

SOLIDWORKS Enterprise PDM Best Practice Workflow

SOLIDWORKS Electrical 2015

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Revision 4.0



3DEXPERIENCE

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Revision History

Rev #	Date	Description
1.0	Dec 2013	Document created.
2.0	Jan 2014	Document revised.
3.0	Feb 2014	Document revised.
4.0	Mar 2014	Document revised.

1. Introduction

SOLIDWORKS Electrical 2014 service pack 02 or higher includes new enhancement features for working with SOLIDWORKS Enterprise PDM.

In addition to the Update to PDM method available in previous versions of SOLIDWORKS Electrical, version 2014 sp02 or higher provides controlled project workflow by allowing users to check in and out projects to or from SOLIDWORKS Enterprise PDM vaults.

These enhancements provide users with improved ways to better regulate project design, take advantage of project level SOLIDWORKS Enterprise PDM version control, and introduce a way of collaborating on projects in a multi-site environment, or where users need to regularly work on site.

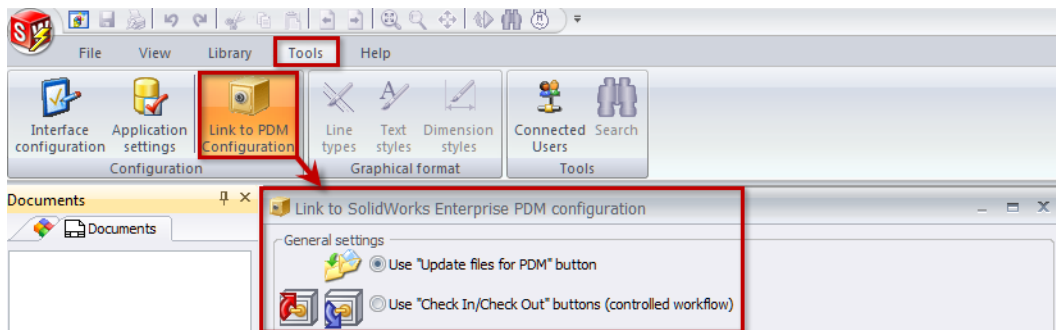
This document provides details on the new recommended workflows for using the SOLIDWORKS Electrical Enterprise PDM link and update tools.

2. Selecting a Method for Linking Data to SOLIDWORKS Enterprise PDM

There are two methods available for linking data to SOLIDWORKS Enterprise PDM:

- *Update Files for PDM*
- *Check In/Check Out* (referred to as the controlled workflow)

Users can access these methods from the **Tools > Link to PDM Configuration**.



The Update Files for PDM method allows users to select project data and export it to an SOLIDWORKS Enterprise PDM vault. This method operates as previous versions of SOLIDWORKS Electrical.

This method is best for sites where multiple users are working on *a single shared Microsoft SQL Server hosting the SOLIDWORKS Electrical databases*, and where all team members are using SOLIDWORKS Electrical.

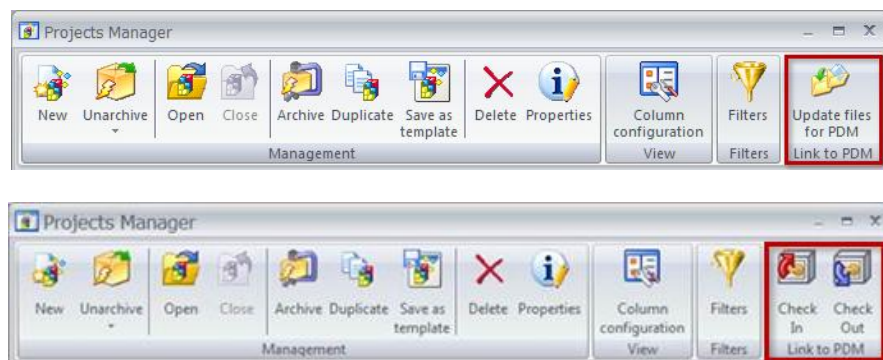
The Check In/Check Out method allows users to check out projects directly from SOLIDWORKS Enterprise PDM and to check them back in. Use it for sites where users work on *different databases*, and want to manage and collaborate on projects by checking data in and out of a shared SOLIDWORKS Enterprise PDM vault. The Check In/Check Out method is also a good choice when networks are slow, when users often have to work outside of their offices (for example on customer sites), and when managers want tight control over which engineers have access to project designs.

NOTE:

Read this the document in its entirety before you select a method.

You can only use one method. You should not change to the other method without first consulting with your local reseller.

When you select a method, you can then use **Check In** and **Check Out** data or **Update Files for PDM** vault from the **Projects Manager**.



3. Project Data Stored in the SOLIDWORKS Enterprise PDM vault

SOLIDWORKS Electrical projects generate data that is useful for manufacturing, purchasing and assembly of electrical designs. SOLIDWORKS Electrical can also pack an entire project into a consolidated zip archive for version and revision history or transport to another SOLIDWORKS Electrical project database. Since SOLIDWORKS Electrical projects and files can only be opened and viewed in SOLIDWORKS Electrical, data used for other areas of the business must be exported.

A SOLIDWORKS Enterprise PDM vault from SOLIDWORKS Electrical can store information such as:

1. Project archive files

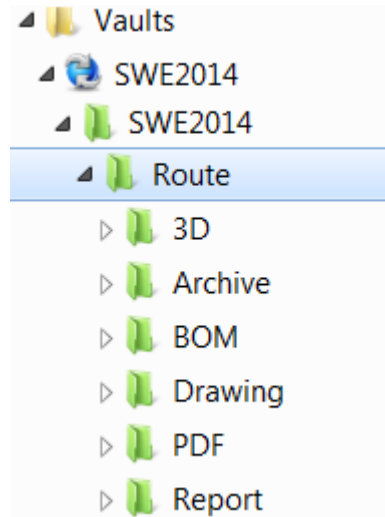
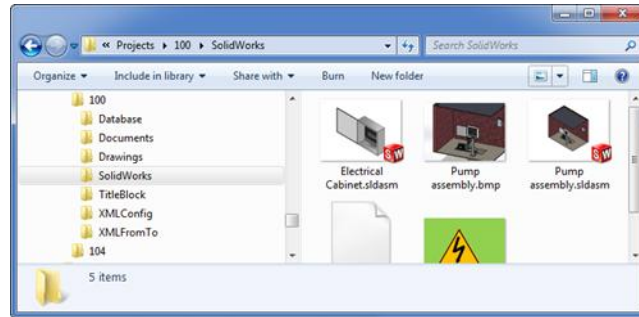
SOLIDWORKS Electrical project archives contain a complete snapshot in time of the project files and database. The archives include SOLIDWORKS models related to the project if using SOLIDWORKS Electrical 3D. Project archives have the extension *tewzip*.

2. Drawing sets in DXF or DWG format

Drawing sheets in an Electrical project can be exported to PDF, DWG and DXF formats. Utilize these output formats for sharing schematic content with all collaborators who do not have SOLIDWORKS Electrical. From the *Import/Export* tab in an active project, select *Export PDF files* or *Export DWG files*. You can also convert to the *DXF* format by using the *Export DWG files* command.

3. 3D SOLIDWORKS parts and assemblies

For users of SOLIDWORKS Electrical 3D, projects also contain SOLIDWORKS assemblies and parts that should be controlled. No export is required from these files unless other derivative files are preferred for document control or PDM such as STEP, IGES or Parasolid. These files reside in the project folder structure under the SOLIDWORKS folder. For example, the SOLIDWORKS assemblies and other files shown in the image below are in project ID 100.



4. Intelligent PDF files

An intelligent, navigable 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.

5. Project level report sets

Reports such as Bills of Materials, wires, cables, harnesses and PLCs can be exported in text or Microsoft Excel formats for downstream use. These can be exported from the *Reports Tool* on the *Project* tab for any active project.

A set of BOMs are also automatically generated as CSV (Comma Separated Value) file types in the BOM subdirectory.

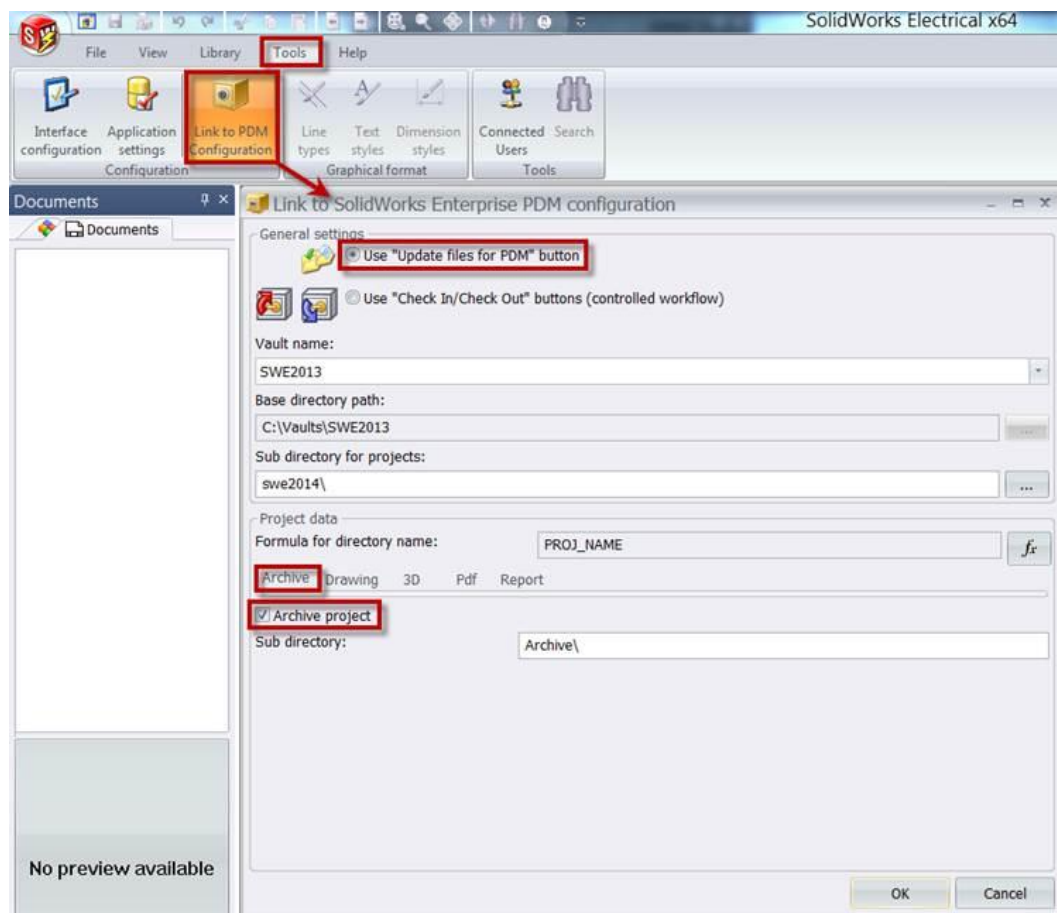
4. Switching from Update Files for SOLIDWORKS Enterprise PDM to Check In/Check Out

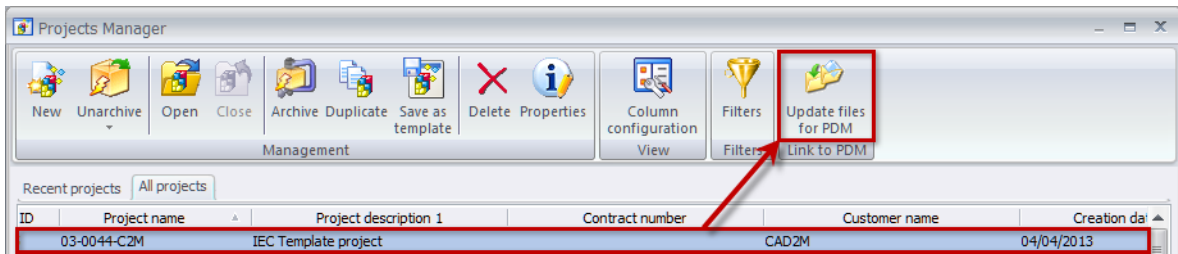
As previously stated users *cannot interchangeably* use the Update Files for PDM method and the Check In/Check Out method.

If you have been using the Update Files for PDM method and want to change, you must archive all project data, delete all projects, and then change to Check In/Check Out.

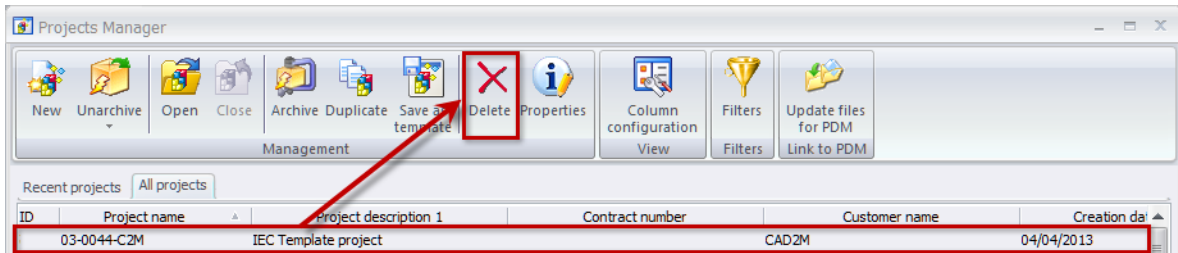
To ensure data integrity, follow these steps:

1. Click **Project data** > **Archive project** to update all projects to SOLIDWORKS Enterprise PDM.

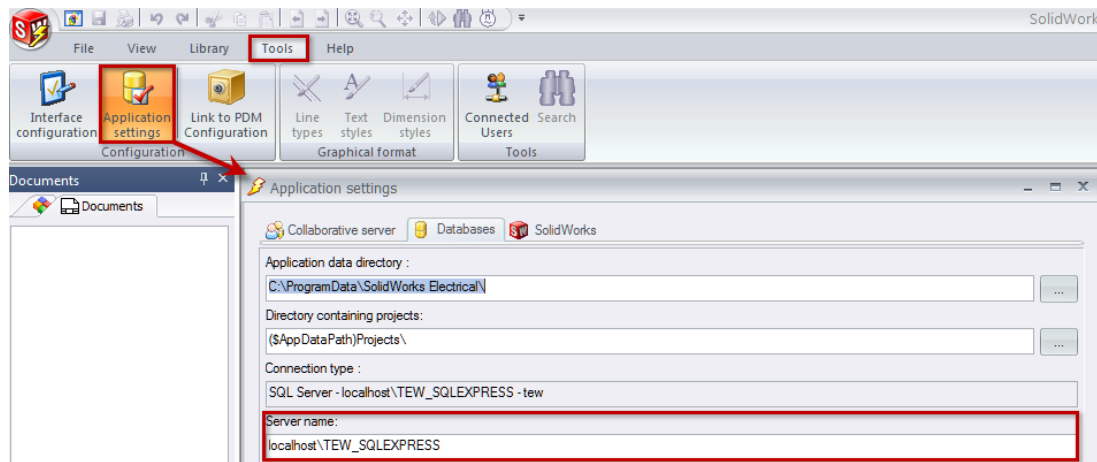




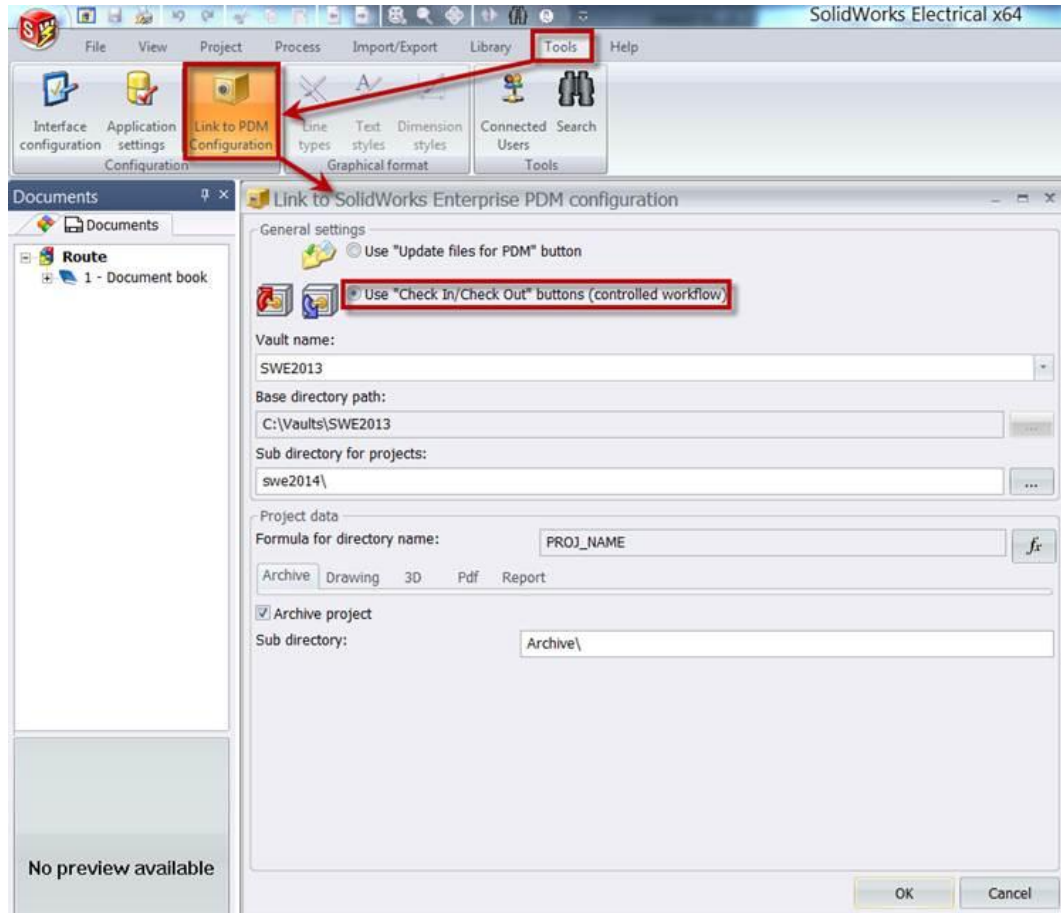
2. Click **Delete** to remove **all** projects.



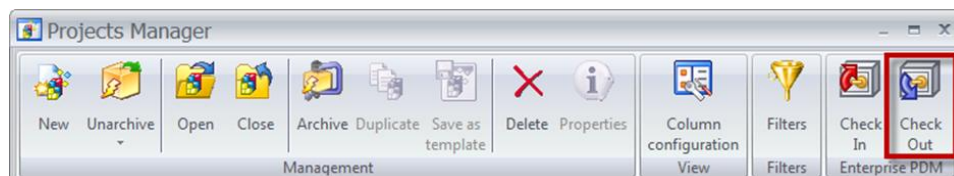
3. When required, select and connect to a SQL Server instance hosting the Electrical database in **Application Settings**.

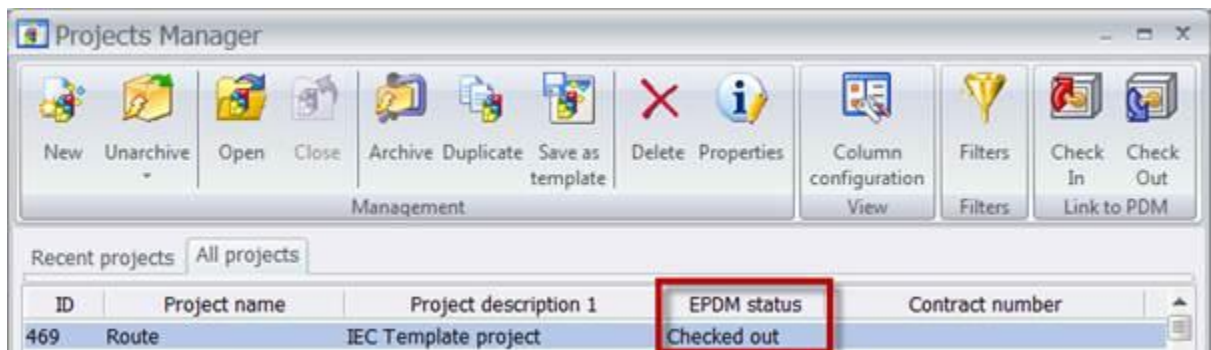
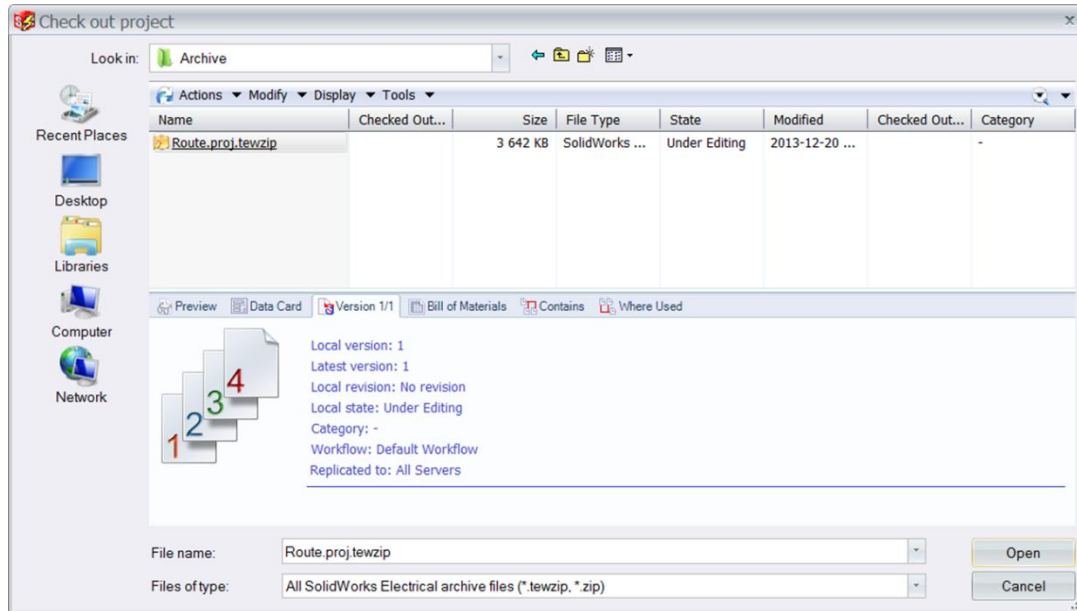


4. From **Tools**, select **Link to PDM Configuration**, and select **Use Check In/Check Out buttons**.



5. Click **Projects Manager > Check Out** to check out projects from the vault.

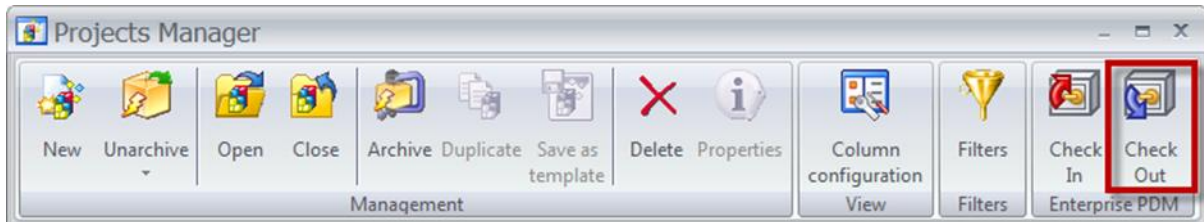




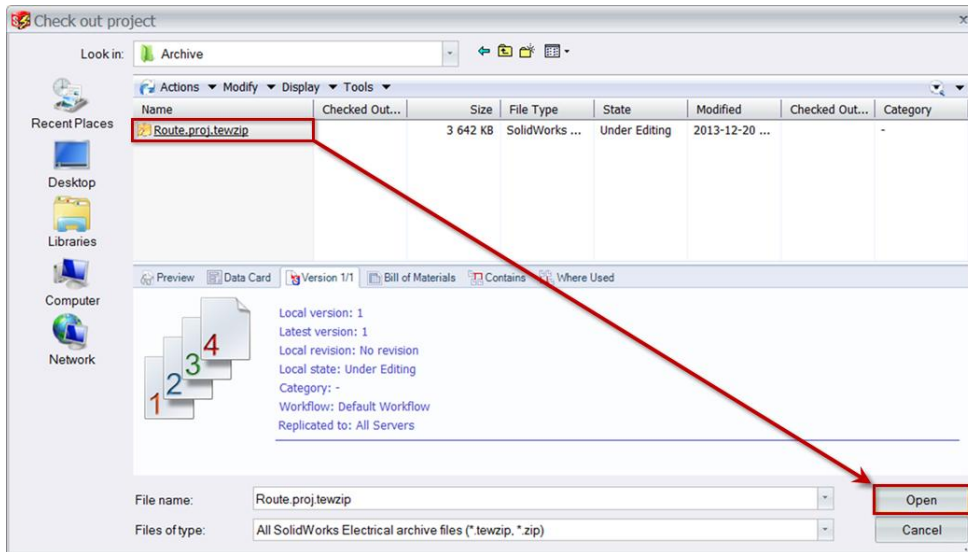
5. Using Check Out

Check Out allows users to check out projects directly from a SOLIDWORKS Enterprise PDM vault. When a project that has been checked out it cannot be accessed by any other user for modification until it has been checked in again as a completed project, unless users are working on the same SQL database and are connected to the same collaborative server.

1. To check out a project and make it available for modification on a local system, select the **Projects Manager > Check Out**.



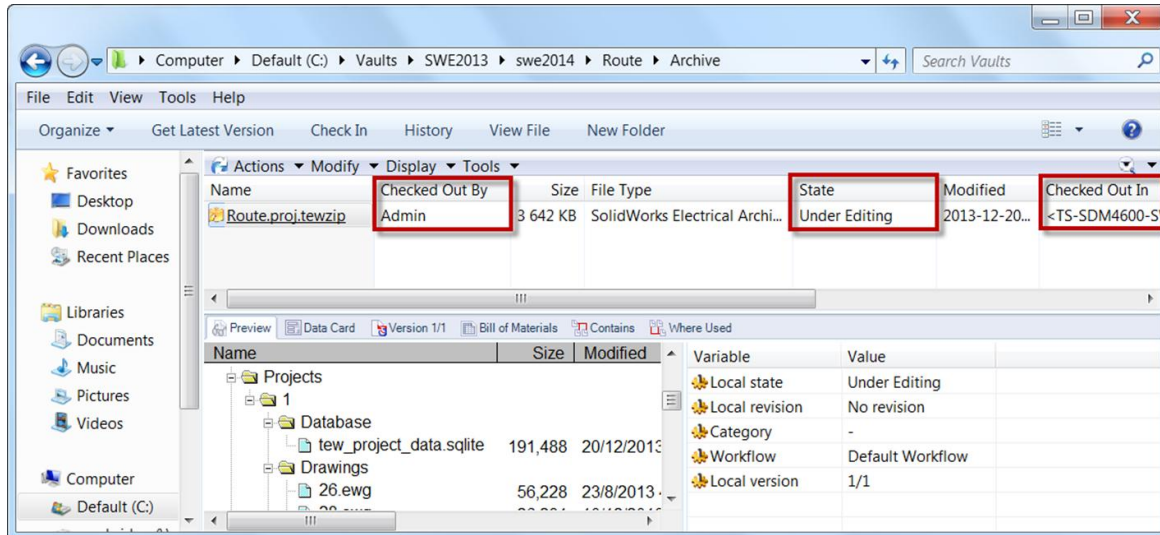
2. In the **Check Out project** dialog box, browse to the SOLIDWORKS Enterprise PDM Vault SOLIDWORKS Electrical **Project** folder **Archive** subdirectory.



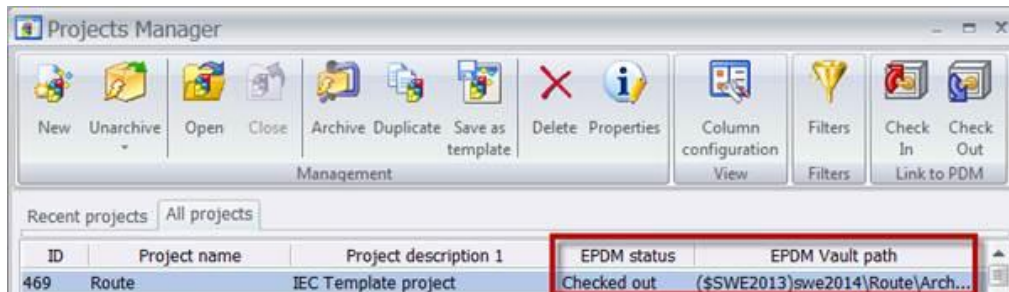
3. Select the project *tewzip* file and click **Open**.

The project is checked out of the SOLIDWORKS Enterprise PDM vault and unarchived. The vault displays the following information:

- the user that currently has the project checked out
- the user's computer where the project is checked out
- the current state of the project



The Projects Manager also displays the SOLIDWORKS Enterprise PDM status and vault path.

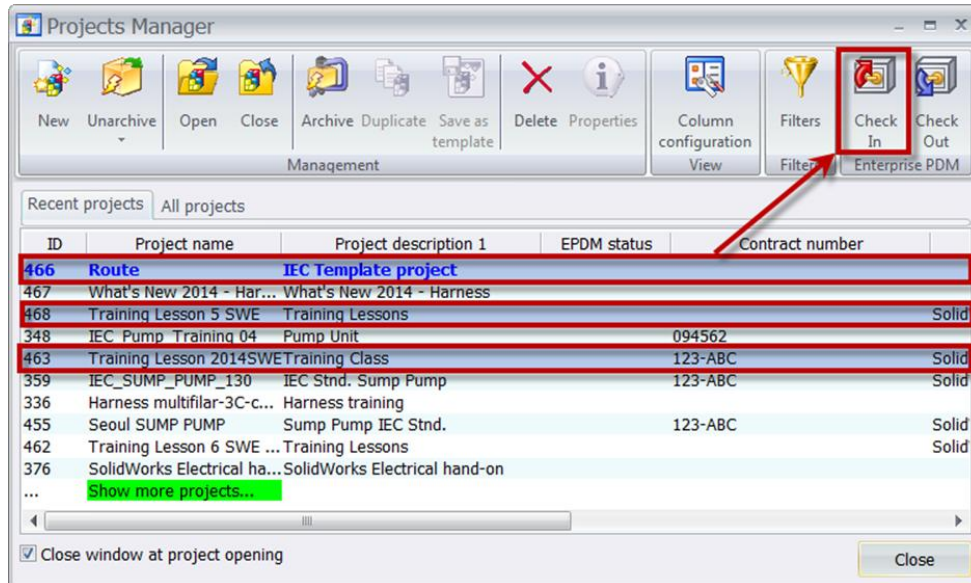


As the project is unarchiving from the vault, the standard **Merge library elements** command displays, where users can select to either **Update data** and access the **Unarchiving projects wizard**, or **Do nothing** to unarchive without updating the application data.

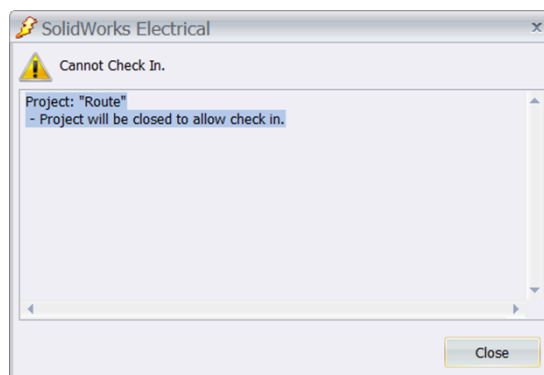
6. Using Check In

Users can check in at any time either to create a backup (while keeping it active on the local system) or to indicate the project is complete.

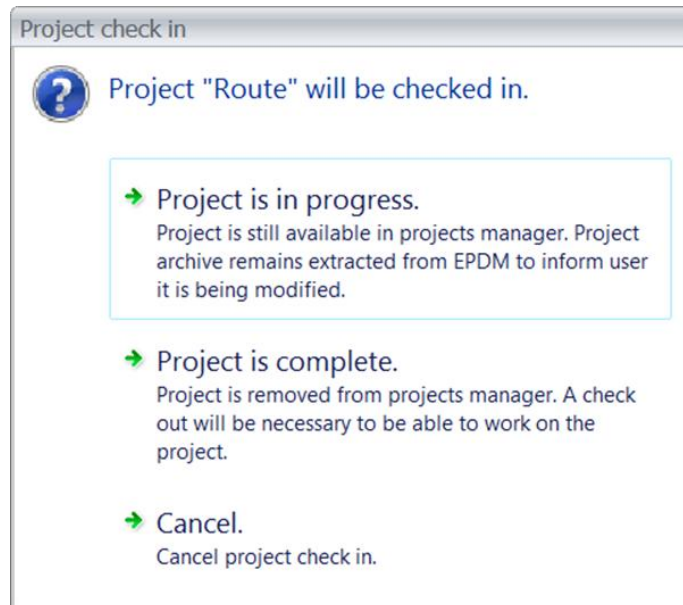
1. From the **Projects Manager**, users can check individual or multiple projects into the SOLIDWORKS Enterprise PDM Vault by selecting the project(s) and clicking **Check In**.



2. Only closed projects can be checked into the SOLIDWORKS Enterprise PDM vault. If a project is open, a warning displays. Click **Close** to automatically close the project and proceed with the check in.



3. In the **Project check in** dialog box, select whether the project is still *in progress* or *is complete*.



If you select the **Project is in progress** option, the current project data is updated to the vault, but the project remains active in SOLIDWORKS Electrical and checked out.

Selecting the **Project is complete** option checks in the project data to the vault, automatically removes the project from the SOLIDWORKS Electrical manager, and makes it available on the vault for other users to check out.

7. SOLIDWORKS Electrical Enterprise PDM Limitations

The two PDM methods cannot be used interchangeably in the SOLIDWORKS Enterprise PDM vault

The Update Files for PDM and Check In/Check Out methods cannot be used interchangeably in the same SOLIDWORKS Enterprise PDM vault. You can only select one method for managing PDM data and should not change the method once it has been used for a SOLIDWORKS Enterprise PDM vault.

Check in of projects can fail when using a SQL Server database that does not use a named instance

The SQL Server hosting the SOLIDWORKS Electrical databases must use a named SQL instance (not the default instance) to ensure that all project data and exported reports check in to the SOLIDWORKS Enterprise PDM vault properly.

Deployments with multiple SOLIDWORKS Electrical clusters attached to the same SOLIDWORKS Enterprise PDM vault can lose data on certain check in failures

If a process checks in all project files, but the project remains in the SOLIDWORKS Electrical database and is shown as checked out in the Project Manager, there is the potential for losing data. Data loss can occur if another SOLIDWORKS Electrical cluster makes modifications to the project while the first SOLIDWORKS Electrical cluster continues to modify the project. These types of check in failures can happen if there are temporary connection issues with the collaborative server, or other check in issues noted in this document.

Potential for data loss when multiple projects with the same name are checked into the SOLIDWORKS Enterprise PDM vault

If two users attempt to check in a project with the same name, data could be overwritten. This error can happen if the warning about the identical file name is displayed to one of the users too late in the check in process.

Users should not manually add archive files to the SOLIDWORKS Enterprise PDM vault

The check in functionality in SOLIDWORKS Electrical creates a file structure for exported reports. If a user adds a file manually to the vault that is not in the expected file structure, this could leave an orphaned copy of the archive file checked out at the original location. However, a workaround is available to address this situation.

Users are not able to open or check out earlier versions of a project inside a vault

Users can only check out the latest version of a file from a SOLIDWORKS Enterprise PDM vault. Once a project is checked out and extracted to the Electrical database, users cannot access previous versions.

There are two potential workarounds to this issue. A user can manually check out the archive file, get the desired version, overwrite the cached version, manually check in the file to create a new version, and check out the file from within SOLIDWORKS Electrical. Alternately, a user could get the previous version, copy the archive file to a location outside of the vault, and unarchive the file in SOLIDWORKS Electrical.

SOLIDWORKS Electrical cannot be used with SOLIDWORKS Enterprise PDM vaults that use unique file names

When SOLIDWORKS Enterprise PDM has the option enabled to restrict duplicate file names, the check in of projects fails. Most of the reports are exported with the same file names for each project. Even if the project name is unique, the reports will fail to check in. **No warnings that the check in has failed are displayed!** This is especially critical when using the option *Project is in progress*, because there is no indication that the check in failed and the files are intended to be shown as checked out. Users may not see that their files were not versioned correctly.

There is no integrated BOM functionality with SOLIDWORKS Enterprise PDM

A BOM file is exported with the check in as a report. There is no way to create a BOM in SOLIDWORKS Enterprise PDM from the BOM file.

You cannot enter comments about the version during check in

The check in command does not give the user the ability to enter a comment about the version. SOLIDWORKS Electrical ignores the SOLIDWORKS Enterprise PDM setting that requires users to enter version comments.

Archive files that are not projects cannot be managed inside the SOLIDWORKS Enterprise PDM vault

A number of archive types cannot be managed in the SOLIDWORKS Enterprise PDM vault. These files cannot be checked out in SOLIDWORKS Electrical if they are manually added to the vault.

Manually checked out files cannot be opened in SOLIDWORKS Electrical; archives cannot be checked out if any of the report files are checked out to other users

If any of the files associated with an archive have been manually checked out, the Check Out command fails to open the archive from within SOLIDWORKS Electrical. All files for an archive must be checked into the vault to check out and open the archive in SOLIDWORKS Electrical.

SOLIDWORKS Enterprise PDM status is not taken from the status of the files in the vault

If users manually check in files from inside the vault, the SOLIDWORKS Enterprise PDM status column in the Projects Manager is not updated, and the status is not checked until the check in command is run.

Check out for archives is restricted to the subdirectory stated in the PDM configuration tab

Some companies work with multiple subdirectories for their electrical projects. In these companies, users have to manually update the subdirectory path before they can check out projects from sub directories that are siblings or parents to the current subdirectory path.

This creates a risk that new projects could be checked into the wrong vault location, since users will not see the current configuration location in the check in dialog box.

SOLIDWORKS Electrical can check files into the wrong SOLIDWORKS Enterprise PDM vault; check out for archives is restricted to the SOLIDWORKS Enterprise PDM vault stated in the PDM configuration tab

Users cannot work in multiple vaults at the same time. If users need to check out files from a different vault, they have to manually update the PDM configuration settings to point to the correct vault and subfolder each time they move to a different vault.

If a user does not update the configuration to the correct vault before check in, or attempts to check in multiple projects checked out from different vaults, the check in will check all projects into the current vault noted in the PDM configuration settings. If the project does not currently exist, this creates a new project directory structure. This situation creates a greater risk that projects could be checked into the wrong vault, since users do not see the current configuration location in the check in dialog box. **You should only use SOLIDWORKS Electrical with only one SOLIDWORKS Enterprise PDM vault.**

The workaround for this issue is to check out the projects from the incorrect vault, change the PDM configuration to the correct vault, and check in the projects again. Users will need to manually delete the projects from the incorrect vault.

In the case where a project of the same name existed in the vault and the files were overwritten, the user should rollback the files to the version before the last check in procedure to restore the correct project files.

Check in does not create any reference structures

When projects are checked into the SOLIDWORKS Enterprise PDM vault, no reference structures are created between the archive zip file and the project report files. If a report is moved from the project directory, you will not be able to identify the project to which it belongs.

Archives cannot be opened or checked out by dragging and dropping items from the File explorer into SOLIDWORKS Electrical

The drag and drop functionality for SOLIDWORKS Electrical runs the unarchive command by default, but the unarchive command is disabled for files inside the SOLIDWORKS Enterprise PDM vault. Drag and drop commands displays a warning that users should use the Check Out command from within SOLIDWORKS Electrical.

Updates to mapped variables in exported report files in the file vault are overwritten on subsequent check ins of the project

If a user updates the data for any mapped variable for report files, these changes are not propagated back into the file in the archive on check out. The changes are lost when the project is checked in again.

PDM functionality does not work with Workgroup PDM

There is no default data card for SOLIDWORKS Electrical archive files

There is no data card for the *tewzip* file format.

Users do not have access to reference structures during check in or check out

Users do not have a way to see what files are checked in or checked out during each process.

SOLIDWORKS Electrical does not report which users have projects checked out and on which computers the projects reside

SOLIDWORKS Electrical does not report which users have projects checked out and on which computers the projects reside. This situation can create issues because only the user who checked out the project can check it back in. Note, that the project must also be checked in on the same machine where it was checked out.