January, 2026

MWElectrical User Manual

Managing your electrical engineering designs (Schematics, PCB, and PCBA) within MechWorks PDM.

Table Of Contents

Introduction

MWElectrical is a powerful solution focused on incorporating an environment's electrical engineering designs within its MechWorks PDM environment. This integration marks a significant step forward in streamlining the design-to-production workflow, enabling engineering teams to work more efficiently, reduce errors, and accelerate time to market.

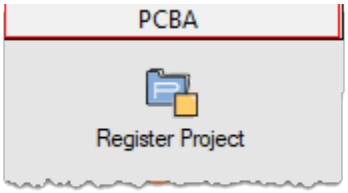
MWElectrical will allow users to easily:

- Register designated electrical projects into MechWorks PDM.
- Track specific electrical data, such as PCBA, PCB and SCHEMATIC numbers.
- Seamlessly associate and replace legacy electrical files with a current electrical project.
- Attach and manage electrical project related files, such as BOMs and technical documentation.
- Configure and require specific attachments for revision approval of electrical projects.
- Automatically archive attachments for previous revisions of electrical projects.

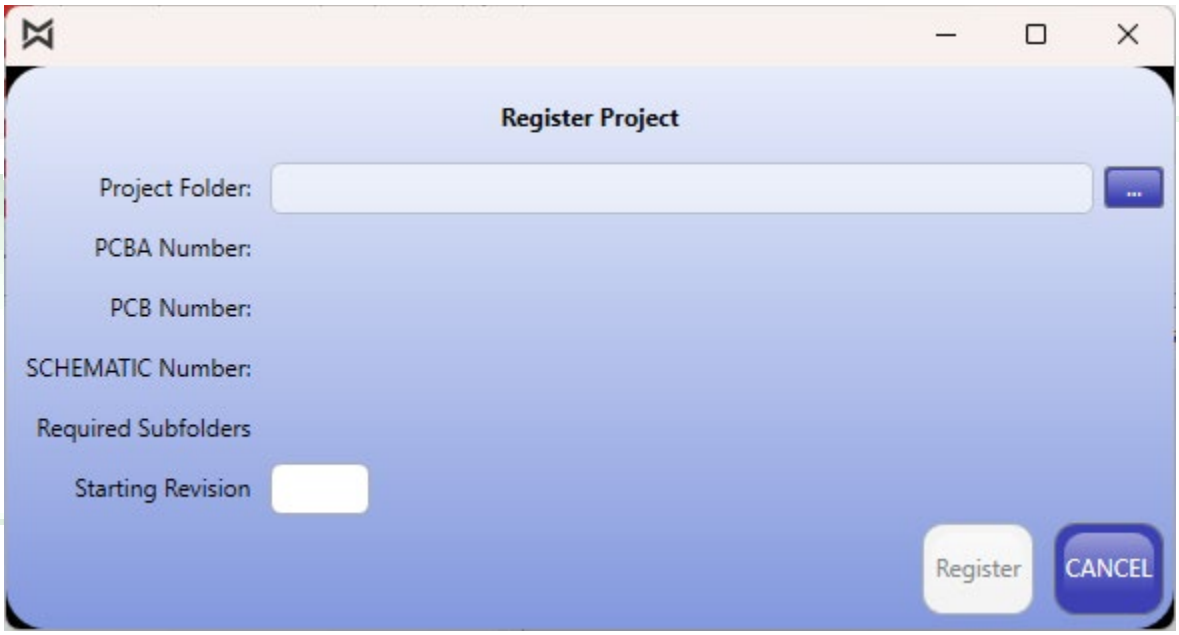
At its core, MWElectrical creates a seamless integration between MechWorks PDM and your electrical design platforms, ensuring that electrical design data is managed, searchable, and is properly revision controlled. Whether you're an electrical engineer, project manager, or part of a manufacturing team, this integration empowers you with real-time visibility, traceability, and control over your PCA, PCBA, and Schematic designs.

Register Project

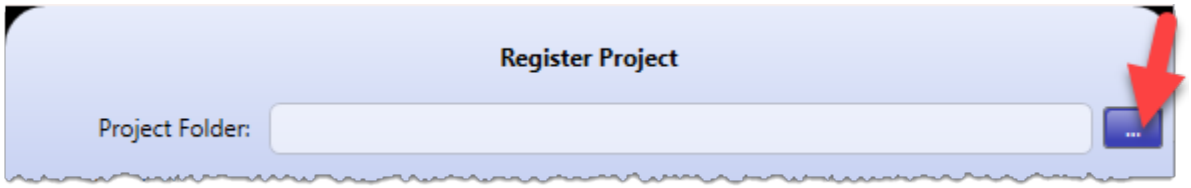
After the initial installation of the application, use its [**Register Project**] utility (seen below within the default **PCBA** shortcut bar) to define your organization's specific rules.



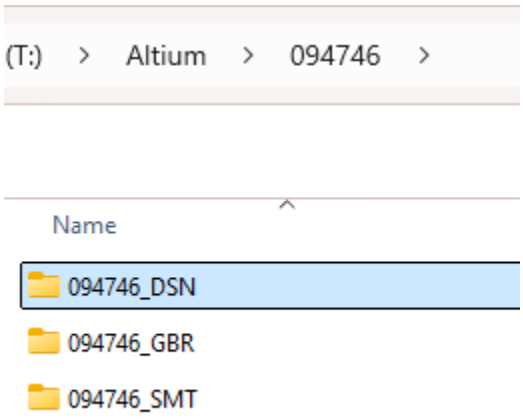
The **[Register Project]** tool displays the following dialog:



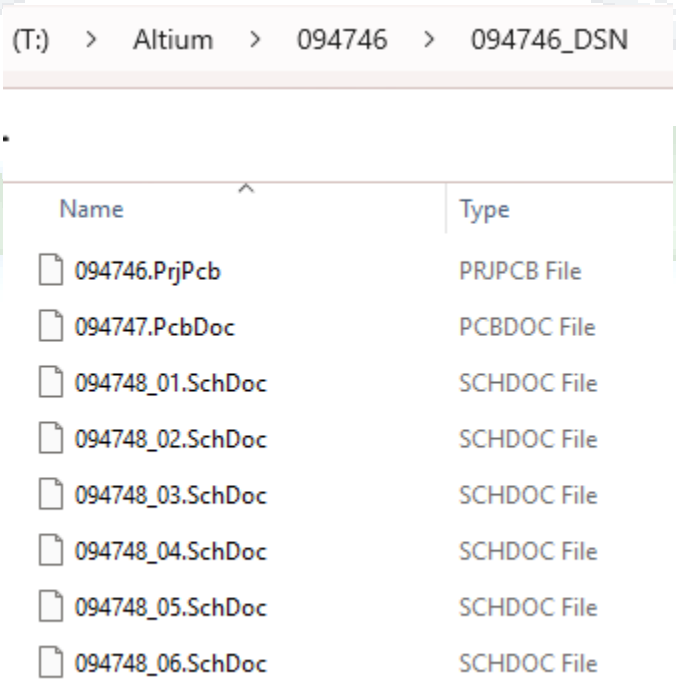
To register an electrical project, first browse to the working project's folder which should be located on a shared network drive:



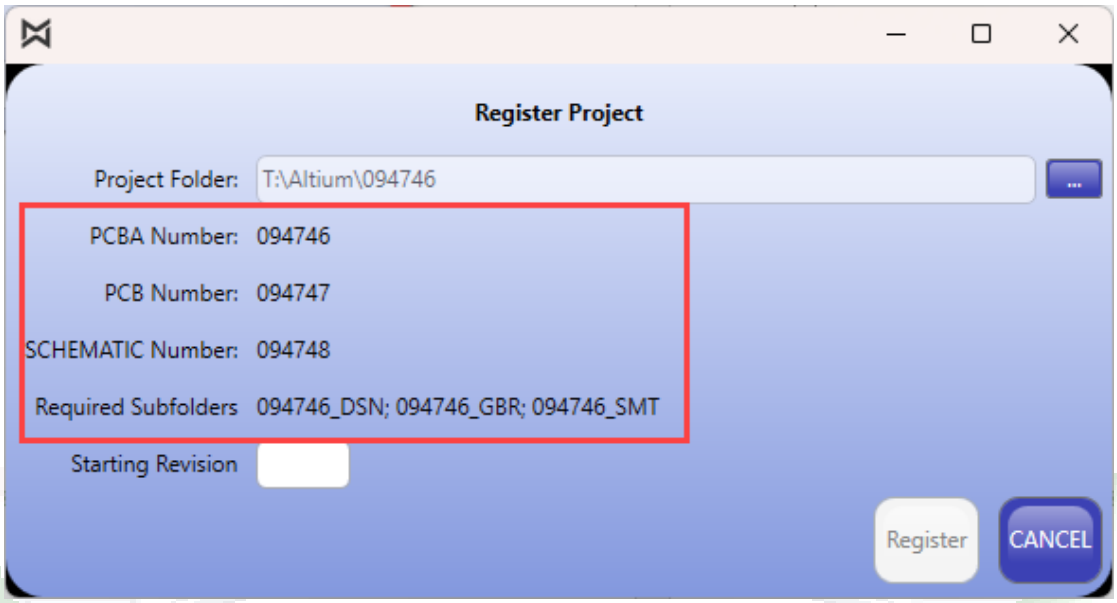
Referencing the following example, the parent electrical project **094746** exists in the **T:\Altium** folder with three designated sub-folders, "**_DSN**" for its board design files, "**_GBR**" for its related Gerber files for manufacturing, and "**_SMT**" for its related surface mount technology files (assembly files).



For this example, its "**_DSN**" sub folder (named **094746_DSN**) contains the following electrical design files:

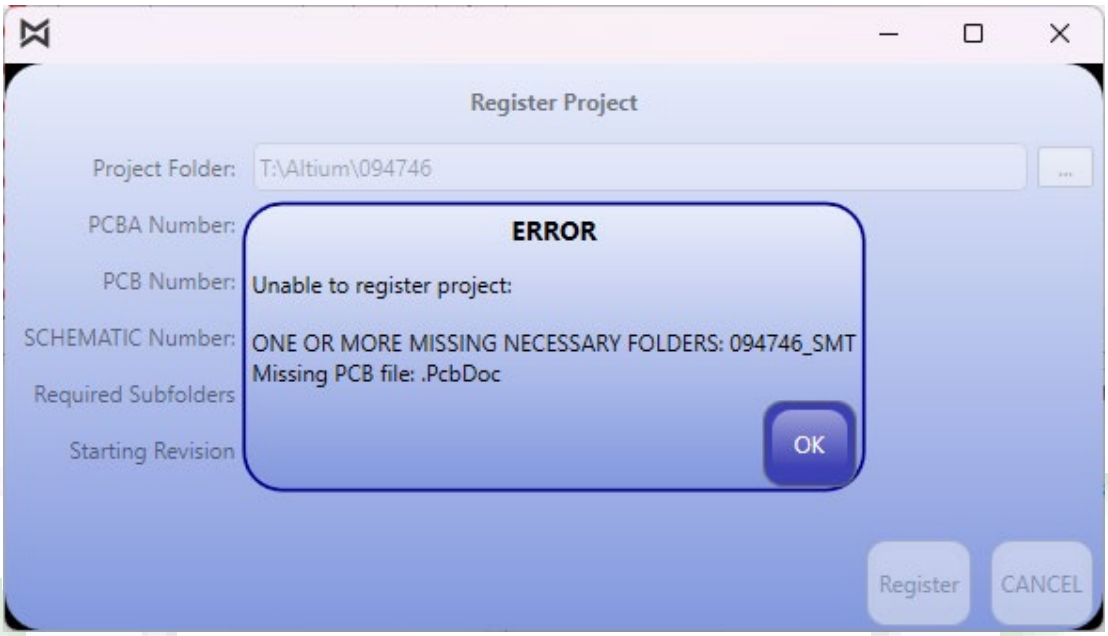


Selecting the **T:\Altium\094746** folder for this example allows the **[Register Project]** tool to analyze its folder structure, extracting its necessary data as follows:

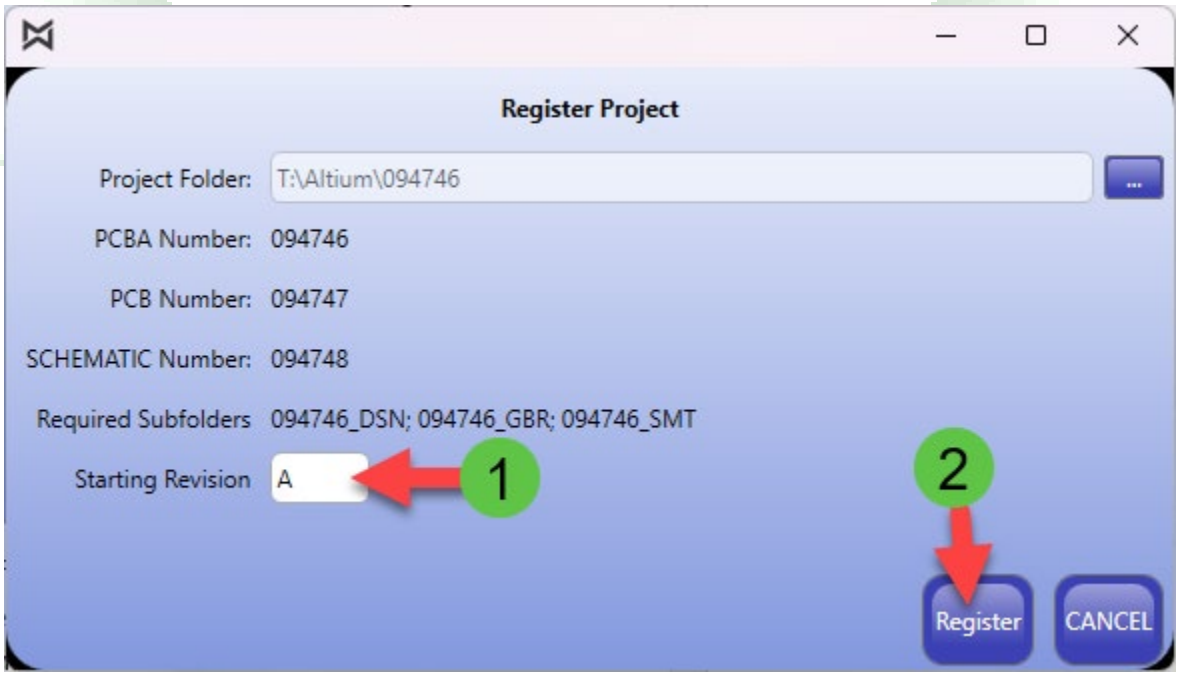


- ***PCBA Number***: This number is extracted from the selected "**_DSN**" sub-folder and its "**.PrjPcb**" file.
- ***PCB Number***: This number is extracted from the selected "**_DSN**" sub-folder and its "**.PcbDoc**" file
- ***SCHEMATIC Number***: This number is extracted from the selected "**_DSN**" sub-folder and its "**.SchDoc**" files.
    - NOTE The "**_DSN**", "**_GBR**", and "**_SMT**" sub-folders are required and must include the number of the project as a prefix as seen within the previous page.

NOTE: The three specific files and the three specific sub-folders MUST exist. If any of them are missing, then the **[Register Project]** utility will display an error dialog and it will not allow the project to be registered:

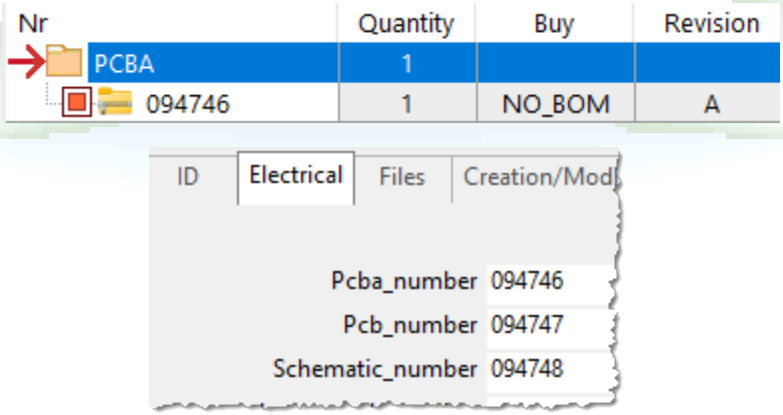


Finally, enter a starting revision level to allow the project to be registered using the assigned revision:

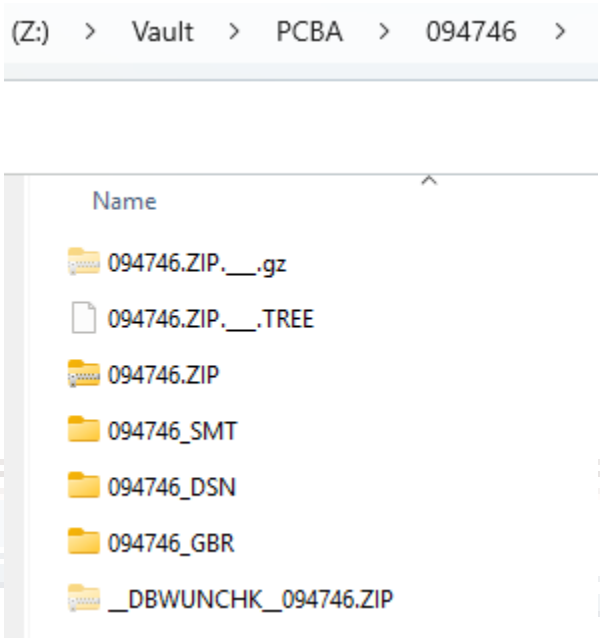


When the **"Register"** button is select, the tool will do the following:

1. Create a .ZIP file of the source project folder and its contents, "**094746.ZIP**" for this example.
2. Copy the created "**.ZIP**" file into a designated subfolder within the vault's structure for the managed electrical designs. For this example, the "**.ZIP**" file is copied into the "**Z:\Vault\PCBA\094746**" folder which is based on some additional administrative configurations.
    a. This root folder is configured by the administrator in the Options
3. Register the copied "**.ZIP**" file in the parent project in MechWorks PDM, the project "**PCBA**" in this example.
    a. This parent project is configured by the administrator in the Options
4. Update the metadata for the new record, setting the following fields within its record:
    a. PCBA Number
    b. PCB Number
    c. SCHEMATIC Number
    d. Revision
5. Executes a [**CHECK-OUT**] against the new record to the current user
    a. NOTE: Checking out the record extracts the .zip file into its parent folder (see the "Life Cycling Electrical Projects" chapter on page 21)



Finally, because the process automatically executed a [**CHECK-OUT**] of the electrical project files, the project folder within Windows File Explorer is automatically opened to the secured folder within the vault:

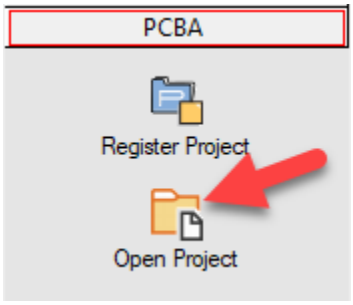


NOTE: Each electrical design must have a unique parent folder based on its determined ***PCBA Number***, so the folder structure within the vault could look similar the following:

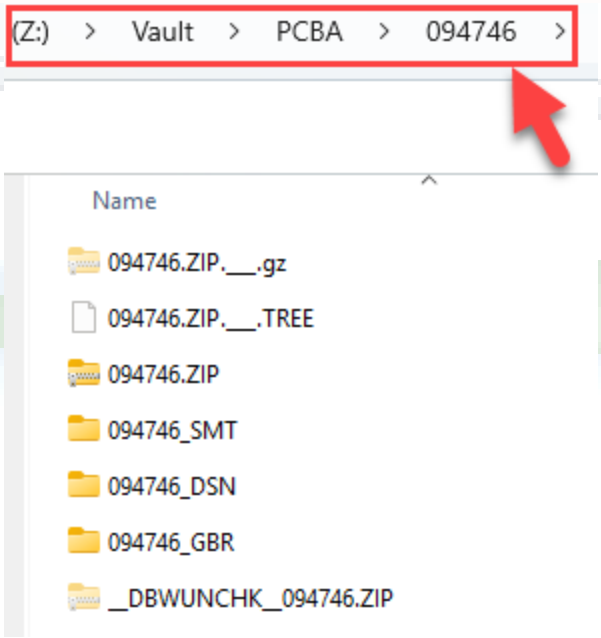
- Z:\Vault\PCBA\094746\
- Z:\Vault\PCBA\094747\
- Z:\Vault\PCBA\094748\
- Z:\Vault\PCBA\094749\
- Z:\Vault\PCBA\094750\

Open Project

Electrical users will use the [**Open Project**] utility (seen below within the default **PCBA** shortcut bar) to access its design files. The resulting action is directly impacted based on the [**STATE**] of the selected record:

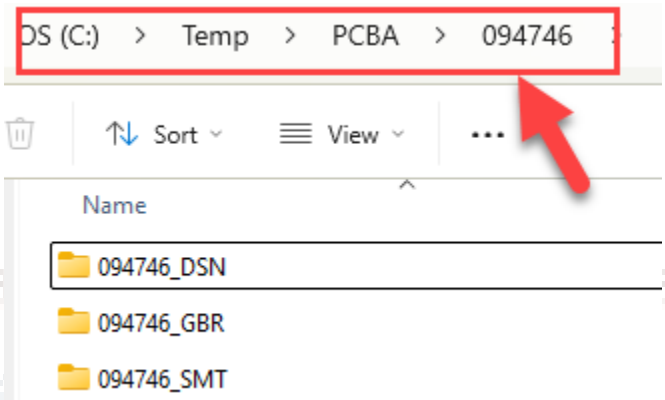


If the selected electrical project is [**CHECKED-OUT**] to the current user, then the **[Open Project]** utility will open the vault's project folder in Windows File Explorer:



If the selected electrical project is **not** [**CHECKED-OUT**] to the current user, then the **[Open Project]** tool will do the following:

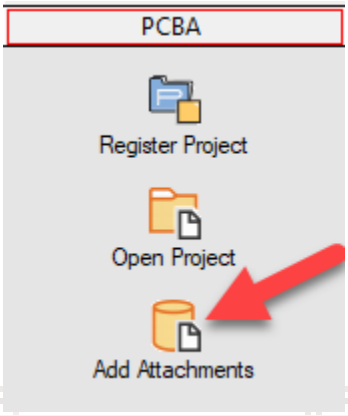
1. Copy the registered "**.ZIP**" file to the local viewing folder.
    a. NOTE: The local viewing folder is configured by the administrator in the Options
2. Extract the copied "**.ZIP**" file into the local viewing folder
3. Open Windows File Explorer within the local viewing folder created by extracting the "**.ZIP**" file:



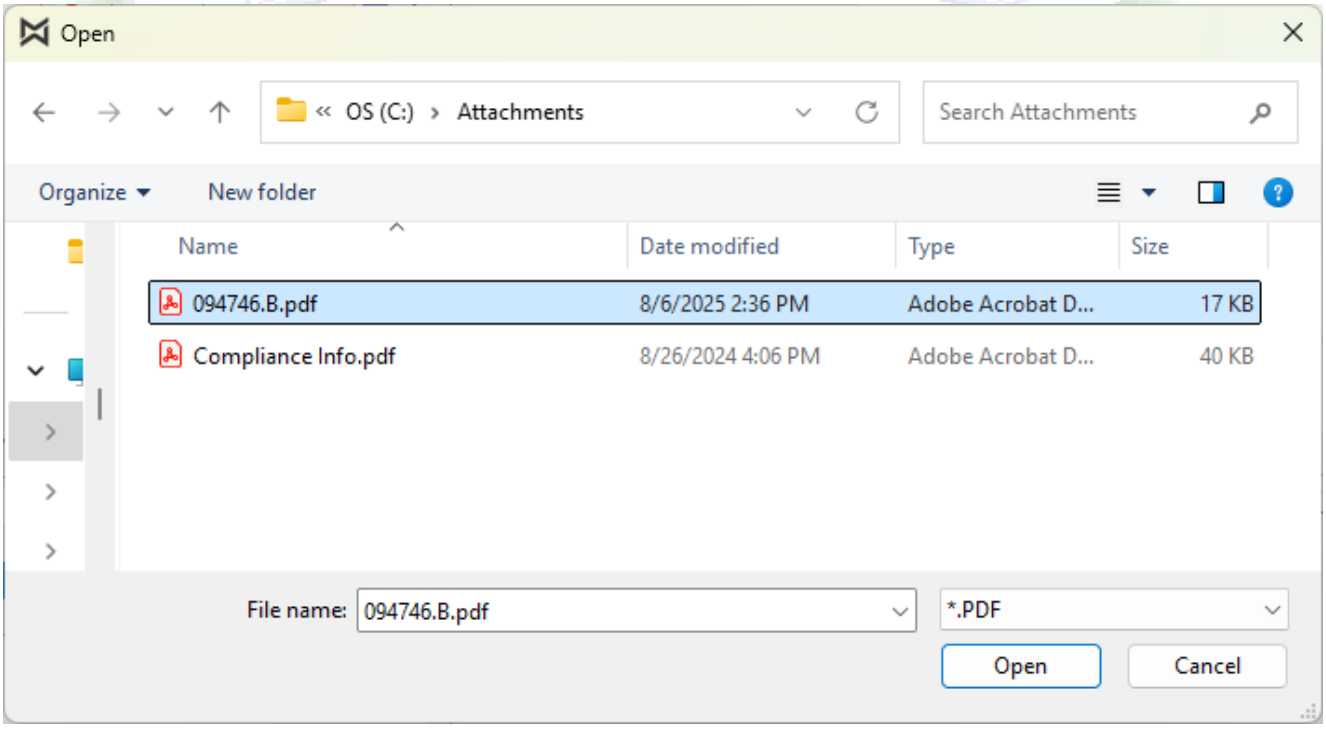
NOTE: The **[Clean Up Local Cache]** utility can be used to delete all the files in this local viewing folder (as noted later within this manual). Obviously if the user has files opened within applications when they attempt to use the [**Clean Up Local Cache**] utility, then Windows will display a dialog letting the user know that the file is locked.

Add Attachments

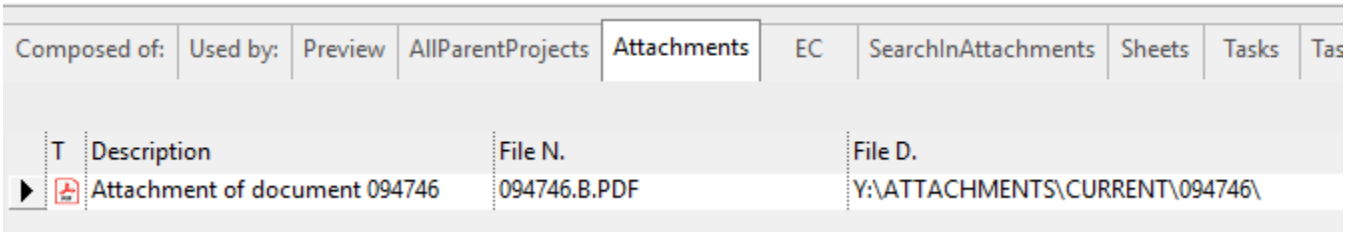
The default **PCBA** shortcut bar also includes a **[Add Attachments]** utility that is specific for the electrical project design process:



Using the noted [**Add Attachments**] utility will display a dialog similar to the following, allowing the user to select the files to become attachments within MechWorks PDM for the selected electrical project record. The



Selecting its [**Open**] command will copy the selected attachment(s) to the assigned Attachment Destination Folder (configured by the administrator within the Options) and attach them to the selected electrical project:

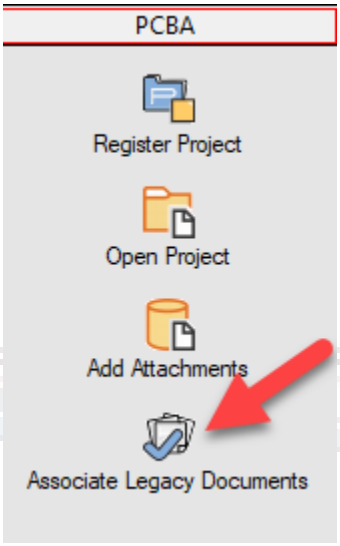


NOTE: In addition to the Attachment Destination Folder, the administrator can use the options to configure which types of documents may be added as attachments as well as the source folder to initialize the Select Attachments dialog from.

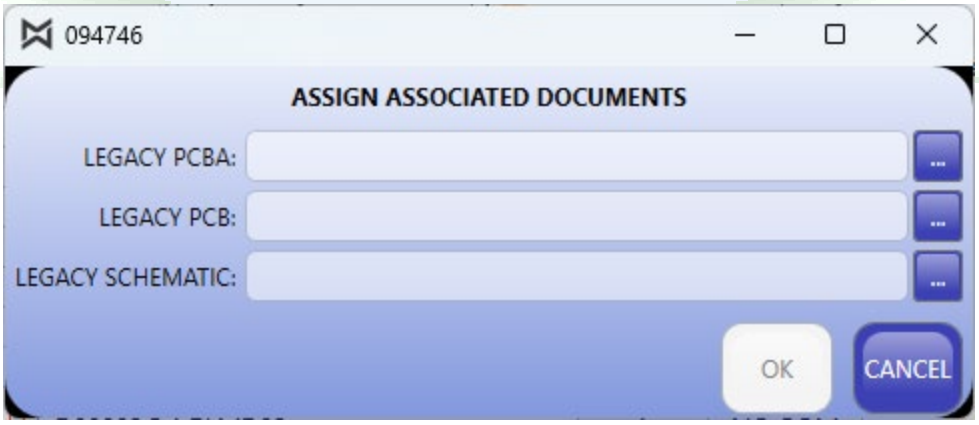
Further NOTE: A custom script is available for more applying customer specific rules to be applied for determining a complex destination folder structure for these attachments within MechWorks PDM. For example, some customers need the current active attachment files to be relocated to an archived folder structure when newer attachments are assigned.

Associate Legacy Documents

The default **PCBA** shortcut bar also includes an **[Associate Legacy Documents]** utility for supporting the process of replacing registered legacy files with new electrical design files (perhaps the electrical CAD application has changed):



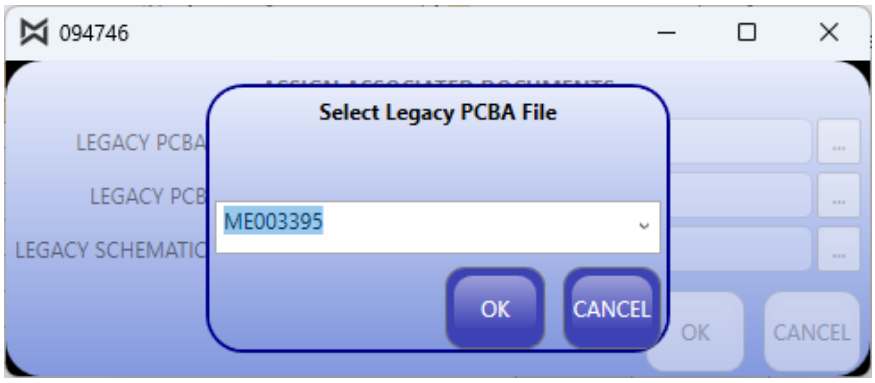
The **[Associate Legacy Documents]** utility will display a dialog for identifying registered legacy documents to be replaced with the selected electrical project:



NOTE: The types of documents that can be considered as legacy documents are controlled by the administrator in the Options. For this example, registered "**.DWG**" files can be identified as legacy documents.

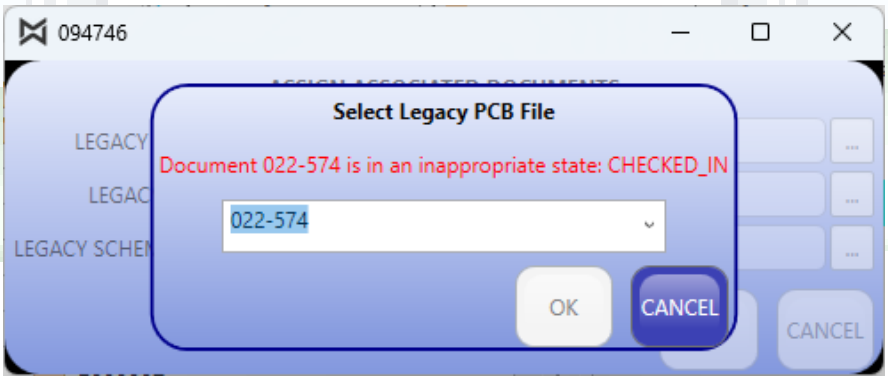
FURTHER NOTE: The registered legacy documents must be in a [**RELEASED**] state before being associated with a current electrical project, as this procedure will automatically force them to become [**OBSOLETE**].

Selecting the Browse button next to one of the three Legacy Numbers will allow the user to identify a registered Legacy document:

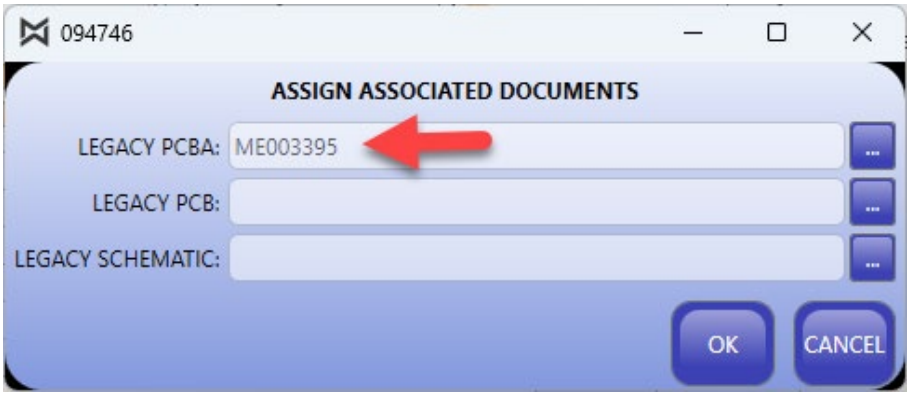


Users can type the [**ID**] of the registered Legacy Document, or they can use its drop-down menu to select from a list of defined matching Legacy Documents (based on some Options).

If the identified Legacy Document is not in a [**RELEASED**] state, the dialog will display an error message notifying the user that the document must be [**RELEASED**] before it can be set as a Legacy Document:



If the selected Legacy Document is in a [**RELEASED**] state, it will be assigned within the selection dialog:



The user can identify up to three Legacy Documents, then continue by selecting the [**OK**] button.

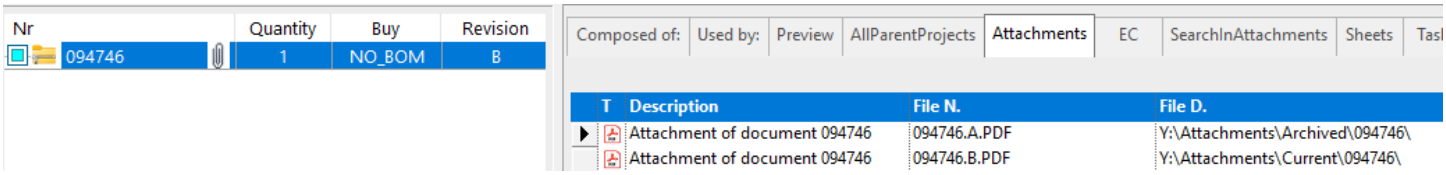
Please note that there is **not** an [**UNDO**] command for this procedure and that it is the user's responsibility to ensure that they are properly identifying the legacy registered documents.

When the [**OK**] button is selected, the application will begin the process of associating each selected registered Legacy Document, which involves the following:

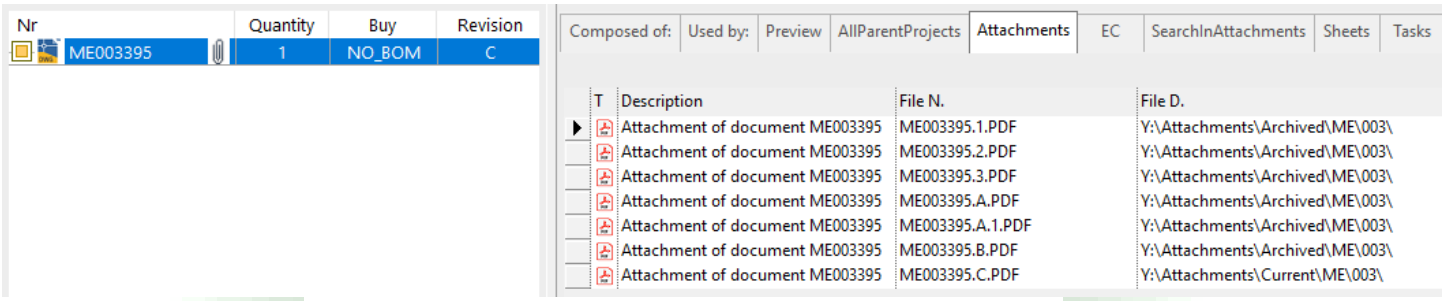
- Update the selected electrical project's record by populating its fields with the assigned [**LEGACY PCBA**], [**LEGACY PCB**] and [**LEGACY SCHEMATIC**] assigned values.
- Update the attachments of the registered Legacy Document(s), if they exist, to be assigned as attachments to the new/selected electrical project.
- Place the registered Legacy Document(s) in an [**OBSOLETE**] state
- Optionally, update the [**DESCRIPTION**] of the registered Legacy Document(s) for cross reference purposes.
    - The format for the [**DESCRIPTION**] is controlled by the administrator in the Options
- Optionally, a custom script can be called to take further action when associating the legacy document(s), such as moving the legacy attachments to an archive folder.

Additionally, if the basic MechWorks PDM options are configured accordingly (by including these fields during search processing), searching for a legacy document will display the new/current electrical project. (See the Installation manual for more information about configuring the MechWorks PDM options this way.)

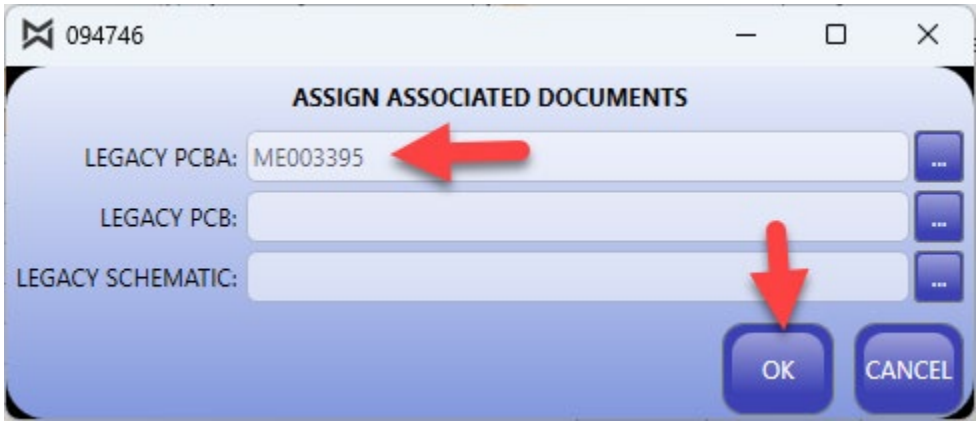
For example, the new/current electrical project of **094746** is currently [**CHECKED-IN**] at [**REVISION**] B, with two associated attachments (as seen within the following figure):



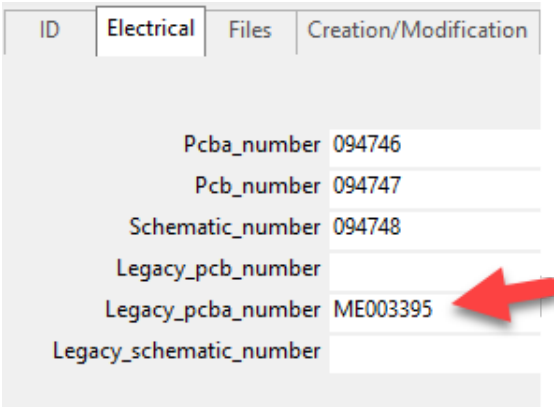
The legacy registered document, **ME003395** is currently [**RELEASED**] at revision **C**, with several attachments (as seen within the following figure):



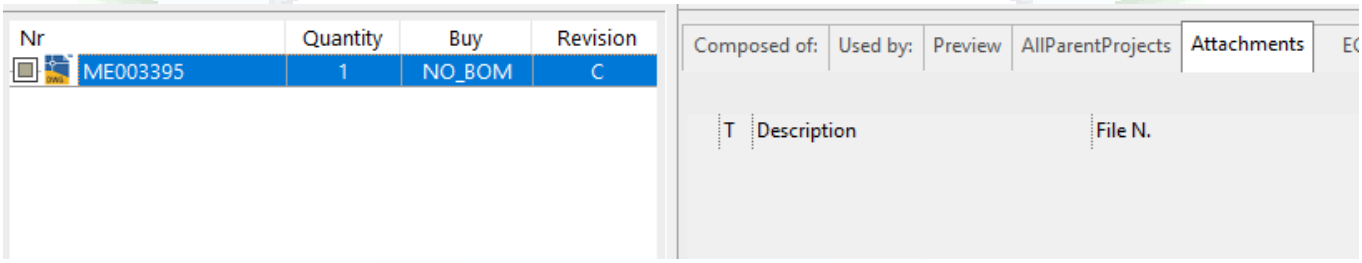
The user then uses the **[Associate Legacy Documents]** utility to associate the legacy **ME003395** record as the [**LEGACY PCBA**] document to the new electrical project record of **094746** (notice the upper-left corner of the following dialog):



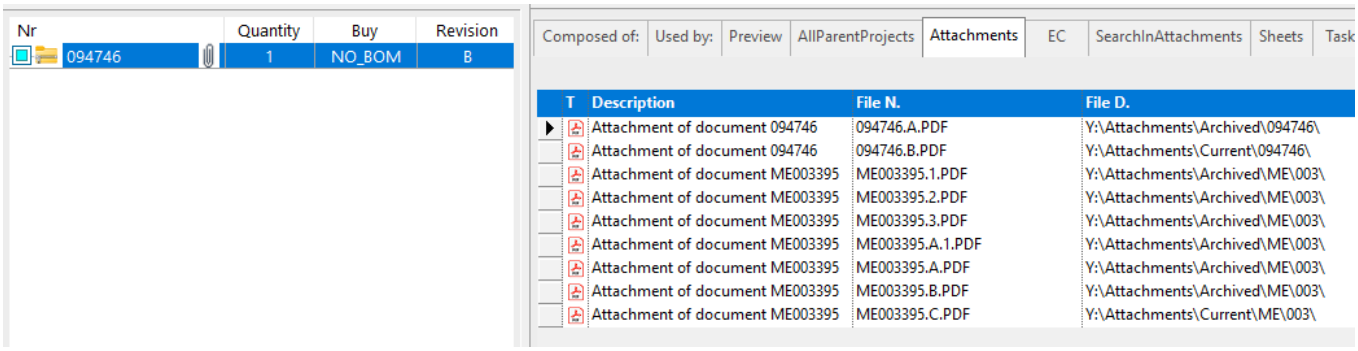
After the application finishes associating the legacy **ME003395** document to the new/current electrical design project record of **094746**, the [**LEGACY PCBA**] number within the **094746** record has been updated accordingly:



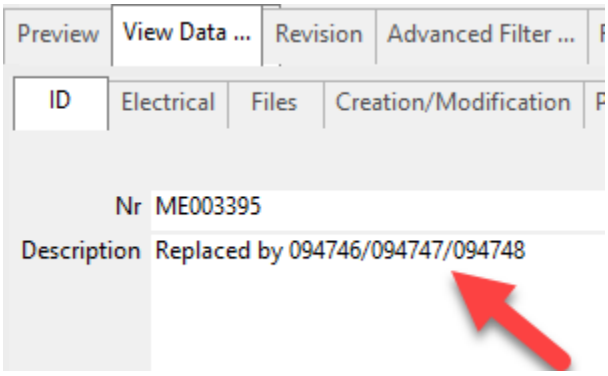
The [**STATE**] of the legacy **ME003395** record has been changed to [**OBSOLETE**] and its attachments have been reassigned:



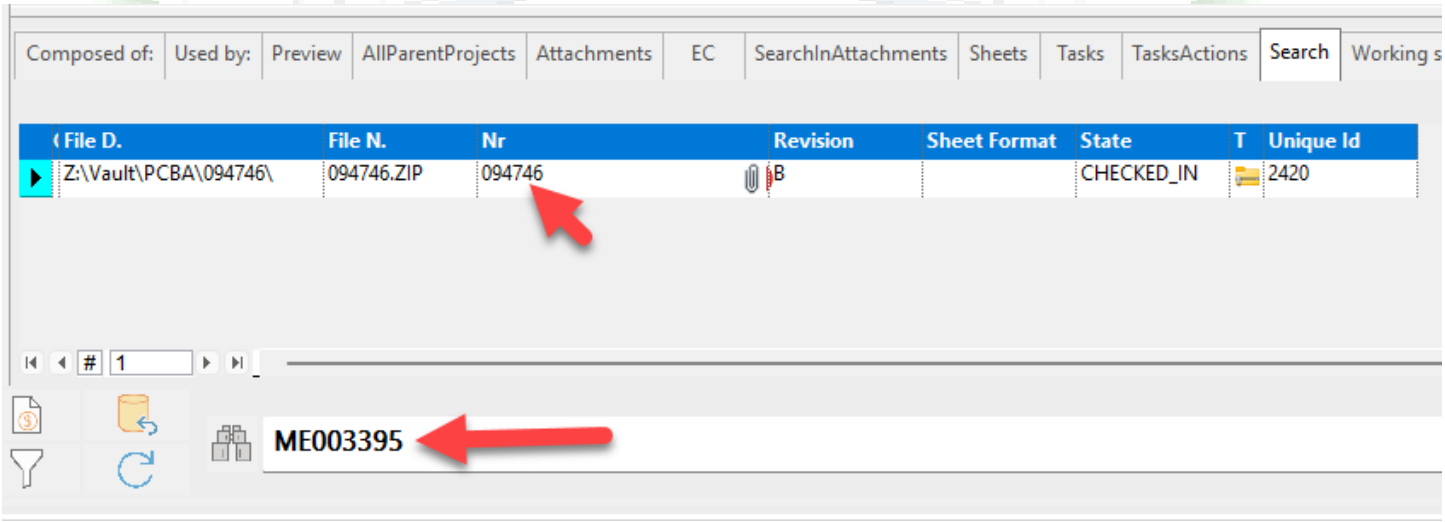
The attachments that were assigned to the legacy [**ME003395**] record have been automatically assigned to the **094746** record:



Based on the configured Options, the description of legacy **ME003395** record has been updated to reflect the new/replacement **PCBA**, **PCB**, and **SCHEMATIC** numbers:

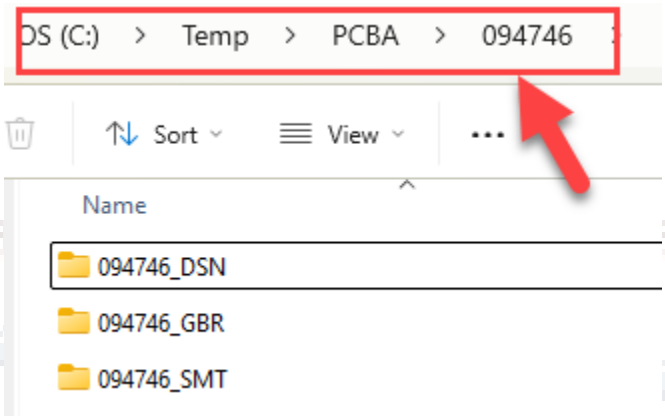


Finally, searching for the legacy **ME003395** value results in the new design file, **094746** being returned so users know that the electrical design has been migrated from an older CAD format to the newer CAD format:

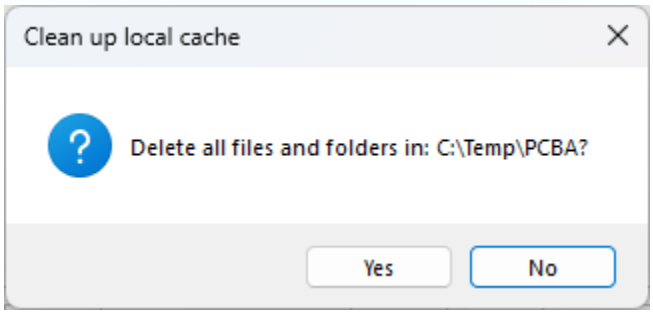


Clean Up Local Cache

As noted on a previous page, the **[Open Project]** utility will extract the electrical project's files to a local viewing folder if the selected electrical project is not [**CHECKED-OUT**] to the current user. The location for the local viewing folder is defined by the administrator using the Options. It will be something similar to "**C:\Temp\PCBA**" as seen within the following figure so it becomes a locally cached structure for the user. Referencing the following figure, the user selected the **094746** electrical record to be opened using the [**Open Project]** utility while it was not [**CHECKED-OUT**] to them, resulting in the following extracted structure:



The local cache location for viewing of "**C:\Temp\PCBA**" in this example is configured by the administrator in the Options. It is the administrative Option that addresses the assigned **[Clean Up Local Cache]** location, so when a user uses the [**Clean Up Local Cache**] utility, it will displays a confirmation dialog:



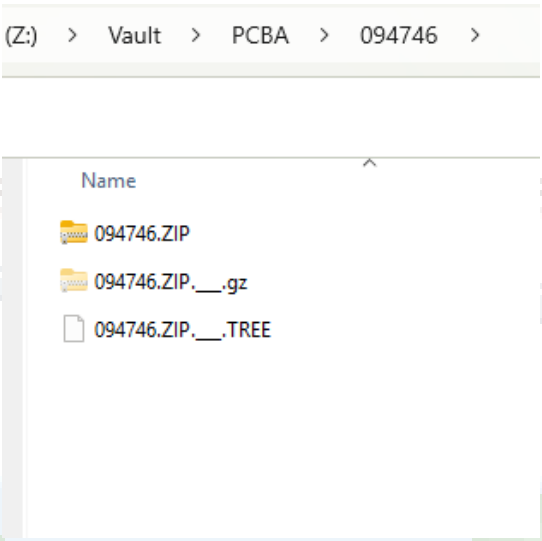
If the user selects [**Yes**], then all files and folders within the noted local cache folder structure that are NOT opened within an application will be automatically deleted. For this example, everything within the "**C:\Temp\PCBA**" folder will be deleted.

Life Cycling Electrical Projects

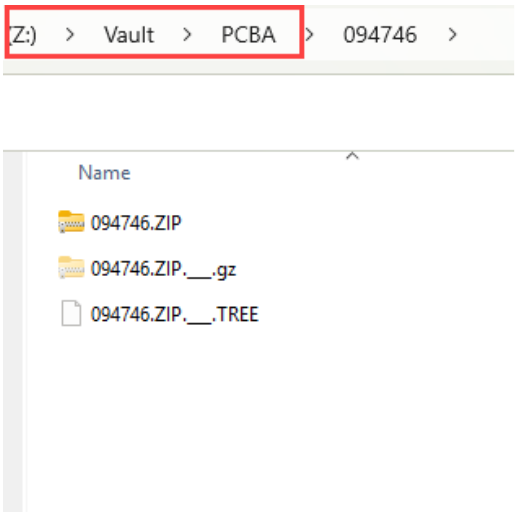
Checking In and Out:

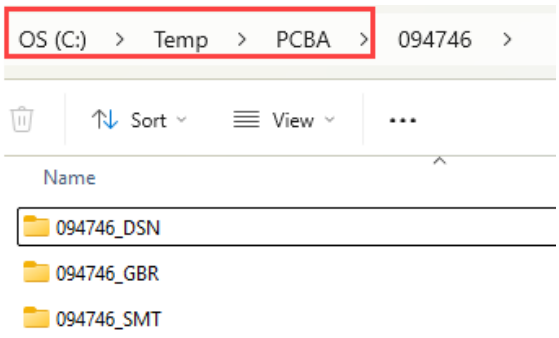
NOTE: The following functionality is dependent on using the OnCheckOut.LST and OnBeforeCheckIn.LST scripts. For more information about configuring these scripts, see the Installation manual.

Each electrical project is registered as a single "**.ZIP**" file, containing all the necessary files for the electrical project. When the "**.ZIP**" file is in a [**CHECKED_IN**] or a [**RELEASED**] state, the ".**ZIP**" file is NOT extracted within the vault folder structure, and all its electrical design files are contained within the registered "**.ZIP**" file:

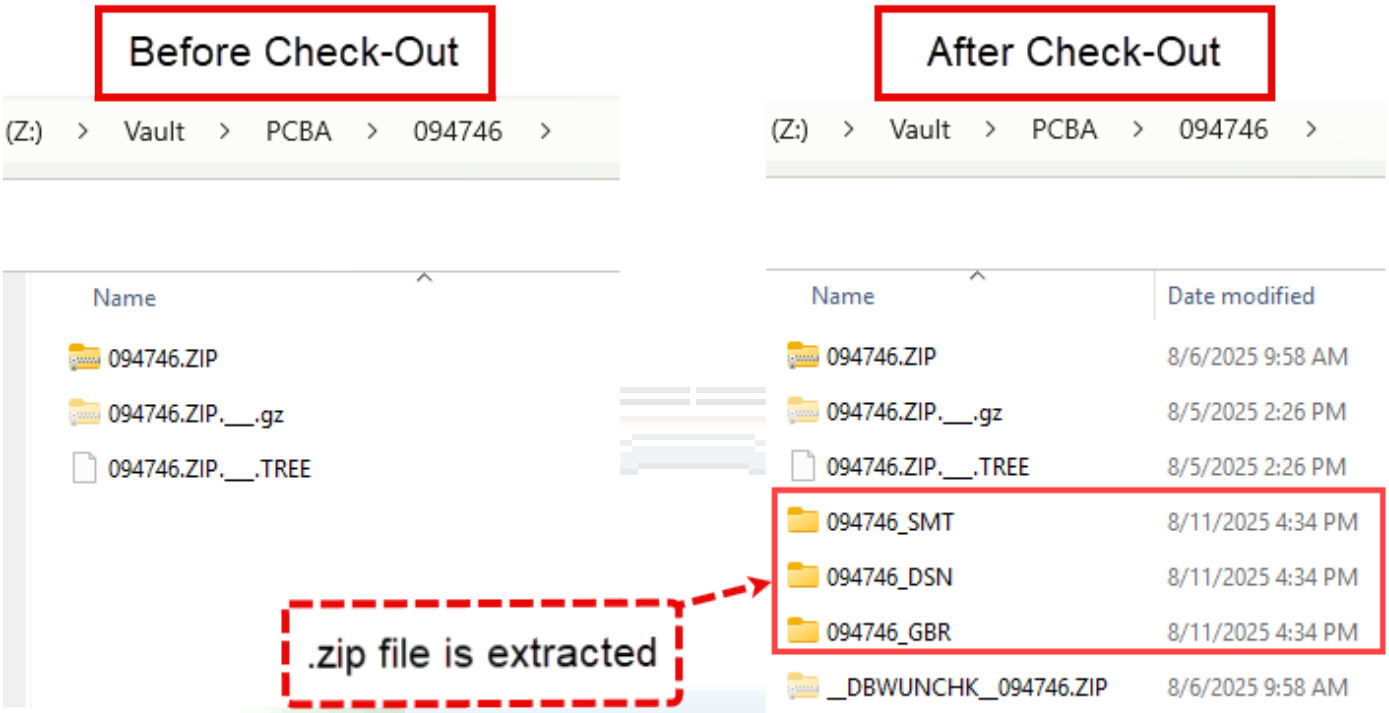


When the "**.ZIP**" file is in a [**CHECKED_IN**] or a [**RELEASED**] state, using the **[Open Project]** utility will extract the ".**ZIP**" file to the local cache folder (NOT within the network vault folder):



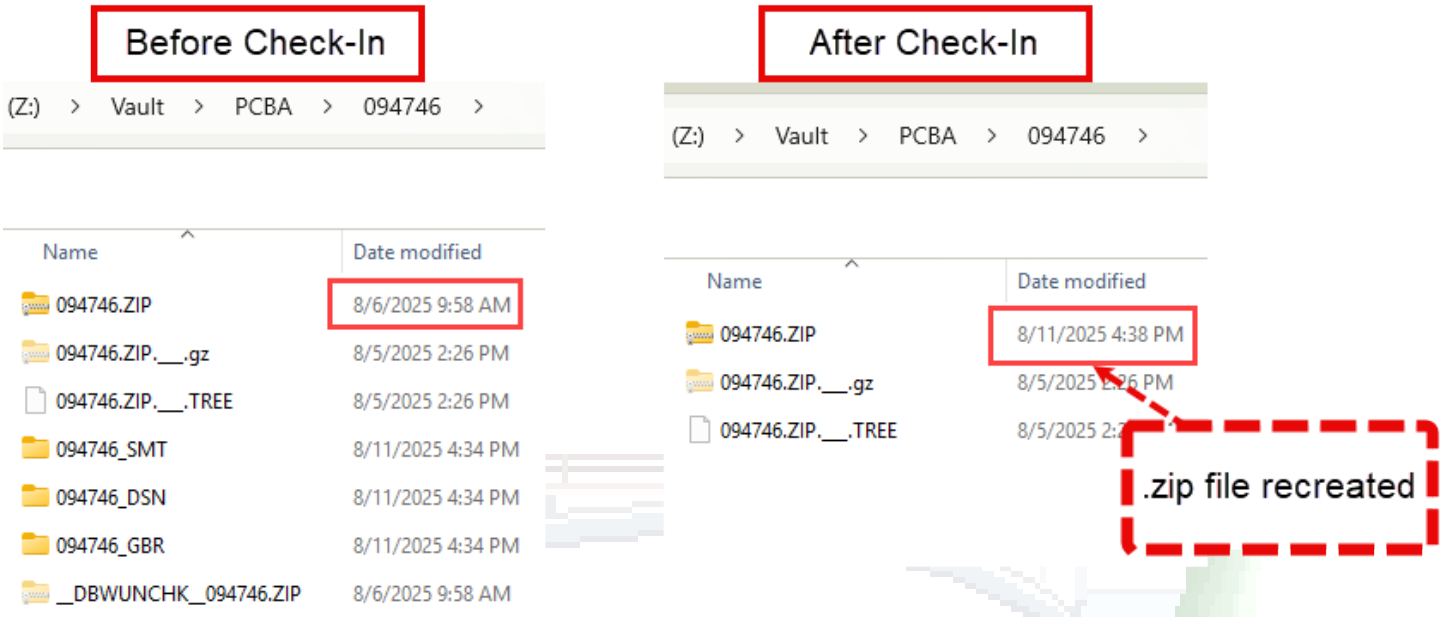


During a Check-Out action, the "**.ZIP**" file is automatically extracted in the network vault location for the user that is executing the Check-Out action:



When the user who has the electrical project [**CHECKED-OUT**] uses the **[Open Project]** utility, that user will be directed to the network location where the extracted files are. If any other user opens the project using the **[Open Project]** utility, the files will be copied to that user's local cache folder and the electrical project will be opened from the local cache location. In this way, only the user with the electrical project [CHECKED-OUT] will be modifying the electrical files in the network vault.

During a Check-In operation, the "**.ZIP**" file is recreated from the files in the vault location. In this way, all modified files in the vault location are included in the updated .zip file:



Approving:

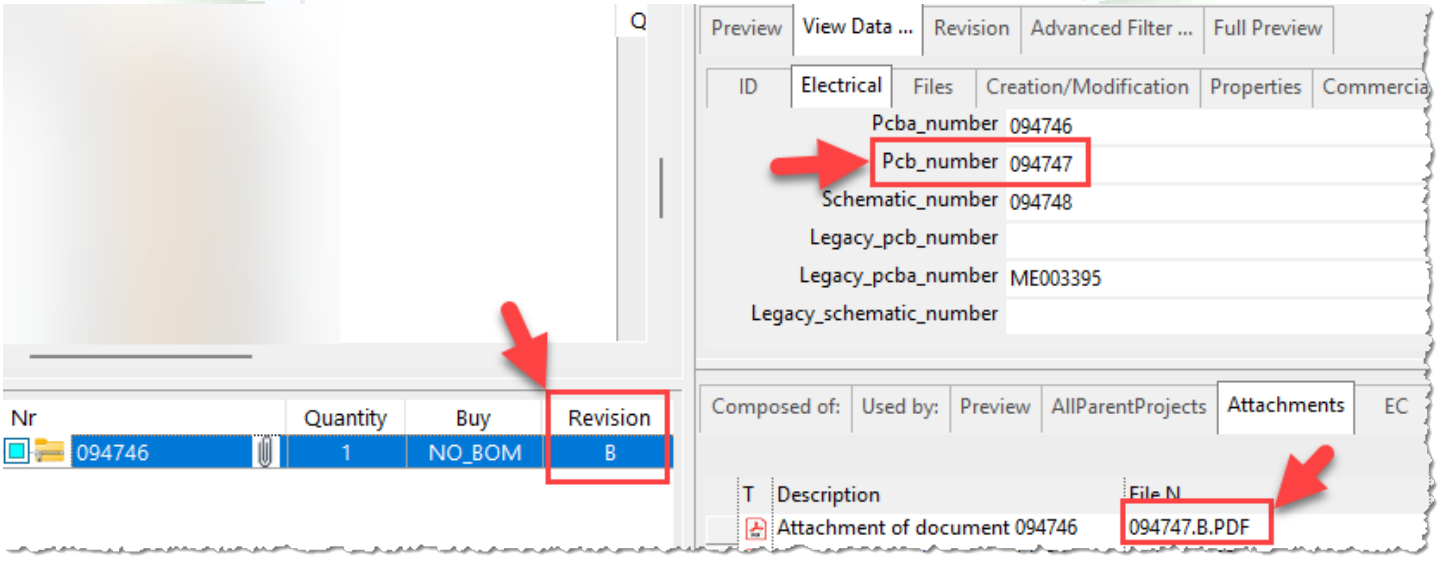
NOTE: The following functionality is dependent on using the OnOkRev.LST script. For more information about configuring this script, see the Installation manual.

Optionally, the administrator can configure required attachments for approving revisions of electrical projects. For more information about how to configure required attachments, please see the Administration manual.

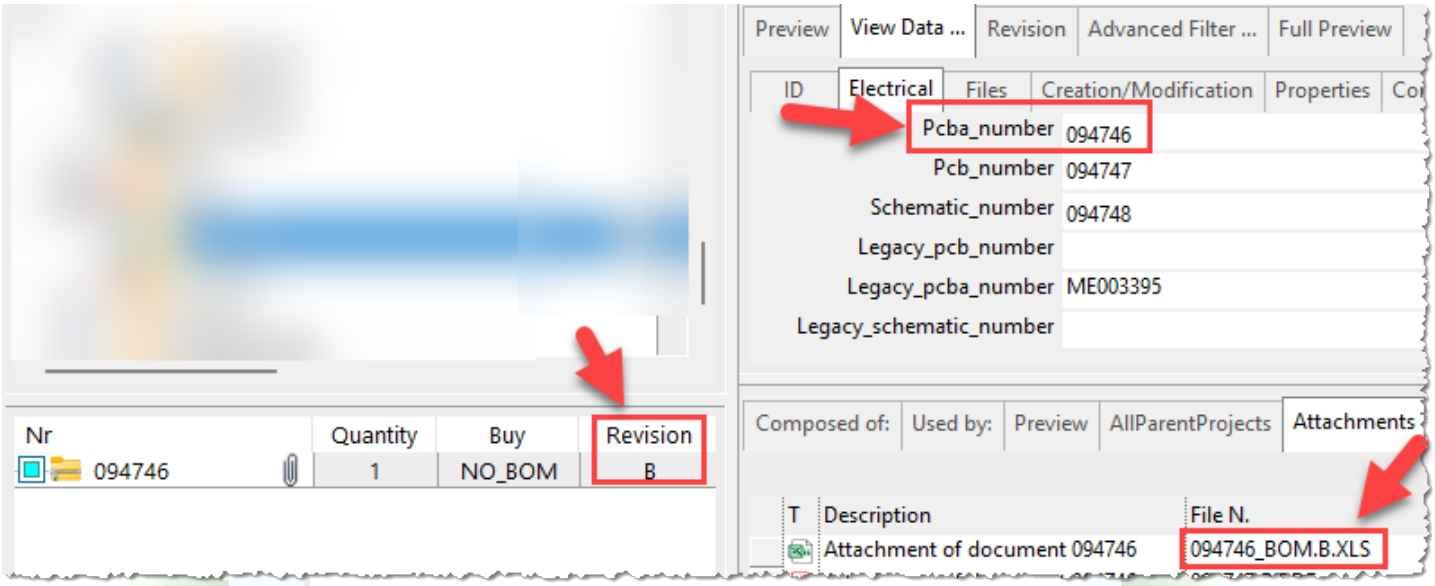
For this example, the administrator has configured the environment to require 3 attachments before an electrical project can be approved:

- One .pdf file of the PCB.
- One .pdf file of the SCHEMATIC.
- One .xls BOM of the PCBA.

For non-BOM attachments, the name of the file will be determined by the assigned number (such as the [**PCB NUMBER**] for the **PCB** .pdf) along with the current [**REVISION**] assignment. For the following example, at [**REVISION**] **B**, electrical project **094746** must have an attachment named "**094747.B.PDF**" to match the combination of its assigned [**PCB NUMBER**] and its assigned [**REVISION**] value:



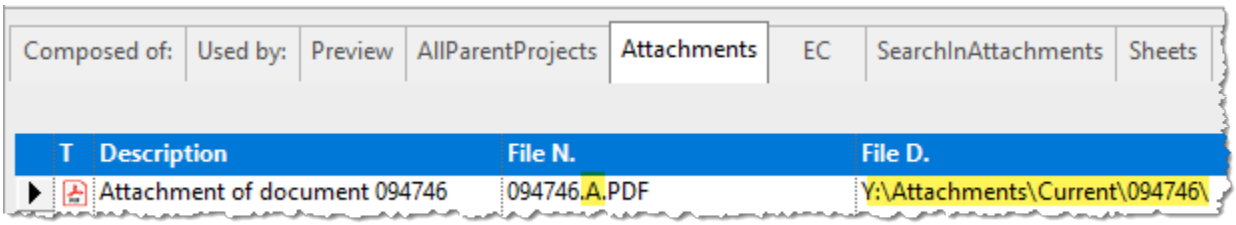
For any BOM attachments, the name of the file must be the associated number (such as the [**PCBA NUMBER**]) followed by "**_BOM**", along with the assigned [**REVISION**] value. For the following example, at [**REVISION**] **B**, electrical project **094746** must have an attachment called "**094746_BOM.B.xls**" to match the PCBA number and revision level:



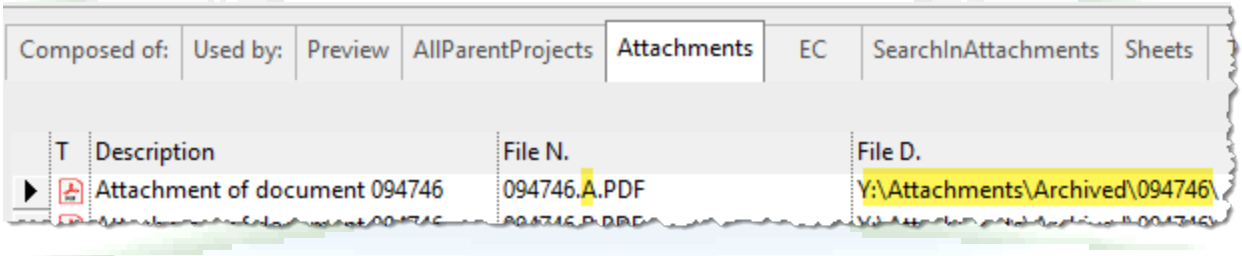
If a user attempts to [**APPROVE**] an electrical project which is missing one or more required attachments, an error message will be displayed, and the approval process will not be allowed to continue:

Once all required attachments are attached, the electrical project will be able to be approved. When it is approved, any attachments for previously approved revisions will be moved from the current attachments folder to the archived attachments folder.

For the following example, before approving the electrical project which is currently in a [**CHECKED-IN**] [**STATE**] at [**REVISION**] **B**, its previously assigned attachments of the previous revision (**A**) are in the "active attachments" folder highlighted in yellow as "**Y:\Attachments\Current\094746**":



The action of approving revision **B**, moves the previous attachments of revision (**A**) automatically to the designated "archive" folder highlighted in yellow as "**Y:\Attachments\Archive\094746**":



NOTE: The current and archived locations are configured by the administrator using the Options.