

Administration Tool

SOLIDWORKS Electrical integration
for
SOLIDWORKS PDM

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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.

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.
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 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 "{COMPONENT.Type}_BOM.swebom.cvd"
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>

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. 11) 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. 12) dialog where the rule is edited.</p>

Tree panel	Property panel	Description
<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. 11) 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. 11) 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. 12) 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 <code>.swebom</code>	Check In

Tree panel name	SOLIDWORKS Electrical	SOLIDWORKS PDM	Used by integration function
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.

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.

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.

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.

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. 11) that helps to fill the property value.

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`.