should be changed.

Example

The initial synchronization should take place between the SOLIDWORKS Electrical library and SOLIDWORKS PDM. After that, changes to the to library components should rely on changes made in SOLIDWORKS PDM. For this to work, the direction for the first synchronization run is set to **SWPDM < SWE** and for all later runs then changed to **SWPDM > SWE**.

Rules for changing the mapping direction:

- **Mapping rule should be one-to-one.** If a complex expression is used, for example when an attribute value on one side is made of a combination of several attributes, or contains a mixture of static text and attributes, then the mapping rule cannot be reversed.
- **Both sides of the mapping rule should be editable.** Some variables in SOLIDWORKS Electrical are read-only and therefore can't be changed. The same is true for file names or number generators in SOLIDWORKS PDM.
- **No duplicate mapping is allowed.** It is not allowed to define several mapping rules on one attribute.

3.1.4.1 Add/Remove Electrical Project Data Files

You can choose which SOLIDWORKS Electrical project data files will be exported to SOLIDWORKS PDM during **Check-In**.

Overview

You may do not want to export all possible SOLIDWORKS Electrical project data files during Check-In. For example, export of PDFs takes significant time and can be turned off to speed up the process of saving data to the SOLIDWORKS PDM vault.

To switch off the export of some project data files, right-click the element of the **Files** tree and select **Remove** from the pop-up menu. You can switch on the export later, by choosing **Add** from the pop-up menu and selecting the data file from drop-down list.



You can not switch off the export of Purchased Parts and Cables.

3.1.4.2 Export of arbitrary files

An electrical project can contain a variety of supplementary files : documents, reports, pictures. It is possible to add them to the SOLIDWORKS PDM vault control during **Check-In**.

Procedure

1. Right-click on the **Files** tree and select **Add**.
→ The **Add file export configuration** dialog opens.
2. Select any of the **Other Object** configurations in the dialog. There are 10 configurations which you can use to setup your own rules.
→ Selected **Other Object** will be added to the tree.

3. Define a **Naming Rule**.



The name of the file you can see in the SOLIDWORKS Electrical is not the actual file name, but its "Mark" property. If you want the file name in SOLIDWORKS PDM to be the same as you see it in the design, use the "{SWEFILE.Mark}" formula. The formula "{SWEFILE.SweNameWithoutExtension}" on the other hand, can be used if you want to use the real file name from the SOLIDWORKS Electrical data directory.

4. **Optional:** Specify a sub folder name to store files.
5. Define a **File Mask**. A file mask will be applied to the names of the files included in the project. Files matching the provided mask will be added to the SOLIDWORKS PDM vault. You can use file name wildcards separated by the pipeline sign ("|") to specify which files should be exported.

Result

Example of the configuration used to export all **JPEG** files:

- **Naming Rule:** {SWEFILE.Mark}
- **Sub folder :** JPEG
- **File Mask:** *.jpg|*.jpeg



Avoid the situation when the same original file will fall into more than one export rule. This is not supported and the **Check-In** will fail.

3.1.4.3 Rule Expression Editor

Helps to define complex naming rules for objects used in the integration process.

Overview



The **Rule Expression Editor** places specifically formatted wildcards into the field from which the editor was called. Wildcards are inserted at the cursor position within the field. Multiple wildcards can be used in one rule at the same time.

Example

Field content: {PART.Manufacturer}_ {SWPDM.SerNo:New Serial Number}

There are two wildcards enclosed in braces, separated by an underline "_". If the cursor is within the first wildcard, the expression {PART.Manufacturer} can be edited. If the cursor is within the second wildcard, the expression {SWPDM.SerNo:New Serial Number} can be edited. If the cursor is not within a wildcard, a new wildcard is created and inserted at the cursor position.

Description of the user interface

Field name	Description
Content Object	Defines the name of the object. The available objects depend on the currently selected folder or file object.
Content Property	Defines the name of the property. The available properties depend on the selected content object.
Property Value	Defines the property value. The value can be plain text or from a list.  This field is visible not for all properties.
Generator Name	Defines the name of the SOLIDWORKS PDM number generator.  This field is visible only for properties which refer to a number generator.

3.1.4.4 Edit Mapping Rule

Defines the attribute mapping between SOLIDWORKS PDM and SOLIDWORKS Electrical.

Overview

Depending on the mapping direction, the target object is always located in the top part of the dialog, whereas the source object is always in the bottom part. Target and source are separated by the text **Will be equal to**.

Example

The following example shows the source with static text and wildcard combined.

```
French name: {PROJECT.Property:DescriptionFr}. English name:  
{PROJECT.Property:DescriptionEn}
```

See the Administration Guide for more details and samples of mapping rules.

Description of the user interface

Field name	Description
Target <object> variable	Defines the target variable, consisting of two values: <ul style="list-style-type: none">■ The variable name, represented by a list of possible attributes to choose from.■ An optional qualifier, depending on the mapping direction, represented by a list of values to choose from or and edit field. It is displayed only for properties which require additional parameters.
Source object variable	Defines the source variable. Here it is a simple edit field to enter a constant value or some rules with wildcard. A combination of constants and wildcard is also possible. Clicking } opens the Rule Expression Editor (p. 16) that helps to fill the property value.

3.1.5 Import/Export Configuration

The integration configuration can be exported to the file system and re-imported.

Overview

The configuration of the integration is stored inside the SOLIDWORKS PDM vault and is not easy to access. The export/import functionality of the configuration can be used when you need to migrate settings between vaults, backup them or send them for support in case of issues.

To export the configuration, click **Export**. Select a folder and click **OK**. A file named `swe-swpdm.config` will be created in the specified directory.

To import the configuration, click **Import**. Select a folder containing the `swe-swpdm.config` file and click **OK**.

 It is also possible to import configurations from previous integration versions. To do so, select a folder containing the legacy configuration files: `swe.config` and `epdm.config`.

3.2 Attribute mapping example

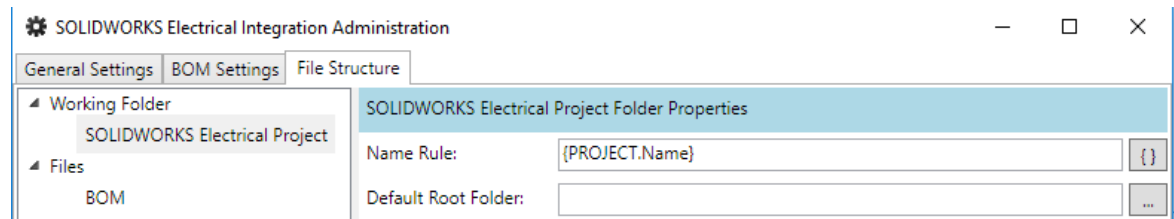
The goal is to set up two new mapping rules. The first rule maps the customer name in the SOLIDWORKS Electrical project to the field **Customer** in the data folder card in SOLIDWORKS PDM. The second rule maps

the field **Location** in the data folder card in SOLIDWORKS PDM to the customer address in the SOLIDWORKS Electrical project.

Procedure

1. Open the SSA Tool from SOLIDWORKS PDM Administration Tool and select **File Structure > Working Folder/SOLIDWORKS Electrical Project**

→ In the dialog, only the **Name Rule** is already configured. The rule states, that the SOLIDWORKS PDM project folder is named with the SOLIDWORKS Electrical project name.



2. Configure the first mapping rule from SOLIDWORKS Electrical to SOLIDWORKS PDM:

- a) In the mapping rule section, click **From SW Electrical To SW PDM**.

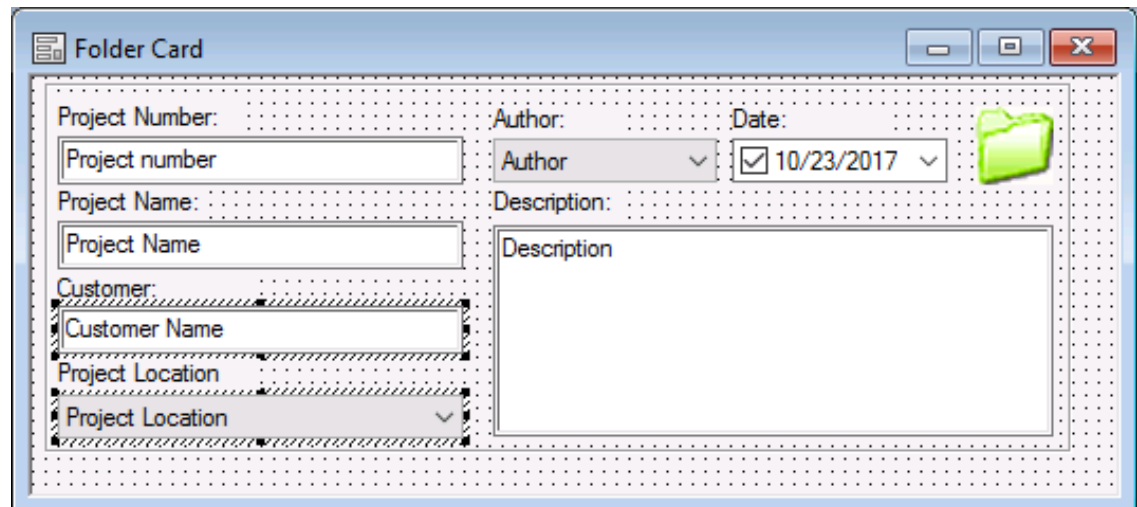
→ The dialog **Edit Mapping Rule** opens.

- b) In the target section (SOLIDWORKS PDM variable), select the only possible value **Variable** from the list.

- c) From the list to the right, select the variable **Customer Name**, that represents **Customer** in the data folder card.

To find out the correct variable name in SOLIDWORKS PDM, start the SOLIDWORKS PDM Administration Tool and navigate to the **Folder Card** definition.

Open the folder card and note the variable name **Customer Name** as used above from the editable field. Also note the variable **Project Location** for the second mapping rule.

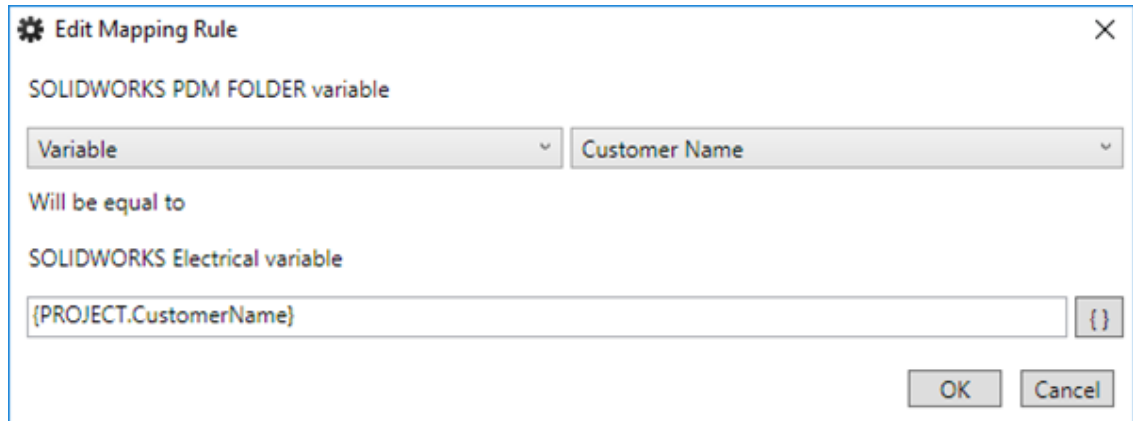


- d) In the source section (SOLIDWORKS Electrical PROJECT variable), click {}.

→ The **Rule Expression Editor** opens.

- e) Select **Content object = PROJECT** and **Content property = CustomerName**

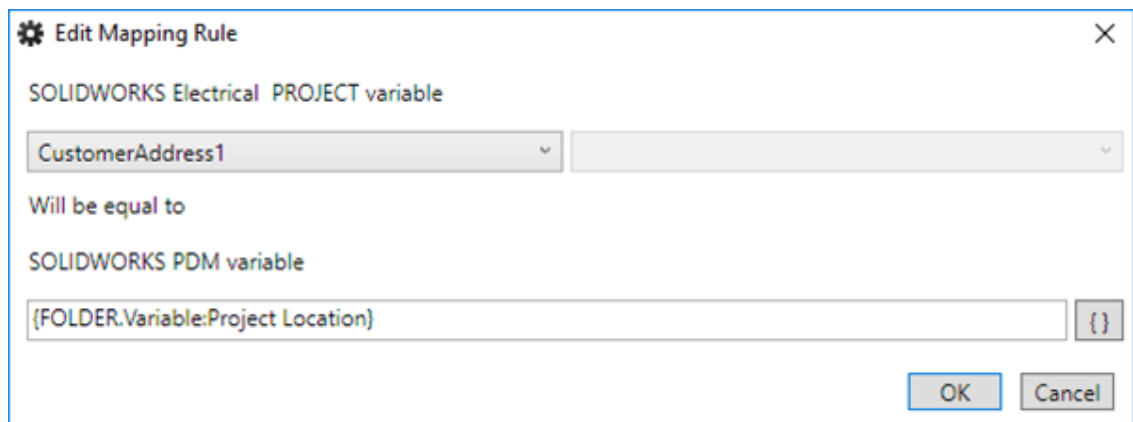
- f) Click **OK**.
→ The dialog shows the complete rule.



- g) Click **OK**.
→ The new rule is added to the mapping rule list.

3. Configure the second mapping rule from SOLIDWORKS PDM to SOLIDWORKS Electrical:

- a) Click **From SW PDM to SW Electrical**.
→ The dialog **Edit Mapping Rule** opens.
- b) In the target section (SOLIDWORKS Electrical PROJECT variable), select the value **CustomerAddress1** from the list. This property does not have any additional attributes.
- c) In the source section (SOLIDWORKS PDM variable), click {}.
→ The **Rule Expression Editor** opens.
- d) Select **Content object = FOLDER**, **Content property = Variable** and **Property value = Project Location**.
- e) Click **OK**.
→ The dialog shows the complete rule.



- f) Click **OK**.
→ The new rule is added to the mapping rule list.

Result

The new attribute synchronization is configured. Each time **Check In** or **Open** is executed, SOLIDWORKS Electrical project attributes are synchronized with the SOLIDWORKS PDM folder and vice versa.


3.3 Special cases of attribute mapping

In mapping rules, some SOLIDWORKS Electrical attributes require additional parameter to be configured correctly.

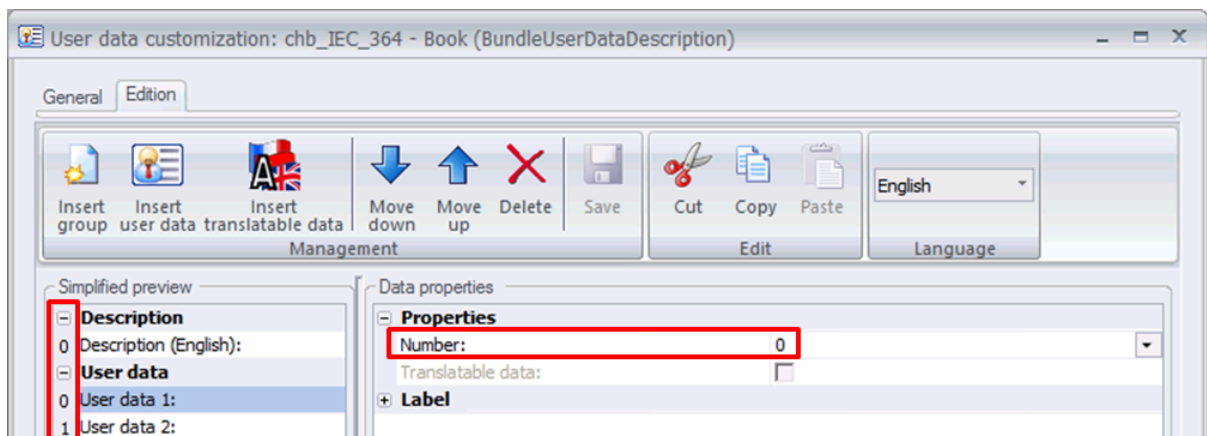
Translated Data and User Data attributes

The following SOLIDWORKS Electrical attributes require additional parameter:

SWE attribute name	Format	Example
DescriptionLang	DescriptionLang:<lang_code>	DescriptionLang:en
TranslatedData	TranslatedData:<int_num>	TranslatedData:2
TranslatedDataLang	TranslatedData:<int_num>:<lang_code>	TranslatedData:2:en
User Data	UserData:<int_num>	UserData:2

 In the mapping rules, the internal number of enumerable attributes is used and not the number from the representation label.

To obtain the internal number, right-click the book in SOLIDWORKS Electrical and select **Properties > Customize**. The internal number is displayed in front of the label, but is also displayed in the window **Data properties** to the left when selecting the label.



DBField attribute

This is a special attribute which can be used by advanced users with good knowledge of the SOLIDWORKS Electrical database structure. It can be used to access attributes which are not listed in the dropdowns, but exist in the database and their table column is known. The syntax is `DBField:<db_field_name>`. For example, by using the formula `PART.DBField:cre_lib_name` in the configuration of purchased part attributes, you can get the name of the library the part belongs to. The following table shows how integration objects are mapped to the SOLIDWORKS Electrical database tables.

Object	Database	Table
PART	tew_catalog	tew_catreference
CABLE	tew_app_data	tew_cablereference
DRAWING	tew_project_data_N	tew_file

Object	Database	Table
BOOK	tew_project_data_N	tew_bundle
LOCATION	tew_project_data_N	tew_location

4 Update

4.1 Modifying installation

Complete these steps to modify an existing installation and add for example new components or remove existing.

About this task

During modification, no existing files are overwritten and only missing files are added.



You can install, modify, repair, or uninstall also in silent-mode. See [Silent-mode installation](#) (p. 30) for more information.

Procedure

1. Close all open applications related to the integration.
2. Start the current installer `Setup-*.exe` with administrator rights. If this file is no longer available, start `Setup-*.msi` directly from the directory `C:\ProgramData\XPLM Solution GmbH\packages`. Alternatively, you can also start the installer from Windows:
 - a) In the system settings, go to *Apps & Features* and search for the entry **Dassault Systemes SOLIDWORKS Electrical Setup**.
 - b) Select the entry and click **Modify**.
 - Visual C++ runtimes are checked/installed again and the installation wizard appears.
3. Click **Next** to start the wizard.
 - The step *Modify, repair or remove installation* appears.
4. Click **Modify**.
 - The step *Installation path* appears.
5. In this step, you cannot change the installation path, but applying overlay packages or making backups are possible.
 - If another product is already installed, you cannot change the installation directory. The product installed first defines this path for other products.
 - If you have received an overlay package from Dassault, or want to apply your own, enable the option **Apply custom files after installation** and enter the path to the overlay. Overlays contain customized files that overwrite the installed files as the last step. See [Working with overlay packages](#) (p. 34) for more information.
 - To backup the existing installation, enable the option **Backup current installation**. Define the backup scope first. Then click **Browse** and select a backup directory.
 - ⚠ Always create a backup when you update or change an installation. This makes it easier to compare and merge the files later.
6. Click **Next** and update components, if required.
 - The step *Ready to install* appears.

7. To start installation, click **Install**.



During installation, a progress dialog is shown with detailed information on each MSI currently installed. If there are installation issues, the process stops and you can examine the error in the lower dialog section. If required, click **Save to log** and send the log file to support@xplm.com for further investigation. This new progress dialog is available from installer versions 24.3.3.606 onwards.

For older installers, a common CMD window with the progress is shown. Do not click into this window or installation cannot proceed. To continue if you clicked in, press **Enter** or **Esc**.

8. To close the wizard after installation, click **Finish**.

Result

The installation is modified. Start the product and verify everything works as expected.

4.2 Repairing installation

Complete these steps to repair an existing installation if the product does not work correctly, for example fixing missing or corrupt files, or incorrect shortcuts and registry entries.

About this task

During repair, existing files are overwritten and components registered again.



You can install, modify, repair, or uninstall also in silent-mode. See [Silent-mode installation](#) (p. 30) for more information.

Procedure

1. Close all open applications related to the integration.
2. Start the current installer `Setup-*.exe` with administrator rights. If this file is no longer available, start `Setup-*.msi` directly from the directory `C:\ProgramData\XPLM Solution GmbH\packages`. Alternatively, you can also start the installer from Windows:
 - a) In the system settings, go to *Apps & Features* and search for the entry **Dassault Systemes SOLIDWORKS Electrical Setup**.
 - b) Select the entry and click **Modify**.→ Visual C++ runtimes are checked/installed again and the installation wizard appears.
3. Click **Next** to start the wizard.
→ The step *Modify, repair or remove installation* appears.
4. Click **Repair**.
→ The step *Installation path* appears.
5. In this step, you cannot change the installation path, but applying overlay packages or making backups are possible.
 - If another product is already installed, you cannot change the installation directory. The product installed first defines this path for other products.
 - If you have received an overlay package from Dassault, or want to apply your own, enable the option **Apply custom files after installation** and enter the path to the overlay. Overlays contain customized

files that overwrite the installed files as the last step. See [Working with overlay packages](#) (p. 34) for more information.

- To backup the existing installation, enable the option **Backup current installation**. Define the backup scope first. Then click **Browse** and select a backup directory.



Always create a backup when you update or change an installation. This makes it easier to compare and merge the files later.

6. Click **Next**.

→ The step *Ready to install* appears.

7. To start installation, click **Install**.



During installation, a progress dialog is shown with detailed information on each MSI currently installed. If there are installation issues, the process stops and you can examine the error in the lower dialog section. If required, click **Save to log** and send the log file to support@xplm.com for further investigation. This new progress dialog is available from installer versions 24.3.3.606 onwards.

For older installers, a common CMD window with the progress is shown. Do not click into this window or installation cannot proceed. To continue if you clicked in, press **Enter** or **Esc**.

8. To close the wizard after installation, click **Finish**.
9. **Optional:** Carefully compare and merge the changes from the backup directory with the newly installed files.

Result

The installation is repaired. Start the product and verify everything works as expected.

4.3 Updating installation

Complete these steps to update an existing installation.

About this task

Dassault strongly recommends using appropriate services for an update. This ensures that existing functionality and modifications are correctly transferred to the new product. Contact <https://www.solidworks.com/support/> for assistance.



You can install, modify, repair, or uninstall also in silent-mode. See [Silent-mode installation](#) (p. 30) for more information.

Procedure

1. Close all open applications related to the integration.
2. Start the new installer `Setup-*.exe` with administrator rights.
→ Visual C++ runtimes are checked/installed again and the installation wizard appears.
3. Click **Next** to start the wizard.
→ The installer detects an existing installation and shows a message.
 - If the existing installation is compatible with the unified installer, you can proceed. At the start of the installation, the old components are removed first and the new ones are installed afterwards.

How to identify if the existing installation is already compatible with the unified installer technology?

- The directory `C:\ProgramData\XPLM Solution GmbH` exists.
- The registry entry `HKLM\SOFTWARE\XPLM Solution GmbH\{00000000-0000-0000-0000-000000000000}` exists.
- If the existing installation is not compatible with the unified installer, it will first be uninstalled completely. Before you continue, manually back up the existing installation directory `<SWPDM INSTALL DIR>\CAD Integration`. Then proceed with installation.



In both cases, files with the prefix `customer_` are not affected by the update. All other files are overwritten with the new files.

4. Click **Next**.

→ The step *License agreement for end-users* appears.

5. Accept the license agreement and click **Next**.

→ The step *Installation path* appears.

6. If an existing and compatible installation was found, you cannot change the installation path in this step, but applying overlay packages or making backups are possible.

- If another product is already installed, you cannot change the installation directory. The product installed first defines this path for other products.
- If you have received an overlay package from Dassault, or want to apply your own, enable the option **Apply custom files after installation** and enter the path to the overlay. Overlays contain customized files that overwrite the installed files as the last step. See [Working with overlay packages](#) (p. 34) for more information.
- To backup the existing installation, enable the option **Backup current installation**. Define the backup scope first. Then click **Browse** and select a backup directory.



Always create a backup when you update or change an installation. This makes it easier to compare and merge the files later.

7. Click **Next** and update components, if required.

8. To start installation, click **Install**.



During installation, a progress dialog is shown with detailed information on each MSI currently installed. If there are installation issues, the process stops and you can examine the error in the lower dialog section. If required, click **Save to log** and send the log file to support@xplm.com for further investigation. This new progress dialog is available from installer versions 24.3.3.606 onwards.

For older installers, a common CMD window with the progress is shown. Do not click into this window or installation cannot proceed. To continue if you clicked in, press **Enter** or **Esc**.

9. To close the wizard after installation, click **Finish**.

10. Carefully compare and merge the changes from the backup directory with the newly installed files.

Result

The installation is updated. Start the product and verify everything works as expected.

5 Uninstallation

5.1 Removing installation

Complete these steps to remove an installation.

About this task

To uninstall an installation, you need the setup and component MSIs. If you installed correctly, they are located in the directory `C:\ProgramData\XPLM Solution GmbH\packages`.



The Windows uninstall feature is not supported. However, you can start the MSIs from Windows and use the installer's uninstall function.



You can install, modify, repair, or uninstall also in silent-mode. See [Silent-mode installation](#) (p. 30) for more information.

Procedure

1. Close all open applications related to the integration.
2. Start the current installer `Setup-*.exe` with administrator rights. If this file is no longer available, start `Setup-*.msi` directly from the directory `C:\ProgramData\XPLM Solution GmbH\packages`. Alternatively, you can also start the installer from Windows:
 - a) In the system settings, go to *Apps & Features* and search for the entry **Dassault Systemes SOLIDWORKS Electrical Setup**.
 - b) Select the entry and click **Modify**.
→ Visual C++ runtimes are checked/installed again and the installation wizard appears.
3. Click **Next** to start the wizard.
→ The step *Modify, repair or remove installation* appears.
4. Click **Remove**.
→ The step *Remove of the installation* appears.
5. Click **Remove** to remove the installation.
6. To close the wizard after installation, click **Finish**.
7. Check the directory `<SWPDM INSTALL DIR>\CAD Integration` for leftover directories and files, and delete them manually.
8. Right-mouse click on **Add-ins > SOLIDWORKS Electrical Integration Add-in** and select **Remove** from the menu.

Result

The installation is removed.

6 Troubleshooting

6.1 Logs and error messages

Errors are logged.

If during the execution of the integration errors occur, the message **Process completed with errors. Please refer to the log file for details.** appears. Also link to open the log file is displayed. The log file directory by default is set to `%TEMP%\SWE-SWPDM\Logs`. Log settings can be changed manually by editing the file `%APPDATA%\SOLIDWORKS\SOLIDWORKS PDM Integrations\log.config`. This file will be created after the first execution of any integration modules.

Configuration parameters

Parameter name	Description
Destination directory	Defines the path to the log files folder. Variables are possible to specify relative paths, see list below.
Severity to log	Defines the log levels: <ul style="list-style-type: none">■ Error: Output error-handling messages.■ Information: Output informational messages, warnings, and error-handling messages.■ Off: Output no tracing and debugging messages.■ Warning: Output warnings and error-handling messages.■ All: Output all debugging and tracing messages.
Delete logs older than (days)	Defines how many days the old log files are stored.

Destination directory variables

Variable	Expands to	Remarks
<code>%EXEDIR%</code>	Integration installation directory	
<code>%SYSTEMDRIVE%</code>	Windows installation drive	
<code>%PROGRAMFILES%</code>	<code>C:\Program Files</code>	
<code>%PROGRAMFILES(X86)%</code>	<code>C:\Program Files (x86)</code>	Only in 64-bit OS
<code>%COMMONPROGRAMFILES%</code>	<code>C:\Program Files\Common Files</code>	
<code>%COMMONPROGRAMFILES(X86)%</code>	<code>C:\Program Files (x86)\Common Files</code>	Only in 64-bit OS
<code>%PROGRAMDATA%</code>	<code>C:\ProgramData</code>	
<code>%USERPROFILE%</code>	<code>C:\Documents and Settings\{username}</code>	
<code>%TEMP%</code> or <code>%TMP%</code>	<code>C:\Documents and Settings\{username}\Local Settings\Temp</code>	

Variable	Expands to	Remarks
%WINDIR%	C:\Windows	

7 References

7.1 Silent-mode installation

You can also install this Dassault product in silent-mode. Silent-mode has the advantage that you can easily install the product from a batch file without showing the installer GUI. Alternatively, you can start the installer with preset options, allowing it to be installed in a controlled manner by the user or by other automated installation routines.


The installer packages are all of type Windows Installer (MSI) and require corresponding parameters for silent-mode installation.

The Visual C++ runtimes are normal executables. Always install all x64/x86 runtimes that come with the installer package.

Understanding installer structure

When you start an installation using the installer, required files are copied first to the directory `C:\ProgramData\XPLM Solution GmbH` and are executed from this location.

```
XPLM Solution GmbH
├──cmd
├──log
└──packages
```

- `cmd`: Contains the batch files `Setup-*_admin.bat` and `Setup-*_user.bat`.
 - The file `Setup-*_user.bat` contains the copy commands for the required MSIs from the original location to the directory `C:\ProgramData\XPLM Solution GmbH\packages`.
 - The file `Setup-*_admin.bat` installs the individual MSIs from this new location with the parameters as defined in the installer.
 - `log`: Contains log files of each installed component.
 - `packages`: Contains copies of all MSIs used for installation, modification or uninstallation.
-  Use the definitions in the files `Setup-*_user.bat` and `Setup-*_admin.bat` as the basis for a silent-mode installation. The command line calls already contain the required component MSIs and parameters as selected in the installer.

General command line calls

Installing Visual C++ runtimes:

```
vcredist_*.exe /quiet
```

Uninstalling Visual C++ runtimes:

```
vcredist_*.exe /quiet /uninstall
```

Installing or modifying MSIs:

```
msiexec /i <fileName>.msi /quiet <parameter>=<value>
```

Repairing MSIs:

```
msiexec /i <fileName>.msi /quiet INSTALLMODE=Restore
```

Uninstalling MSIs:

```
msiexec /x <fileName>.msi /quiet REMOVE_SECURE=1
```

Using preset options in the installer (Setup-*.exe):

```
Setup-*.exe <parameter>=<value>
```

Using preset options in the installer (Setup-*.msi):

```
msiexec /i Setup-*.msi <parameter>=<value>
```

Creating a batch file for silent-mode installation

This example is intended as a general guideline for creating an installation script in silent mode. It assumes that the MSIs used for the installation are stored on a network share.

1. On a test computer, extract the main archive and start the file `Setup-*.exe` with administrator rights.
2. Select required components and settings, and finish installation.
3. Copy the entire contents from the extracted archive to a network share, for example `\\myShare`.
4. Create a new batch file locally, for example `silent.bat`.
5. Add the installation commands for the C++ runtimes to this file, for example:

```
REM *** install c++ runtimes ***  
\\myShare\vcredist_14.38.33130.0_x64\vcredist_14.38.33130.0_x64.exe /quiet  
\\myShare\vcredist_14.38.33130.0_x86\vcredist_14.38.33130.0_x86.exe /quiet
```

6. Go to the directory `C:\ProgramData\XPLM Solution GmbH\cmd`.
7. Open the file `Setup-*_user.bat` and copy all *robocopy* commands into your batch file. Change the path in the first argument (the source) to point to the network share, for example:

```
REM *** copy from network share to client ***  
robocopy "\\myShare\packages" "C:\ProgramData\XPLM Solution GmbH\packages" ←  
Core_23.0.0.538.msi  
...
```

8. Open the file `Setup-*_admin.bat` and copy the command line calls for installing the MSIs into your batch file. Change the path of the MSI to point to the network share, for example:

```
REM *** installing msi ***  
msiexec /i "\\myShare\packages\Core_*.msi" /passive ←  
CALLED_BY=Setup-* ←  
INSTALLDIR="C:\Program Files\XPLM Solution GmbH\" ←  
BATCH_ADMIN="C:\ProgramData\XPLM Solution GmbH\cmd\Setup-*_admin.bat" ←  
GUI_LOG_FILE="C:\ProgramData\XPLM Solution GmbH\log\Setup-*_gui.log" ←  
JAVA_JNI=0 ←  
JAVA_JNI_X86="" ←  
JAVA_JNI_X64="" ←  
...
```



In the above example, line breaks were inserted to show readable content. Usually, each `msiexec` call would be on one line. You can further clean-up each call by deleting the information marked red, as it is not required in your batch file.

9. To test silent installation, use a clean client computer, copy the batch file `silent.bat` to it and run it with administrator rights.

Parameters usage

- If no parameters are defined, default settings apply. In the following tables, default settings are underlined>.
- If you use parameters in the scope of `Setup_*.exe/msi`, use them with the provided prefix, for example `COR_JAVA_JNI`.



You cannot use the setup files `Setup_*.exe/msi` for silent-mode installation. For this you must use the individual component MSIs. However, you can use parameters in `Setup_*.exe/msi` to preset options when installing in GUI-mode.

- Use parameters without prefix to define settings within the scope of component MSIs, for example `Core_*.msi`.
- Use the following parameters to control either GUI-mode or silent-mode installation:
 - `none`: GUI-mode
 - `/quiet`: Silent-mode without GUI
 - `/passive`: Silent-mode with additional progress indication

Parameter for silent-mode & GUI-mode

The following parameters apply to silent mode as well as to the presetting of options in GUI mode.

Table 5: Component MSI & Setup EXE/MSI

Prefix	Parameter	Value	Description and use
	INSTALLDIR	Path to a valid directory	Defines the path of the installation directory. Available in all MSIs.

Prefix	Parameter	Value	Description and use
	INSTALLMODE	<u>C</u> hange Restore	Defines the installation mode after the installation is already completed and the installer is restarted. <ul style="list-style-type: none"> ■ Change = Modify ■ Restore = Repair Available in all MSIs.
	REMOVE_SECURE	<u>0</u> 1	Enables uninstallation. This corresponds to Remove in the installer. Available in all MSIs.
PDM	VERSION	Version as shown in installer	Defines the SOLIDWORKS PDM version. Available in EnterprisePDM MSI and all Setup EXE/MSI using this component.

Parameter for GUI-mode

The following parameters apply exclusively to the presetting of options in GUI-mode and not to silent-mode.

Table 6: All MSIs

Parameter	Value	Description and use
BACKUP_FILES	Path to a valid directory	Defines a location for the backup.
BACKUP_TYPE	FULL CONFIG	Defines the backup scope. <ul style="list-style-type: none"> ■ FULL = full backup of <SWPDM INSTALL DIR>\CAD Integration. ■ CONFIG = backup of configuration directory only, for example <SWPDM INSTALL DIR>\CAD Integration.
CUSTOM_FILES	Path to a valid directory	Defines the path to a directory containing an overlay package with custom files to be copied after installation.

Table 7: Setup-PDMProfessional-SolidworksElectrical EXE/MSI

Parameter	Value	Description
PDM_SWE	<u>0</u> 1	Enables SOLIDWORKS Electrical.

7.2 Working with overlay packages

An overlay package usually contains custom files with a modified configuration. You can select an option in the installer to apply an overlay as the last step of the installation process, copying the custom files over the installed files.

About this task

Overlays are best applied in the installer, but are also possible to be applied in silent-mode. Complete the following steps to learn more about overlays.

Procedure

1. How to create an overlay:

- a) Install the product on a client computer using the installer.
- b) After installation, configure the product as required.
- c) Create the directory `overlay` under the extracted installer structure:

```
├─overlay
├─packages
├─vcredist_*_x64
├─vcredist_*_x86
└─Setup-*.exe
```



You can also execute an additional script while applying overlay packages. To do this, create the directory `script` in your overlay directory, and add the batch file `custom.bat`. The script will be executed automatically when the overlay is applied. This is a generic mechanism and works everywhere in an overlay package. Just make sure that you keep the directory and script name as described.

- d) Copy the modified files from the directories under `<SWPDM INSTALL DIR>\CAD Integration` to `custom_files`, creating also their relevant parent directories as in the original location, for example `myOverlay\CAD Integration`.

→ Overlay package is created.

2. How to test the created overlay package in GUI-mode:

- a) Start `Setup-*.exe` or any of the MSIs and click **Modify** to change the installation.
 - The step *Installation path* appears. Because the directory `custom_files` exists in the installer structure, the option **Apply custom files after installation** is already selected and the path to the overlay package is automatically detected.
- b) Complete installation.
 - The content of the overlay package is first copied with *Robocopy* to `%ProgramData%\XPLM Solution GmbH\custom_files`. From there it is copied to `<SWPDM INSTALL DIR>\CAD Integration`.
- c) Check if content of the overlay package was correctly applied in `<SWPDM INSTALL DIR>\CAD Integration`.

→ Overlay package is working.

3. How to test the created overlay package in silent-mode:



There are no parameters for applying an overlay package in silent mode because the copy operation is not triggered by an MSI. You can only use certain parameters to preselect options in the installer. However, you can use the *Robocopy* commands created during application of an overlay package in GUI-mode and use these commands in your own batch file for silent-mode installation.

- a) Go to the directory `%ProgramData%\XPLM Solution GmbH\cmd`.
 - b) Check the latest batch file `Setup-*_user.bat` corresponding to the installer version for the first copy command of the overlay package to `%ProgramData%\XPLM Solution GmbH\custom_files`.
 - c) Copy this command to your own batch file.
 - d) Check the latest batch file `Setup-*_admin.bat` corresponding to the installer version for the second copy command from `%ProgramData%\XPLM Solution GmbH\custom_files` to `<SWPDM INSTALL DIR>\CAD Integration`.
 - e) Copy this command to your own batch file.
 - f) Test your batch file on a clean client computer.
- Overlay package is working.

4. How to manually integrate an updated overlay package from Dassault into an existing installation:

- a) Extract the overlay package and check the directory structure.
 - b) Copy the directory that contains the modified files to the directory `<SWPDM INSTALL DIR>\CAD Integration` and its related sub-directory, for example:

`C:\temp\myOverlay\CAD Integration >> <SWPDM INSTALL DIR>\CAD Integration`
 - c) Check the copied files and start the integration.
- Overlay package is working.

Result

You understand how to create, test and apply overlay packages.