cadence

OrCAD Engineering Data Management User Guide

Product Version 8.5.0

For Cadence/OrCAD 25.1, 24.1, 23.1

May 2026

Table of Contents

1 Introduction 1
1.1 Getting Started 2
1.2 Engineering Data Management 6
1.3 Using Engineering Data Management 6
2 User Settings and Customizing Views 9
2.1 EDM Settings 9
2.2 Home Page 10
2.3 Profile 13
2.3.1 Local Workspace 17
2.4 Customizing Views 18
3 Administration Guide 21
3.1 Manage Templates 21
3.1.1 Manage Project Templates 22
3.1.2 Manage BOM Templates 29
Connection to CIP 30
DSN BOM Validation 33
Configure BOM Template Format 34
3.1.3 Manage Release Templates 41
3.2 Manage User Accounts, Groups, and Roles 44
3.2.1 Adding and Modifying Roles 45
Default Roles 46
Role Permissions 48
3.2.2 Adding and Modifying Users 53
3.2.3 Adding and Modifying User Groups 56
3.3 Configuration 58
3.3.1 General Configuration 59
Customizable Fields 60
Email Notification 63
Build BOM Search 65
Show EDMClient Download Link 65
Warning Durations 66
3.3.2 Capture Configuration 66
Schematic Title Block Settings 67
Promoted Schematic Symbol Library Locations 70
Netlist Generation Configuration 72
Occurrence Properties 73
OrCAD Capture Preferences 74
3.3.3 PCB Editor Configuration 74
Board Title Block Field Mappings 75
Promoted Footprint and Step Model Locations 78
Rules for Footprint Promote and Delete 81
3.4 Vault Setup 82
3.4.1 Repository Server Type 83

3.4.2 File System Vault Setting ... 85
3.4.3 SVN Vault settings ... 86
3.4.4 Quick SVN Setup Steps for VisualSVN ... 90

4 Project Overview (All Users) 95

4.1 Organization of Projects ... 96
4.2 View Projects (Dashboard) ... 97
4.3 View Project Details ... 100
4.3.1 Capture Project View ... 102
Capture Designs in a Capture Project ... 107
4.3.2 Footprint Project View ... 109
4.3.3 Schematic Symbol Project View ... 118
4.3.4 General Project View ... 120
4.3.5 Team Design Detail View ... 125
4.3.6 Team Library View ... 131
4.4 Local State Definitions ... 136

5 Manage Projects (Project Leader Tasks) 141

5.1 Project Structure and Options ... 142
5.2 Add and Edit Projects ... 143
5.2.1 Add Project ... 144
5.2.2 Edit Project and Group ... 150
5.3 User Assignments ... 156
5.4 Add, Edit, and Delete Folders ... 158
5.4.1 Add Folder ... 159
5.4.2 Add Schematic Symbol Library Folder ... 164
5.4.3 Edit Folder ... 165
5.4.4 Delete Folder and File ... 171
5.4.5 Edit File Properties ... 175
5.5 Activate/Baseline Project ... 179
5.6 Import Design and Board Files ... 181
5.7 Manage Footprint Projects ... 185
5.7.1 Import Footprints and Step Models ... 185
Sync DRA to PSM files ... 191
5.7.2 Promote Footprints and Step Models ... 191
5.7.3 Delete Footprint and Step Model Files ... 193
5.8 Manage Team Design Folders ... 195
5.8.1 Add New Pages to a Flat Design Folder ... 196
5.8.2 Add New Schematics and Pages to Hierarchical Design Folder ... 200
5.8.3 Import Schematics and Pages ... 205
5.8.4 Edit Schematic/Group and Page properties ... 208
5.8.5 Delete Schematic/Group and Page ... 212
5.8.6 Freeze Team Designs ... 213
5.9 Manage Team Library Folders ... 215
5.9.1 Import Symbols ... 216
5.9.2 Edit Symbols ... 218
5.9.3 Delete Symbols ... 220
5.9.4 Promote Symbols ... 221

5.10 Delete Project Revision ... 223
5.11 Copy Project to New Revision/Name/Group ... 225
5.12 Package For Release ... 229
5.12.1 Create a Release Package ... 229
5.12.2 Review Release Package Contents ... 233

6 Project, Folder and File History 241

6.1 File and Folder Icons ... 243
6.2 History Version Log ... 245
6.3 File Differences ... 247
6.4 Team Design History ... 250
6.5 History From Versions Prior to 5.0 ... 252

7 Check Out 253

7.1 Check Out Files ... 254
7.2 Check Out Managed Capture Files ... 256
7.3 Check Out Footprint Files ... 259
7.4 Check Out Team Design Pages ... 261
7.5 Frozen Design Check Out ... 264
7.6 Check Out Team Library Symbols ... 265
7.7 Cancel Check Out ... 267

8 Check In 271

8.1 Check In Files and Folders ... 271
8.2 Check In Managed Design Files ... 280
8.3 Check In Managed Board Files ... 284
8.4 Check in Footprint Files ... 288
8.5 Check In Team Design Pages ... 291
8.6 Check In Team Library Symbols ... 295

9 Search 299

9.1 BOM Search ... 299
9.2 Advanced Search ... 301
9.3 Saving Search Criteria ... 304

10 Reference Designator Management For Hierarchical Team Designs 306

10.1 Reference Designators in Hierarchical Designs ... 307
10.2 EDM RefDes Commands ... 308
10.2.1 Clear ... 309
10.2.2 Restore Lock ... 310
10.2.3 Find Duplicates ... 311
10.2.4 Remove Duplicates ... 312

10.2.5 Apply Lock RefDes ... 313
10.2.6 Compare LockRefDes ... 314
10.2.7 Clear Compare ... 315

11 Lock and Restore Occurrence Properties 316

12 EDM Client Installation Guide 319

12.1 Client Installation and Setup ... 319
12.1.1 Required Information ... 319
12.1.2 Installing EDM Client ... 319
12.1.3 Updating EDM URL ... 323
12.1.4 Updating Cadence Version ... 325
12.1.5 Verify EDM Installation ... 326
12.2 Troubleshooting Client Installation Problems ... 326
12.2.1 Trouble with EDM Menu Item in Capture ... 327
12.2.2 Trouble with EDM Menu Item in PCB Editor ... 329
12.2.3 EDM Fails to Connect to PCB Editor ... 330
12.2.4 Incompatible Version Error After Log In ... 330
12.2.5 License Error After Log In ... 331
12.2.6 EDM Shows Blank Screen ... 332

13 Appendix 335

13.1 Definition of EDM Terms ... 335
13.2 File Type Associations ... 346
13.3 Hierarchical Designs ... 347
13.4 Warnings and Errors ... 349
13.4.1 Open File Warnings ... 349
13.4.2 Checkout Feature Warnings ... 350
13.4.3 Checkin Feature Warnings ... 352
13.4.4 Unable to Create Project ... 354
13.4.5 Unable to Download ... 355
13.4.6 Unable to Import ... 356
13.4.7 Unable to Check Out ... 357
13.4.8 Unable to Complete Check In ... 359
13.4.9 Unable to Promote or View Promoted Location ... 364
13.4.10 Unable to Rename ... 369
13.4.11 Unable to Backup and Clear Local Workspace ... 372
13.4.12 Unable to Add Vault ... 372
13.4.13 Browser Caching or Database not reachable issue ... 373
13.4.14 Other Warnings and Errors ... 374
13.5 EDM Shows Blank Screen in Capture ... 376
13.6 Technical Support ... 378
13.7 Modifying PDF Printer Setup ... 379

1 Introduction

OrCAD® Engineering Data Management (EDM) is an application that enables project leaders and users to manage all engineering design and data files. The management of files can be achieved directly from inside OrCAD Capture or from the use of the EDM Desktop application that opens as either a standalone application or from your PCB Editor.

EDM empowers project leaders, engineers, and designers with the following work capabilities:

- Assignment of files, designs, pages, schematic symbols, and PCB footprint files to selected engineers and designers.
- Tracking of project status from a standard web browser.
- Control of changes to designs and other data files.
- Management and promotion of footprints and step models to a central library.
- Management and promotion of individual schematic symbols from Capture library files (.OLB) to a central library.
- Usage of previously-created designs as a starting point for new projects.

Project Leaders can configure Team Designs that support:

- Concurrent design development where individual engineers and designers check out/check in schematic pages, library parts, or groups of pages while simultaneously working on a single Capture design.
- Management and removal of duplicate reference designator as well as the ability to lock and restore saved reference designators.
- Design freeze process that enables variant information to be added to a design file. This process also supports back-annotation.

EDM can be opened from three different user interfaces: a supported web browser; the Capture plug-in that enables you to open EDM from within Capture; and the EDM Desktop application that enables you to open EDM without Capture and connect to PCB Editor. The Capture plug-in and EDM Desktop application requires the installation of the EDM Client.

The EDM web browser enables you to:

- View projects and statuses
- Open PDF project files
- Update user assignments, statuses, and properties of projects
- Configure templates, (excluding functionality requiring Capture or PCB Editor)
- Configure EDM properties
- etc.

The EDM Capture plug-in, when opened from within Capture, enables you to:

- Check in and check out files, pages, and symbols
- Automatically generate related Capture files and reports
- Promote symbols
- Access Team Design functionality

- etc.

EDM Desktop application allows you to:

- Check in and check out files, footprints, and step models
- Automatically generate related board files and reports (when connected to PCB Editor)
- Promote footprints and step models
- Create release packages
- etc.

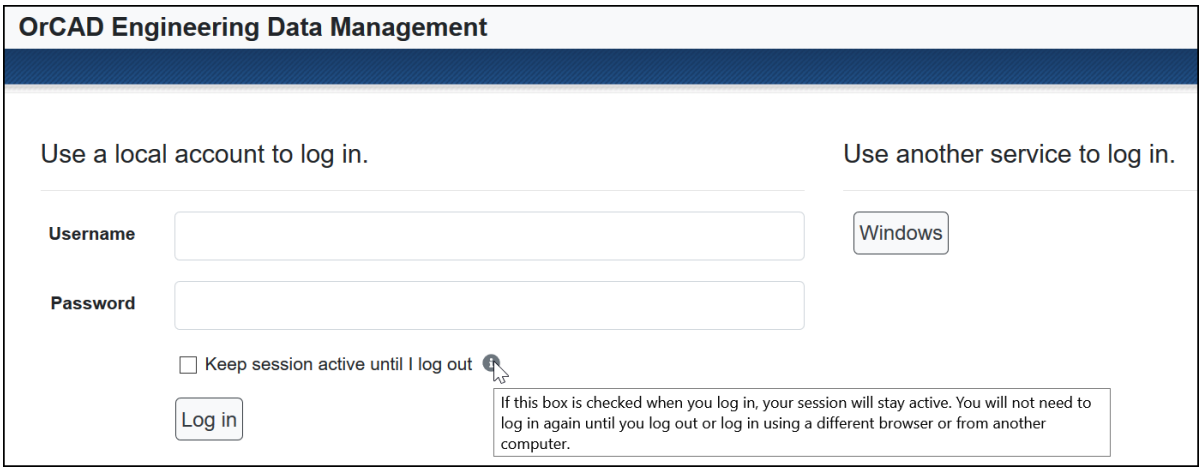
1.1 Getting Started

You can access EDM functionality from *any of three different user interfaces*. Functionality provided by each is limited by capabilities of the tool used to open the user interface.

1. *Web browser* - Enables limited configuration, viewing of projects, files, and status, and download of project history.
2. *Capture plug-in* - Enables you to open EDM from within Capture, configure features pertaining to Capture schematics and libraries, and check out and check in with auto-generation of Capture related files.
3. *EDM Desktop application* - Enables you to open EDM without Capture, configure features pertaining to PCB board and libraries, check out and check in with auto-generation of PCB related files, and create a package for release of your project.

The Capture plug-in and EDM Desktop application user interfaces are available after the EDM Client is installed.

When starting EDM from your browser, enter the EDM URL http://<servername>/EDM into your browser where the <servername> is the hostname of your web application server. Then log into EDM with the username and password that your Administrator has assigned to you. If your username is a Windows domain account, simply click on the Windows button.

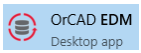


EDM Log In

Note: If you are unable to see the EDM login screen, please contact your administrator or IT team who installed EDM.

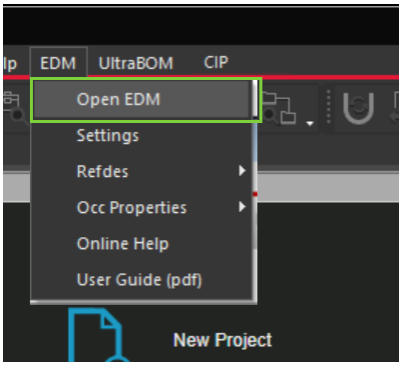
If you wish to remain logged into EDM, you may select the checkbox to “Keep session active until I log out.” Your session then remains active within your browser until you either log out or log into a different browser or computer. You are required to log in each time you switch between Capture and the EDM desktop application.

Launching the EDM Client

After the EDM Client (Key) is installed on your PC and the EDM URL is configured, you can launch the EDM Desktop application by double-clicking the EDM shortcut 

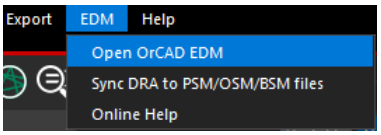
Note: The EDM shortcut is created during installation.

Open EDM from within OrCAD Capture by selecting **EDM > Open EDM** as shown in the image below.



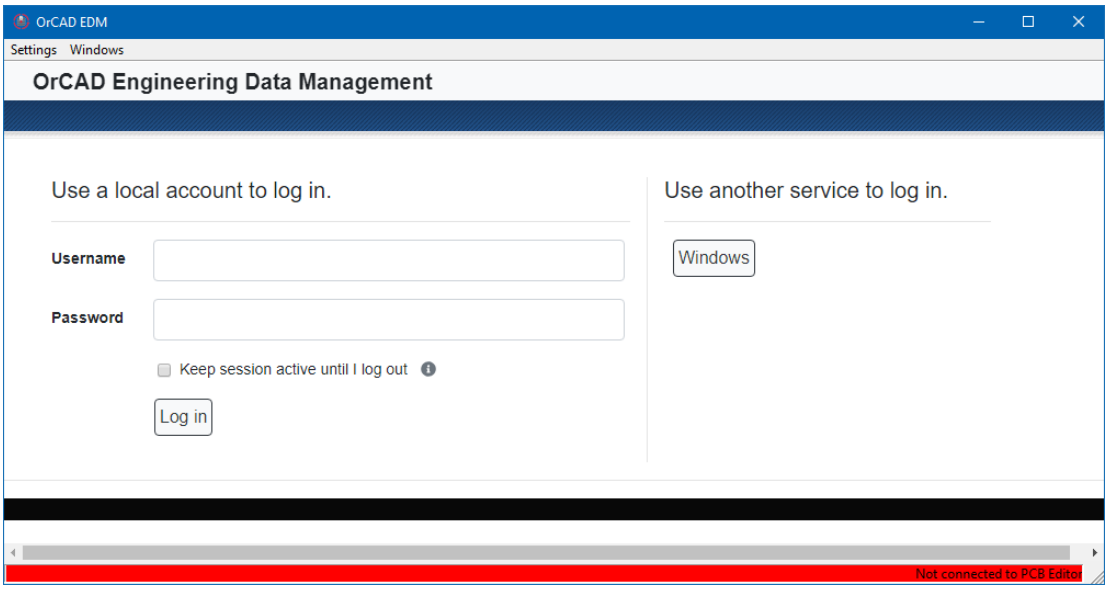
Select EDM > Open EDM from OrCAD Capture

To open the EDM Desktop Application from your PCB Editor select: **EDM > Open OrCAD EDM**.



Select EDM > Open OrCAD EDM from PCB Editor

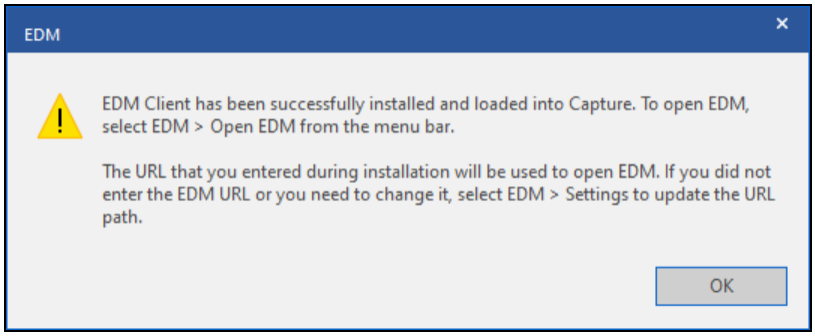
The EDM desktop automatically connects to your PCB Editor when both are opened. The EDM Desktop application uses PCB Editor functionality when importing or checking PCB Editor files, such as footprints, step models and board files. As shown in the screen capture that follows, the EDM Desktop application displays a RED bar at the bottom of the screen when it is disconnected to the PCB Editor. Review troubleshooting information when EDM is unable to connect to PCB Editor.



Desktop Log in

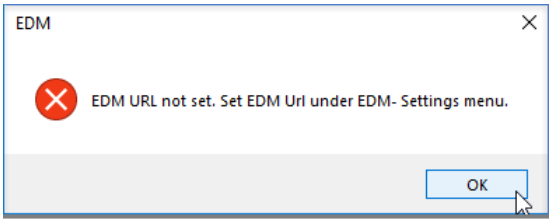
You can continue to log in following the same procedures that are followed when you log in from your browser.

When you first launch OrCAD Capture after the installation of EDM, a pop-up window opens that notifies you of the successful installation. This pop-up window only displays once without reappearing after re-installation. Click **OK** to continue.



EDM Client Success Pop-up

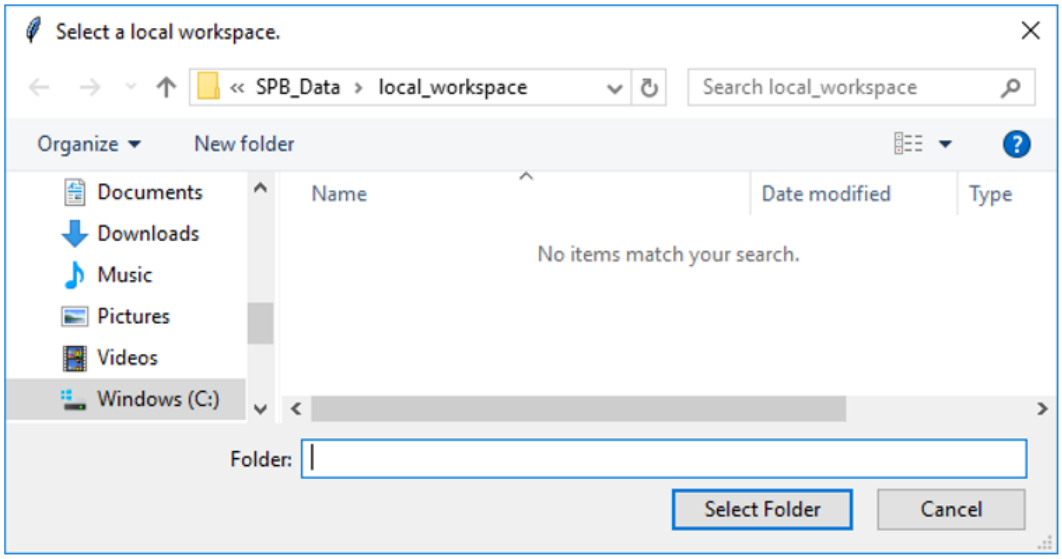
If your EDM URL was left blank during installation, the following dialog box displays, indicating you need to configure the EDM URL.



EDM URL Not Configured

After your first successful login (from either inside Capture or your PCB editor) you will be prompted to browse to select a folder to define your local workspace. The local workspace is a location on your computer where you save the design files you are developing. The location of your workspace must be defined before you proceed.

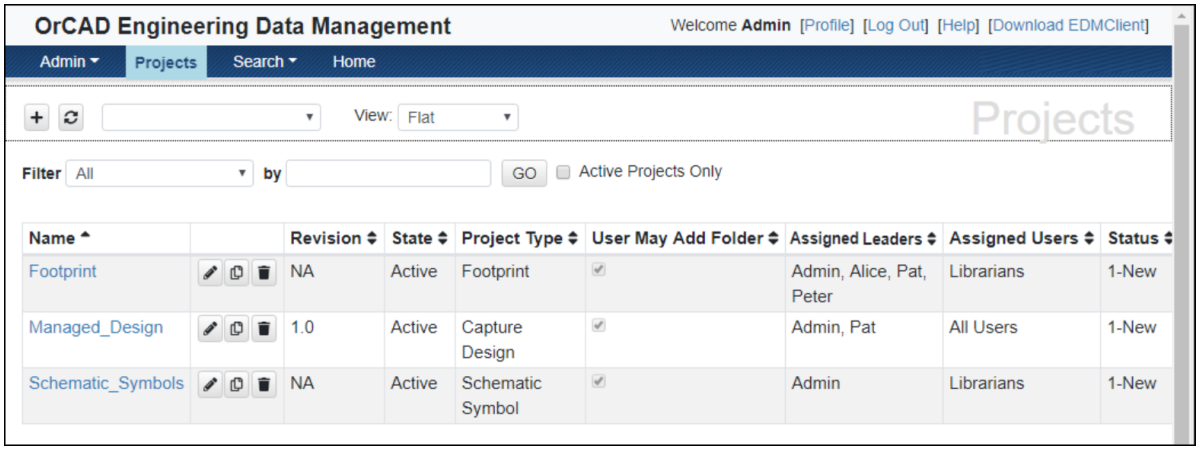
The screen that prompts you to select a local workspace is shown below.



Browse to Select Local Workspace

The Project Dashboard

After setting up your local workspace, the Project Dashboard appears. The Project Dashboard shows all projects that are assigned to you. All users have access to the Projects, Search, and Home menus while users with permissions pertaining to Non-Project Roles settings additionally have access to the Admin menu.



Example of EDM Project Dashboard

If you do not see projects in your Project Dashboard, you have not yet been assigned to a project.

1.2 Engineering Data Management

Schematics, symbols, footprints, bills of materials, and board files are among a few of your engineering data that are required to build PWB assemblies for your products. Effective management of all your design files and versions throughout various stages of development can increase productivity and reduce time to market.

In general, Engineering Data Management systems provide the following basic functionality:

- Control of who can check in and check out designs and files from the vault.
- Tracking of versions so that users can tell whether they have an out-of-date copy. When updated files are checked in, older versions are not discarded or lost.
- The status of the files can be updated and reviewed by users.
- Change history of designs and files are maintained as users check files in. Older versions of design files may be retrieved at any point in time.

The OrCAD EDM user interface is integrated into OrCAD Capture, enabling engineers to check in and out design files from within the Capture design environment. In addition to the basic engineering data management functionality, EDM offers the following benefits:

- Managers who may not use the design tools can monitor the status of multiple projects from a web browser.
- Designs requiring multiple engineers to work on a single design can be added as a Team Design.
- Menu item is available in PCB Editor to launch EDM.
- Users checking in a design file can have EDM automatically insert comments for the number of parts and nets within the design file.
- BOM files may be automatically created and added as part of the check in.
- Netlist files may be automatically created and added as part of the check in.
- PDF design exports may be automatically created and added as part of the check in.

1.3 Using Engineering Data Management

EDM is designed for day-to-day management of engineering data. There is no need to wait for milestones to submit engineering data into a vault. EDM is integrated directly into OrCAD Capture CIS and enables easy daily check in and check out of your files into the vault. Besides file management, EDM enables the management of designs that require multiple engineers to work in a team environment. New users to EDM should follow the getting started information to enable EDM on their system.

All team members working on an EDM project should always open EDM or enable RefDes when working on any portion of the design.

A typical flow for projects using EDM is described in the following stages:

1. Project creation - includes defining project structure:

a. Project Structure - structure of projects inside EDM is flexible and enables a project leader to manage the project different ways. Things that may influence a project structure include: design team size (will Team Design be required), type of design (Hierarchical or Flat), similarity to an existing project (can a project template be reused). For Team Designs, pages and schematics may be added to the project.

b. Assignments and configuration of structure - if a project leader needs tight control of the project structure, you may configure the project to disallow users to create sub-directories. Folders can be configured to restrict check in of specific file types. Users can be assigned to pages and files.

2. Activate the project for check out and check in of files and content.
3. Development phase (Schematic and PCB design and validation) - during this phase, engineers and designers with EDM perform daily check out of files and designs to work on, then check the files and designs back in at the end of the day. For Team Design projects, a project leader may also need to clean reference designator using the refdes feature. (Refer to Team Design process below).
4. Baseline the project - baseline can be thought of as creating checkpoints in your project or used to mark points in which the project is finished and ready for release.
5. Create new project revision - restarts the project in a new branch.

Team Design Process

1. Users who see pages assigned to them can check out their assigned pages. Users can download all pages not assigned to them, but pages will be read-only. Download of all pages will help with synchronization of the project.
2. Users will work on their assigned pages during the day. When users are at a good stopping point, or at the end of the day, users should check in all checked out pages. Early sharing of work will help with on-going communications.
3. Project leaders can bring the design together and annotate Ref Des at the project level. This can be done as many times as needed. Locking down reference designator for parts in a design that are firm will enable early layout for those parts of the design. The entire project should be checked out.
4. Project leaders will check in the entire project if changes have been made.
5. When the design has reached the point that PCB layout may begin, the PCB designer can get a copy of the entire design as read-only and start preliminary placement, while the engineer continues on the design.
6. When all the engineers are done, the project leader will perform final Ref Des annotations, Netlist, CIS Variant management, DRC check, and any reviews, per company process. Once this is done, the project leader will check in the entire design and freeze the design.
7. The full layout can be performed on the frozen design at this point. The PCB designer will check out the frozen design.
8. When the layout is completed, the PCB designer will create a back annotation file, if changes were made that require back annotation.
9. The project leader can check out the frozen design, perform back annotation, and check the design into EDM.

The project would be completed at this time. If any changes need to be made to separate pages in the future, the frozen project can be copied to a new revision. The new revision will separate all the pages and the entire process starts again.

2 User Settings and Customizing Views

There are numerous settings users can customize. The configuration of these settings include the following required and optional settings.

EDM Settings

The Capture menu item for EDM, EDM URL and postscript interpreter settings may be configured. If you want to include an EDM tab in Capture, URL configuration is required. The PDF generation setting is required only when you want EDM to auto-generate PDF files when you check in a managed design.

Home page

The Home page lists links to checked out files and projects you have most recently accessed. If desired, you can set the Home page as the first page to open after login.

Profile

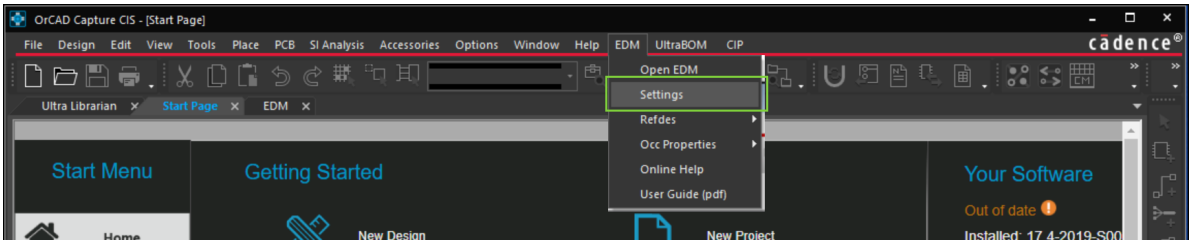
Your Profile dialog screen includes input fields that allow you to change your password (if not using Windows Authentication), define the location of your local workspace, and select your log in page.

Views

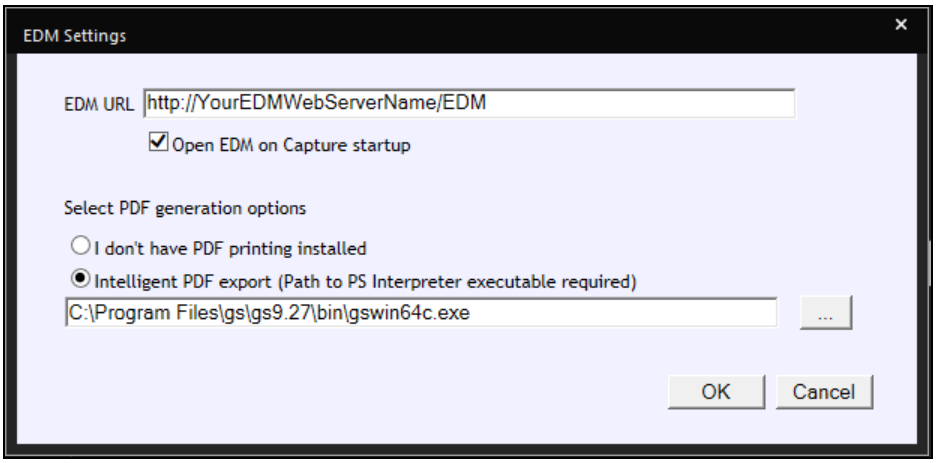
Each user may configure their views to show or hide attributes, as well as change the order in which items display.

2.1 EDM Settings

The EDM Settings dialog is available from the Capture menu: **EDM > Settings**. You can open the Settings dialog to configure the EDM URL, set up PDF printing and set up an external tool to perform differences on text files. The correct EDM URL is required to open EDM from Capture or the EDM Desktop application.



The EDM Settings dialog appears.



If you are changing the EDM URL, simply enter the URL into the field.

To have EDM automatically open when you start up Capture, select the checkbox, **Open EDM on Capture startup**.

PDF Printing for Capture Design Projects

To set up PDF printing for Capture Design Projects from the Capture > EDM > Settings dialog:

Select the radio button **Intelligent PDF export** and enter the path to the PS Interpreter, or use the Browse button to select the postscript interpreter, then click OK.

Note: A pre-requisite for PDF Printing is a Post script Interpreter executable such as Ghost script (gswin32x.exe or gswin64c.exe) or Acrobat (acrodist.exe).

If you receive a warning message that states the "PS driver/print not found" you can click OK to open the "Add Printer" window. Instructions for resolving PDF printing issues are provided in the section, Modify PDF Printing Setup.

Next time you check in a managed schematic design file, you will be able to automatically generate a PDF file as described in Check In Managed Schematic Files.

To change the default paper size for the OrCADEDMPS printer

1. From the Printers & Scanners window, scroll to locate the PS printer that you want to configure and click the **Manage** button.
2. Open the Printing preferences link and select the Paper/Quality Tab. Under Paper Options select the Size, then click **OK**. EDM uses the default settings for the printer.

2.2 Home Page

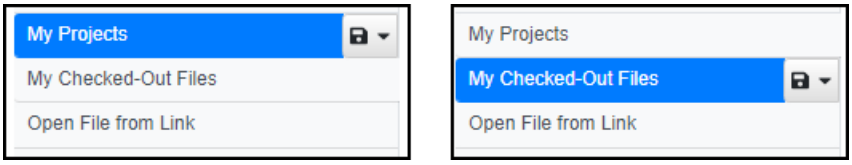
The **Home** page provides access to a list of your most recently accessed projects and checked out files. From the Home page you can organize the list order, open a project's detail view, download multiple project files, or export the list to a CSV or Excel file as explained in the instructions that follow. Additionally, you may also open a file and directly to a project from an EDM link provided to you.

Viewing projects, checked out files or open file from a link

Click on the desired section to show **My Projects**, **My Checked-Out Files**, or **Open File from Link**, as described below.

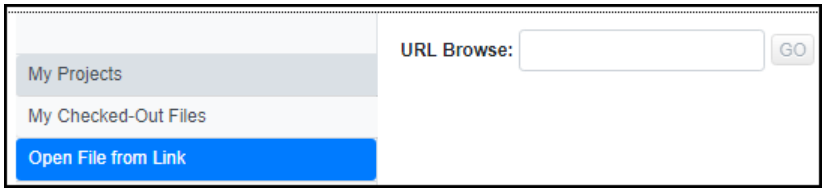
- **My Projects** – lists your pinned projects and those you have most recently accessed. List details include the Project Name, Revision, Last Accessed, Status, Assigned Leaders, Assigned Users, Project Group, and Vault. You can click a project listed to open the project directly from your Home Page.

- **My Checked-Out Files** – lists files that you currently have checked out. The list details include the Item Name, Date Checked Out, Item Type, Item Status, Project Name, Revision, and Assigned Leaders. You can click the name of a checked out item to open the project folder where you can access the item.



Select My Projects OR Select My Checked-Out Files

- **Open File from Link** – Enter the link for an EDM file (obtained by viewing the history of a file) in the text box and click **GO**. EDM will open the project that the file belongs and the file will be opened using the file association set up on your computer.



Organize Lists of Projects

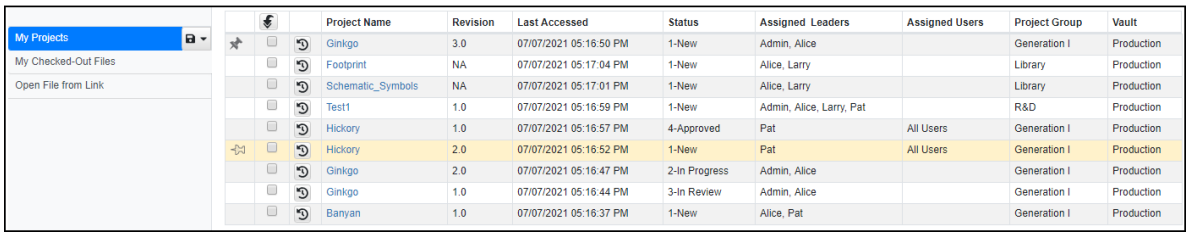
You can pin and unpin projects to manage the content and order of the **My Projects** list. When you pin a project, you move the project up the list. This provides quick and easy access to frequently accessed projects.

To pin a project

1. Select **My Projects**.

2. Hover your cursor over a row in the displayed list. The row becomes highlighted and the pin icon appears to the left of the **Project Name** checkbox.

3. Click the pin icon of the project you want to pin. The pin icon is added and the project row appears at the bottom of your pinned projects list as shown in the following figure:



Home Page My Projects Pinned List

To unpin a project

1. Select **My Projects**.
2. Click the **pin** icon of the project you want to unpin. The **pin** icon is removed, and the project row is moved to the bottom of your **My Projects** list.

To access Project Details

In the Project Name column of either **My Projects** or **My Checked-Out Files**, click a project name to go directly to Project Details View for that project.

To download projects to your local workspace

1. Select **My Projects**.
2. Select the checkboxes in the rows of the projects that you want to download.
3. Click the **Full Project Download** button at the top of the column. All project files for the selected projects are downloaded to your local workspace.

To review history of a project

1. Select **My Projects**.
2. Select the **History** button of the project you want to review. The project history information flyout opens on the right of the display.

Note: This process is the same as going to each project's Project Details View and doing Full Project Download successively.

Export a List of My Projects or My Checked-Out Files

A list can be generated in Excel or .CSV format that contains the information shown on the Home page of your **My Projects** or your **My Checked-Out Files** list.

To create and save a list

1. Select either **My Projects** or **My Checked-Out files**.
2. Click the **Export Report** button and select either **Excel** or **CSV** from the drop-down menu. The file selector displays.

3. Select a download location and click **OK**. The list is saved to the location you specified.

2.3 Profile

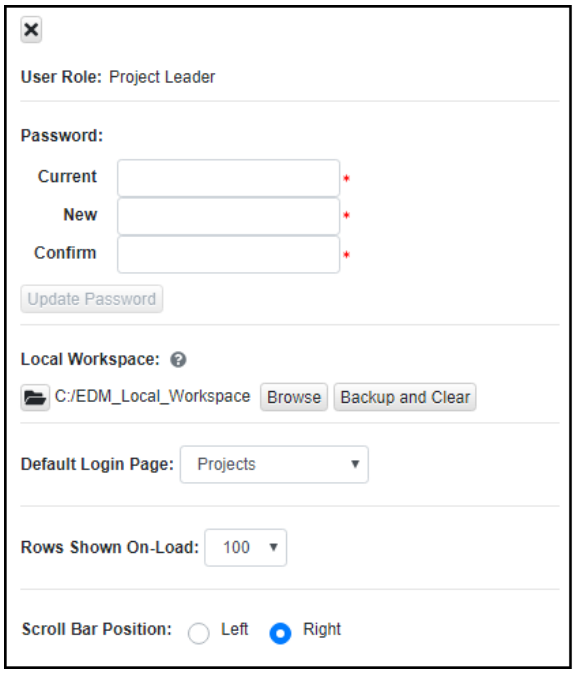
The **Profile** link shows how your EDM client is configured.



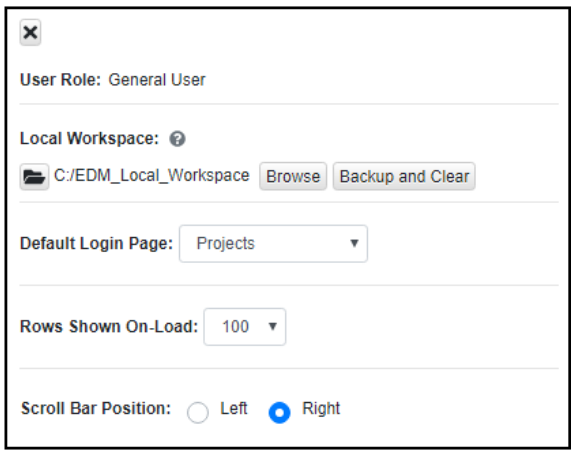
The Profile dialog that opens allow to see and configure the following:

- View your assigned EDM role
- Change your password
- Update your local workspace location
- Select a default login page
- Select number of rows to show on load
- Configure Scroll bar location

Click the **Profile** link in the upper right corner of the EDM page to open the Profile dialog and start making changes. The first image shows the profile for a non-Windows user. The second image shows the profile for a Windows user.



Profile for Non-Windows User



Profile for Windows User

View assigned EDM role

Your assigned role determines the permissions of actions that are available to you while logged in EDM. Default roles in EDM are Administrator, Project Leader, Project Leader with All Copy and General User. New roles may be added or permissions changed for these roles by your EDM Administrators. Contact your company EDM Administrator to found out your specific permissions for your role.

Change Password

To change your password (for Non-Windows Users only):

1. Click the **Profile** link in the upper right corner of the **EDM** page.
2. Type in your current password and then type your new password into both the **New** and **Confirm** text boxes.
3. Click the **Update Password** button.
4. Click the **Close** button to close the **Profile** link.

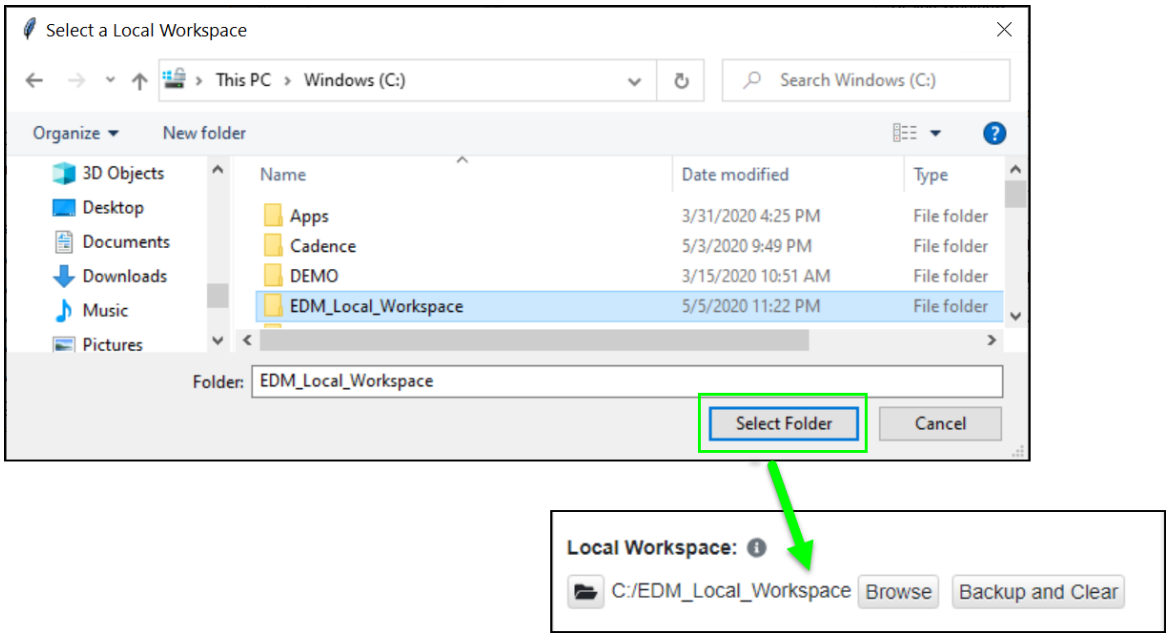
Note: When your user account is set up as a Windows user, the option to change your password is not visible.

Update Local Workspace Location

Your local workspace is a holding area on your PC where changes can be made to files before you check them into the EDM Vault. You may open directly to your local workspace by selecting the open folder button. You may create a backup and clear your local workspace when you encounter conflicts between your local workspace and the EDM Vault.

To change the Local Workspace:

1. Click the **Profile** link in the upper right corner of the EDM page.
2. Click the **Browse** button. A **Browse for Folder** window appears.
3. Select an empty folder that you want to use as your new local workspace. You may create a new folder or select the default folder created.
4. Click **OK** to save the local workspace. Or click **Cancel** if you do not want to save the changes.
5. Click the **Exit** button to close the **Profile** link.



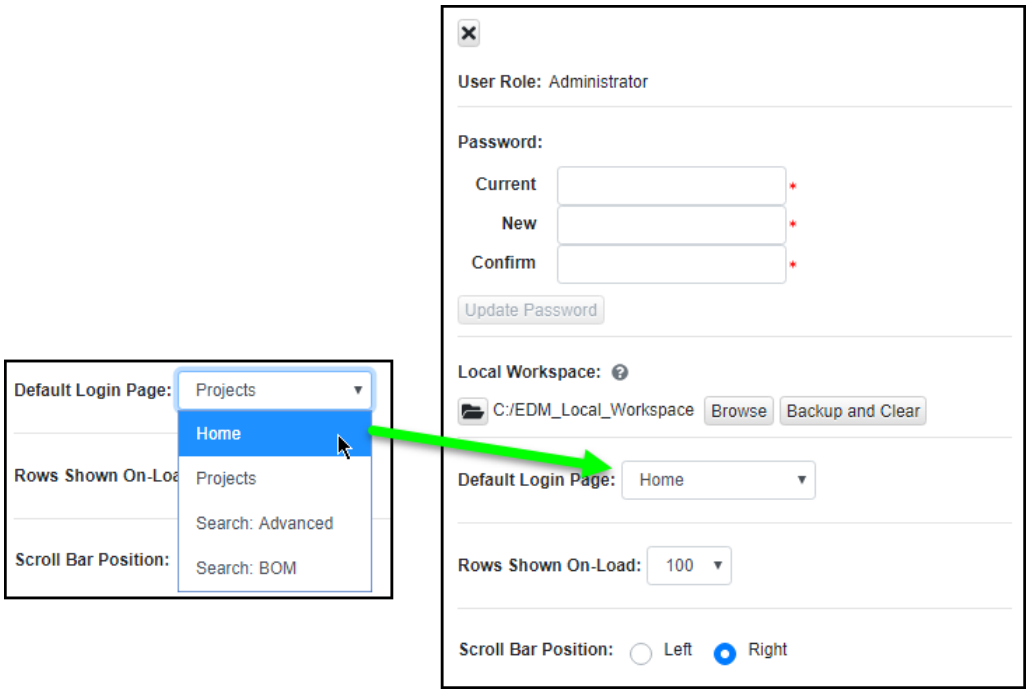
Select a Local Workspace

Select Default Login Landing Page

The default login page is the page that displays after you log into EDM. You can select to have Projects, Home, Search: Advanced, or Search: BOM as your default.

To select a login page:

1. Click the **Profile** link in the upper right corner of the EDM page.
2. Click the **Default Login Page** drop-down list. A list of pages are shown in the drop-down list.
3. Select a page from the drop-down list.
4. Click the Exit button to close and save your selection. When you log in the next time, the page you selected displays.



Profile Options for Default Login Page

Select Number of Rows To Show in the Project Dashboard and Project Detail Views

You can adjust the number of rows initially loaded and shown when the project dashboard or project details view opens. Scrolling to the bottom of your file and folder list enables you to view the next set of rows in the project. If you would like to change the number of rows to load and show more or fewer row, select the drop list number next to **Rows Shown On-Load** in your profile. The number of rows loaded will affect how quickly the project opens and updates as you scroll down the page. If your page is updating slowly, you may reduce the number of columns by customizing your view.



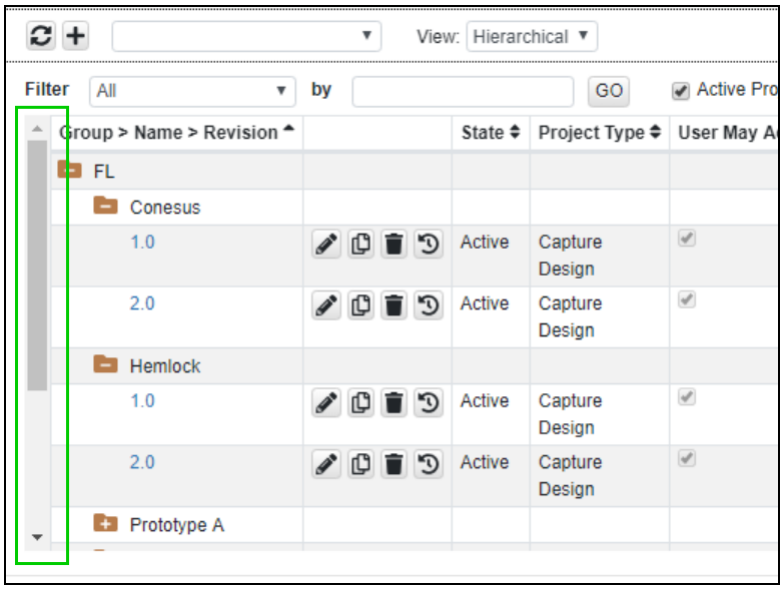
Configure Scroll Bar Location

You may personalize the placement of the scroll bar in your Project Dashboard and Project Detail views. The Scroll Bar in the Project Dashboard and Detail views by default are placed on the right side of the page.

To change the scroll bar location to the left side, select the **Left** radio button next to **Scroll Bar Position** in your profile.



A following example shows the scroll bar on the left side of a sample dashboard.



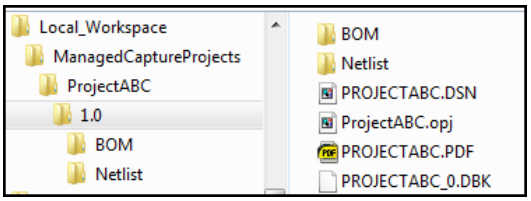
2.3.1 Local Workspace

The local workspace is a location on your local system where copies of projects are placed as a holding area for the work you are developing. When you download items from the vault, the items are placed into your local workspace. As you make changes to checked out items, your updates are saved to your local workspace.

After your first successful EDM login (from inside the Capture plug-in or Desktop application), you will be prompted to browse to and select a folder for your local workspace. The location of your workspace must be defined before you proceed. Your local workspace can be changed as needed.

The structure of the local workspace is the same as the structure in the vault. The structure consists of the following: **Local Workspace > Project Group > Project Name > Project Revision > Project Folders/Files**. See Organizing Your Projects for a description of each project level.

Example: Project Group **ManagedCaptureProjects** contains **ProjectABC** Revision **1.0**, which has project folders and files that belong to it, as shown in the following figure:



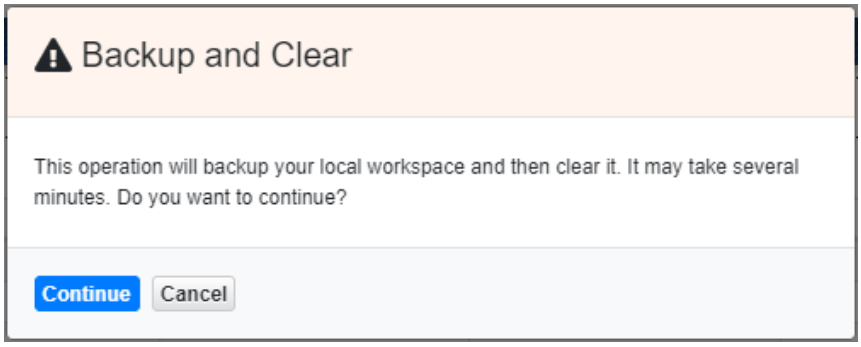
Local Workspace Structure

The **Backup and Clear** option under the Local Workspace section of the Profile enables you create a backup of your local workspace, then clear its contents. This may be needed to clear any conflicts between your local workspace and the EDM Vault.

When you click Backup and Clear, a confirmation pop-up window will appear, warning you that the operation could take several minutes depending on the size and quantity of files and folders. Click the Continue button to proceed or click the Cancel button to cancel the operation.



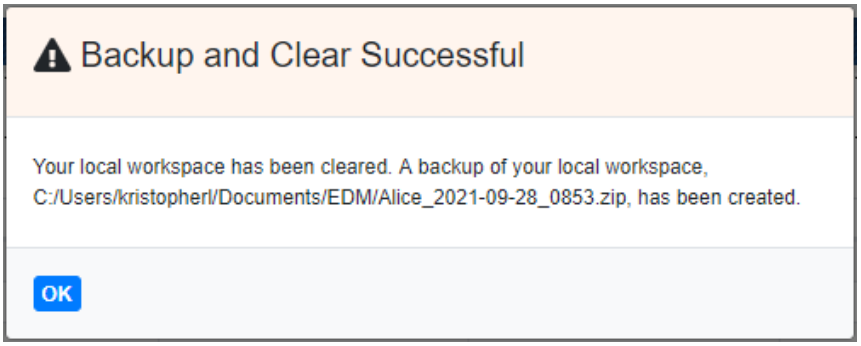
Profile > Local Workspace > Backup and Clear Button



Backup and Clear Confirmation

If you proceed, **Backup and Clear** performs the following steps:

1. Backup - EDM compresses your entire local workspace and places the compressed .zip file in the EDM folder in your Documents directory (%userprofile%\Documents\EDM). The backup zip file name contains the date and time of the backup (<LocalWorkspaceFolder>_YYYY-MM-DD_HHMM). - e.g. Alice_2021-09-28_0853.zip
2. Clear - EDM deletes the entire content of your local workspace. Any files that are opened and locked by application will prevent the successful clearing of your entire local workspace and leave some files and folders.
3. When the operation is completed successfully, a success pop-up window will appear, showing you the backup file name and location. Click OK to close the window.



Backup and Clear Successful

Note: If you see an Unable to Backup and Clear Local Workspace error message, you will need to delete the contents of your local workspace manually. Repeating the **Backup and Clear** action immediately after clicking it once will backup an empty or partial local workspace.

2.4 Customizing Views

You may customize the view of your project dashboard, project details, or team folder details. You can hide or unhide columns and modify the column order.

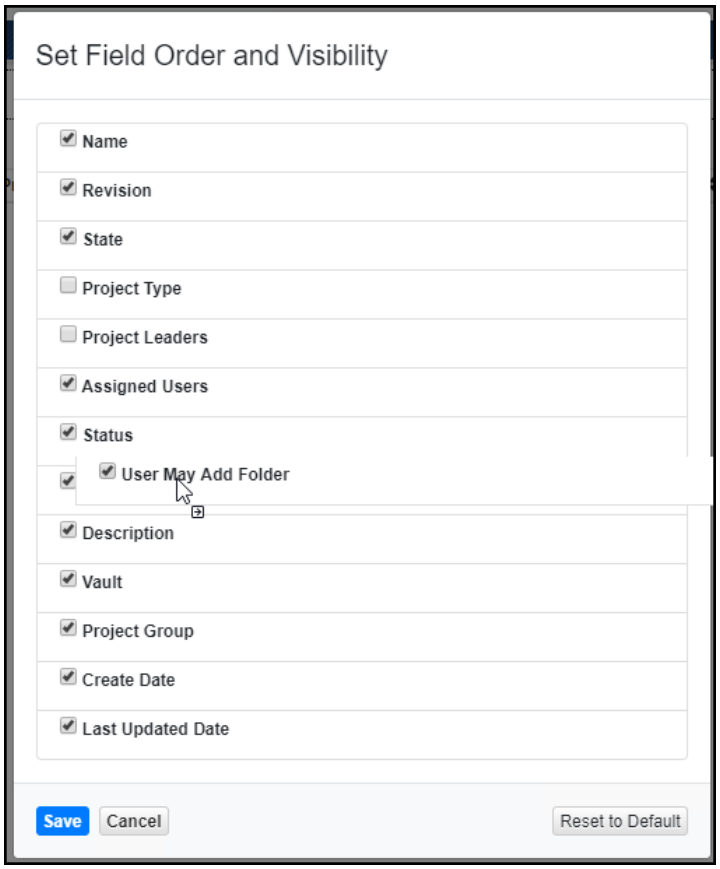
To change the order or visibility of a column

1. Click the Gear, icon located in the upper right corner of the projects view.



Project Dashboard With Gear

A **Set Field Order and Visibility** pop up opens. An example of the Project dashboard fields is shown in the following figure.



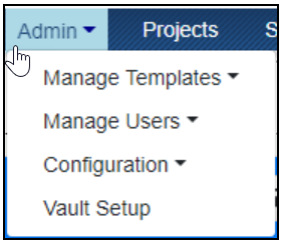
2. Deselect a checkbox next to a column name to hide that column. Or select a checkbox to display the column.

3. Select and drag the column name to move it to a different position. The order of column names from top to bottom becomes the left-to-right column order of your project details.

4. Click the **Save** button when you are done customizing the view.

3 Administration Guide

Engineering Data Management Administration features are available from the Admin menu. Only users whose assigned role have Non-Project Roles permissions have access to the **Admin** menu. The **Admin** menu includes options that enables a user with appropriate permission to Manage Templates, Manage Users, Configuration, and Vault Setup. Users with appropriate permission have access to menu items that they have permission to see.

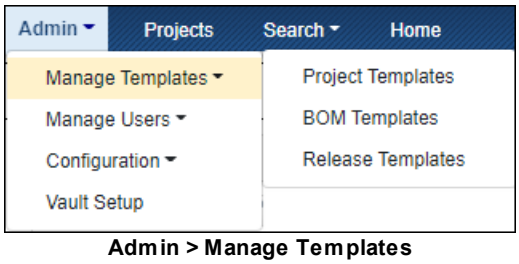


All Admin Menu Options

- **Manage Templates** - allows users whose role has the Manage Templates permission to create Project, BOM, and Release templates for use when creating new projects, checking in managed designs and auto-generated BOM files, and packaging contents for release.
- **Manage Users** - allows users whose role has the Manage Users permission to add, delete, update, and disable user access to the EDM database. This option also allows Administrators to create and manage User Group membership.
- **Configuration** - allows users whose role has the Configuration permission to configure EDM settings, such as Status, Revision, Customizable User Fields, Title Block Mapping, Email Notifications, Promoted Library Locations, and Promote Rules for management of Footprint projects.
- **Vault Setup** - allows users whose role has the Vault Setup permission to set up or change the vault and server locations.

3.1 Manage Templates

Templates help ensure format consistency across multiple projects. Users assigned roles with the permission to manage templates may create project, BOM, and release templates from any of the three EDM user interfaces: EDM URL accessed through your browser, Capture, or the EDM Desktop.



Admin > Manage Templates

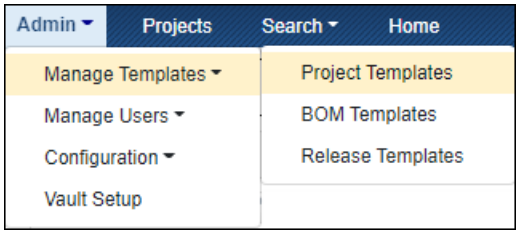
Three types of templates may be configured in EDM: Project Templates, BOM Templates, and Release Templates.

1. Project Templates provide an easy way to start your projects using a pre-configured folder structure.
2. BOM Templates allow you to configure DSN BOM Validation and helps standardize auto-generated BOM output from a managed design.
3. Release Templates help you standardize release packages. Templates may be defined to include different files and output formats as needed for archival purposes, internal review, delivery to contract manufacturers, etc.

3.1.1 Manage Project Templates

Users assigned roles with the permission to manage templates have the permissions required to manage project templates. Project templates help maintain consistent folder configuration and structure across multiple projects. Once you successfully configure a project template, you can easily create a new project that is a copy of a template. This can significantly reduce setup time.

Select **Admin > Manage Templates > Project Templates** to configure Project Templates.



Accessing Project Templates

Two default project templates come pre-installed. The pre-installed project templates may be copied or modified for use.

Project templates may be configured as one of two types: managed Capture Design or General project.

Capture Design Project Information

Capture Design projects are one of the four types of projects that can be created in EDM. Capture Design projects track relationships between the managed schematic design (BOM, Netlist, PDF) and managed board (DB Doctor, EDM Validation, PDF). This allows associated files to be automatically generated when design and board files are checked in.

Design and board files may be imported into either a project template or a new project. When you create a Capture Design, you can select a project template to quickly create the project structure and properties.

Capture design projects include a Capture design file called the managed design, whether imported or auto-created during download. Only one design file can be designated as the Managed Capture Design. A team design may also be designated as the managed design. The managed design designation takes effect after the team design is frozen.

Files related to a Capture Design project are automatically generated when you Check In

the managed design. The options for auto-generating files are presented when you select the managed design for check in. Auto-generating managed files upon check in keeps them in sync with the managed Capture design. The managed design is associated to related files by file names. If the file name of a managed design is changed in EDM after check in, the association between the managed design or board and associated files becomes out-of-sync. To update the file relationship, check out/check in the project files to regenerate the files with the updated managed name. To avoid or eliminate confusion, delete files that were checked in prior to the name change.

Capture Design projects include folder designations for auto-generated files. By default, folder designations are at the project folder level until you change the designation when you Add a Folder or Edit a Folder. The folder designation consist of the following:

- **DSN**

  The folder with this designation is used for the Managed Design. After Capture project is created, the project will not have any files. Before working on a project, you will need to download the project structure by clicking on the **Full Project Download** button. This creates the .DSN (managed design) and the .opj (Capture project file). The .opj file created by EDM contains the configured locations for the netlist and board files. To obtain an updated .opj file after changes are made to the project's folder designations, delete your local .opj file and perform a Full Project Download. The name of the managed design file is identified as a folder property. The managed design file is identified with a blue star (★) next to the filename. Check in of a managed design file from Capture enables the auto-generation functionality. Auto-generation of a BOM or PDF file creates files with the filenames configured in EDM. Changes to the BOM or PDF filenames must be made in the EDM project to retain the relationship of the managed design to these files. Checking in BOM or PDF with different filenames disassociates the relationship to the managed design.

- **BOM**

  The folder with this designation is used for the auto-generated BOM file based on the selected BOM Template(s). The BOM file that is autogenerated by default is given the same file name as the managed design. The file extension is either .BOM or .xlsx depending on the configured BOM Template. Changing the BOM file name disassociates the BOM from the managed design.

- **Netlist**

  The folder with this designation will be used for the auto-generated Netlist files. Three text files with your schematic design nets will be created and placed into this folder.

- **PDF**

  The folder with this designation will be used for the auto-generated PDF of the schematic. The PDF filename by default is given the same name as the managed design with a file extension of PDF. Changing the PDF file name will disassociate the PDF from the managed design. Auto-generating a PDF file of the schematic requires initial PDF Printer Setup.

- **BRD**

  The folder with this designation will be used for the Managed Board. The name of the managed

board file is identified in a folder property. The file with the managed board file name is identified with a blue star (★) next to the filename. Check in of a managed board file from the EDM desktop application enables auto-generation functionality. Auto-generated reports (DB Doctor, EDM Validation) are placed into a Reports folder and the auto-generated PDF file is placed in the designated folder for the board file. These files are automatically created/updated when the board file is checked in or when manually running auto-generation.

Release

The folder with this designation is used for files that are packaged for release to manufacturing. Selection of the Package for Release option allows you to select files for creation. The auto-generated files are compressed and saved in a zipped file that is saved in the project directory. Package for Release may include the auto-generated files: IPCs-2581, ODB++, Artwork, IPC 356, Fabmaster, Testprep, NC Route, NC Drill, and Pick and Place.

General

Projects created as a General project may contain standard or Team folders. With the exception of Team folders discussed in Managed Team Design Folder, all files and folders function in a similar manner.

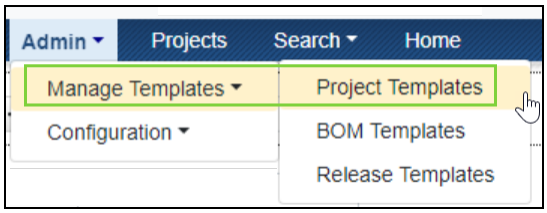
The instruction described below explain how assigned leaders can manage projects by creating, modifying, deleting, and copying templates. The creation of a project template is a two step process: first create a template, and then structure the project by adding folders.

Create Template

General instructions for creating a template and project structure are provided below. A detailed description of each field is described in the section topics, Manage Projects and Add Folder.

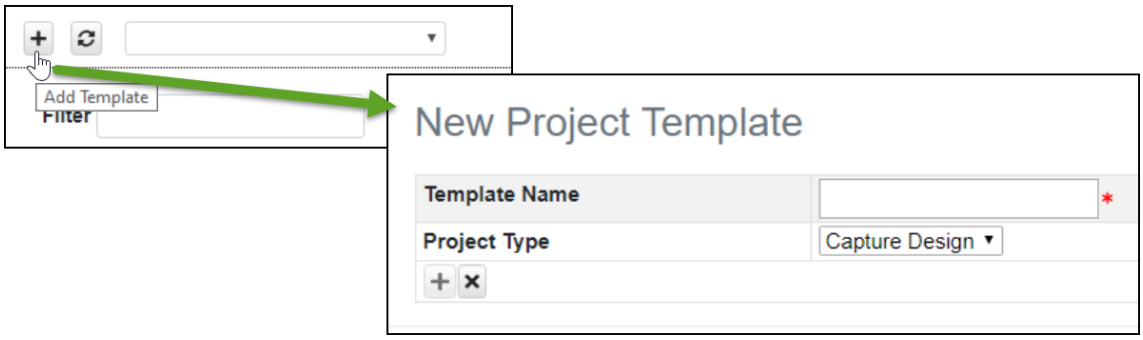
Step 1 - Create Project Template

1. Select **Admin > Manage Templates > Project Templates**.



Project Templates menu selection

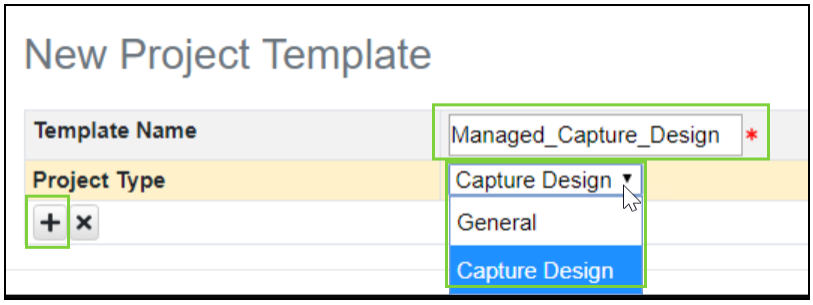
2. Click the **Add Template** button to open the New Project Template form.



Opening the New Project Template Form

3. Enter a name and select a **Project Type** from the drop list menu. The name you enter must be a unique template name and conform to Windows folder naming conventions.

   The image that follows, for example, shows the entry of the project name, Managed_Capture_Project and the selection of Capture Design as the project type.



New Project Template Form

4. Click the **Add** button to save the new project template. Once a new project template is named and saved, a message displays indicating successful template creation. This indicates you are ready to add structure to your template.

Step 2 - Add structure to your template

The process for adding structure to a project template is similar to the process for creating a project and adding, editing, and deleting folders. Any template folder you add may be configured as a Team or a standard folder. The Team folder enables you to assign individual design pages or symbols to different users who are working on the same design. You can configure the Team folder as a Flat or as a Hierarchical Team Design. You may also configure a Team folder to be a Team Library.

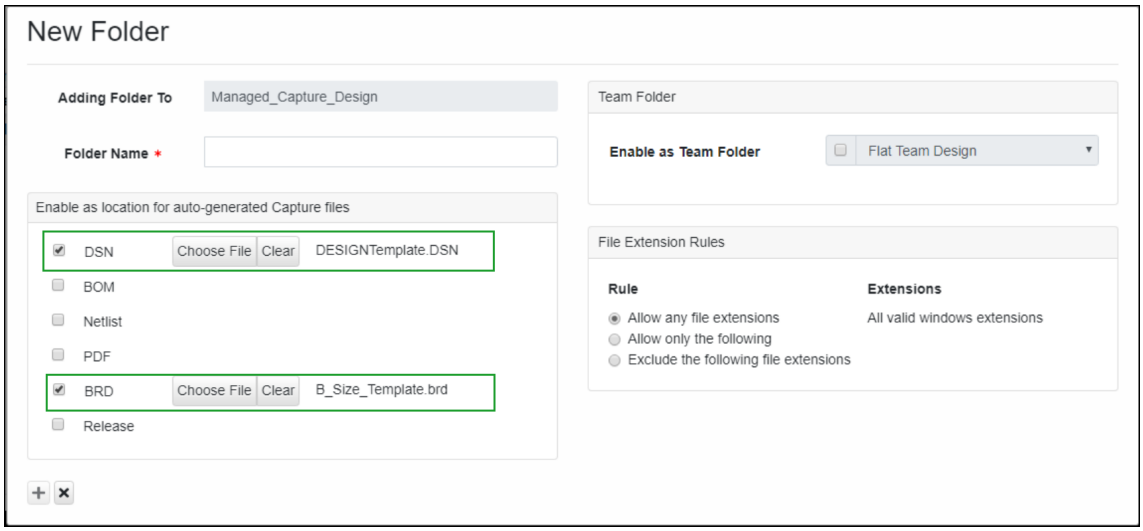
1. Open the project template and click the **Add Folder** button that is located to the right of the template name. This opens the New Folder form. The image below, for example, shows the Add Folder button for the template named, Managed_Capture_Project.



Add Template Folder Form

2. Enter information into the New Folder form. The red asterisk indicates required information.

   The **New Folder** form for a Capture Design Template includes options for auto-generating files as shown in the figure below. When you add a folder that is designated to store DSN or BRD files to a Capture Design template, you may browse to select a DSN or BRD file that will be used as a template.



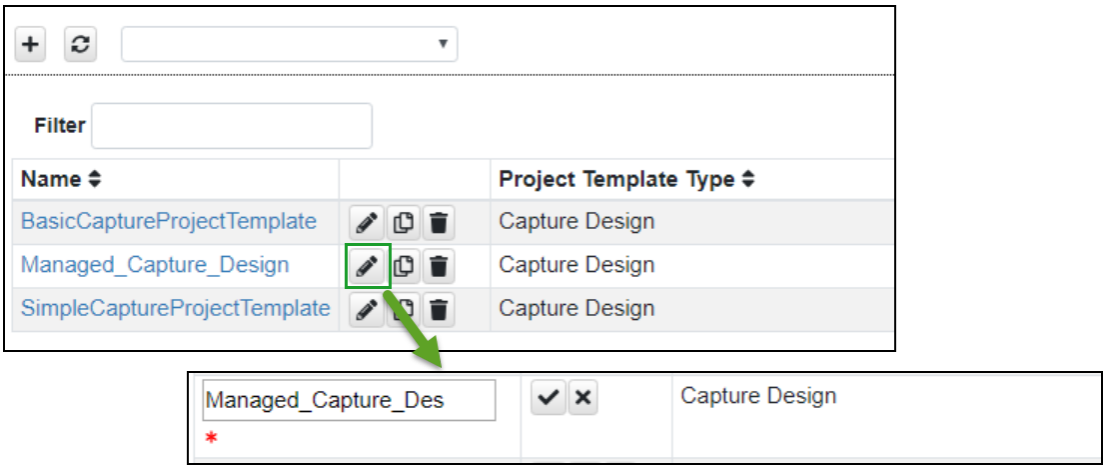
"Choose File" Buttons to Select a DSN or BRD Template

   A detailed description of each folder setting in the New Folder form is provided in the section topic, Add Folder.

3. When you have entered all the information into the form, click the **Add** button to save the configuration of the folder you are adding to the template.

Edit Template

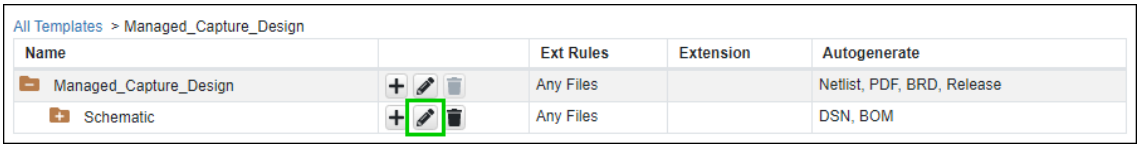
The template name may be changed after creation by selecting the **Edit** button located next to the Template name. Changes to a template or structure update without affecting projects already created from the template.



Edit Template Button for a Capture Design

To edit a template folder

1. To edit the name or properties of a template folder, click the **Edit** button. You may edit a template name and the name of a template folder as desired.

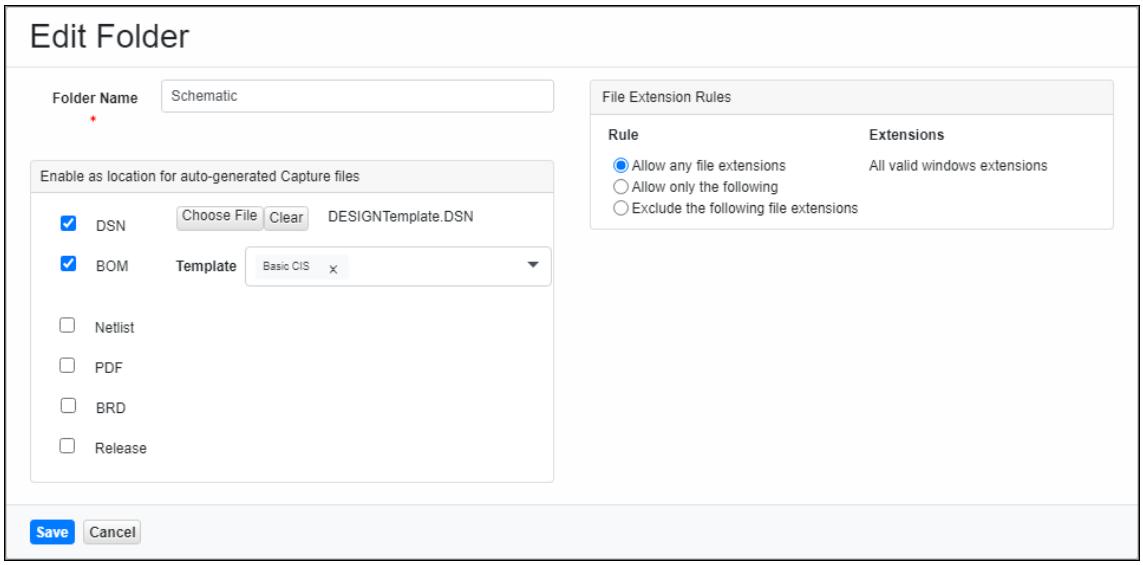


Edit Template Folder

2. Modify the Edit Folder form as desired.

When you are editing a template folder for a managed Capture Design project template, you may change the folder designation for auto-generated files as well as DSN or BRD file templates. You may also change the assigned BOM Template(s) on the assigned BOM folder. Your modifications only apply to new projects at the time of creation. You may also edit file extension rules assigned to the template folder. The process for editing rules is the same process you follow when editing a project. This process is described in the section topic Edit Folder.

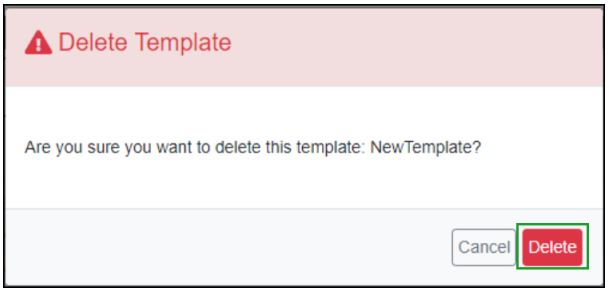
To change the configuration of a folder from a standard folder to a Team folder or vice versa, you need to remove and add the folder with the desired setting.



3. Click the **Save** button to save your modifications. Or click the **Cancel** button to discard your updates.

Delete Template

To delete a template, click the **Delete** button that is next to the template name. If you are deleting a template folder, click the delete button next to the folder name. A confirmation dialog pops up to confirm the deletion. Click the **Delete** button to confirm.

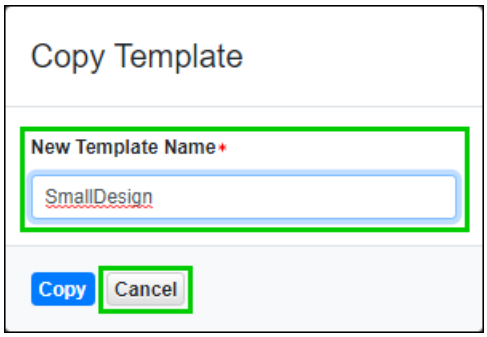


Delete Template

Copy Template

Copy to a New Template is useful when you create a new project template that has a structure similar to an existing template. When you copy the existing template to a new template, only minor modification will be required.

To copy to a New Template, click the **Copy to New Template** button. The **Copy Template** pop-up will display. Enter a **New Template Name**, then click the **Copy** button.



Copy Template

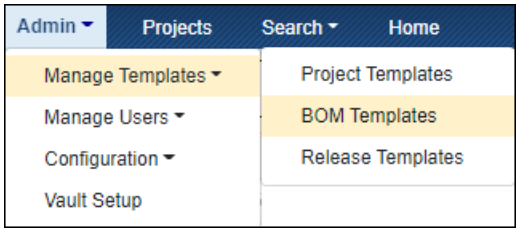
3.1.2 Manage BOM Templates

BOM Templates are used for auto-generated BOMs in Capture Design projects. Options that may be configured in the BOM Template include: the BOM format (Capture or CIS), RefDes format (Delimited, Compact, or Single), desired parametric field data, etc. The Capture BOM format can be configured and applied to any managed design. The CIS BOM format only applies to managed designs created with Capture CIS. This format is required to auto-generate BOMs for designs with CIS Variants.

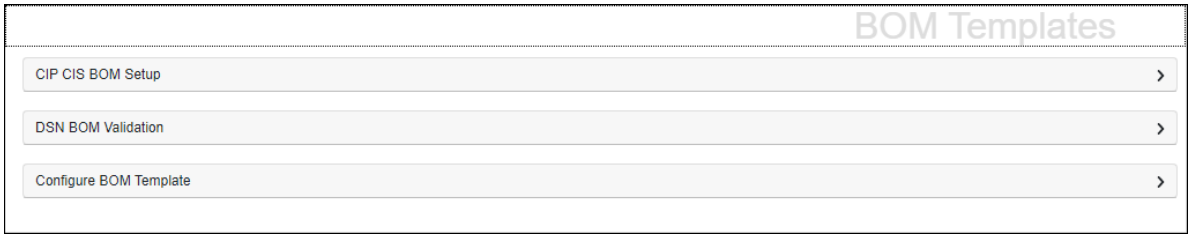
Two BOM Templates are included with the EDM installation: "Basic Capture" and "Basic CIS." You can use a template "as is" or you can modify the settings of an existing template. You can update/overwrite an existing template or save to a new template name. You may define as many templates as you need and select which ones to assign to a given project's designated BOM folder. You may select or change the BOM template assigned to a project when you are adding or editing the project's designated auto-generated BOM folder.

The functionality of the Basic CIS BOM template is integrated with OrCAD CIP. If you do not have valid connection to CIP when you attempt to apply a CIS BOM template, EDM uses local properties transferred to the design file. If you are using CIS variants, the variant information is retrieved from the design and each variant BOM can be auto-generated when the managed design is checked into the vault.

Select **Admin > Manage Templates > BOM Templates** to create, update, and manage BOM Templates and their related settings.



Accessing BOM Templates



Admin > Manage Templates > BOM Templates

BOM Template creation consists of three sections:

1. CIP CIS BOM Setup - Configure EDM's connection to CIP. This is required to Configure DSN BOM Validation and if you wish to Configure BOM Format using CIP. *Skip this step if you do not have CIP.
2. DSN BOM Validation - Configure DSN BOM Validation options for managed designs, which can run validation options against a design before it and its related auto-generated files are checked in. *Skip this step if you do not want to configure the field and value(s) for the CIP Restricted Field Value Report.
3. Configure BOM Template - Configure the format and related options for BOM Templates.

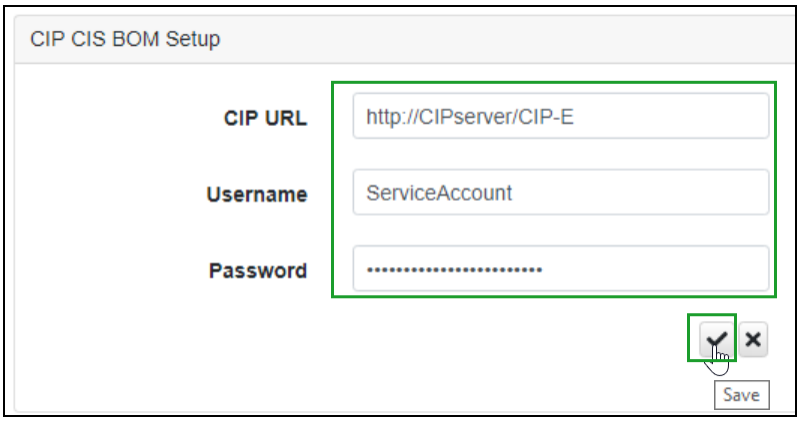
3.1.2.1 Connection to CIP

BOMs created with EDM can pull attribute data directly from OrCAD CIP. Fields configured in CIP are used to configure your BOM Template. EDM retrieves field names from CIP when you save the CIP username and password. List of fieldnames may be refreshed when needed.

Compatible versions of CIP, version 17.2.5 or greater, are required for successful configuration. If you have a CIP version that is less than 17.2.5, skip the steps presented in this sub-topic and continue to Configure BOM Format.

To configure CIP information

1. At the CIP CIS BOM Setup, enter the URL for your CIP.
2. Enter the username and password of the user who is configuring CIP information. **Note**: You need to verify the permissions of this user to ensure they are able to view the required BOM properties.
3. Click the **Save** button to save the information.



Configure CIP for BOM Template

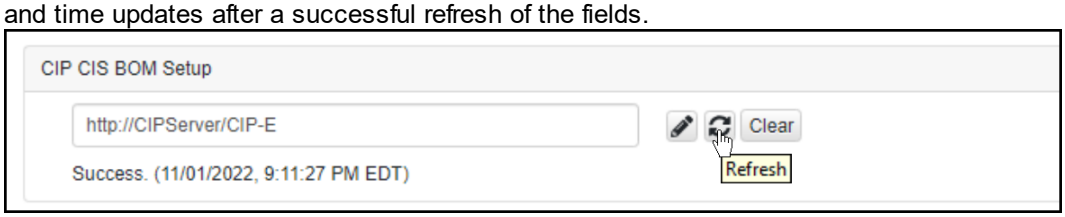
4. EDM logs into CIP and retrieves all the fields from CIP that will be used to configure the BOM template. After successful login, the username, encrypted password and list of fieldnames are saved. The date and time of fieldname retrieval are displayed.



Successful CIP Configuration

Changing CIP information

- To update CIP fieldnames for your BOM Template configuration, click the **Refresh** button. The date and time updates after a successful refresh of the fields.



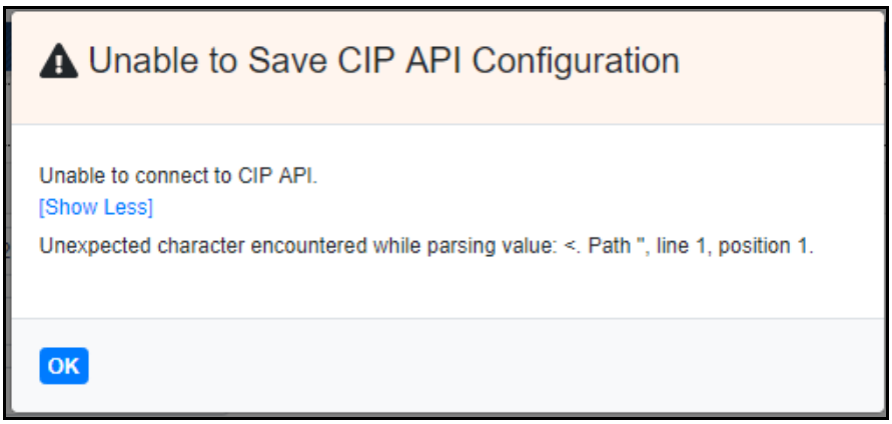
- To change the CIP URL, Username or Password, click the **Edit** button.
- To remove the CIP URL, login information and retrieved fieldname, click the **Clear** button. A warning will be provided with a list of templates configured to use CIP field when you attempt to clear CIP configuration as shown in the following sample figure.



Configuration Errors

Some of the possible errors associated with CIP configuration are provided below:

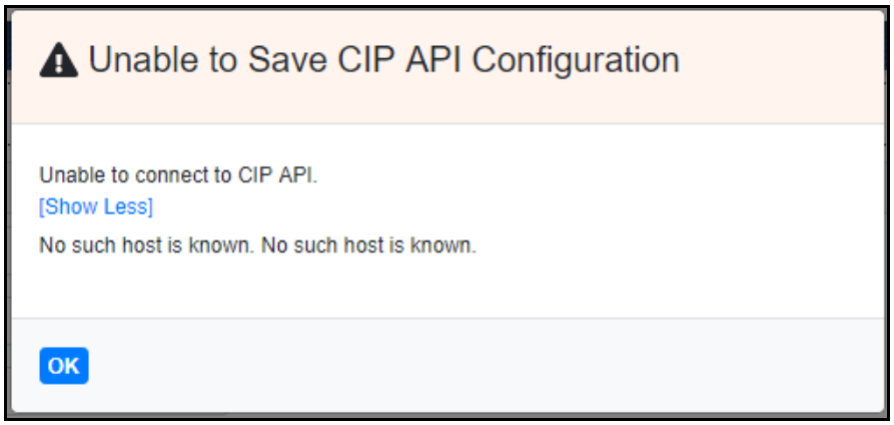
- CIP Version is incompatible - When the CIP version is incompatible, the following error message opens when you attempt to save the update: *Unable to Save CIP API configuration. Unable to connect to CIP API*.



CIP API Error - CIP Version Incompatible

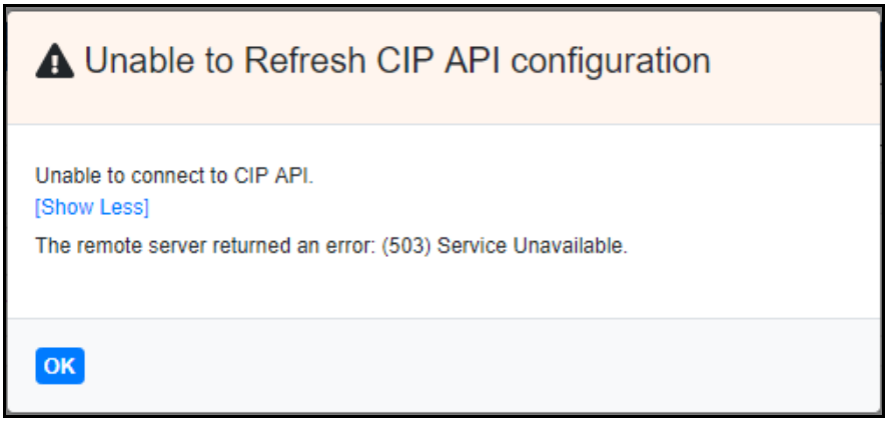
Click **OK** to dismiss the message, then click the **Cancel** button to cancel the CIP settings. You will need to install CIP version 17.2.5.19 and/or obtain the proper URL and credentials to continue with CIS BOM Template. This is required if you want to use CIP database information for your auto-generated CIS BOMs.

- CIP Server name is incorrectly entered - When the server name in the URL is incorrectly entered, the error shows as *No such host is known*. Click **OK** to dismiss the message, then enter the correct URL.



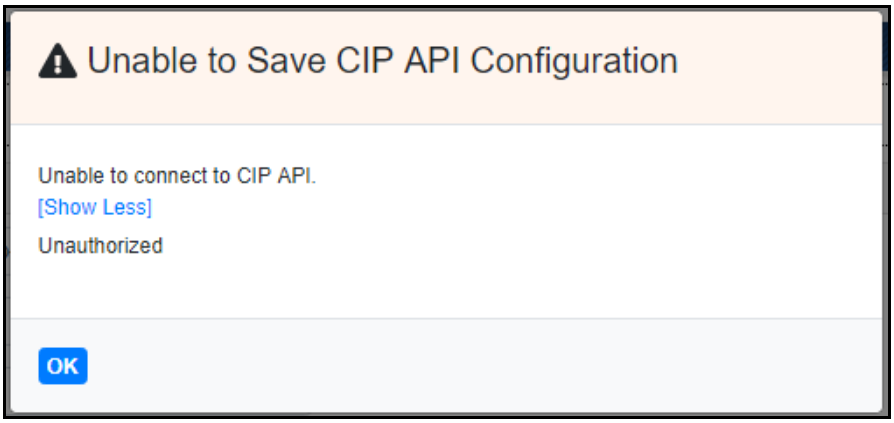
No such host is known

- CIP Service is Unavailable - If you get an error from BOM generation, click Refresh at the CIP CIS BOM Settings to verify that the connection to CIP and user credentials are active. If the CIP service is no longer running, the following error opens when you perform a refresh: *The remote server returned an error: (503) Service Unavailable*. Click **OK** to dismiss the message then verify that the CIP web site is up and running.



CIP Service is Unavailable

- CIP Username or Password is incorrectly entered - If you get an error from BOM generation, click Refresh at the CIP CIS BOM Settings to verify that the connection to CIP and user credentials are active. If the user credentials changed after saving, the following error message opens when you attempt to refresh: *Unauthorized*. Click **OK** to dismiss the message then **Edit** the Username and Password to enter the updated credentials.



Username is Unauthorized

3.1.2.2 DSN BOM Validation

When checking in a managed design file, you may want to run a validation check against the design from which the BOM file(s) will be generated.

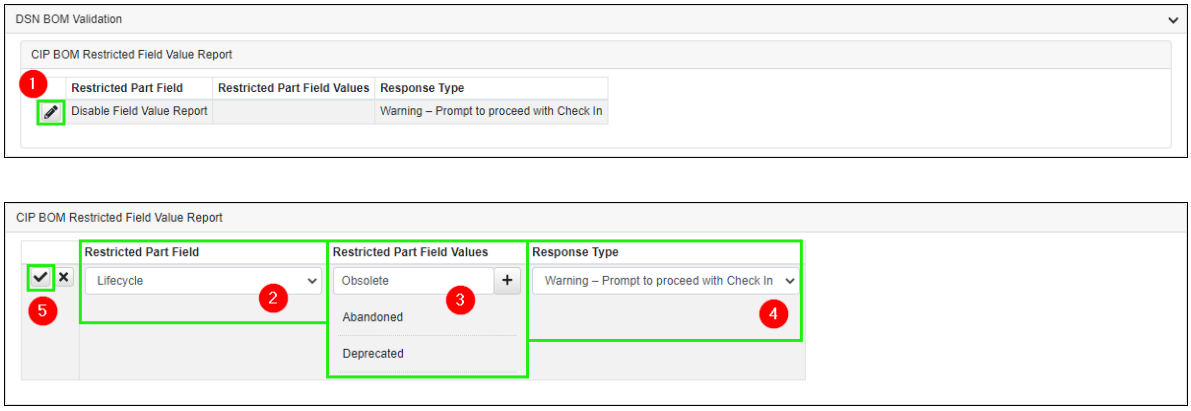
CIP BOM Restricted Field Value Report

The CIP BOM Restricted Field Value Report allows you to warn or prevent check in of a managed DSN and its auto-generated files (BOM, PDF, etc.) when **any** of the configured **CIP Restricted Part Field Values** are found in the **Restricted Part Field**. Before you can use the report, it must be configured as shown below.

You can select a CIP field and enter **up to five** restricted field values. You'll also need to choose which response type to use, either a **warning** with option to proceed with check in, or

an **error** that prevents the check in from continuing until the parts are replaced or the restricted field values are updated. When any of the field's restricted values are found in the managed design during check in, a CIP BOM Restricted Field Value Report pop-up window will appear, listing the CIP parts with restricted field values. The path forward depends on your configured response type.

Note: This feature requires you to configure EDM's connection to CIP so it can access CIP's list of fields to display.



Steps to Configure CIP BOM Restricted Field Value Report

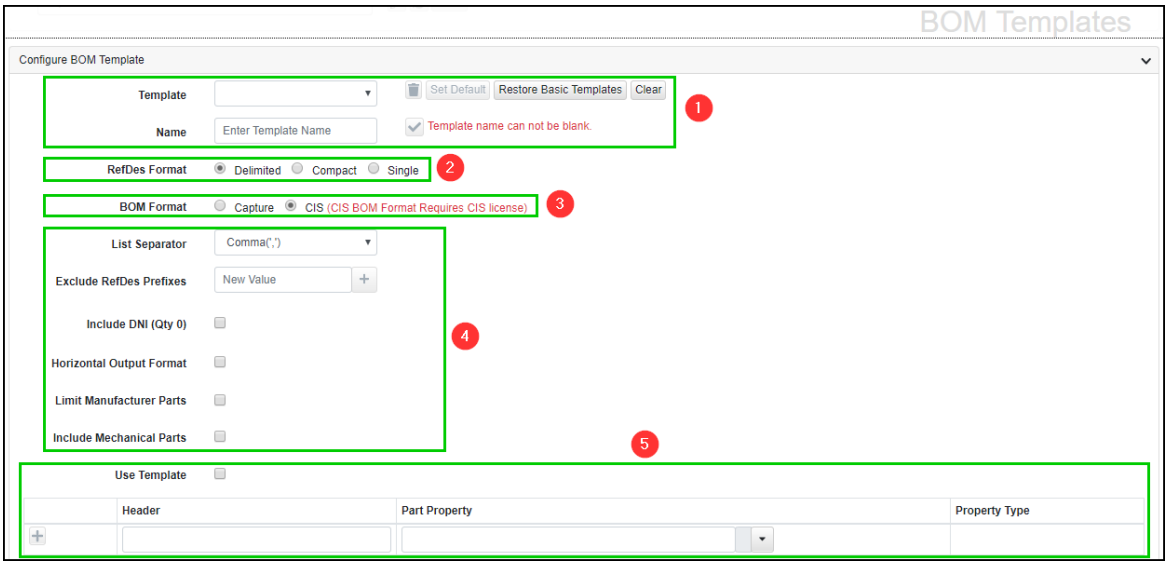
Each call-out number in the image above references a description of the screen area below.

To configure CIP BOM Restricted Field Value Report

1. Click the **Edit** button to begin. This report is disabled by default, as the Restricted Part Field is set to **Disable Field Value Report**.
2. Select the desired **Restricted Part Field** from the list of your CIP fields shown. If you do not see your CIP fields, you may need to update EDM's connection to CIP.
3. Enter **Restricted Part Field Values** one at a time, hitting Enter or clicking the **Add Picklist Item** button for each value entered. Repeat the process as needed, up to a maximum of 5 values. To remove a value, hover over it and click the Delete Picklist Item button.
4. Select the **Response Type** from the list.
   a. **Warning - Prompt to proceed with Check In** - This option will present the report and **prompt** the user to choose whether to proceed with the check in and auto-generation of files. This is the default response.
   b. **Error - Prevent Check In** - This option will present the report and **prevent** the user from proceeding with the check in and auto-generation of files until the issues are resolved.
5. Click the **Save** button to save your configuration.

3.1.2.3 Configure BOM Template Format

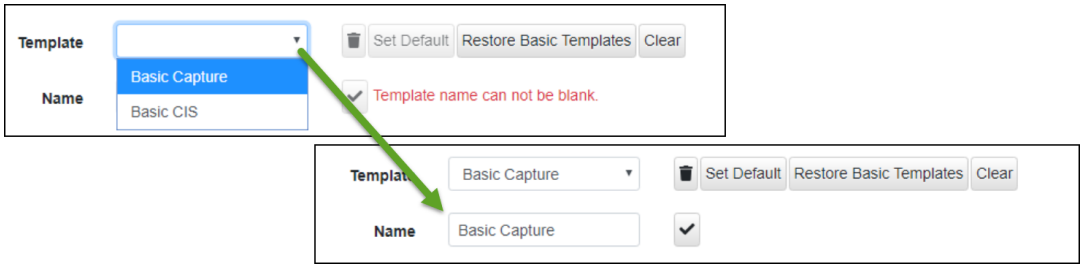
The figure below shows a "blank" BOM Template as it appears when you click the **Clear** button and select the **CIS** BOM Format.



Each call-out number in the image above references a description of the screen area below. When you create or modify a BOM Template, you need to specify settings in each of these sections.

1) Select an existing template that you want to edit, save, or rename as a new template

Select a saved BOM Template from the **Template** drop-down list as shown. There are two templates installed with EDM by default: **Basic Capture** and **Basic CIS**.



Select BOM templates

When you select a **Template** from the list, its name populates the **Name** text box. Template settings and the mapped property configuration display in the lower part of the form.

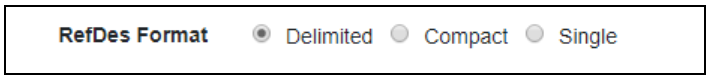
To save settings as a new template, type a new name into the **Name** text box and click the save button. You can reconfigure and rename the template if desired. You can save the template as your default BOM template by clicking the Set Default button.

When you attempt to save your configured settings to an existing template name, you are asked if you want to overwrite the settings. If you delete or modify a pre-installed template (**Basic Capture** or **Basic CIS)** and want to restore the original default template settings, click the **Restore Basic Templates** button.

To clear settings from the existing form, click the **Clear** button.

2) Select the Reference Designator (RefDes) Format

Choose a **RefDes Format** that determines whether BOM parts are listed by each occurrence of a part per line or grouped when used multiple times.



Reference Designator Options

The BOM Format options include the following:

- **Single:** Shows each reference designator on a single line.
- **Delimited** - Shows each part on a single line and reference designators separated by a comma. (e.g. C1,C2,C3,C6)
- **Compact** - Shows each part on a single line and contiguous reference designators as a range. (e.g. C1-C3,C6)

3) Select the BOM Format

Choose a **BOM format** to define whether **Capture** or **CIS** functionality is used during BOM creation.



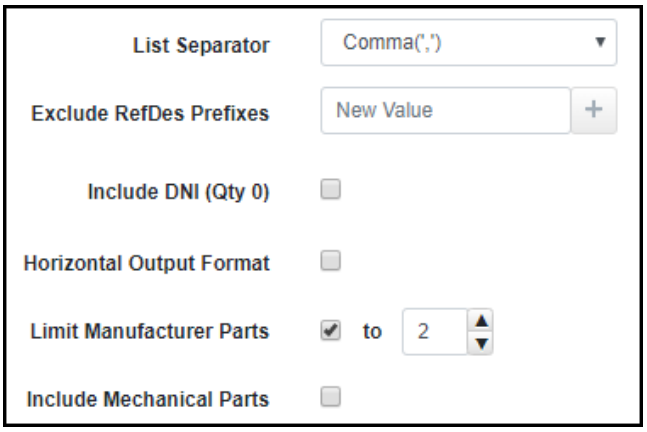
BOM Format Options

Parts and properties for a BOM may be created using the following formats:

- **Capture** - A BOM Template using the **Capture BOM** format does not require a Capture CIS license from the user generating the BOM during check in. BOM property values are derived exclusively from properties on the Capture design. Variant BOMs will not be generated.
- **CIS** - A BOM Template using the **CIS BOM** format requires a Capture CIS license from the user generating the BOM during check in. BOM property values are derived from the Capture design, which includes those transferred to design using CIS. When CIP is configured and enabled, BOM property values are transferred from CIP. The **CIS** BOM Format is required for generation of CIS Variant BOMs.

4) Specify Options for CIS BOM Format

When CIS BOM format is selected, additional configuration options appear as shown in the following figure. Configure the CIS options as desired.



CIS BOM Format Options

CIS BOM format options include:

- **List Separator** - Configured list separator will be used as the reference designator delimiter/separator in the CIS BOMs. The options are **Comma** or **Space**.
- **Excluded RefDes Prefixes** - Specifies which reference designator prefixes, if any, will be excluded from the CIS BOMs.
- **Include DNI (Qty 0)** - Specifies whether parts designated as **Do Not Stuff** (Do Not Install) in CIS design variants will be included in the CIS Variant BOMs with the quantity of **0** (zero). When unchecked, **Do Not Stuff** parts are simply excluded from the CIS Variant BOMs.
- **Horizontal Output Format** - Specifies whether to use a horizontal (separate column) output for CIP parts with multiple associated MFR/MFRPNs, keeping all of the manufacturer and distributor data for the part on a single row.
- **Limit Manufacturers** - The CIP CIS BOM Setup is required to enable this option. When checked, the number of manufacturer parts to include in the CIS BOMs is configurable. Specify the number of **Manufacturer Parts to Include** with each part on the CIS BOMs by clicking the up/down arrows or entering a valid value. When unchecked, all manufacturer parts are listed in the CIS BOMs.



- **Include Mechanical Parts** - Specifies whether to include mechanical parts in the BOM output for CIP parts with associated Mechanical Parts. The output is similar to the CIP BOM Export option to **Include Mechanical Parts** and the CIS BOM option to **Output Mechanical Part Data: Mechanical parts only**.

5) Map schematic part properties to BOM headings

The appearance of your BOM output may be a basic column list of part information in CSV format. Or you can enable **Use Template** to use a company BOM file as a template file to configure your BOM output.

Format column headers of BOM output

To create BOM output as a basic column list of parts without using a company template:

a) Enter a heading name into the **Header** field, which will determine the column header/name in the BOM.

b) Map the corresponding **Part Property** to the header name that was entered for the row.

c) Click the **Add** button to add the **Header** to the BOM template.

d) Click the **Save** button located next to the template name to save the template.

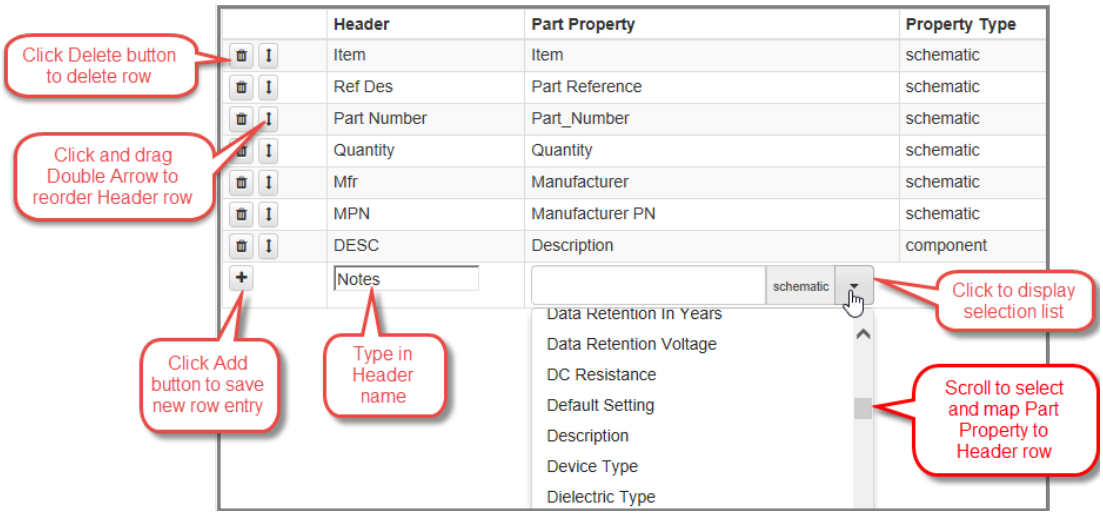
The figure below shows mapped headings and part properties that were mapped without use of a company template file. The Property Type column identifies the data source used to create the BOM. For example, data retrieved from the schematic is identified as schematic.

	Header	Part Property	Property Type
	Item	Item	schematic
	Ref Des	Part Reference	schematic
	Quantity	Quantity	schematic
	Value	Value	schematic

Reviewing Header Row Configuration of Your BOM Template

You can delete a header from your BOM template by clicking the **Delete** button in the row. You can re-order the **Header** rows by selecting the Double Arrow button and dragging the row to a new position in the Header column.

As shown in the image below, the last row of the **Part Property** column has a selection arrow that presents drop-down list you can use to add a new **Header** row to your BOM. When CIP is configured, the **Part Property** column lists CIP field names and their associated level to which they belong (Component, Manufacturer, or Distributor) in CIP. In addition, the Component View Name can be selected from the list, which will include the CIP View (e.g. Capacitors, Connectors, etc.) for each part shown on the BOM.



Mapping Part Properties to BOM Template Header

Format BOM output using a company BOM template file

To use a company BOM file to format your BOM template:

a) Select the **Use Template** checkbox.

b) Click the **Choose File** button, browse and select your template file, and click **Open**.

c) Click the **Upload** button.



The **Header Row** and **Data Row** number selections are displayed, along with the column and row information from the uploaded file is shown below. A configurable **Title Block Properties** section appears towards the bottom of the page.

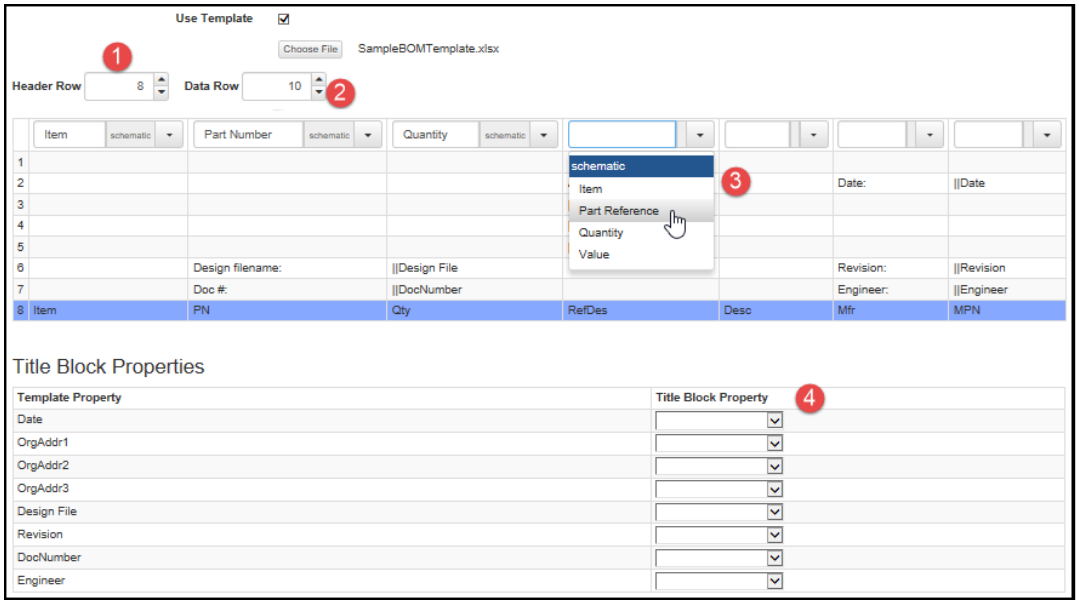
d) Verify and update the field mapping for BOM output as described below.

1. **Header Row** - identify and map the row in your template file that contains the column header names.
2. **Data Row** - identify and map the row in your template file where the BOM items should start when a BOM is auto-generated.
3. **Part Properties/Fields** - map the schematic or CIP field that you want to associate to each BOM header column. If you are mapping a property that is unique to your design, enter the name manually or select a property from the drop-down list. (call out 3, below).
4. **Title Block Properties** - map the Title Block Properties you want to automatically include in your BOM output.

   Note: If you want to transfer information from your Title Block to your BOM, you need

to enter two consecutive pipe characters "||" and the title block property name into an area of your BOM template. Sample BOM Template files are available through **EDM Online Help** files (e.g. in Capture, click EDM > Online Help).

The following figure shows how standard Capture properties/fields (Item, Part Reference, Quantity, and Value) are selected from the drop-down list. When CIP is configured, the list of properties from CIP are visible. Properties that are specific to your design can be entered manually by clicking in the text box above the pick list.

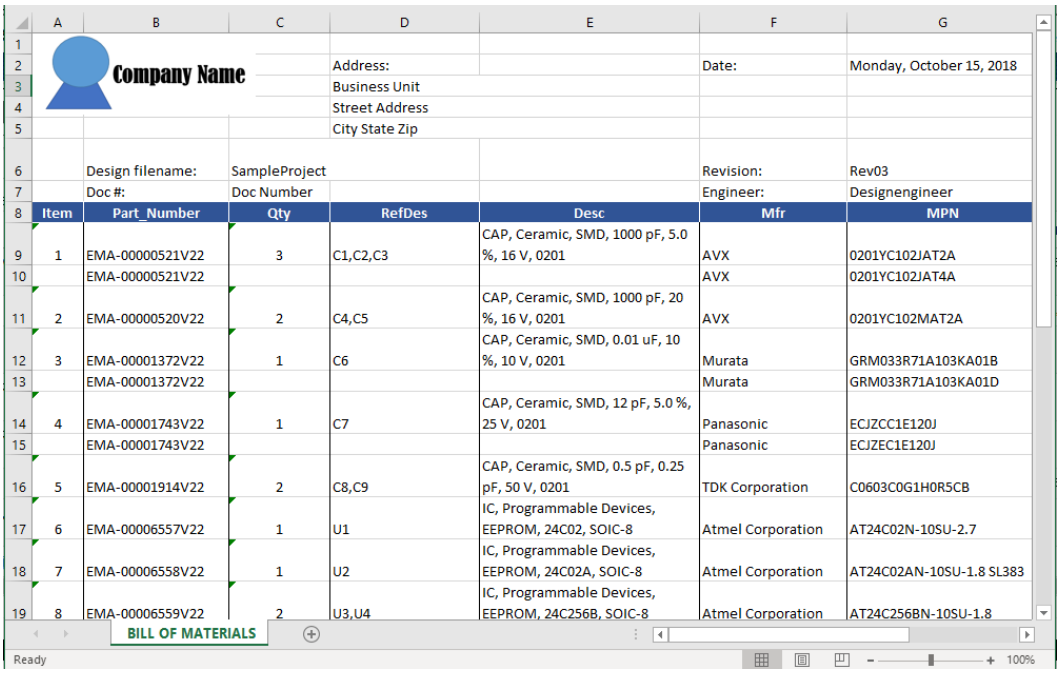


Note: Be sure Title Block Settings are configured before you attempt to configure the Title Block information of your BOM template. The following warning message appears when Title Block configuration settings are undefined or missing. This information is configured from the menu selection: **Admin > Configuration > Capture.**



Sample BOM output generated from a template file

The following example shows sample BOM output that is limited to two manufacturers per (company) part number.



	A	B	C	D	E	F	G
1							
2		Company Name		Address:		Date:	Monday, October 15, 2018
3				Business Unit			
4				Street Address			
5				City State Zip			
6		Design filename:	SampleProject			Revision:	Rev03
7		Doc #:	Doc Number			Engineer:	Designengineer
8	Item	Part_Number	Qty	RefDes	Desc	Mfr	MPN
9	1	EMA-00000521V22	3	C1,C2,C3	CAP, Ceramic, SMD, 1000 pF, 5.0 %, 16 V, 0201	AVX	0201YC102JAT2A
10		EMA-00000521V22				AVX	0201YC102JAT4A
11	2	EMA-00000520V22	2	C4,C5	CAP, Ceramic, SMD, 1000 pF, 20 %, 16 V, 0201	AVX	0201YC102MAT2A
12	3	EMA-00001372V22	1	C6	CAP, Ceramic, SMD, 0.01 uF, 10 %, 10 V, 0201	Murata	GRM033R71A103KA01B
13		EMA-00001372V22				Murata	GRM033R71A103KA01D
14	4	EMA-00001743V22	1	C7	CAP, Ceramic, SMD, 12 pF, 5.0 %, 25 V, 0201	Panasonic	ECJZCC1E120J
15		EMA-00001743V22				Panasonic	ECJZEC1E120J
16	5	EMA-00001914V22	2	C8,C9	CAP, Ceramic, SMD, 0.5 pF, 0.25 pF, 50 V, 0201	TDK Corporation	C0603C0G1H0R5CB
17	6	EMA-00006557V22	1	U1	IC, Programmable Devices, EEPROM, 24C02, SOIC-8	Atmel Corporation	AT24C02N-10SU-2.7
18	7	EMA-00006558V22	1	U2	IC, Programmable Devices, EEPROM, 24C02A, SOIC-8	Atmel Corporation	AT24C02AN-10SU-1.8 SL383
19	8	EMA-00006559V22	2	U3,U4	IC, Programmable Devices, EEPROM, 24C256B, SOIC-8	Atmel Corporation	AT24C256BN-10SU-1.8

BILL OF MATERIALS

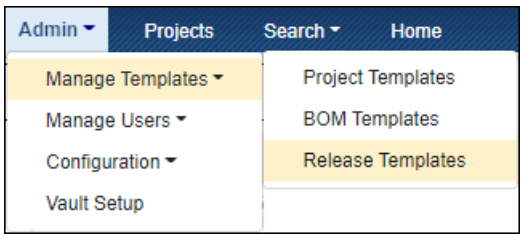
Ready 100%

Example of BOM Configuration

3.1.3 Manage Release Templates

Release Templates are used when Capture Design projects are packaged for release. Release templates ensure consistency of information for release packages. Multiple templates may be created and saved. Saved settings from the templates may be selected and used when creating a package for release for a project. Options saved in templates auto-populate when you select a release template during the package for release process.

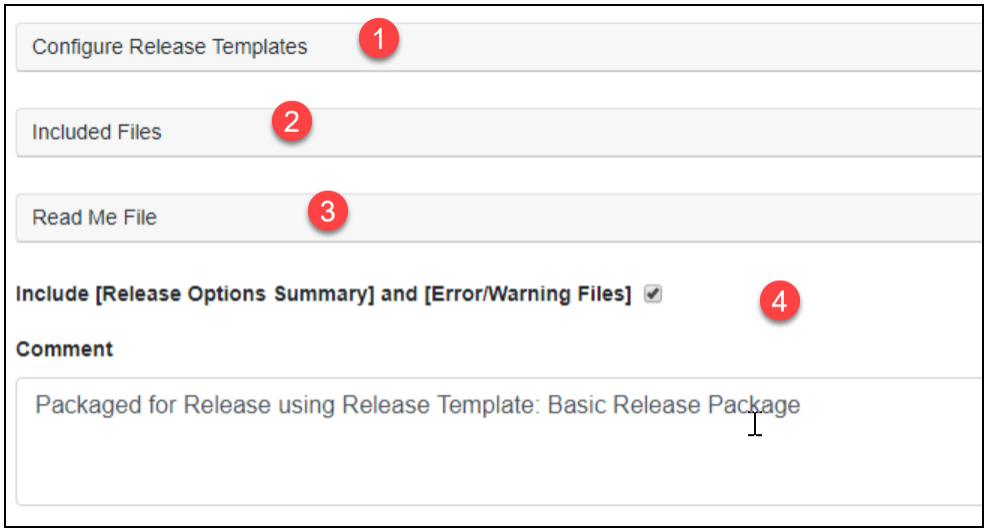
Select **Admin > Manage Templates > Release Templates** to configure Release Templates.



Accessing Release Templates

Release Template configuration options include:

1. Configure and save Release Templates
2. Select files to include in Release Package
3. Configure Read Me File
4. Configure default options for logs and comments



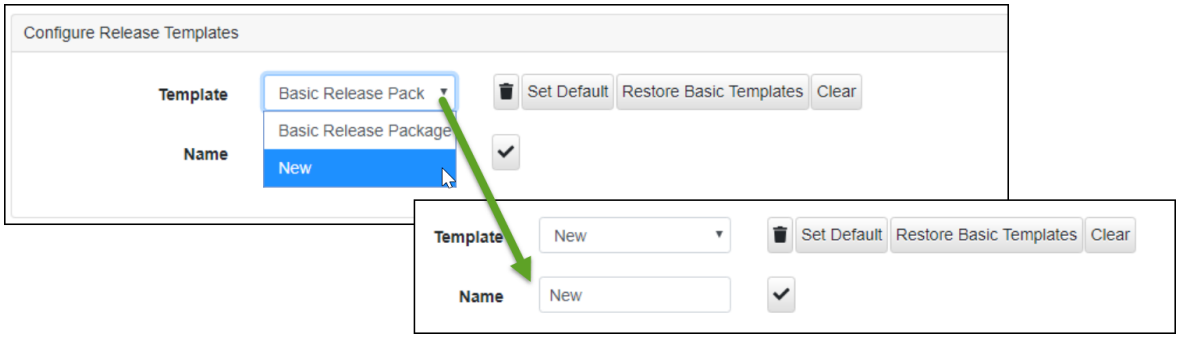
Configuration Options for Release Template

1) Configure and Save Release Templates

You may create a new template or edit an existing template. The Basic Release Package Template is pre-installed with EDM. You can use it "as is" or modify settings and save as a new template.

Options available to configure a Release Template include the following:

- **Template** - Drop-down list containing all saved release templates. Selecting a template from the list updates the user interface to show settings of the saved template.
- **Name** - The name in which the template will be save when you click the **Save** button.
- **Delete** - Deletes the template name shown in the Template drop-down list.
- **Set Default** - Sets the Template name shown in the Template drop-down list as the default in this Manage Release selection and for the designated Release folder of newly created projects.
- **Restore Basic Templates** - Restores the Basic pre-installed template to the original default settings.
- **Clear** - Clears the current settings in the form.



Select Release Package templates

To select a saved template, select from the drop-down list in the **Template** field and select a saved template.

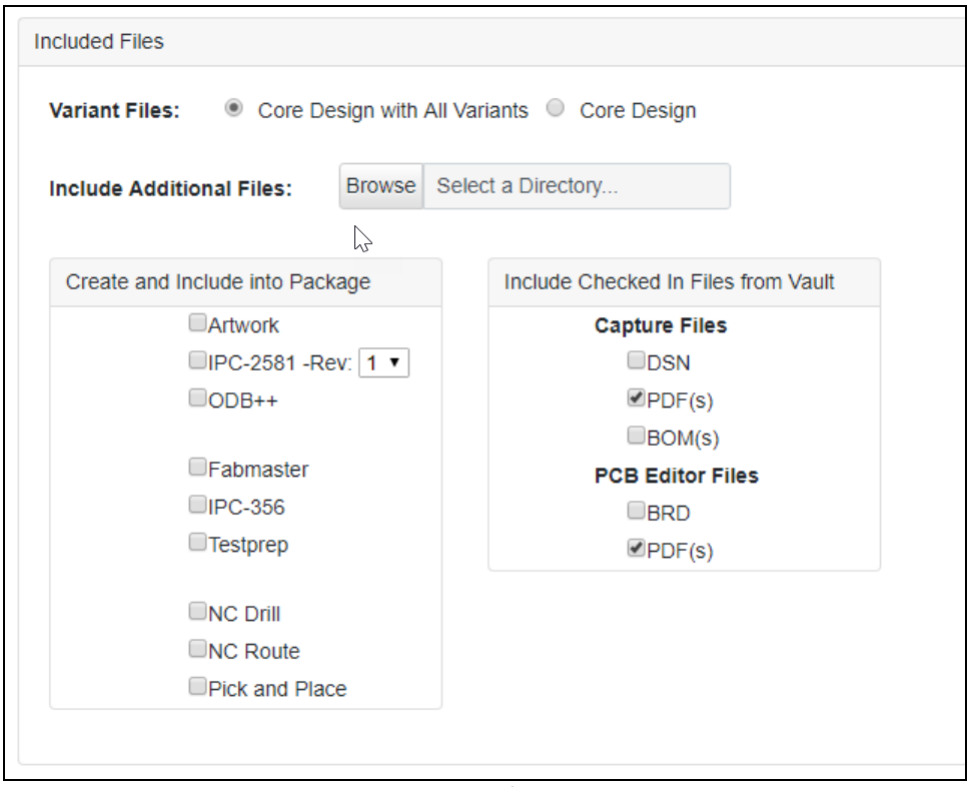
When you select a template from the menu, the template name populates the **Name** text box. Template settings are loaded and displayed.

To save settings as a new template, type a new name into the **Name** text box and click the **Save** button.

2) Select files to include in Release Package

You may define the Release Template to include the following types of files:

- **Variant Files:** Selection for inclusion of variant information from designs
- **Additional Files**: A directory consisting of files external from the EDM project may be specified. This location may be changed when creating a package for release.
- **Checked in Files from Vault**: Files checked in and maintained in the EDM project. Select files to include for your template configuration. A latest checked in version of the selected files will be included. These files will not be recreated.
- **Create and Include:** Manufacturing files to be created from existing project files. Selected items will be created when the saved template is applied.

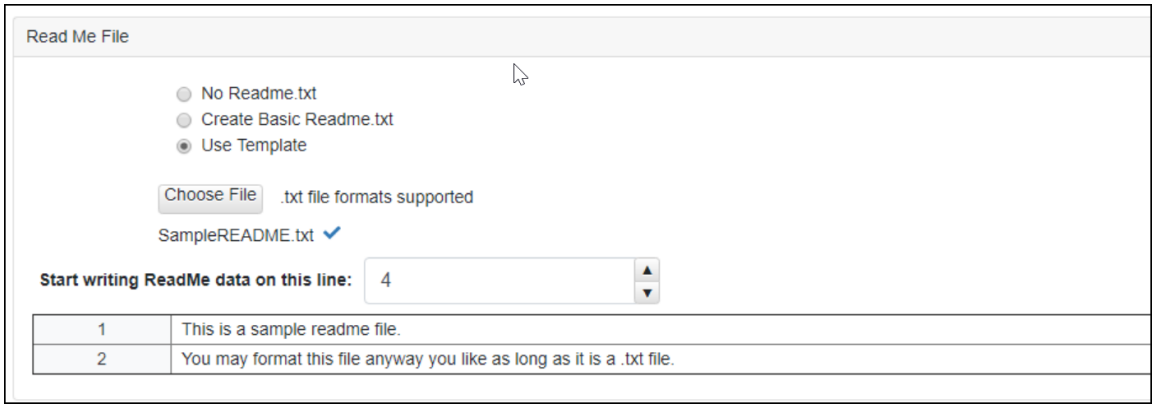


Files to Include for Release

3) Configure Read Me File

A Read Me file containing information of files included in the release package may be included in the Template. The following options are available to configure Read Me file.

- **No Readme.txt** - No Readme.txt file is created when the release template is applied.
- **Create Basic Readme.txt** - A basic Readme.txt file is created each time a release package is created using the configured template. A listing of files for the release package is included in the Readme.txt file.
- **Use Template** - The use Template option allows you to select and import a template. Your template file must be a text file format. After selecting and uploading your template, you may select the first line in which EDM auto-populate the file listing.



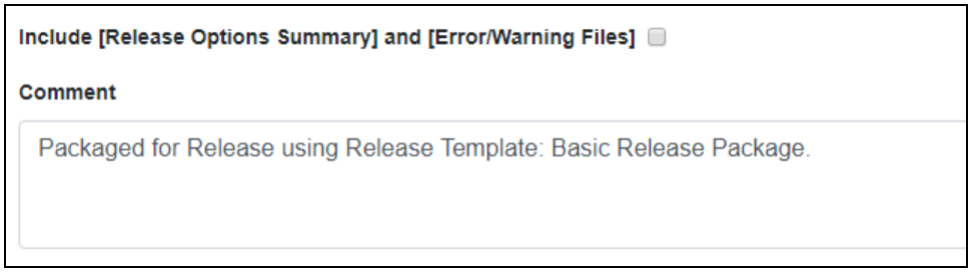
Read Me File Options

4) Configure default options for logs and comments

A summary of options selected for the creation of release package as well as warning and error logs may be included in the release package.

To configure the template to include logs and selections, select the checkbox for **Include [Release Options Summary] and [Error/Warning Files]**.

You may enter a default **Comment** to be used as the start for all comments when creating a release package using a Template.



After configuring the template options, you may Name and Save them as a new template or update/overwrite the currently-selected template.

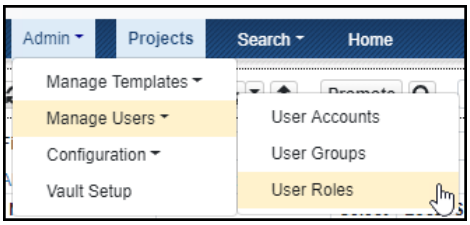
3.2 Manage User Accounts, Groups, and Roles

Only users who are assigned a role with the permission to Manage Users are able to access the **Manage Users** options. Users with Manage Users privileges can:

- Create and manage roles (Four default roles are preloaded)

- Add users
- Assign role-based permissions to users
- Disable users
- Monitor users who are logged into EDM
- Add user groups
- Manage user membership for user groups (add, change, delete users)

To access the User Accounts, User Groups, and User Roles menu options, an Admin selects **Admin** > **Manage Users**.



Accessing Manage Users Menu

3.2.1 Adding and Modifying Roles

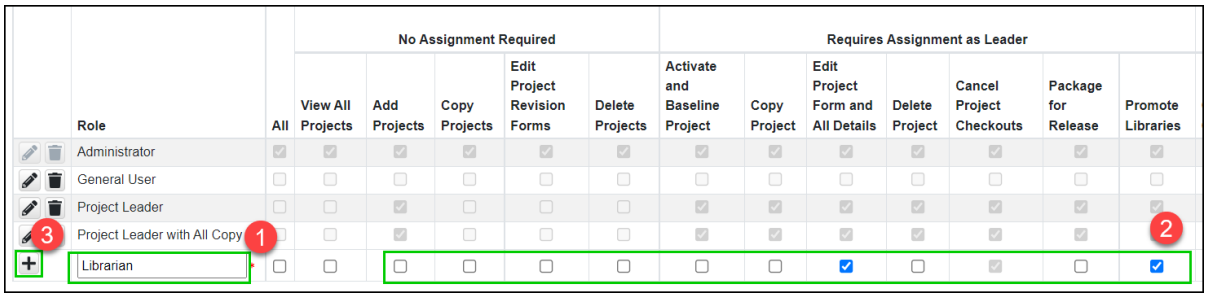
Role administration can be performed from the **Admin > Manage Users > User Roles** menu option. Roles can be added, modified or deleted from this menu. EDM permissions are divided into groups of tasks that pertain to non-project configurations and setting, all projects regardless of assignment, tasks when assigned as leader on a project, general tasks when assigned as a user. Any of these groups of permissions can be assigned to any role.

Roles can be created with any name. A user whose role has the Manage Users permission can grant permissions to a role when the role is created or modified at a later time.

Add Role

To add a new role,

1. Type the name of the role into a blank table row.
2. Select the permissions for the role. Multiple checkbox may be checked when selecting some of the checkboxes. Detailed information on permissions can be found in the Roles Permissions section.
3. Click the **Add** button. 



Modify Role

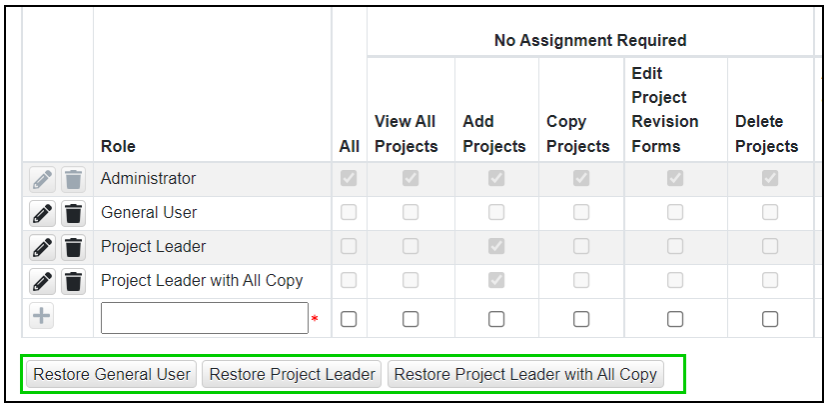
To modify role name or permissions to be granted with a role:

1. Click the **Edit** button next to the role you wish to modify.
2. Edit the role name or change permission .
3. Click the **Save** button to save changes. Or you may click on the **Cancel** button to dismiss changes.



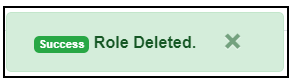
Restore Default Roles

The default roles may be modified. If modified and you need to restore these roles to their default settings, click on the applicable Restore button at the bottom as shown in the following figure.



Delete Role

Only roles that hasn't be assigned to users may be deleted. To delete a role, click on the **Delete** button. After you confirm deletion of the role, a success message displays and the role is deleted.



If any user are assigned to the role, an **Unable to Delete Role** message displays.



3.2.1.1 Default Roles

Four Default Roles are available. The following list describes the default roles of the **General User**, **Project Leader**, **Project Leader with All Copy**, and **Administrator** users during project development.

These predefined roles can be modified by anyone who is granted the Define Roles permission. To restore any of these roles to the default settings, click on the applicable Restore button at the bottom of the Admin > Manage Users > User Roles menu option. The Administrator role is a system role that cannot be changed by anyone.

1. **General User:**
    - View projects assigned to the user
    - View details for projects assigned to the user
    - Download items (files, pages, symbols, or footprint files) for projects in which the user is assigned
    - Check Out items (files, pages, symbols, or footprint files) specifically assigned to the user
    - Check In items (files, pages, symbols, or footprint files) that have been checked out by the user
    - Cancel Check Out of items (files, pages, symbols, or footprint files) that have been checked out by user
    - Perform all RefDes functionality except Lock Reference Designator
    - Perform all occurrence property functionality except Lock Occurrence Properties

2. **Project Leader:**
    - Same permissions as a user with **General User** role when assigned as **Project User** on a project.
    - When assigned as a **Project Leader** on a project, they will have the following additional permissions:
        - Create/Modify/Delete project templates
        - Create/Modify/Delete BOM Templates
        - Create/Modify/Delete Release Templates
        - Update EDM configuration using Admin >Configuration
        - Create/Modify/Delete projects, folders, Team Design folders, and Team Library folders
        - Delete items (files, pages, symbols, or footprint files)
        - Modify properties on items (files, pages, symbols, or footprint files)
        - Change project state (Activate and Baseline projects)
        - Copy project revisions to new revisions
        - Import design or board files into Capture projects in which the user is assigned
        - Create/import/delete schematics and pages in a Team Design folder
        - Import schematic library files
        - Import footprint files and step models
        - Add other leaders to projects (only if already assigned as a leader)
        - Assign/change users to projects and items (files, pages, symbols, or footprint files)
        - Promote schematic symbols to team libraries
        - Promote footprint and step model files

- Cancel Check Out of any items (files, pages, symbols, or footprint files) that have been checked out by any user
- Generate release packages for projects
- Lock Reference Designator
- Lock Occurrence Properties

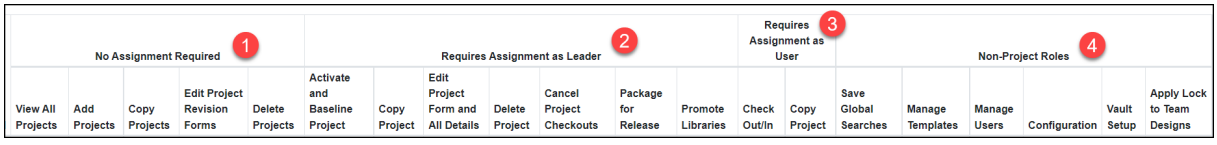
3. **Project Leader with All Copy**
   - Same permissions as a user with **General User** role when assigned as **Project User** on a project
   - Same permissions as a user with **Project Leader** role when assigned as **Project Leader** on a project
   - Copy project revisions to new revisions when assigned as **Project User** or **Project Leader** on a project

4. **Administrator**
   - Same permissions as a user with **General User** role when assigned as **Project User** on a project
   - Same permissions as a user with **Project Leader** role when assigned as **Project Leader** on a project
   - Same permissions as a user with **Project Leader with All Copy** role when assigned as **Project User** or **Project Leader** on a project
   - View all projects, regardless of Project User and Project Leader assignment
   - Update Project Leader and Project User assignments for any user on any project, regardless of Project User and Project Leader assignment

3.2.1.2 Role Permissions

Role Permissions are grouped into four categories.

1. **No Assignment Required** - Users granted permissions in this group can perform the selected action regardless of project assignment. Users granted any of the permissions in this group, other than View All Projects, can be assigned as a leader to a project.
2. **Requires Assignment as Leader** - Users granted permissions in this group can only perform the selected action when assigned to the project as a leader. Users granted any of the permissions in this group can be assigned as a leader to a project.
3. **Requires Assignment as User** - Users granted permissions in this group can only perform the selected action when assigned to the project as a user or a leader.
4. **Non-Project Roles** - Users granted permissions in this group can perform the selected action regardless of project assignment. The permissions in this group correspond to configuration and other setup that is independent of projects.



All users are able do the following:

- View the list of projects (in the Project Dashboard) in which the user is assigned.
- View Project Details for projects in which the user is assigned.
- Download items (files, pages, symbols, and footprint files) for projects in which the user is assigned.

The remaining actions users can take are determined by specific role permissions and project/item assignments.

Permissions that do not require assignment to a project

▼ View All Projects

A user with this permission is able to:

- View all projects in the project dashboard
- View the project details of all projects (i.e. open all projects)
- Be assigned to projects as a user, but may not perform actions which Require Assignment as User, such as **Check Out/In,** unless explicitly granted.

A user with only this permission is a read-only user who can review all projects, but may not contribute to them.

▼ Add Projects

A user with this permission is able to:

- Add Projects
- Be assigned to projects as a leader (i.e. appear on the list of available Assigned Leaders)
- Be assigned as an Available Project Leader for a vault, which is required to create projects within a given vault.

Additional permissions that will also be granted with **Add Projects** are:

- Activate and Baseline Project
- Edit Project Form and All Details
- Cancel Project Checkouts

▼ Copy Projects

A user with this permission is able to:

- Copy any project revision, regardless whether the user is assigned to the project.
- Be assigned to projects as a leader (i.e. appear on the list of available Assigned Leaders)

Additional permissions that will also be granted with **Copy Projects** are:

- View All Projects
- Edit Project Form and All Details
- Cancel Project Checkouts

▼ Edit Project Revision Forms

A user with this permission is able to:

- Edit Project Group Names

- Edit Project Names
- Edit any project revision and assign self as a leader
- Be assigned to projects as a leader (i.e. appear on the list of available Assigned Leaders)

Additional permissions that will also be granted with **Edit Project Revision Forms** are

- Edit Project Form and All Details
- Cancel Project Checkouts

▼ Delete Projects

A user with this permission is able to:

- Delete any project revision
- Be assigned to projects as a leader (i.e. appear on the list of available Assigned Leaders)

Additional permissions that will also be granted with **Delete Projects** are

- View All Project
- Edit Project Form and All Details
- Cancel Project Checkouts

Permissions that require assignment to a project as a leader

▼ Activate and Baseline Project

A user with this permission when assigned as a leader to a project is able to:

- Activate a project to enable check in and check out of files, symbols, pages or footprints.
- Baseline a project to prevent users from checking in or checking out files, symbols, pages or footprints.

Additional permissions that will also be granted with **Activate and Baseline a Project** are

- Edit Project Form and All Details
- Cancel Project Checkouts

▼ Copy Project

A user with this permission when assigned as a leader to a project is able to:

- Make a copy of any project revision that the user is assigned.

▼ Edit Project Form and All Details

A user with this permission when assigned as a leader to a project is able to:

- Create/Modify/Delete folders, Team Design folders and Team Library folders
- Delete items (files, pages, symbols, or footprints files)
- Modify properties on items (files, pages, symbols, or footprints files)
- Import design and board files into Capture projects (only when Check Out/In permission has also been granted)
- Create/import/delete schematics and pages in a Team Design folder
- Import schematic library files
- Import footprint files and step models
- Assign other users as leaders to projects (only if already assigned as a leader for the project)
- Assign/change users to projects and items (files, pages, symbols, or footprint files)

Additional permissions that will also be granted with **Edit Project Form and All Details** are

- Cancel Project Checkouts

▼ Delete Project

A user with this permission when assigned as a leader to a project is able to:

- Delete project in which the user is assigned as a leader

▼ Cancel Project Checkouts

A user with this permission when assigned as a leader to a project is able to:

- Cancel Check Out of any items (files, pages, symbols, or footprint files) that have been checked out by any user

▼ Package for Release

A user with this permission when assigned as a leader to a project is able to:

- Generate release packages for projects

Additional permissions that will also be granted with **Package for Release** are

- Check Out/In

▼ Promote Libraries

A user with this permission when assigned as a leader to a project is able to:

- Promote schematic symbols to team libraries
- Promote footprint and step model files

Permissions that require assignment to a project as a user

▼ Check Out/In

A user with this permission when assigned to a project either as a leader or user is able to:

- View projects assigned to the user
- View details for projects assigned to the user
- Download items (files, pages, symbols, or footprint files) for projects in which the user is assigned
- Check Out items (files, pages, symbols, or footprint files) specifically assigned to the user
- Check In items (files, pages, symbols, or footprint files) that have been checked out by the user
- Cancel Check Out of items (files, pages, symbols, or footprint files) that have been checked out by user

▼ Copy Project

A user with this permission when assigned as a user to a project is able to:

- Make a copy of a project that the user is assigned

Users with only this copy permission will be unavailable to be added as a leader. When making a copy, users who can also be added as leader to a project may add themself as a leader for the project copy.

Permissions not related to projects

- Save Global Searches

  A user with this permission is able to create and save global searches

- Manage Templates

  A user with this permission is able to create/modify/delete

  - Project Templates
  - BOM Templates
  - Release Templates

- Manage Users

  A user with this permission is able to:

  - Add and delete roles
  - Modify permissions on each role
  - Add/modify/delete user groups
  - Add/modify/delete user accounts (accounts may only be deleted if the account was never used in EDM)

- Configuration

  A user with this permission is able to:

  - Configure user fields
  - Configure Email Notifications (after it has been enabled as part of the installation process)
  - Disable EDM Client download link
  - Configure Titleblock property mappings for Capture Design and PCB Editor Design files
  - Define network locations to promote Schematics Symbols, PCB Footprint and Step Models
  - Define location and file name of Netlist Generation Configuration file
  - Define occurrence properties that EDM will need to manage
  - Define page delimiter for auto-numbering of pages in Team Designs
  - Define default behaviors when footprint files are promoted and deleted

- Vault Setup

  A user with this permission is able to:

  - Setup, configure or change the Vault for EDM

- Apply Lock to Team Designs

  A user with this permission is able to:

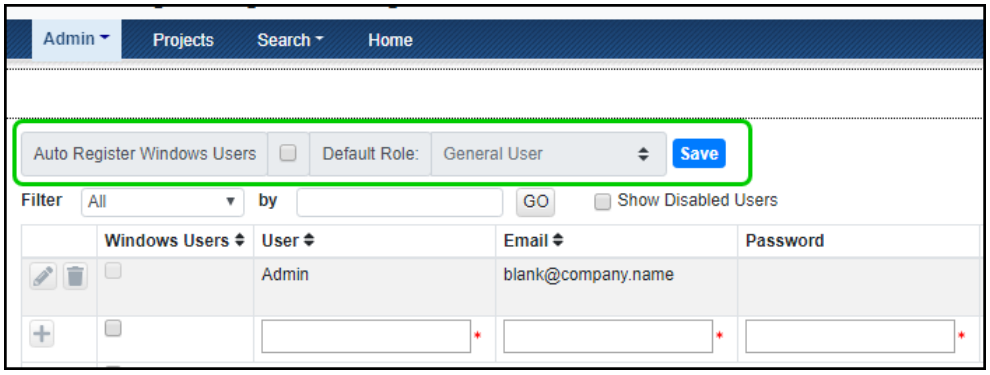
  - Lock Reference Designator
  - Lock Occurrence Properties

3.2.2 Adding and Modifying Users

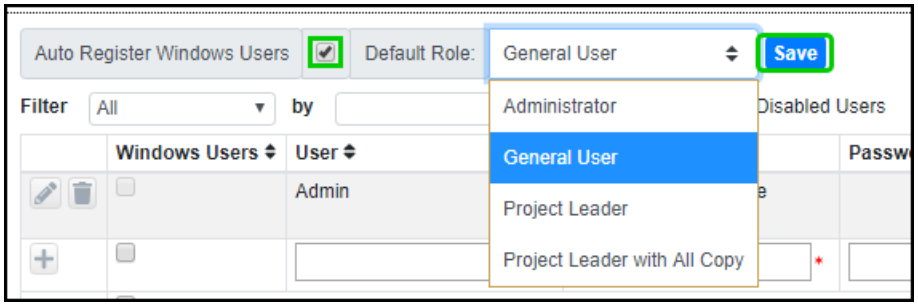
You can enable the Automatic Windows User Registration option or add EDM users manually. Users may be added as an EDM User or a Windows User from the User Accounts menu option. You can modify user information or delete users who haven't been assigned to any project. If there are users who were assigned but are no longer active, you can disable the user.

Enable Automatic Windows User Registration

The Automatic Windows User Registration option allows Windows domain users to automatically add themselves to **EDM** by clicking the Windows authentication button at the EDM log in screen. The first time they do so their Windows domain user will be added to the EDM Users list and their role will be auto-assigned based on the configured default role.



To enable this option, select the **Auto Register Windows Users** checkbox and then select the Default Role from the drop-down list. Click the **Save** button to save the settings.

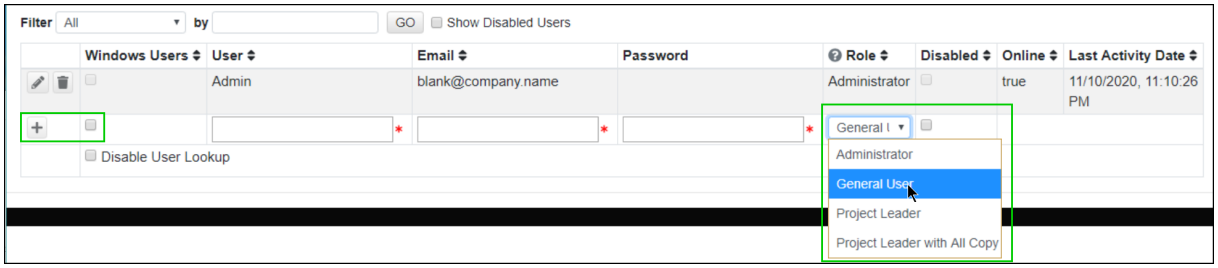


Enable Auto User Account Option

Add Users

To add an EDM user:

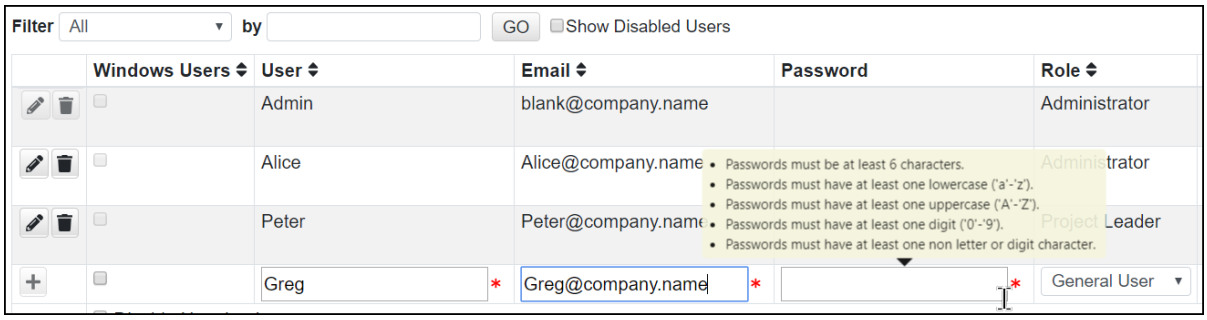
1. Enter the required information into a new line or table row. The required fields are **User** name, **Email** address and **Password**.



Add Users

Note: The email address must conform to standard email address format. The password must meet specific criteria, which displays as you type the password as shown in the *EDM User Password Policy* figure that follows.

2. Select a role for your user. **General User** is preselected by default.
3. Enter all required information into the input fields.
4. Click the **Add** button to add the user.



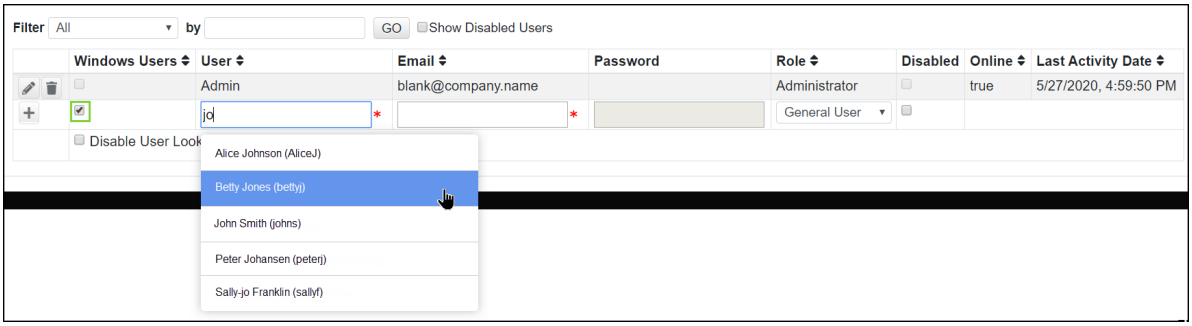
EDM User Password Policy

To add a Windows user:

1. Select the **Windows User** checkbox.
2. Type the user name into the User input field. As you start to type the user name, a list of Windows users display as letters of their name match the letters you type. This enables you to select the desired user from the pop up list.

Note: If there are many Windows Users in your company listing, you can select **Disable User Lookup** and type in the exact Windows username. Additionally, the email address of the user is auto-filled. You do not need to enter the password for Windows users.

3. Select the role for the user. Available Default roles are **Administrator**, **Project Leader**, **Project Leader with All Copy** and **General User** (preselected).
4. Click the **Add** button to save the user.



Add Windows User

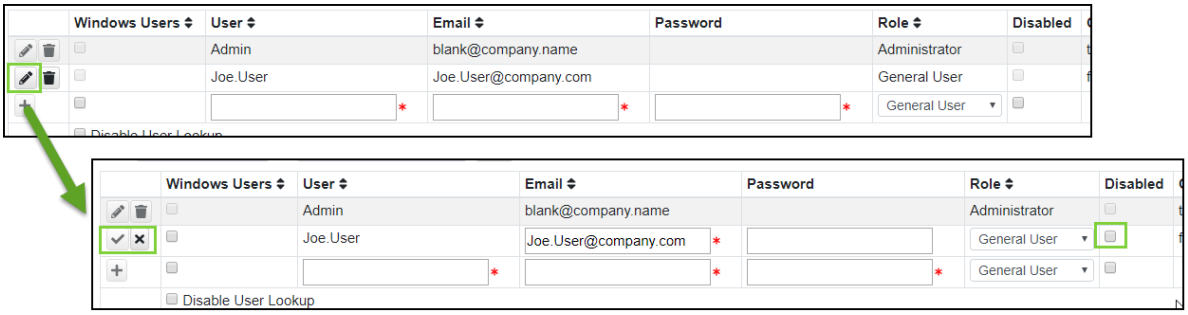
Modify or Disable User

To modify user information:

1. Click the **Edit** button next to the user you wish to modify.
2. Edit the user input fields as desired. You may change any field other than the **User**. You may update **Email** address, **Password**, or user **Role**.
3. Select the **Disabled** checkbox if you want to disable a user.

   Note: Changes must meet the original requirements for that field. Once changes satisfy the field requirements, the **Save** button becomes enabled.

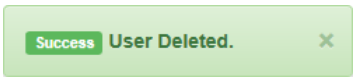
4. Click the **Save** button to save changes. Or you may click on the **Cancel** button to cancel changes.



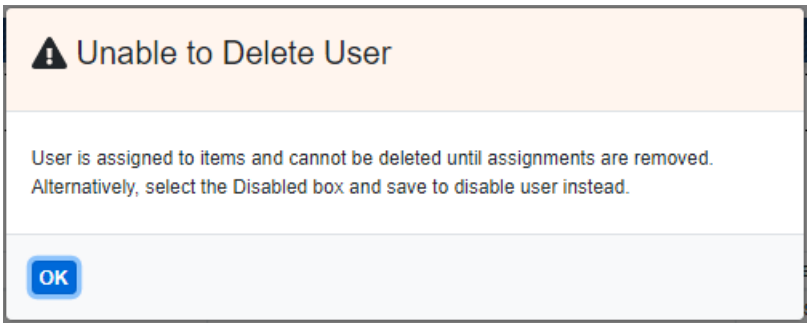
Edit User

Delete User

To delete a user who hasn't been assigned to a project or other items, click on the **Delete** button. The user is deleted and a success message displays.



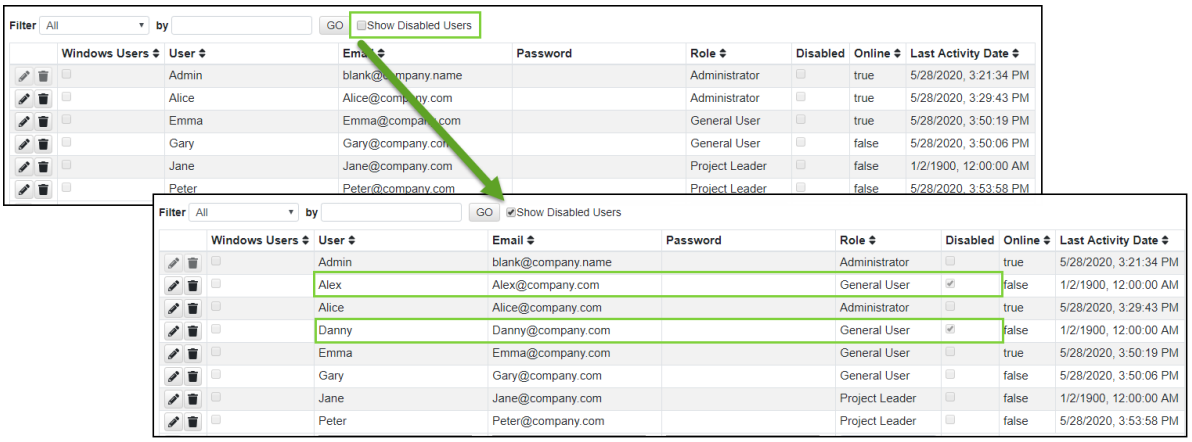
If a user is assigned to a project or items, an **Unable to Delete User** message displays. You may **Disable** the user in lieu of deleting them, otherwise all of the user's assignments need to be removed before deleting the user.



Locate, Sort, or Hide Users

To display or hide disabled users, you may select or deselect the **Show Disabled Users** checkbox. Selecting the checkbox updates the list of users who are disabled as shown in the following figure. To locate a user quickly, type the user name or part of the user's name into the Filter Box and click **GO**.

You may change the appearance of the user list by selecting the column heading to sort the column.



Show Disabled Users Enabled

3.2.3 Adding and Modifying User Groups

From the **User Groups** menu option, you can add and delete user groups. You can also modify the user group name as well as the member assignment to user groups. User groups must be assigned at the user Projects level before being assigned to files or folders.

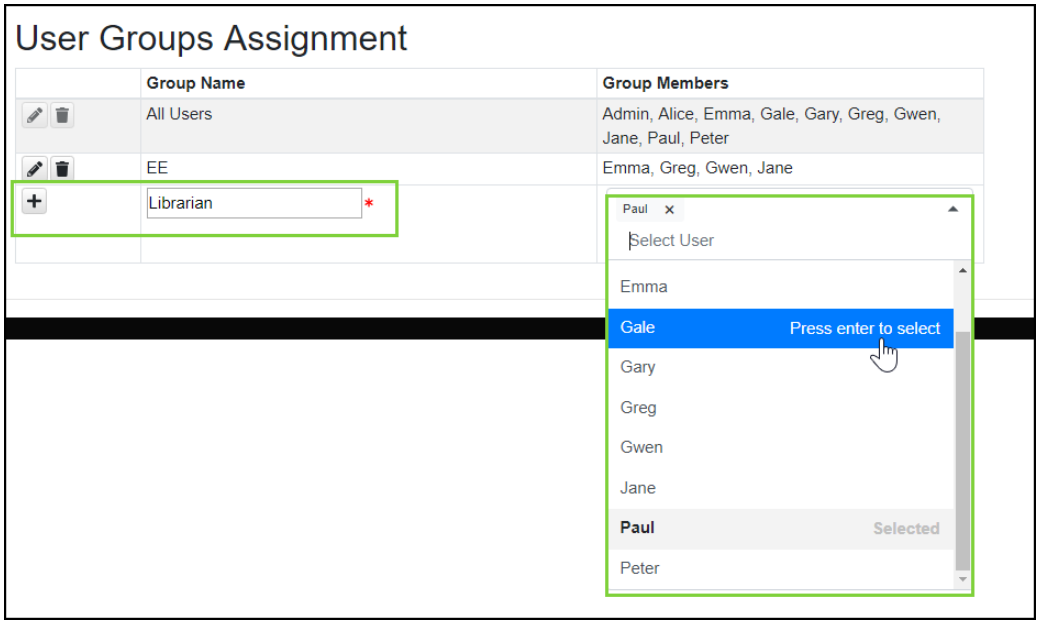
Add User Groups

To add a new user group

1. Enter the required information into the input fields of a new line.
    - Type a name for the new group into the **Group Name** field.

- Click the **Select User** field to select the users you want to assign to the group as shown in the following figure.

2. Click the **Add** button when finished.



User Groups

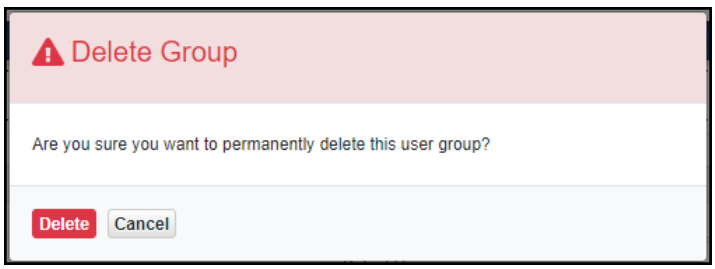
The **All Users** group is a built-in default group that contains all active users in EDM.

Delete User Groups

To delete a user group

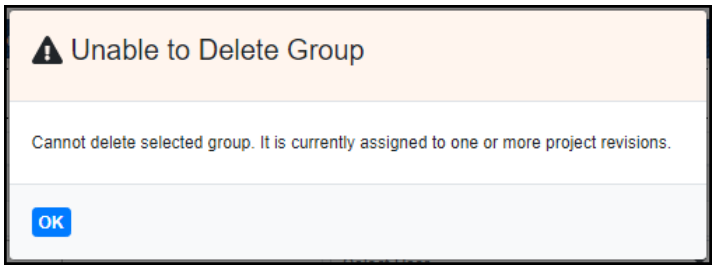
1. Click the **Delete** button next to the group you want to delete. The *Delete Group* confirmation message displays as shown below.

2. Click the **Delete** button to confirm the deletion. Or click the **Cancel** button to cancel.



Delete Group Confirmation Message

Note: You must remove assignments of a group from all projects before deleting the group. When you attempt to delete a user group that still has assigned projects, the following message opens.



Unable to Delete User Group

Modify Group Name or Membership

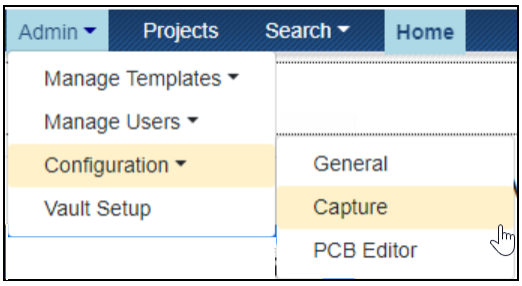
To modify user group name or membership for a group

1. Click the **Edit** button next to the group that you want to modify.
2. Click in the **Group Name** field and type a new name to modify the name.
3. Click in the **Group Members** field to add or remove users.
4. Click the **Save** button when finished. Or click the **Cancel** button to cancel your changes.

3.3 Configuration

EDM configuration settings help you standardize properties of title blocks, libraries, projects, and files.

The **Configuration** menu provides access to settings associated to general functionality of EDM as well as settings that are specific to the PCB Editor and Capture Schematics. Users whose role has been granted Configuration permission can access the General settings from any EDM user interface. You need to open EDM from Capture when you are configuring Capture related settings. When configuring settings associated to PCB Editor, open the EDM Desktop application while connected to your PCB Editor. Many settings need to be configured before feature functionality is enabled. For example, schematic symbols cannot be promoted until the **Promoted Schematic Symbol Library Locations** are defined.



Accessing EDM Configuration

The **Configuration** menu includes the following:

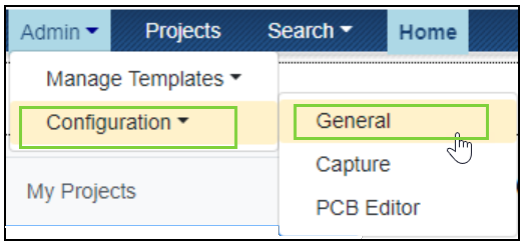
- General Configuration – Customizable Fields, Email Notifications, Build BOM Search, Show EDMClient Download link, Warning Durations.
- Capture Configuration – Title Block Field Mappings, Promoted Schematic Symbol Library Locations, Netlist Configuration, Occurrence Properties and Capture preferences, OrCAD Capture Preferences for auto-generated pages
- PCB Editor Configuration – Board Title Block Field Mappings and configuration of the footprint and step model promote library locations, which may be configured to be a single centralized UNC shared location for all footprint projects or a separate shared location unique to each footprint project.

3.3.1 General Configuration

Users assigned roles with the permission to update Configurations may customize the configuration of user defined fields, setup automated email notifications, and extract BOM information from BOMs created prior to EDM version 4.2.0 to enable BOM search. General settings that are customized are applied to all projects, files, symbols, pages or footprint and step files.

- Customizable Fields – Standard and custom (user defined) fields are available for configuring the properties of footprints, files, projects, team designs, and team libraries.
- Email Notifications – automated email messaging may be setup to notify users when the status of a file or project changes.
- Build BOM Search – index BOMs generated from Capture to enable the BOMs to be searchable.
- Show EDMClient Download Link - show/hide the internal download link for the EDM Client installer that matches the installed version of EDM Server.
- Rename Warning Duration - set the duration of warnings when project or project group names are changed.

Customization of General settings is available from the menu selection: **Admin** > **Configuration** > **General**.



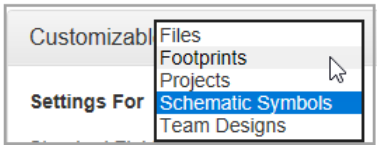
Admin > Configuration > General Menu

3.3.1.1 Customizable Fields

Users assigned roles with the permission to update Configurations can configure EDM settings for a project, footprint, file, team design and team library. A sample of default settings are pre-loaded when installed. You may customize fields to match your company's design processes. Each customizable setting has a Status field and ten user defined fields that can be configured to provide a pick list. The Project setting also includes the Revision field that can be configured.

To edit field configuration

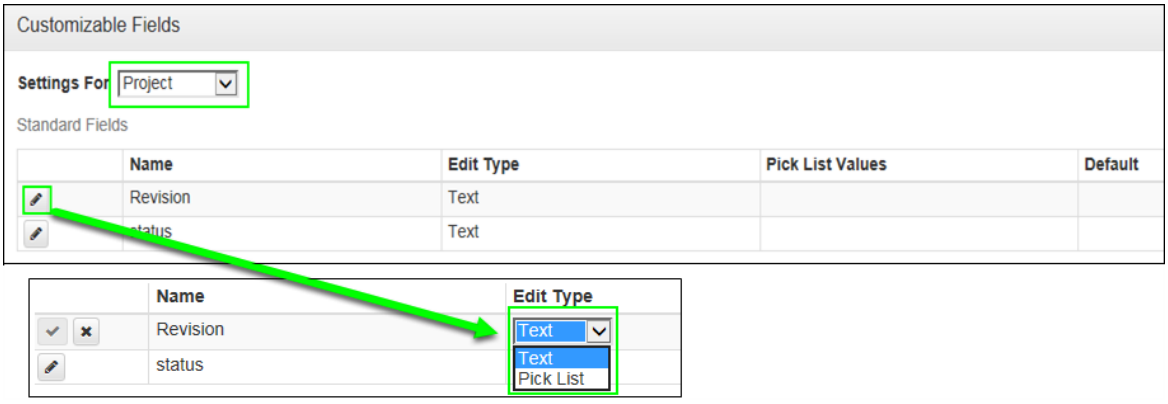
1. Select the **Admin > Configuration > General** menu items.
2. Select the desired option from the drop-down list in **Settings For.** The setting you select determines whether you are changing fields for a File, Footprints, Projects, Schematic Symbols, or pages in the Team Design folder.



Select Settings For

a) To edit the Revision or status field:

i. Click the **Edit** button next to the field you want to configure.



Edit Revision for Project Settings

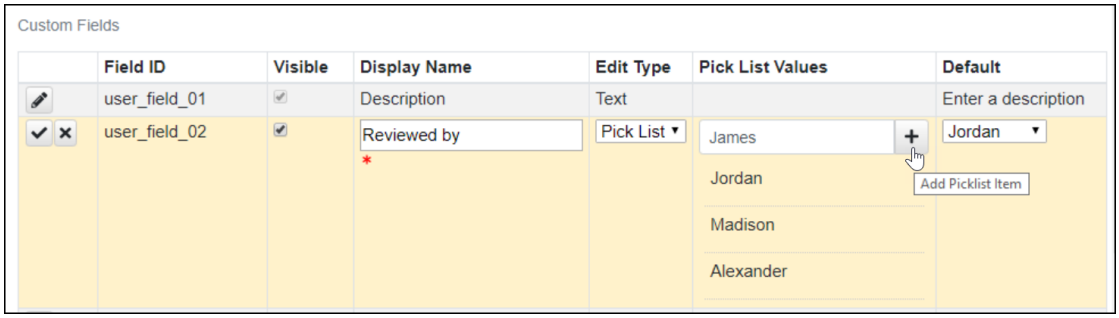
ii. Configure one or more of the following field characteristics:

- **Edit Type** - The edit type controls whether the user is to manually enter text data or choose data from a pick list.
- **Pick List Values** - If you select Pick List as the Edit Type, enter values that are available for a user to choose. Users will only be able to select options in the pick list values when they create a project, or check in a file, footprint, page or symbol.
- **Default** - The default value is the initial value of the data when a new project is created or new item is checked in. Default values are required for Pick Lists. They can also be specified for Edit Types of Text.

iii. When done, click the **Save** button to save your configuration or click the Cancel button to discard changes.

b) To configure one or more attributes of a Custom Field (user_field):

i. click the **Edit** button next to the custom field you want to configure.



Sample User Fields

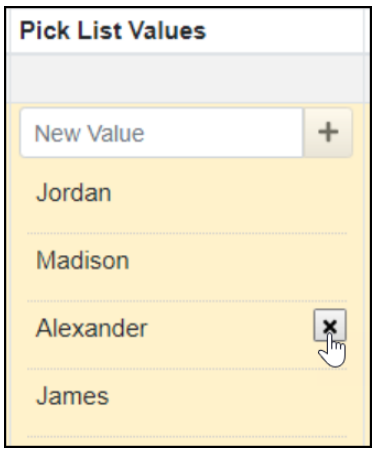
ii. Configure one or more of the following field characteristics:

- **Visible** - enable or disable checkbox(es) to change whether the field displays in the project dashboard, project details,footprint project, team design or team library details views. The configured field will display when you create or edit a project or when you check in a file, footprint, page, or symbol.
- **Display Name** - change the display name to change the appearance of the field name in the project dashboard, project details,footprint project, team design or team library details views.
- **Edit Type** - select an edit type to control whether users manually enter data or choose data from a pick list. The options allowed are Text or Pick List.
- **Pick List Values** - if you are setting up a pick list, enter a new value into the text box and press your keyboard Enter key. Or you may click the "+" button. If

you are using a pick list the first value entered becomes the initial default value unless you select another value from drop-down list of entered values. Users will only be able to select an option from the pick list when creating a project or when checking in a file, footprint, page or symbol.

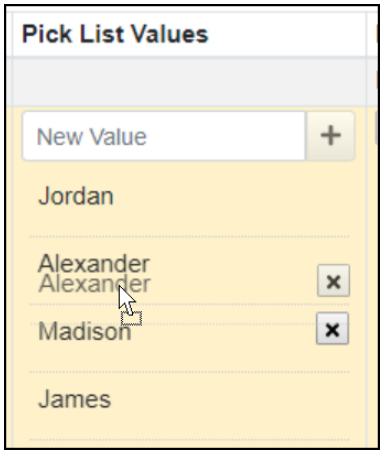
- **Default** - enter a default value that will be the initial value of the data when a project is created or when a file, footprint, page or symbol is checked in. Default values are required for Pick Lists. They can also be specified for Edit Types of Text.

To remove a value from the pick list, hover over the list item and click the **Remove** button .



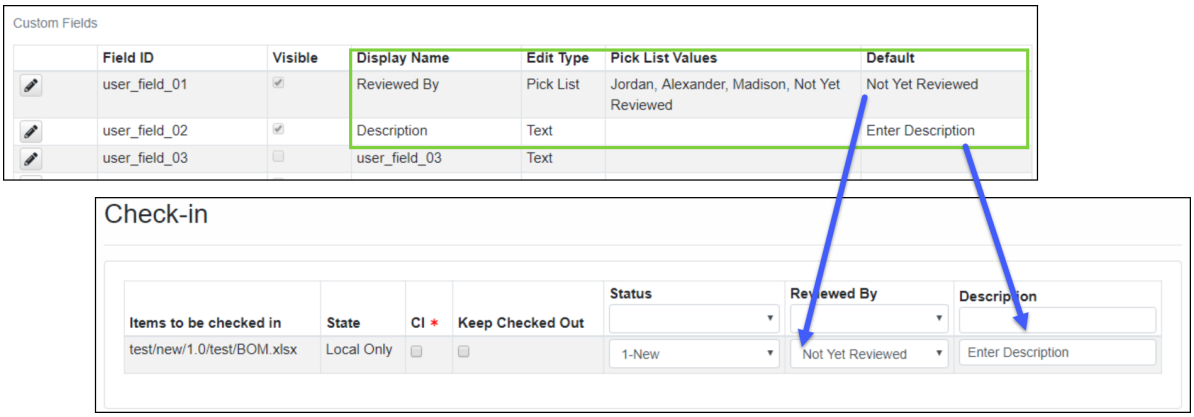
Remove Pick list Item

Select and drag the pick list item to move it to a different position.



c) After you complete your updates, select the **Save** button to save your changes or click the **Cancel** button to discard changes.

The saved User Field will appear the next time you create a project or check in a file, footprint, page or symbol as shown in the example in the following figure.



Sample User Fields Shown for Check In

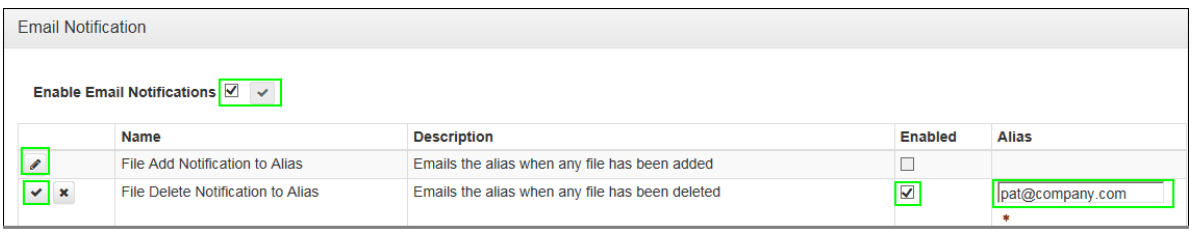
3.3.1.2 Email Notification

The Email Notification feature allows users to receive emails when design pages or projects are added, modified, or deleted from EDM.

Note: Before you begin using Email Notifications, the SQL Server Administrator must enable and configure Database Mail on the server. This setup occurs outside of Engineering Data Management.

Users assigned roles with the permission to update Configurations can configure Email Notifications for individual users or email aliases. To enable Email Notifications, navigate to the **Admin > Configuration > General** menu and scroll down the page to **Email Notification**.

To enable each type of email option, first select the **Enable Email Notification** checkbox and click the **Save** button. Then click the **Edit** button. Select the **Enable** checkbox that is located next to the description and type in the email or alias. To configure an options that supports an Alias, you may use an email distribution alias or an individual email address. Click the **Save** button to save the notification. Or click the **Cancel** button to cancel the option.



Enable Email Notifications

The following table lists and describes all Email Notifications the Administrator can setup in EDM.

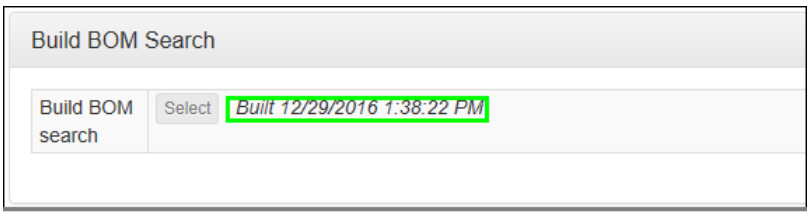
Email Notification Name	Description
File Add Notification to Alias	Emails the alias when any file has been added
File Delete Notification to Alias	Emails the alias when any file has been deleted
File Delete Notification to Assigned Users	Emails assigned users when assigned files have been deleted
File Update Notification to Alias	Emails the alias when any file has been updated
File Update Notification to Assigned Users	Emails assigned users when assigned files have been updated
Library Location Add Notification to Alias	Emails the alias when a promoted library location is added
Library Location Delete Notification to Alias	Emails the alias when a promoted library location is deleted
Page Add Notification to Alias	Emails the alias when any page has been added
Page Add Notification to Assigned Users	Emails assigned users when assigned pages have been added
Page Delete Notification to Alias	Emails the alias when any page has been deleted
Page Delete Notification to Assigned Users	Emails assigned users when assigned pages have been deleted
Page Update Notification to Alias	Emails the alias when any page has been updated
Page Update Notification to Assigned Users	Emails assigned users when assigned pages have been updated
Project Add Notification to Alias	Emails the alias when any project has been added
Project Delete Notification to Alias	Emails the alias when any project has been deleted
Project State Change Notification to Alias	Emails the alias when any project state has been changed
Project State Change Notify Assigned Users	Emails the assigned users when any project state has been changed
Project Update Notification to Alias	Emails the alias when any project has been updated
Project Update Notification to Assigned Users	Emails assigned users when assigned projects have been updated
Promote Symbol to Library Notification to Alias	Emails the alias when a symbol is promoted into a library
TeamDesign Freeze Notification to Alias	Emails the alias when a project is frozen
User Added Notification to Alias	Emails the alias when a user has been added
User Disabled Notification to Alias	Emails the alias when a user has been disabled
User Password Notification to Alias	Emails the alias when a user password has been updated
User Role Updated Notification to Alias	Emails the alias when a user role has been updated

3.3.1.3 Build BOM Search

This feature is only applicable for pre-existing BOMs that were created prior to EDM version 4.2.0. BOMs that were created in EDM versions later than 4.2.0 are automatically searchable and may be retrieved without the indexing capabilities of this feature.

This BOM Search feature allows you to conduct a search for BOM created in EDM. If you have already created BOMs prior to updating to a version of EDM that includes the BOM Search feature, you can index them so that they will appear in BOM Search results by selecting the Build BOM Search **Select** button. The **Select** button may only be clicked once and will be unusable once you have indexed the pre-existing BOMs.

Once you have clicked **Select,** a date and time stamp will display, indicating when the BOMs were indexed.



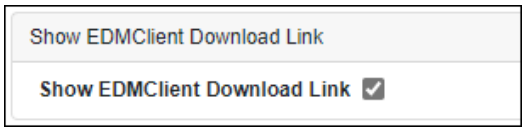
Build BOM Search Created

3.3.1.4 Show EDMClient Download Link

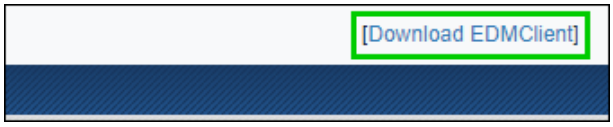
The EDM Client Download Link allows users to download the EDM Client installer that is compatible with the installed version of the EDM Server (Web Application and Database). The download link will be available from the landing/home page before logging in, and from any EDM page when logged in. The link will be enabled by default for new EDM databases, but can be disabled if your company does not want the download to be available for all users.

To enable the EDMClient Download Link

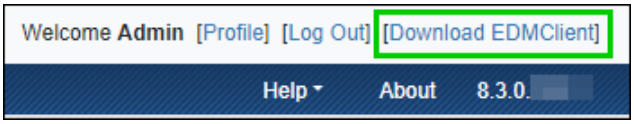
1. Select the **Admin** > **Configuration** > **General** menu, then click the **Show EDMClient Download Link** section to expand it.
2. Check the box to **Show EDMClient Download Link** or uncheck it to hide the link.



After making this change, the **Download EDMClient** link appears when you refresh the page. The download link will be available from the landing/home page before logging in, and from any EDM page when logged in.



EDMClient Link on EDM Landing/Login Page



EDMClient Link on any EDM Page when Logged In

To download the EDM Client for the corresponding installed EDM Server, open EDM from a standard browser and click the **Download EDMClient** link.

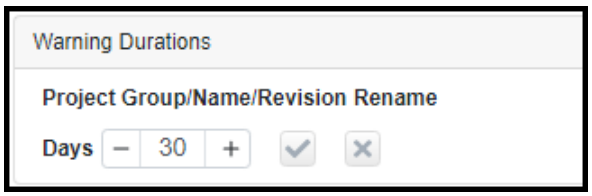
3.3.1.5 Warning Durations

After renaming Project Groups, Project Names, or Project Revisions in EDM, the local workspace on each user's machine will continue to reference the original group, project, or revision names. This may be confusing to users who did not perform the rename operation and were not made aware of it by the user who did. The users will need to open and download the affected project revs before continuing to work on them. EDM will attempt to complete the rename automatically on the local workspace for the user who performed the rename. If it fails, it will inform the user they will need to perform them manually.

To help with this potential issue resulting from a rename operation, EDM provides a notification warning next to Project Groups, Project Names, and Project Revisions that were changed. Refer to Edit Project and Group for more info, including screenshots showing the icon and text displayed for the warning notification.

The duration of the warning is configurable in the **Admin** > **Configuration** > **General** > **Warning Durations** section.

Under **Project Group/Name/Revision Rename**, select the **Plus** + or **Minus** − to increase or decrease the notification duration. Select the **Save** ✓ button to save the setting or ✗ to cancel.



3.3.2 Capture Configuration

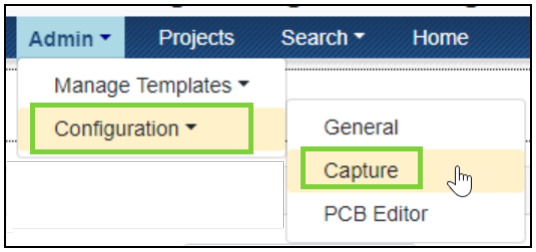
Users assigned roles with the permission to update Configurations who open EDM from Capture may configure title block properties, specify the location of promoted libraries and the netlist configuration file, as well as modify other Capture preferences.

Capture configuration settings that may be modified include:

- Capture Title Block – control title block property mappings
- Promoted Schematic Symbol Library Locations – promote symbols to a centralized library so that all users may have shared access to promoted items

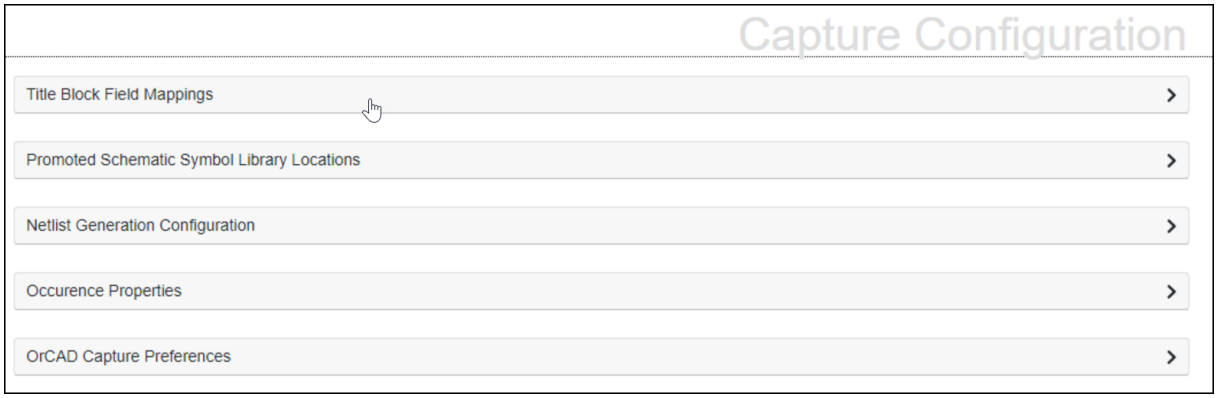
- Configure Netlist Generation – specify the default netlist configuration file for creation of netlists
- Occurrence Properties – specify the occurrence properties for restoration in hierarchical designs
- OrCAD Capture Preferences – control blank page numbering

Customization of Capture configuration settings is available from the menu selection: **Admin** > **Configuration** > **Capture**.



Admin > Configuration > Capture Menu

Configuration options may be collapsed or expanded by clicking on the option titles.



3.3.2.1 Schematic Title Block Settings

EDM allows users assigned roles with the permission to update Configurations to set up the Title Block Field Mappings. Sample default Title Block settings are pre-loaded when installed. You may change settings to match properties and mapping in your company's design Title Block.

Note: This setting can be changed only when EDM is launched inside Capture.

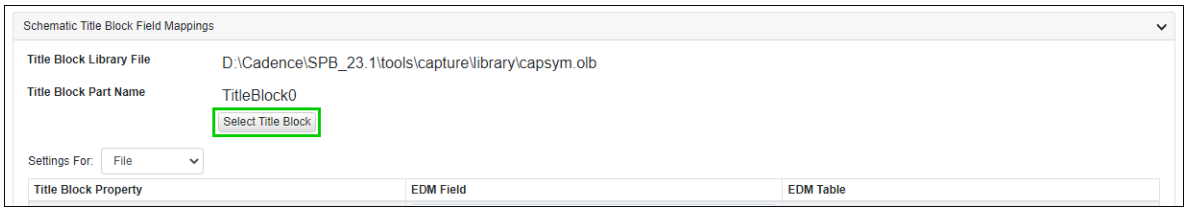
When title block mappings are configured, users may choose an option that automatically updates their design with values from mapped fields. Projects that have Team Design folders have a separate design mapping that are used when a design is checked in.

All users will need to have the same setting in Capture Design Template for Title Block (**Options > Design Template > Title Block tab**). When more than one Title Block template is available for use, be sure to verify that the name of each Title Block property of one template matches the property names of the other.

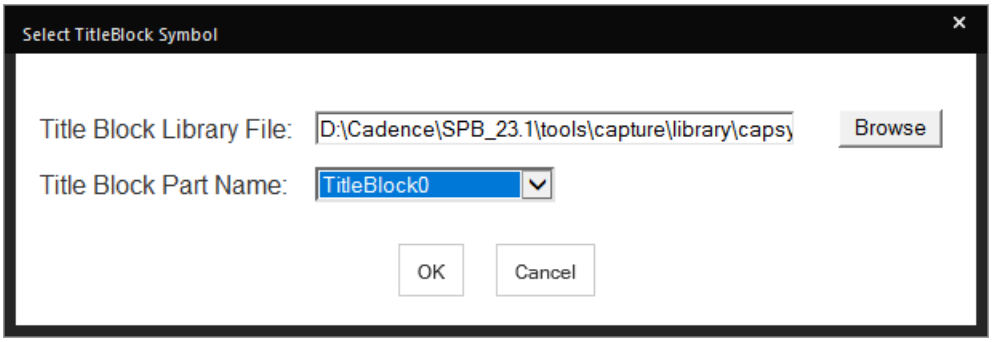
To configure title block mappings, you need to select the title block and then map EDM fields to title block fields.

To change the Title Block

1. Select the menu items: **Admin > Configuration > Capture** and navigate to the section of the page, **Title Block Field Mappings**.

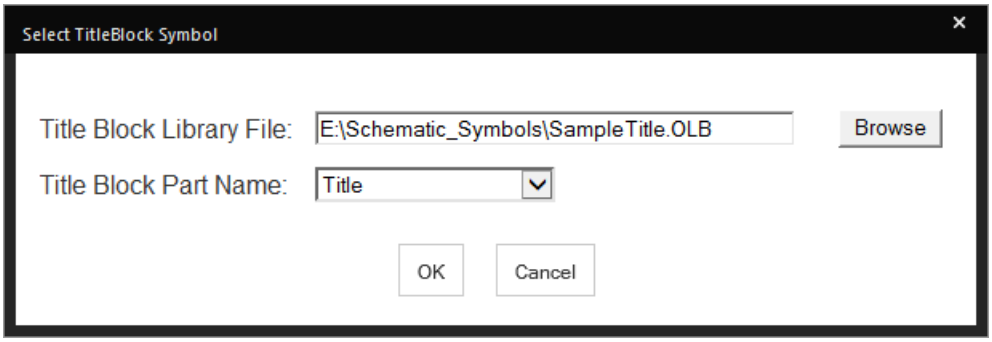


2. Click the **Select Title Block** button. Select browse to select the library file containing your TitleBlock. See the graphic that follows.



Select Title Block

3. Select the TitleBlock within the library for which you wish to map all your projects. Click **OK** to save or click the **Cancel** button to cancel the selection.



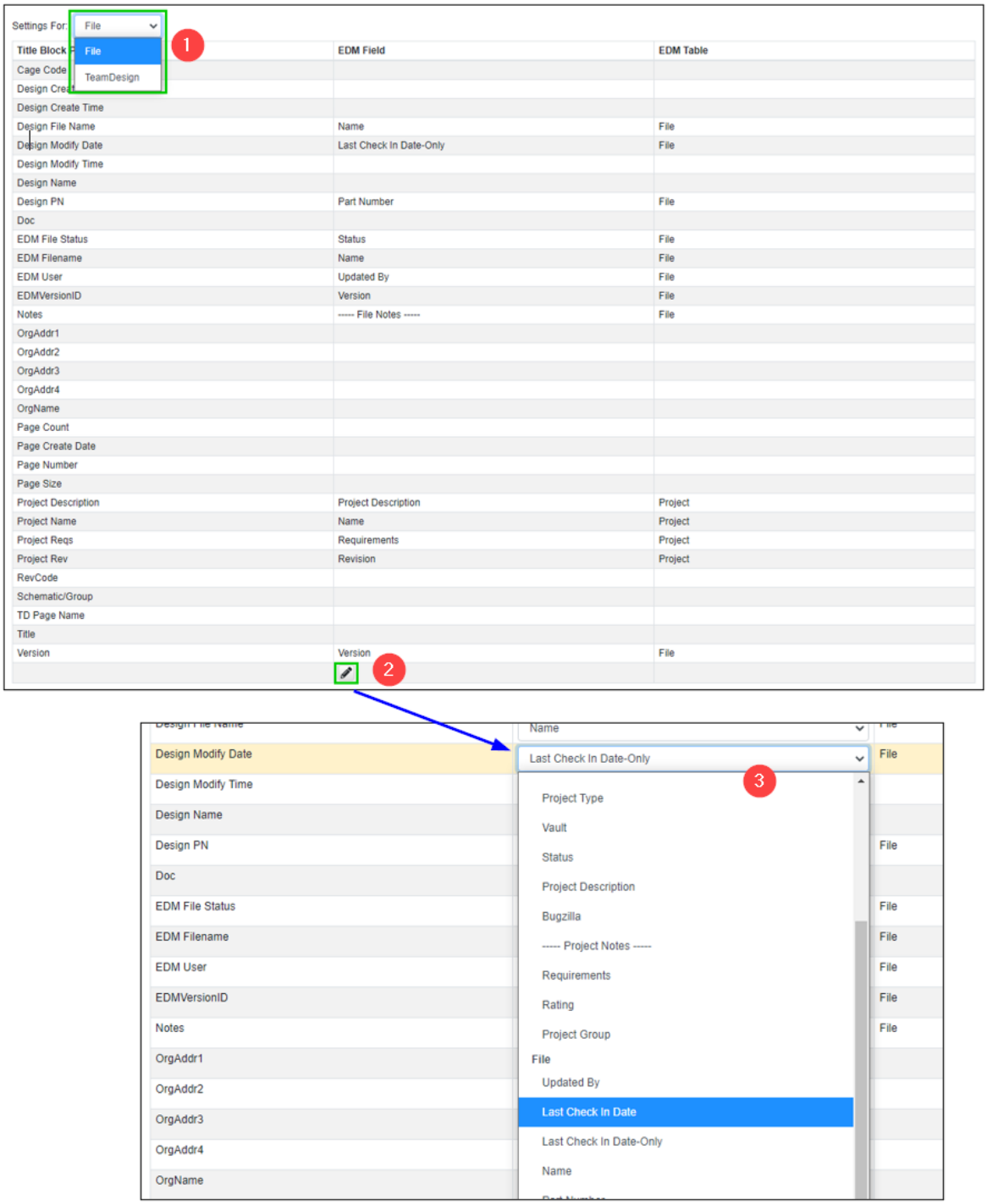
Title Block Library and Block Selection

Once you complete the selection for your Title Block, the properties from the title block you selected are added to the Title Block Property column. EDM fields that may be mapped to your Titleblock include: Project, File and Page. You need to map both Team Design and Single file designs if you plan on using teams for your projects.

To map EDM fields to Title Block Properties

1. Select **Settings For** in the drop-down list for **File** or **TeamDesign**.

2. Click the **Edit** button. The **EDM Field** column displays a drop-down list for EDM fields you may map to each title block field.
3. Select the EDM field next to a property in **Title Block Property** column that you want map. The **EDM Table** column identifies the type of field selected.



Map EDM Fields to Title Block Properties

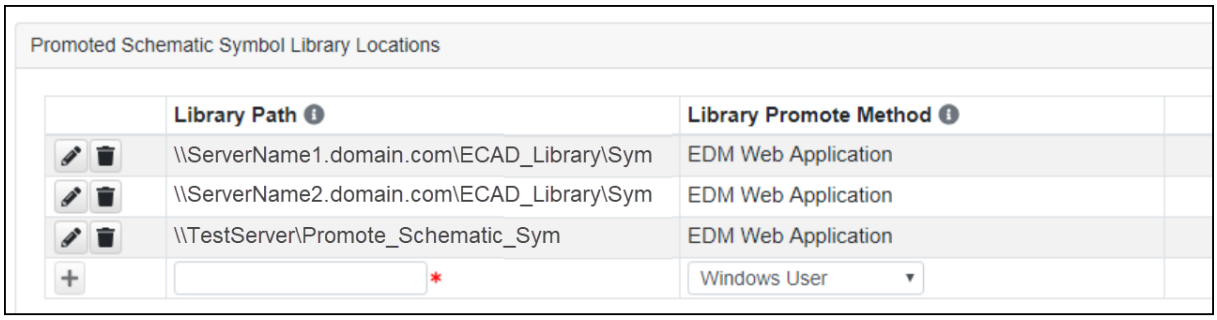
4. Continue to map additional fields following the same procedure.
5. When done, click the **Save** button to save your Title Block selection or click the Cancel button to cancel.

3.3.2.2 Promoted Schematic Symbol Library Locations

Promoted libraries may be viewed as released libraries that are separate from working library projects in the vault. Symbols that are ready to be used across projects are promoted and made available to users.

A Promoted Schematic Symbol Library Location is a path location of a shared or centralized folder that stores promoted schematic symbol library (.olb) files. At least one location must be specified before you can promote symbols to a library. When a symbol is promoted, it is added to all configured promoted library locations. All Capture/CIS users should be pointing to these promoted libraries so they can use the same schematic parts on their designs.

Users assigned roles with the permission to update Configurations can configure the promoted library locations by selecting the **Admin** > **Configuration** > **Capture** menu. Scroll down the page to locate **Promoted Schematic Symbol Library Locations**.



Promoted Schematic Symbol Library Locations

You may select the locations of library and the method to promote symbols.

Promote Method: Two methods are available for promoting Schematic symbols:

- EDM Web Application
  - The EDM Web Application promotes a symbol from the Vault to the Promoted location.
  - When using this method, the EDM Web Application identity will need to have **Full control** permission on the physical path of the configured library location.
  - All other users will only need Read permissions.
  - This is the most secure method and will reduce accidental edits or overwrites to files in the central library.
  - This will be most complex to configure since a System Administrator to the EDM server may be required to configure EDM Web Application identity.
- Windows User

- The credential of the logged in user is used to promote a symbol from the Vault to the Promoted location.
- Users who are assigned to manage the EDM libraries will need to have **Full control** permission on the configured library location.
- All other users will only need Read permissions.
- This method is less secure since users may bypass EDM and accidentally edit library files.
- This method is easier to configure.

Library Path:

You may set up multiple library locations. Multiple locations may be required to locate libraries geographically closer to design engineers. When a symbol is promoted, the symbol is promoted to all configured locations.

If you are changing a Library location in EDM, you may copy the library files from the current location or re-promote all promoted symbols to the new location.

UNC paths are recommended over mapped drives because different users may map the same drive letter to a different physical path. If UNC paths do not work for all users, a mapped drive can be used. Ensure all users have the same mapping for the drive letter.

Note: Do not select a library location that coincides with any of the EDM Vault locations.

To Add a Library Location:

Enter a path. Select a Library Promote Method. Then click the **Add** button. The path and method are validated by EDM. Warnings and errors are provided for problems found during Add. If you are adding a new location, you may copy the library files from an existing location or re-promote all promoted symbols to the new location.



To Edit a Library Location or Method:

Click the **Edit** button. Change path or method. Then click the **Save** button. The path and method are validated by EDM. Warnings and errors are provided for problems found during Save. If you are changing a Library location in EDM, you may copy the library files from the current location or re-promote all promoted symbols to the new location. When changing library locations, be sure to notify all users to change their path to use the new library location.



To Delete a Library

You may delete a path by clicking the **Delete** button that is located in the row you want to delete. This only deletes the path link. You will still need to manually remove the directory to delete the actual library.

3.3.2.3 Netlist Generation Configuration

Netlist Generation Configuration allows you to change the default **allegro.cfg** configuration file used by Capture when a netlist is created.

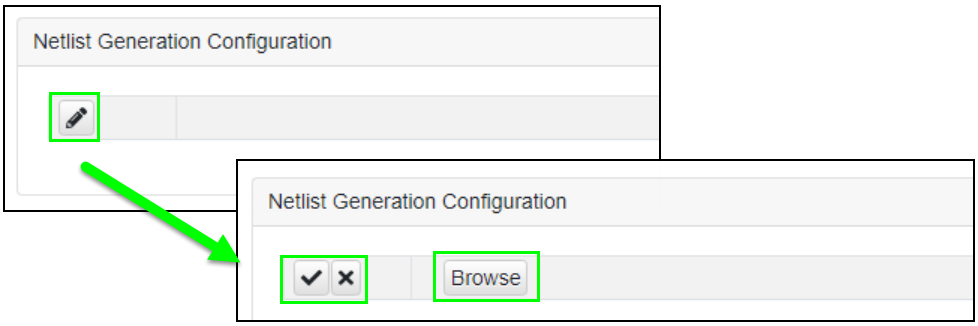
If you do not configure this setting, EDM will use the **allegro.cfg** file located in the same directory (**%HOME%\cdssetup\OrCAD_Capture\24.1.0**) as the user's Capture.ini file. If that is not available, it will fall back to the the **allegro.cfg** file in the **.\tools\capture** sub-directory of your OrCAD installation directory (e.g. **C:\Cadence\SPB_24.1\tools\capture**).

Users assigned roles with the permission to update Configurations can configure Netlist Generation file by selecting **Admin > Configuration > Capture** menu and opening **Netlist Generation Configuration** settings. Once configured, all projects use this setting by default, but that may be cleared or changed by editing the project.

Note: This setting may be configured when EDM is launched inside Capture or from the EDM Desktop application. The configuration of this setting is not available when you open EDM from your browser.

To configure the Netlist

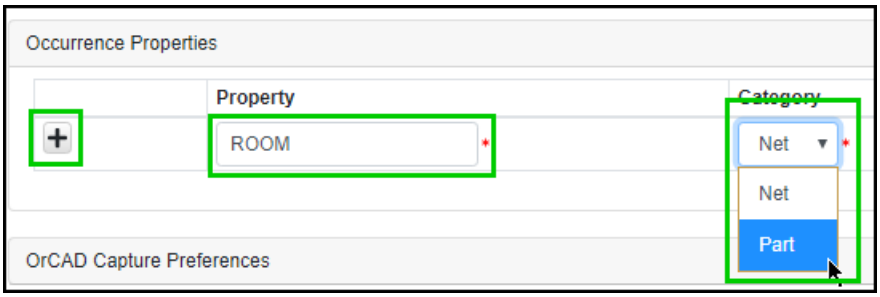
1. Select the **Edit** button.
2. Select the **Browse** button.
3. Navigate to your configuration file and select it. The path updates after you select the file.
4. When done, click the **Save** button to save your Netlist configuration or click the **Cancel** button to cancel.



3.3.2.4 Occurrence Properties

When pages are split and checked into the vault, the concept of Occurrence Properties or their values are lost, which is problematic for hierarchical designs. To ensure Occurrence Properties are restored after you check out a hierarchical design, you may configure Occurrence Properties from the **Admin > Configuration** menu. Refer to Lock and Restore Occurrence Properties for additional explanation.

From the **Admin > Configuration > Capture** menu, scroll down the page to **Occurrence Properties**. To add an occurrence of property names, type in the **Property** name, select the **Category** option, either **Net** or **Part**, then click the **Add** button.



Add Part Occurrence Property



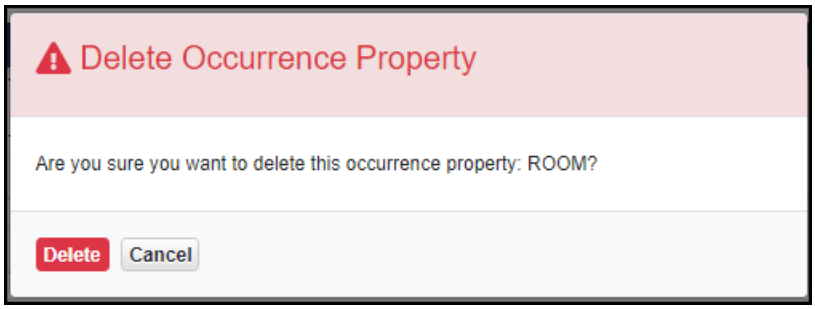
Add Net Occurrence Property

To delete the property, click the **Delete** button.



Delete Occurrence Property

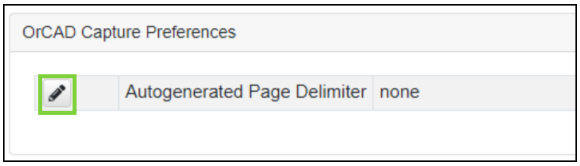
The **Delete Occurrence Property** pop-up will display to verify the deletion. To delete the property, click the **Delete** button. Click **Cancel** to cancel the delete option.



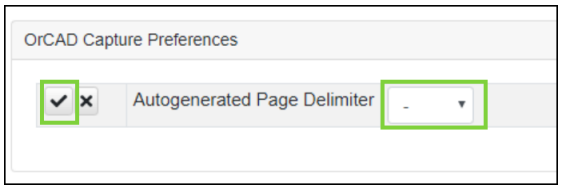
3.3.2.5 OrCAD Capture Preferences

Blank pages can be added to team designs as groups of pages. The pages will be auto-numbered if created as a group. When you create groups of pages, you can set either the hyphen (-) or underscore (_) as the delimiter for the page number. If you do not set a delimiter, none will be used.

Under **Admin > Configuration > Capture** scroll down the page to locate OrCAD Capture Preferences. Click the **Edit** button next to Auto-generated Page Delimiter and select one of the delimiter options. Click the **Save** button to save the delimiter.



Edit Page Delimiter



Select Page Delimiter

After configuration, the pages will be created with the group name followed by a number. For example, if you want to create a memory group of 3 pages with a hyphen for the delimiter (**-**), the 3 pages will be created with the following names: memory-1, memory-2, and memory-3. If you leave the default set to **none**, the 3 pages will be created with the following names: memory1, memory2, and memory3.

Click the **Cancel** button to cancel the selection.

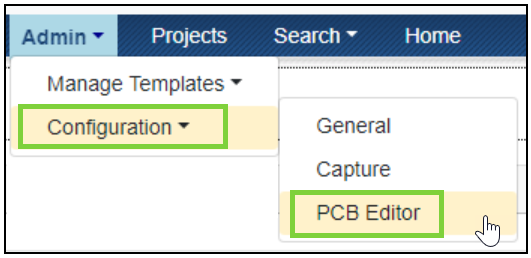
3.3.3 PCB Editor Configuration

Users assigned roles with the permission to update Configurations who open EDM from PCB Editor or with the EDM Desktop may configure title block properties, specify the location of promoted libraries and specify rules for promoting and deleting footprint files.

PCB Editor configuration settings that may be modified include:

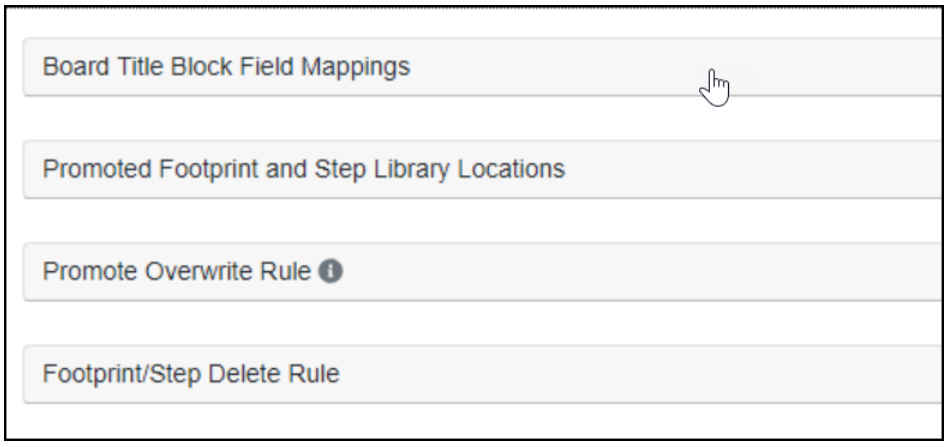
- Board Title Block Field Mapping - control title block property mappings
- Promote Footprint and Step Library Locations - promote footprint and step files to a centralized library so that all users may have shared access to promoted items
- Rules for Footprints Promote and Delete - define default behaviors when footprint files are promoted and deleted

Customization of PCB Editor configuration settings is available from the menu selection: **Admin** > **Configuration** > **PCB Editor** as shown in the image that follows.



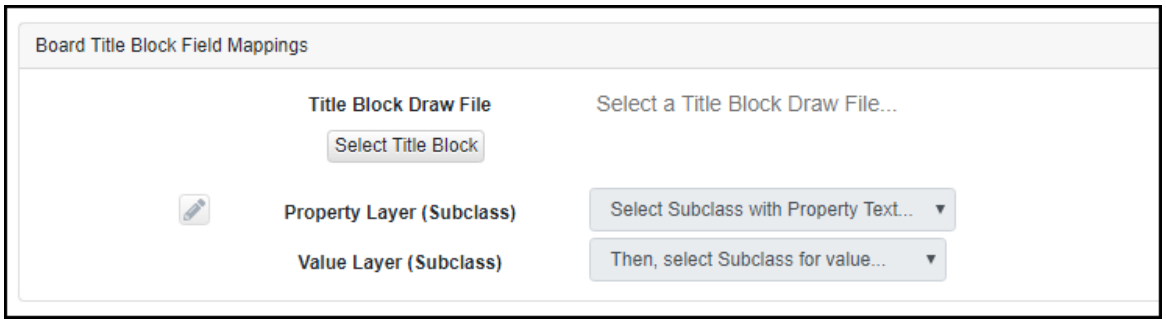
Admin > Configuration > Allegro Menu

Configuration options may be collapsed or expanded by clicking on the option titles.



3.3.3.1 Board Title Block Field Mappings

The update title block functionality uses the title block property mapping defined in this section. Note that the title block format changed with the 6.3.0 release of EDM. You may continue to use the older style format. However, if you need to make any changes, you will need to update your title block to the new format described below.



Configure board title block in EDM requires the following:

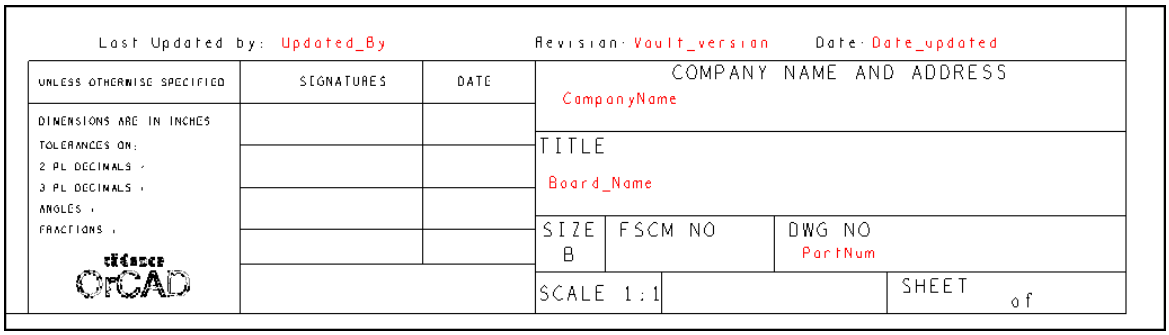
- Add Properties to Your Title Block
- Configure EDM to use your title block and Map EDM fields to title block properties

Add Properties to Your Title Block

You may add title block properties to a symbol draw file (.dra) by adding the following information to your title block symbol draw file.

1. Add a subclass to the DRAWING FORMAT class that will be used to identify your title block **properties** for EDM, e.g. EDM_PROPS.
2. Add a second subclass to the DRAWING FORMAT class that will be used by EDM to place mapped **values, e.g.** TEXT_OUTPUT. During Titleblock update, EDM adds this subclass to the board file if not found. EDM also clears all text in this subclass before placing values to ensure no overlapping text show in the title block.
3. Add text to the **properties** subclass (step 1) which identifies your PCB title block properties. The size and font of the text added to the property subclass will be used by EDM when updating the title block. During Titleblock update, all occurrences of mappings found will be updated. E.g. If you need to have the user who made the update (shown as Updated_By in the following example) appear in several locations in the Titleblock, then you only need to add this property at multiple locations.

The following figure shows an example of a title block symbol file (.dra) with properties added.



Configure EDM to use your title block and Map EDM fields to title block properties

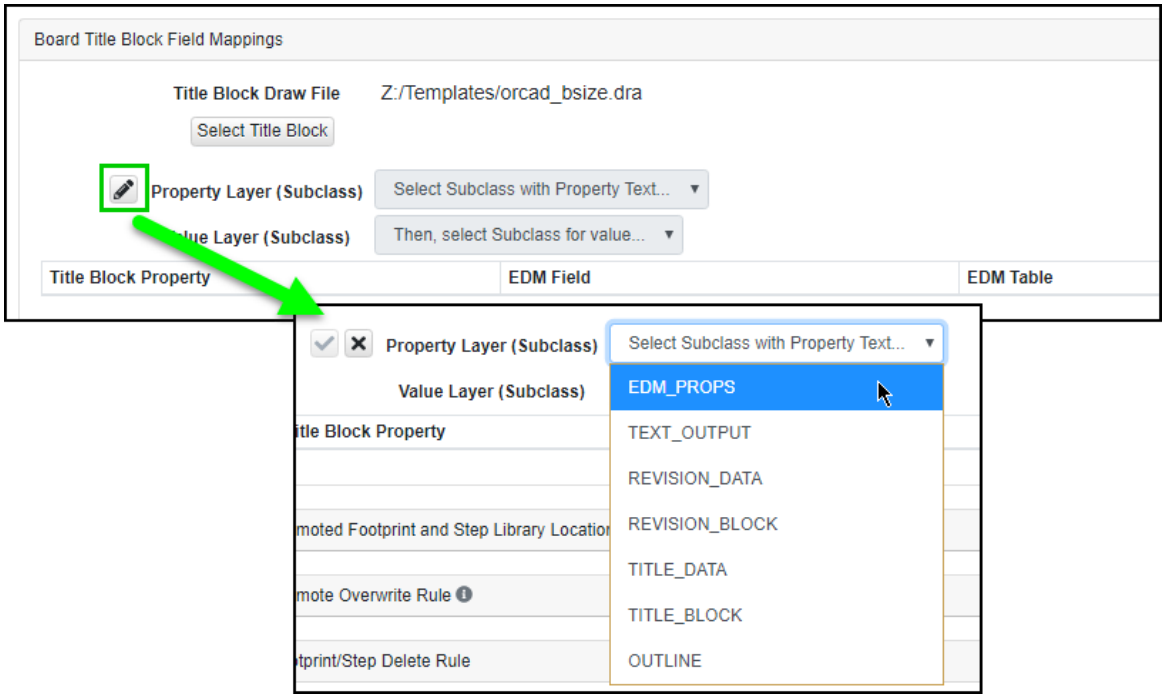
Selecting and mapping EDM fields saves the text property name and corresponding EDM field that will be used when updating the title block during a check in of a PCB board file. To configure the mapping for multiple title block templates, use the same text property name across all title block files. The same mapping applies to all title blocks.

To Configure the Board Title Block:

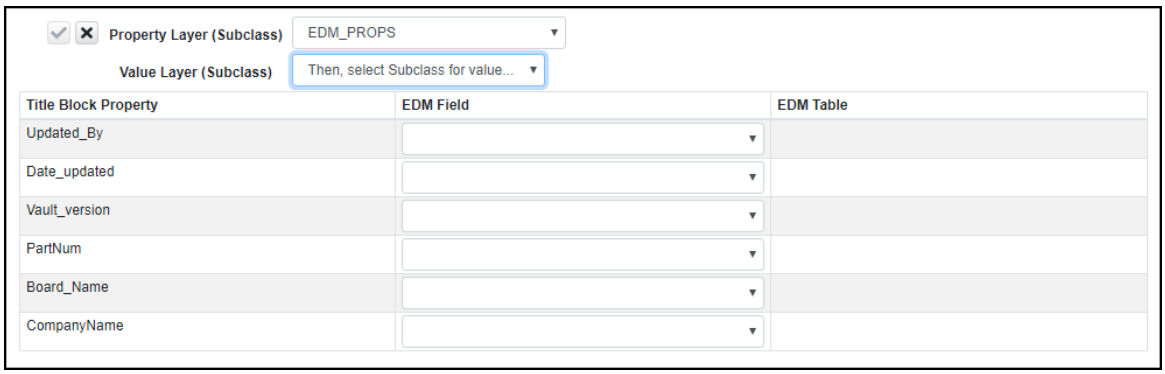
1. Click the **Select Title Block** button and browse to select the title block (.dra) file containing your Title Block. See the graphic that follows. EDM opens the selected title block file to read the subclass information in the DRAWING FORMAT class.



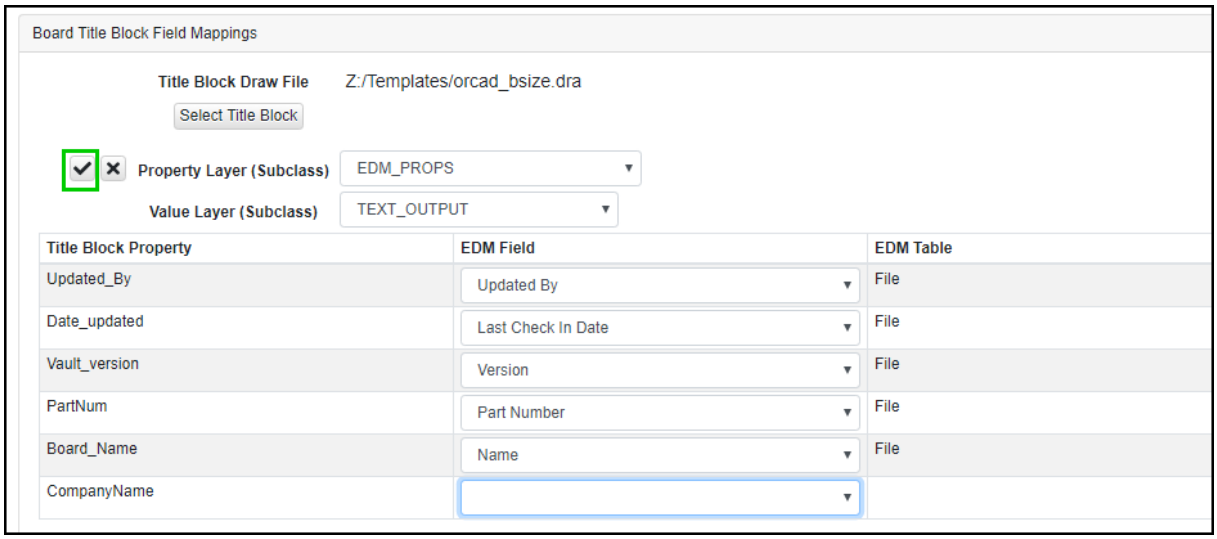
2. The Edit button is enabled after obtaining the subclass information. To configure the subclasses or layers in EDM, click on the **Edit** button.



3. Select the **Property Layer (Subclass)** name containing the properties in your Title Block. EDM obtains all texts in this subclass and updates the display.The property names from the subclass in the title block you selected are added to the Title Block Property column.

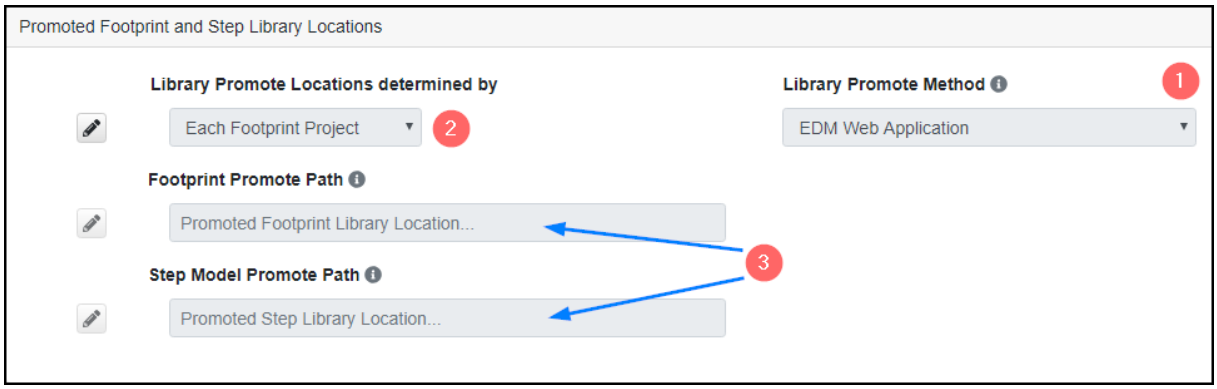


4. Select the **Value Layer (Subclass)** name for EDM to place the mapped field values. Conti
5. Select the **EDM Field** next to a property in **Title Block Property** column that you want map. The EDM Table column will show the type of field selected.
6. Continue to map additional fields.
7. When done, click the **Save** button to save your Title Block selection or click the **Cancel** button to cancel.



3.3.3.2 Promoted Footprint and Step Model Locations

Promoted libraries may be viewed as released libraries that are separate from working library projects in the vault. Footprints and Step Models that are ready to be used across projects (i.e. on boards) are promoted and made available to PCB Editor users pointing to the promoted locations. When a footprint and its associated files are promoted, files are copied from the vault to the configured promote location.



Before promoting footprint or step files, you will need to configure the following settings:

1) Library Promote Method
2) Library Promote Locations determined by
3) Promote Paths for Footprints and Step Models

1) Library Promote Method:

- **EDM Web Application**
  - The EDM Web Application promotes footprints and step files from the Vault to the Promoted location(s).
  - When using this method, the EDM Web Application identity will need Modify permissions on the physical path of the promoted library locations.

- All other users will only need Read permissions on the promoted library locations.
- This is the most secure method and will reduce accidental modifications to files in the promoted library locations.
- This will be most complex to configure since a System Administrator to the EDM server may be required to configure EDM Web Application identity.

- **Windows User**
  - The Windows User logged into the machine and running EDM promotes footprints and step files from the Vault to the Promoted location(s).
  - Users allowed to promote footprints and step files will need their Windows User to have Modify permissions on the physical path of the promoted library locations.
  - All other users will only need Read permissions on the promoted library locations.
  - This method is less secure since users may bypass EDM and accidentally modify files in the promoted library locations.
  - This method is easier to configure.

2) Library Promote Locations determined by:

- **Each Footprint Project**
  - The promoted library locations for footprint and step files is specified when you create/edit each footprint project.
  - Access for each location must be configured as you create each project to allow access. Permissions for access must be configured based on your **Library Promote Method** selection.
  - This provides most flexibility for location of libraries.
  - This option is more difficult to manage especially when using Web Application promote method since each new footprint project library location must be appropriately configured before use.
- **Central Footprint Location**
  - A central location is specified once and promoted footprint and step files will be in sub-directories of the configured location based off the project name and revision.
  - Access is configured once at the central location. Permissions for access must be configured based on your **Library Promote Method** selection.
  - Promoted library location follows a sub-directory structure that is the similar to that of the vault, *<Configured Central Location>\ProjectName\Rev\<footprint project folders>*

3) Promote Paths for Footprints and Step Models:

When the Library Promote Locations is set to **Central Footprint Location**, you need to specify the **Footprint Promote Path** and **Step Model Promote Path.** If desired, a single promote path may be used for footprints and step models. When a footprint file or step model is promoted, they are promoted/copied to the specified promote paths using the specified Library Promote Method.

If you are changing a Promote Path for Footprints or Step Models, you may copy the files and folder structure from the current promote location to the new one, or re-promote all promoted symbols to the new location. If you choose to re-promote all symbols, make sure to use the rule to Always promote selected and related files.

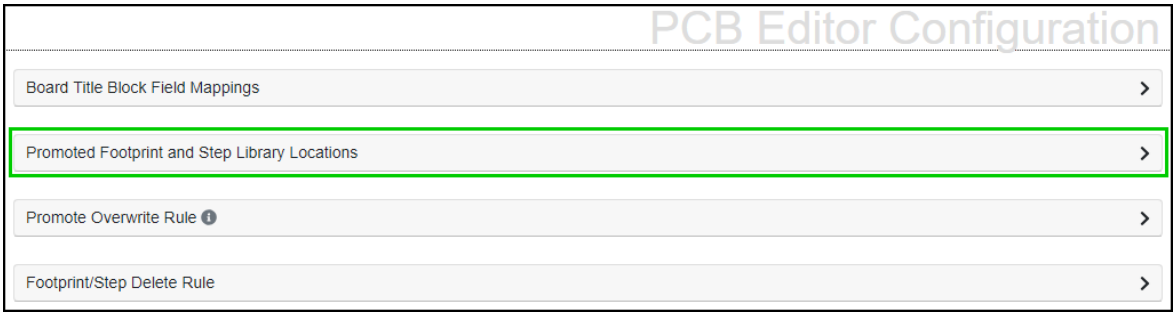
UNC paths are recommended over mapped drives because different users may map the same drive letter to a different physical path. If UNC paths do not work for all users, a mapped drive can be used. Ensure all users have the same mapping for the drive letter, along with the appropriate permissions based on the specified Library Promote Method.

Note: Do not use a promoted path location that coincides with any of the EDM Vault locations. Do not add, modify, or delete files directly in a promoted location outside of EDM.

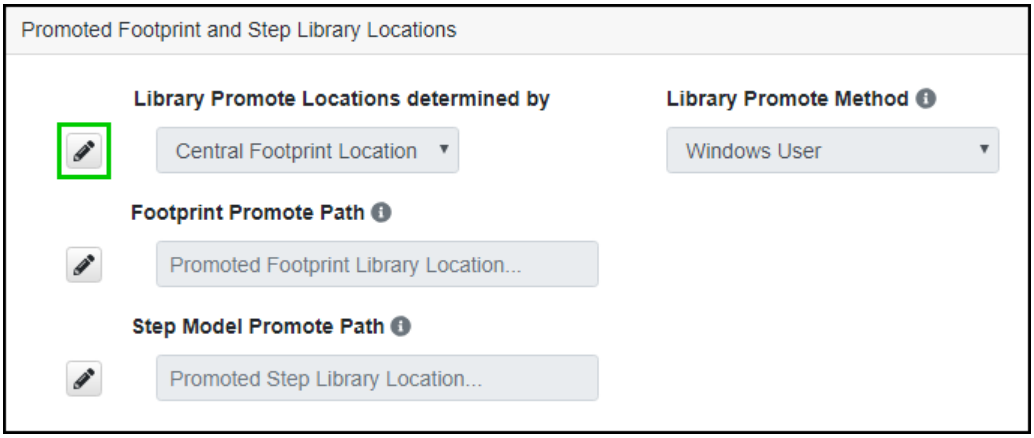
Steps to configure promote methods and promote locations

The following series of detailed steps show how to configure the promote footprint and step model locations.

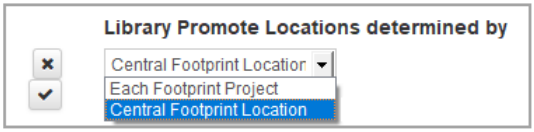
1. Select **Admin > Configuration > PCB Editor** and expand the section called **Promoted Footprint and Step Library Locations**.



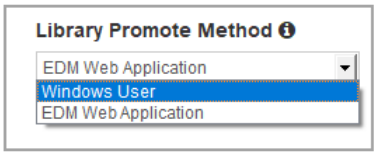
2. Click the Edit button to edit **Library Promote Locations determined by** and **Library Promote Method.**



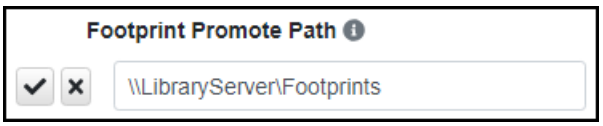
3. Select Central Footprint Location or Each Footprint Project.



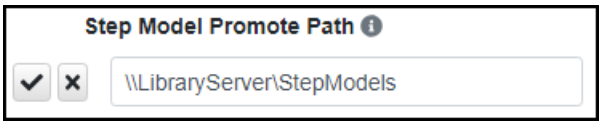
4. Select EDM Web Application or Windows User.



5. Click the Save button to save your entry.
6. Click the Edit button for the Footprint Promote Path and enter the promoted footprint location. The entry should be UNC formatted as: \\servername\folder. If desired you may enter a shared mapped drive location.



7. Click the Save button to save your entry.
8. Click the Edit button for the Step Model Promote Path and enter the promoted step model location. The entry should be UNC formatted as: \\servername\folder. If desired you may enter a shared mapped drive location.



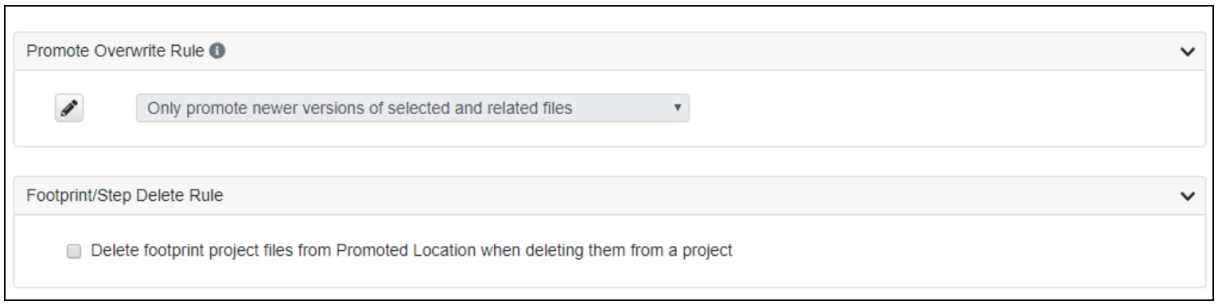
Note: The folder names you enter above need to have the appropriate sharing and security permissions configured as described in Library Promote Method. PCB Editor users need read access to place components footprint files onto board files. EDM web application needs access to show footprints and step models in the promoted locations.

9. Click the Save button to save your entry.

Note: If you are not able to save the settings, ensure that the specified paths exist and that permissions have been provided to the EDM Web Application process. If users are unable to promote footprints or step models, you may need to consult with your IT support or the user who created the shared library locations.

3.3.3.3 Rules for Footprint Promote and Delete

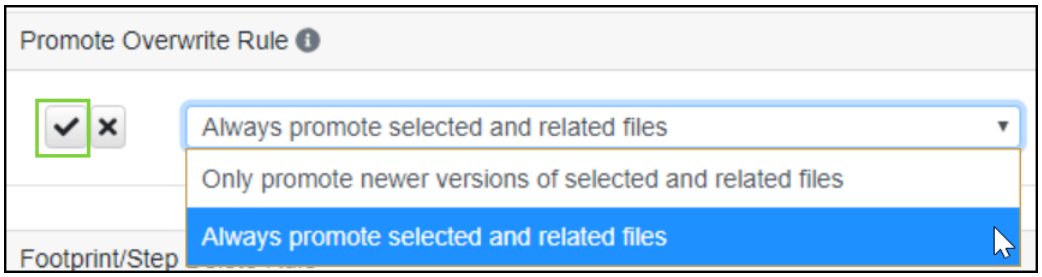
Footprint and step file relationships are acquired during check in and import. When a footprint and corresponding related files are promoted, files are copied from the vault to the configured promote location. You may configure the default behavior or rule for selected and related files when promoting footprints and step files. Additionally, you may configure the default rule when deleting a footprint. The default rules are correspondingly applied when promoting or deleting footprint and step files.



Promote Overwrite Rule options

- **Only promote newer versions of selected and related files** - When a footprint is promoted, only selected and related files with versions in that vault that are newer than versions that have been previously promoted will be copied from the vault to the promoted location. Files that are at the same version will not overwrite existing files in the promoted library location.
- **Always promote selected and related files** - When a footprint or step file is promoted, all files selected and related will always overwrite all files in the promoted library whether versions are the same. If you change your promote library location, you will need to specify this option to re-promote already promoted footprints and step files to the new location.

To change the Promote Overwrite rule, select the **Edit** Button, and select your preference. Then click the **Save** button.



Footprint/Step Delete Rule

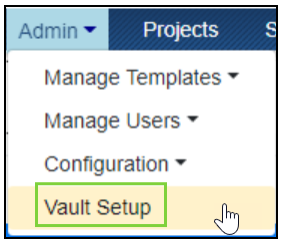
The Footprint/Step delete rule when enabled will automatically delete footprints from both promoted location and the vault. To enable this rule, place a check mark in the checkbox for this rule. A success message appears when completed.

3.4 Vault Setup

Only users assigned a role with Vault Setup permissions are able to configure and view the Vault Setup settings.

Before configuring EDM to use your Vault selection, you will need to have one of the supported Vault options installed and/or preconfigured. You will be unable to complete the Vault Setup in EDM without a fully set up file system or SVN. Supported Vault options are SVN and File System. Consult the EDM Installation guide for set up of your Vault if it has not been set up.

To configure or change vault servers, select **Admin** > **Vault Setup** within EDM.



Accessing Vault Setup

The repository type for EDM by default is set up for SVN. If you are not using an SVN server for your vault, you may change the Repository Server Type.

If you are using an SVN server, you will need to continue with the SVN Vault Settings. If using VIsualSVN for your SVN server, these Quick SVN Setup Steps for VisualSVN may be helpful.

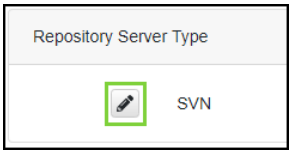
3.4.1 Repository Server Type

The **Repository Server Type** selection is used for configuring the type of Server that EDM will use to create and manage projects, folders, and the check in and check out of all types of files.

Only one type of repository can be configured for use: **SVN** or **FileSystem**. Once a vault is created in a repository location and in use, you are unable to switch the repository server type.

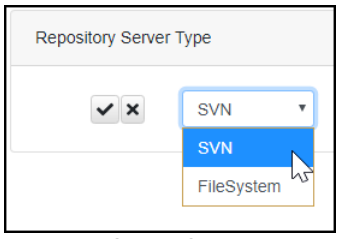
To configure the repository server type

1. Click the **Edit** button.



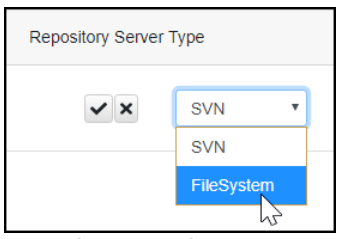
Select Edit

2. Choose either **SVN** or **FileSystem** from the drop-down list as shown in the figure below.



Select SVN

or



Select FileSystem

3. Click the **Save** button to save the Repository Type. Click the **Cancel** button to cancel the Repository Server Type selection.

A comparison of SVN and File System are provided in the following table.

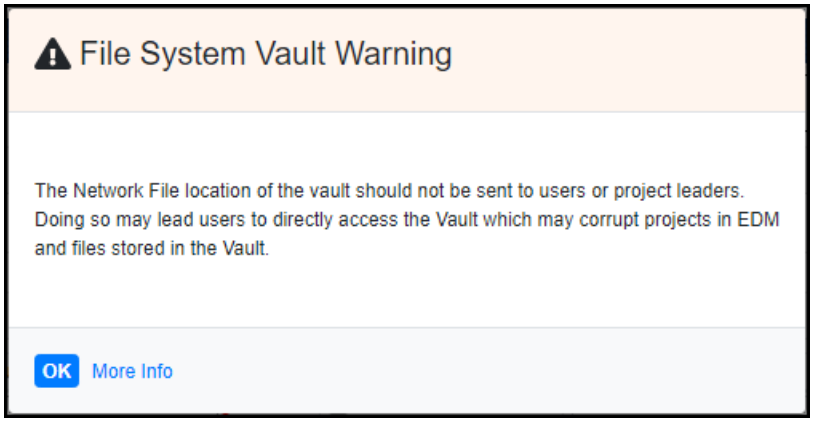
	SVN	File System
Cost	There may be an additional cost for SVN server. Requires installation of SVN Server.	A File System repository can be created and shared using existing Windows File System. No additional installation of repository required.
Installation and setup	Windows installer for SVN server may be downloaded and installed onto a server following information provided by the SVN server provider. User list is managed by SVN server once installed.	Windows group should be created for all EDM Users. The File System directory will need to be set up to provide R/W permission to he EDM web AppPool along with Windows group set up for EDM users.
Security and User access	Users do not directly access files in SVN providing better data integrity and security. Access can be set up using http or https.	All EDM users have the potential to access files in the file system repository directly.
	For companies without Active Directory, users have to be added to both EDM and SVN servers. SVN users if not using active directory will not be able to change their own password.	No additional log in required to access the repository.
	For companies with Active Directory and SVN configured for Windows Integrated, no addition log in required to access the repository.	
Storage and Performance	Storage of files take less space with SVN (binary differentials)	Storage of files will grow quickly with each check in.
	Download, check out, and check in times can take less time.	Download, check out, and check in relies on file copy of larger files which can take longer.*

*Footprint projects tend to be the largest, both in file quantity and total size, and can take the longest time to complete EDM operations such as a Full Project Download, Check In, and so on. Some of the factors that affect file transfer speed for a File System Vault are listed below:

- Client PC's connection to the Vault - e.g. On-site vs. VPN, Wired vs. WiFi, network response time and download speed, etc.
- Number of files in a project
- Size of files in a project
- Virus scan software running on Client PC, Network, or Server(s)
- Hardware specs for Client PC and Server(s) - e.g. RAM, CPU cores, Storage speed, etc.
- Other limitations set by company network on transfer speeds

3.4.2 File System Vault Setting

When the repository server type is configured as FileSystem, you will see a notification that providing the directory location for the vault is not recommended since manual changes to structure or files may corrupt the repository.

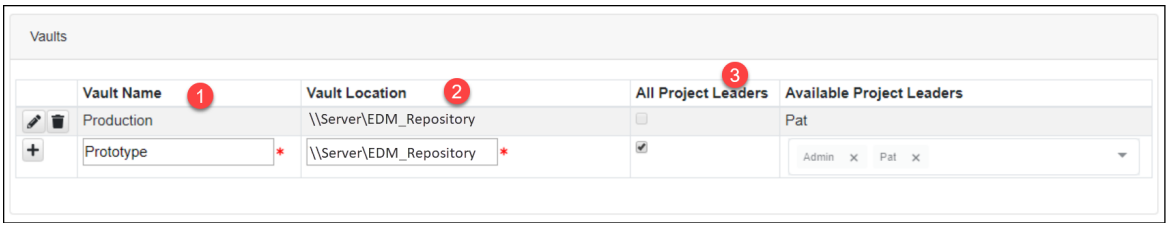


For File System configuration, a directory location to be used as the vault is required. The directory location must be configured to enable **Full control** permission for all EDM users. Additionally, the Web Application identity requires **Full control** permission. A warning shows when the Web Application cannot access the configured location. The structure of the vault is maintained by EDM. Any access to the EDM File System vault should be through the use of the EDM user interface. Directly changing files or structure in the vault may lead to corruption of projects in EDM.

The following information is required to configure the vault settings:

- **Vault Name** - The vault name is the sub-directory name to the Vault Location. If you have multiple locations, each vault name must be unique. EDM does not allow the same vault names across multiple servers. EDM users will need to have **Full control** permission to this location. If a vault is not located on the EDM Web Server, be sure to configure the AppPool identity to run as a Windows user with proper permissions on the vault(s). If any changes are made to the Application Pool (e.g. EDMAppPool), renaming it will ensure that an uninstall does not remove its settings.
- **Vault Location** - The vault location is the directory path to the file location used as the Vault. UNC paths are recommended to avoid problems that may arise from the use of different mapped drive letters among users. If UNC paths do not work for your company, mapped drives may be used. In the example below, \\Server\EDM_repository\Production will be the directory in which EDM projects will be added.
- **All/Available Project Leaders** - Users granted permission to Add Projects may be added to the list of leader who can create projects within the vault. You may individually add users or you can place a check mark in this checkbox to dynamically add **All Project Leaders** to the list of Project Leaders, which enables them to create projects in the specific vault.

  However, you may prefer to individually add Project Leaders to the **Available Project Leaders** column list instead of dynamically enabling all Project Leaders. To do this, click in the Select User text box, then select users from the list of Project Leaders.



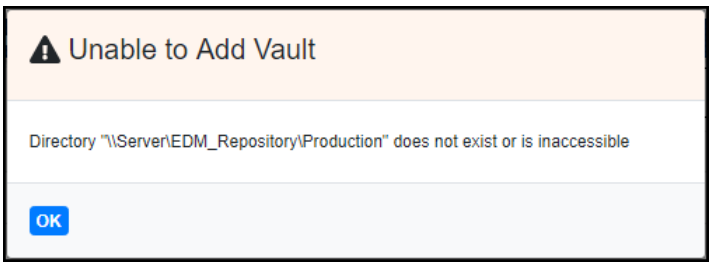
Add Vault FileSystem Server

After entering the Vault Name and Vault Location, click the **Add** button to add the vault.

To modify the vault location or change assignments of user who may add project to this vault, click the **Edit** button, enter the new information, then click the **Save** button.

To cancel the information, click the **Cancel** button. To delete the vault, click the **Delete** button. Only empty vaults may be deleted.

Note: If the vault is not created or the web application can not access the directory, you will receive the following message when you try to save the vault. Check with your Administrator to make sure that the Vault Server is created and that the EDM Web Application identity has access.



3.4.3 SVN Vault settings

When the repository server type is configured as SVN, the following vault settings need to be configured:

- Server Name and URL - identifies the URL for the SVN server.
- Vaults - identifies vaults that EDM will use for EDM projects.
- General Account - General Account required to view and download revision history from the SVN server.

The following section provides detailed information on settings. The quick setup guide provides instructions for a simple setup using Visual SVN Server.

Server Name and URL

To configure the server name and URL, enter the following, required information and click the **Add** button.

- **Server Name** - alias that refers to the vault (SVN) server specified in the Server URL field.

- **Server URL** - URL that EDM uses to create and manage projects, folders, and the check in and check out of all types of files. SVN must be pre-configured to use http protocol. SVN protocol is not supported.
- **Windows Integrated** - specifies how your SVN Server authentication has been configured on the SVN Server. Leave this unchecked if your SVN Server uses SVN or Windows Basic authentication. Check the box only if your SVN Server uses Windows Integrated authentication. When using Windows Integrated authentication, the Web Application identity must also be configured with credential that has access to the SVN Server. With SVN or Windows Basics authentication, users are prompted to enter their credentials when accessing EDM projects. Users may save their credentials. With Windows Integrated authentication, no prompt is provided for user credentials. Consult the EDM Installation Guide for additional information on EDM Web Application setup.



Add Server

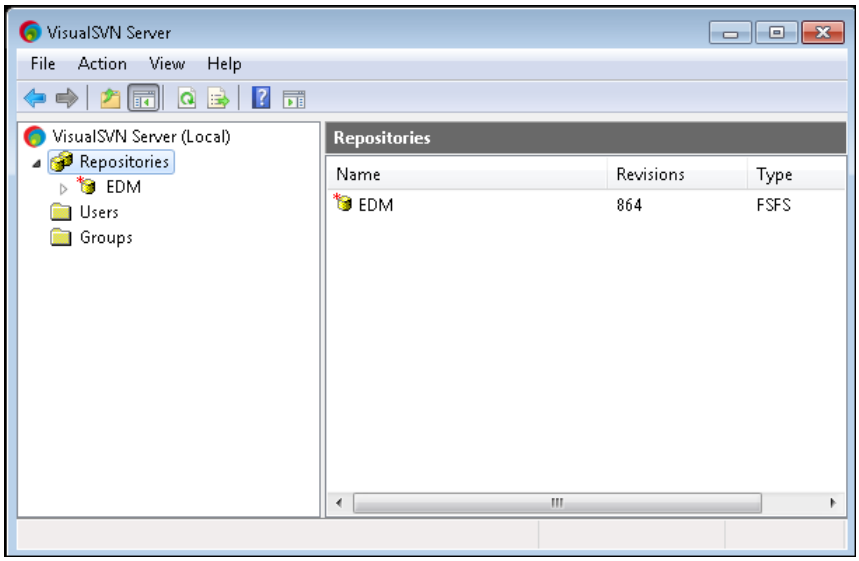
To modify the server information

1. Click the **Edit** button.
2. Enter the desired changes.
3. Click the **Save** button to save the updated info.
   Or, if you want to discard your changes, click the **Cancel** button.

Note: If using SVN authentication, the username added to EDM must also be added to SVN.

Vaults

The vault is a repository on the SVN server as shown in the following figure. The vault itself must be externally created outside of the EDM tools before you proceed.

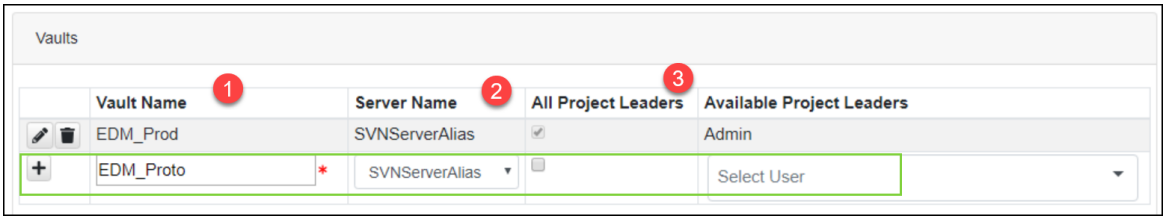


SVN Server Repository

To associate a vault to the server, enter the following field information. This is required field information.

1. **Vault Name** - repository name that is created on the SVN Server. In the example above, "EDM" is the vault name. EDM does not allow the same vault names across multiple servers.
2. **Server Name** - name of a server created in Server Name and URL. If more than one server is added, select the one associated with the specified vault.
3. **All/Available Project Leaders** - Users granted permission to Add Projects may be added to the list of leader who can create projects within the vault. You may individually add users or you can place a check mark in this checkbox to dynamically add **All Project Leaders** to the list of Project Leaders, which enables them to create projects in the specific vault.

   However, you may prefer to individually add Project Leaders to the **Available Project Leaders** column list instead of dynamically enabling all Project Leaders. To do this, click in the Select User text box, then select users from the list of Project Leaders.



Add Vault SVN Server

After entering the Vault Name and selecting the Server Name, click the **Add** button to add the vault.

To modify the vault data, click the **Edit** button, enter the new information, then click the **Save** button.

To cancel the change, click the **Cancel** button. To delete the vault, click the **Delete** button.

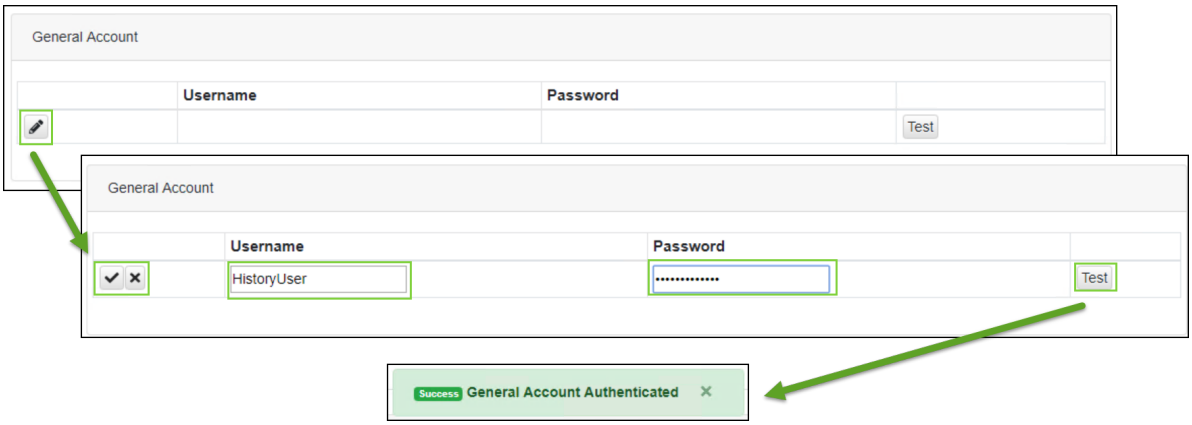
General Account

The General Account section enables you to set up and test the history download functionality that is provided to users who log into EDM from a standard web browser (without the need to install the EDM client). The General Account set up is disabled when the SVN server setup uses **Windows Integrated Authentication.** The General Account must be configured when the SVN Server is set up to use SVN or Windows Basic Authentication.

General Account Setup (not required for Windows Integrated)

If more than one server is used for vaults, the same General User must have access to each vault server. It is recommended that you have a dedicated account configured for the general account. For non-Windows authentication, an SVN user can be created for the general account. Alternatively, for Windows Authentication (non-Integrated), a domain service account can be created for the General Account.

Click the **Edit** button, enter the Username and Password for the General Account, and click the **Save** button. Click the **Cancel** button to discard changes.



Add General Account

General Account Test

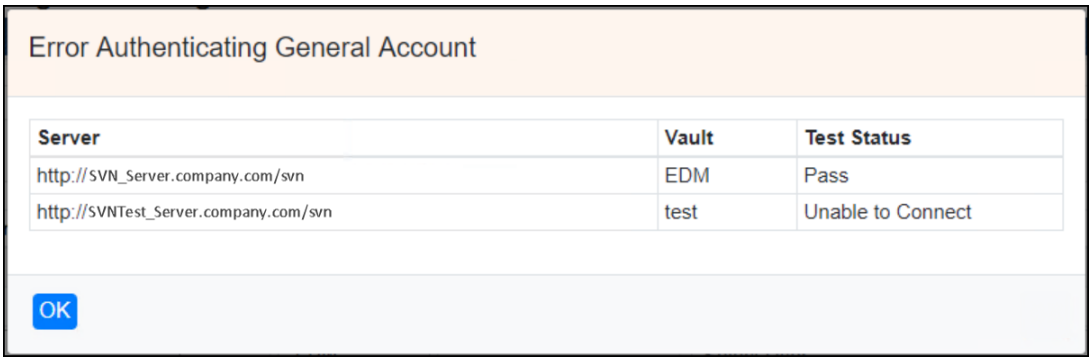
To ensure that the configured General Account has sufficient permissions to perform history functionality, click the **Test** button. A success message opens that notifies you when the credentials are valid.



Test History Functionality for Windows Integrated

When the SVN server setup uses **Windows Integrated Authentication**, the EDM Web Application identity is used to access the SVN server instead of the configured General Account. The EDM Web Application identity is configured using the Internet Information Services Manager AppPool settings. Consult the EDM Installation Guide to complete the AppPool settings of the installation process.

If credentials are invalid for any of the vaults, you will receive an Error Authenticating General Account message with the test status for each Vault.



Fail to Authenticate with General Account

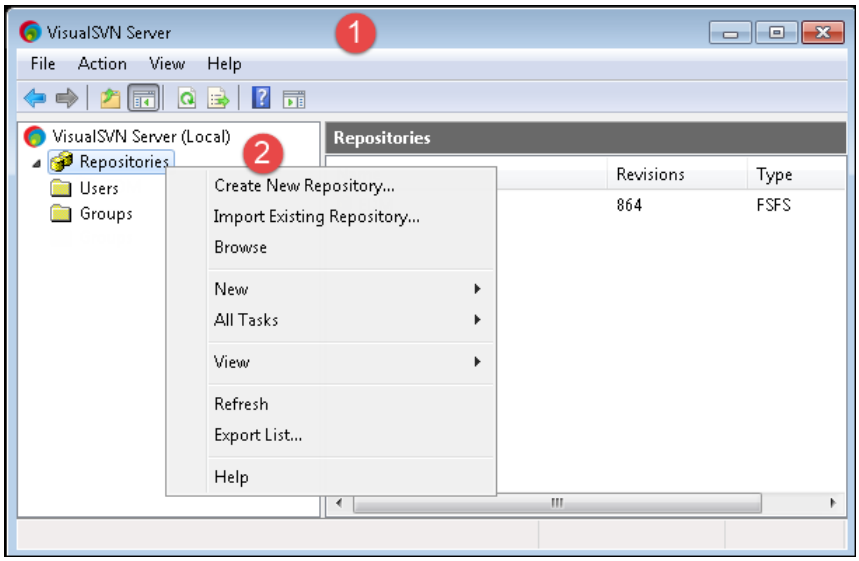
3.4.4 Quick SVN Setup Steps for VisualSVN

This section briefly describes the steps to set up the SVN vault repository using VisualSVN, when the server is configured for SVN Authentication or Windows Authentication.

- Set up SVN vault when SVN server is configured for SVN authentication

To quickly set up the SVN vault after installation

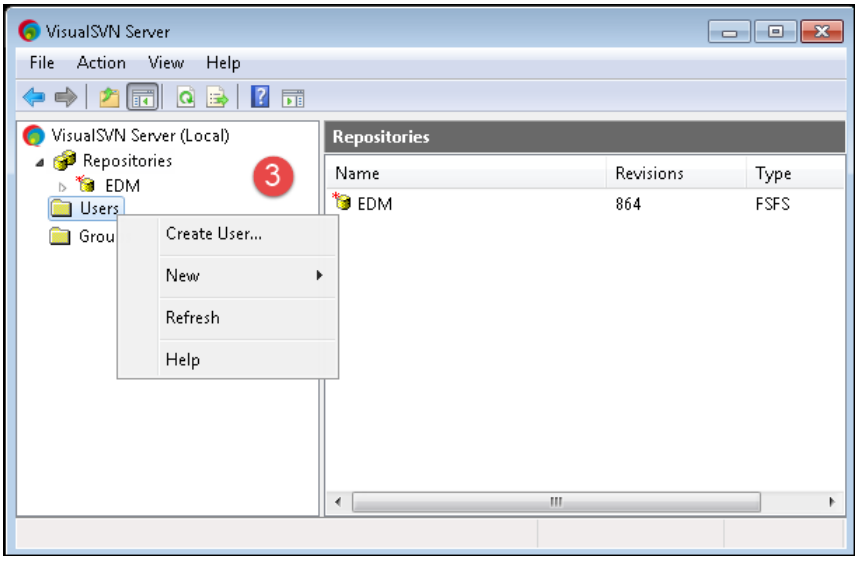
1. Open VisualSVN server.
2. Create the new repository:
    - Right click Repositories and select Create New Repository.
      See call-out 2 in the graphic below.



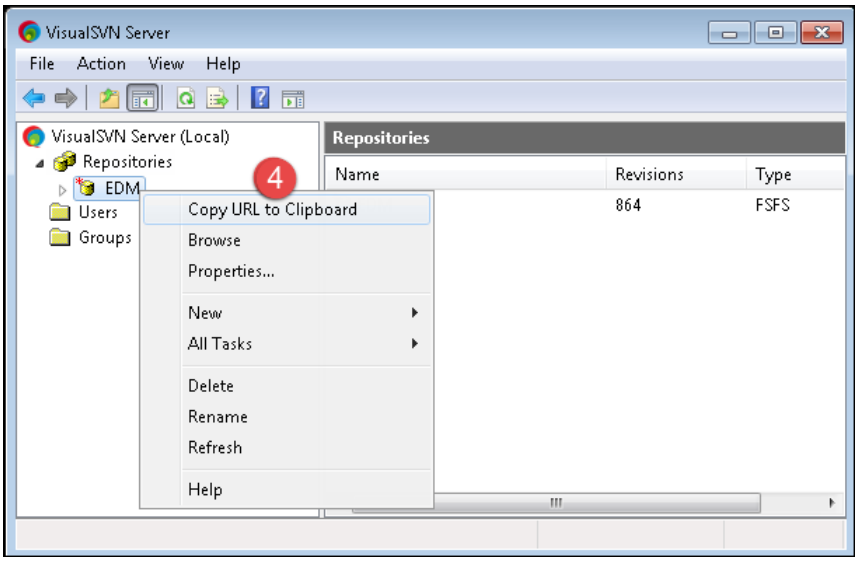
- Enter **EDM** into the Repository Name screen.
- Click **Next** to continue.

3. Create a user with the user name "Admin." (only required when SVN server uses SVN authentication)

   - Right-click the Users folder and select Create User from the pop up menu. See call-out 3 in the screen capture below.
   - Create a user with the user name Admin.
   - Enter a password.



4. Right click **EDM** and select **Copy URL to Clipboard** after the EDM repository is created. The URL becomes your SVN Server URL. See call-out 4 in the graphic that follows.

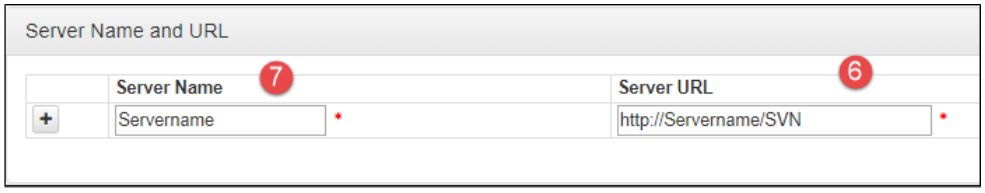


5. Open your browser and navigate to the EDM URL.

   Note: If your EDM web application and SVN server are installed on the same server, you can use http://localhost/EDM.

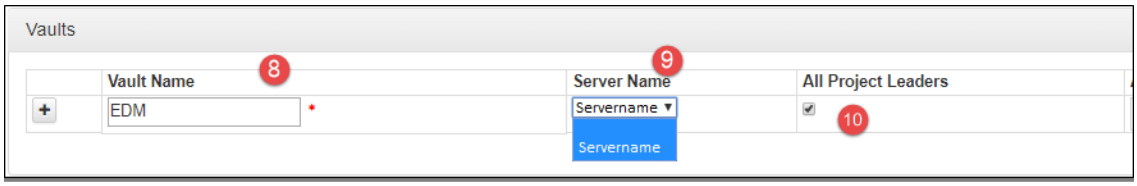
   - Login and select **Admin > Vault Setup**.

6. Paste the SVN server URL from step 4 into the **Server URL** box. (Remember to delete "EDM" from the pasted URL.)



7. Type local into the **Server Name** field and then click the **Add** + button. See call-out 7 in the above graphic.

8. Type "**EDM**" into the **Vault Name** field**.** See call-out 8 in the graphic below.



9. Select the server name. See call-out 9 in the graphic above.

10. Select the checkbox for **All Project Leaders** and then click the **Add** button.

11. Select **Edit** in General Account and type "Admin" for the username and the password that you entered in step 3.

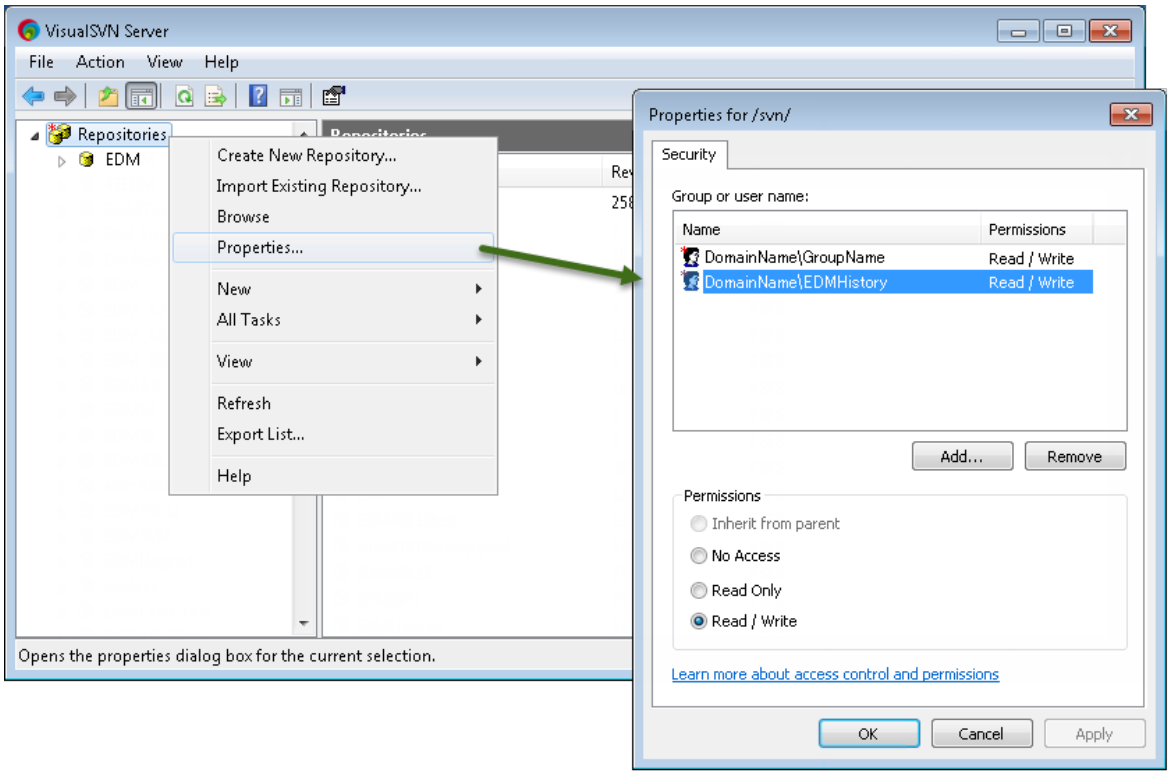


- Set up SVN vault when SVN server is configured for Windows authentication

To configure user permissions for access to SVN

1. Open VisualSVN server.

2. Right click on **Repositories** or repository name and **Properties.**



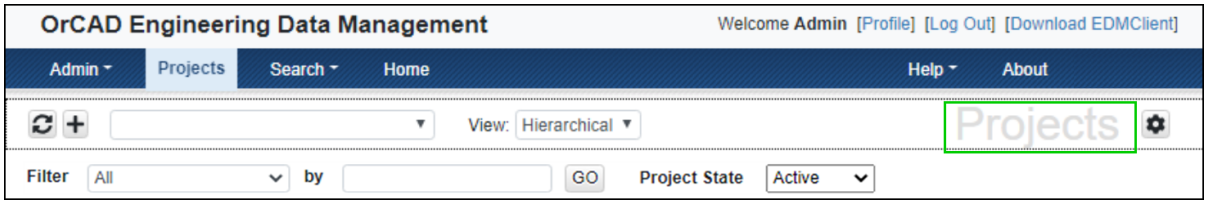
3. Add Windows domain user or domain group with Read/Write access.

4 Project Overview (All Users)

A project provides a structure in which folders and files can be managed and checked into the vault. Users are able to view only the project names in the project dashboard that they are assigned. Users who are granted View All Projects permission are able to see all project names in the project dashboard. The project dashboard and details views may be customized to show or hide column based on your preference. The action buttons that are available to you in Project Dashboard and Project Details views depend on both the permissions granted to your assigned role and project assignment. The project dashboard and all project details may be opened from Capture, EDM Desktop and standard browser. To enable full functionality of each type of project and auto-generated file capabilities, projects details will need to be opened with a specific user interface. The standard browser will provide limited functionality since no client plug in is used.

EDM includes several high level windows that provide different views of project files:

- Project Dashboard View – provides a high level status of all EDM projects and lists all projects and revisions created. Projects viewed in the Project Dashboard may be formatted as a flat list or as an expandable/collapsible hierarchy. A "Projects" watermark is displayed on the upper righthand corner of the Project Dashboard View.
- Project Details View – shows detailed information for a specific project that varies depending upon the project type.



Project Dashboard View

There are four types of EDM Projects, the Capture project, Footprint project, Schematic Symbol project, and General project.

- Capture projects – support schematic and board management and the auto-generation of schematic and board associated files. When opened in OrCAD Capture, enables Schematic functionality. When opened with the EDM Desktop, enables PCB and Release functionalities.
- Footprint projects – are specific to footprint and step model management. When opened with the EDM Desktop enables full footprint management and usage functionalities.
- Schematic Symbol projects – provides a shared location where a project team can easily access the schematic symbols libraries. This type of project is specific for the management of schematic symbols. When opened in OrCAD Capture, enables full schematic symbol management and usage functionalities.
- General Projects – supports check in and check out of all files types without auto-generation or associations of any file types. When opened in either OrCAD Capture or EDM Desktop, enables full project functionalities.

4.1 Organization of Projects

The organization of projects and structures will depend on how you want to manage your projects. You may organize a project to include all content associated with a board, or you may organize projects to match different content from different design stages.

If your process/tools do not enable you to reuse schematic symbols and PCB footprints, you may want to create a folder for all your libraries within each project.

The organization of the vault in EDM is as follows:

- **Vault** - top level in EDM. This must be created first to begin using EDM. Project Leaders assigned to a vault will be able to create projects within the vault.
  - **Project Group** - next level down from vault. Creating projects in different project groups makes it easier to find your projects using the search filter. The Hierarchical View on the View Projects page will display your Project Groups with their associated projects nested beneath them. This allows for better organization of projects, by associating related projects with a relevant Project Group.
    - **Projects** - level in which assignment of users can be made. Any user assigned at the project level will be able to view and download files. Users assigned only at the project level will not be able to check in modified files unless they are specifically assigned to the file. The Flat View on the View Projects page will display all open projects in a list, regardless of Project Group. This provides a high-level look at all open projects.
      - **Revision** - revision of a project which is organized into separate folders.
        - **Folders** - added to projects or project structure. If you wish to create your project using a common folder structure, you can create templates and create projects from templates. You may add as many sub-folders as needed for your project.
          - **Team Designs** - added as folders configured for Team Design. Team Design folders cannot have sub-folders or files under them.
          - **Team/Schematic Symbol Libraries** - added as folders that are configured for either a Team Library or Schematic Symbol Library. Library folders cannot have sub-folders or files under them.

Typically, one vault will be created per site. More can be created and added to Vault Setup if needed.

Although the project structure is flexible and allows you to store multiple boards in a project, we do not recommend using a single project for multiple boards. Instead, we recommend that you create separate projects for each board to contain all design, simulation, layout, and associated files.

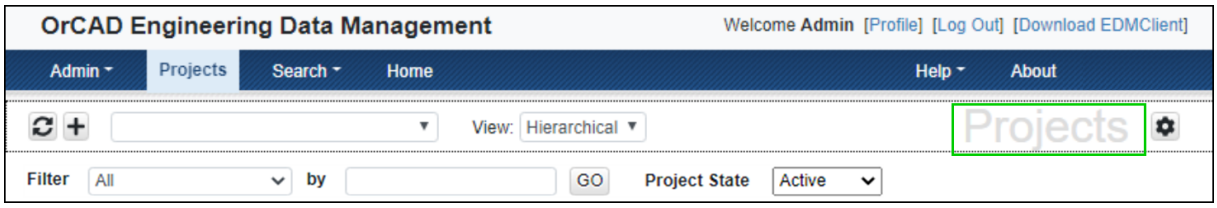
We also recommend that each design that requires a team be created as its own project as well. Symbols and footprints can be added to the project if you need to keep a comprehensive design package. Managing each board as a separate project allows you to keep all files associated with the design together. When the board is ready to be released, the complete project can be zipped for an archive or sent to a contract manufacturer.

Team Library folders, Team/Schematic Symbol Library folders, and footprint folders support library management. Schematic Symbol Libraries can be imported into either a Team Library or Schematic Symbol Library folder. PCB footprint symbols, pads, and step model files can be imported to separate folders of a single Footprint project.

4.2 View Projects (Dashboard)

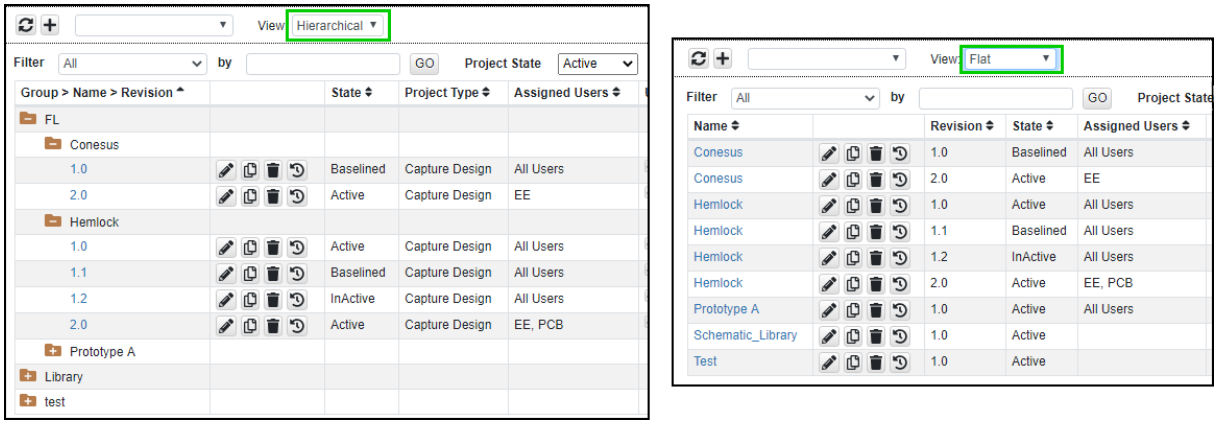
The Project Dashboard provides a high level status of all EDM projects and lists all projects and revisions created, provided you have permission to see them. Projects viewed in the Project Dashboard may be formatted as Flat list or as an expandable/collapsible Hierarchy.

A "Projects" watermark is displayed on the upper righthand corner of the Project Dashboard View as shown in the following figure.



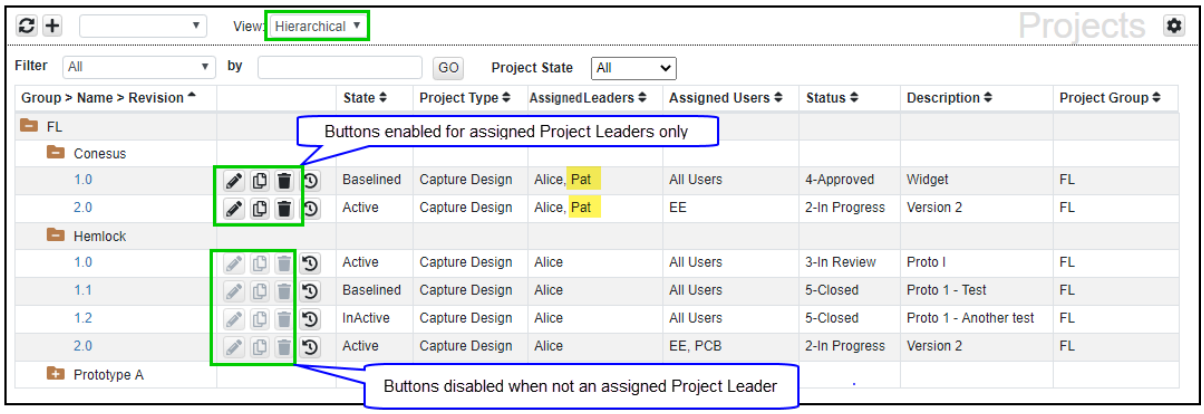
Project Dashboard View

Projects shown in the Project Dashboard depend on permissions granted to your assigned Role and Assignment.



Hierachical and Flat Project Dashboard Views

Users who are assigned roles with the permission to view all projects are able to view all projects regardless of user group assignments. All other users, are only permitted to view projects to which they are specifically assigned as a project leader or project user.

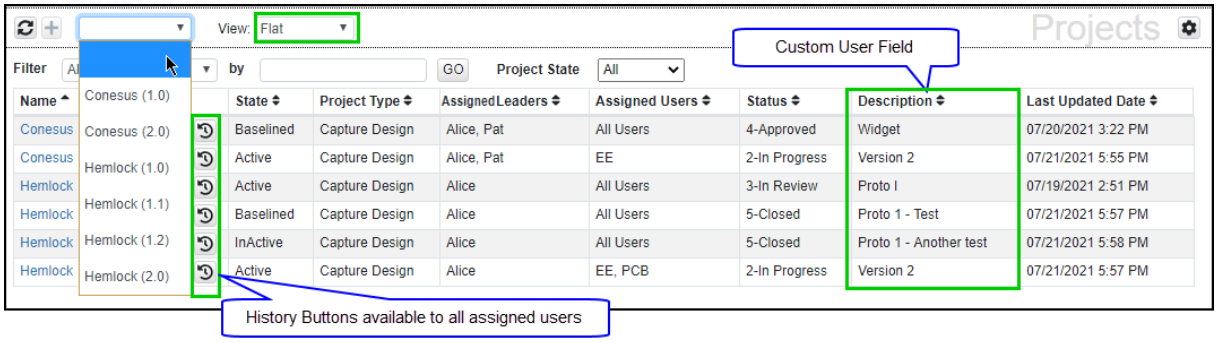


Project Dashboard Hierarchical View for Admins or Project Leader

When users who may be assigned as a Project Leader view the Project Dashboard, the project list displays buttons that allow the Assigned Leader to **Edit** the project properties, **Copy** the project to a New Revision, or **Delete** the project based on the users' granted permission. These buttons are disabled when a user, regardless of their role permission, is assigned as a user. Only Assigned Leaders able to **Edit**, **Copy** or **Delete** projects.

All assigned users, regardless of role permission, may view a projects history from the Project Dashboard as shown in the following figure.

Action buttons that support project management are unavailable to Users without Leader permissions. The following figure is an example of the Project Dashboard viewed by a user without any Leader permission.



Project Dashboard Flat View for Assigned User

The upper left hand corner of the Project Dashboard contains a button to create a project and options that permit you to modify the display view. Project control and View options include the following:



Project Action Buttons and View Options

- **Refresh** button - refreshes the Project Dashboard with updated information. If a new project is added or properties have changed while you have this page open, you must refresh the page before viewing the latest changes.
- **Add** button - provides access to the New Project form and allows a user whose role has appropriate permissions to add a new project to the vault.
- **Project** drop-down list - selects specific projects. You can select a project by clicking a project name in the list. When you select the project, the view updates to show the Project Details View of the project.
- **View** drop-down list - selects either a Flat or Hierarchical View of your projects.
    - **Flat View** displays all open projects, allows you to select a project, and click to open the Project Details View for that project. This view is ideal for displaying a high-level view of every open project.
    - **Hierarchical View** sorts projects by Project Group, allows you to expand the display list of the Project Group. The expanded view includes specific revisions. You can click

a revision to open the Project Details View of that revision. This view also enables you to view related projects within a Project Group.

- **Filter** drop-down list - selects a specific column in which the **Filter** text is applied or select **All** to filter by all columns.
    - **by** - text entered filters the display to list projects that have properties that match the string. For example, users may type their name or a specific project status to show all projects that match the entered name or project status.
    - **GO** button - executes filter when clicked.

- **Project State** drop-down list - filters the displayed projects by their state: **All**, **Active**, **Baselined**, and **InActive**.

The columns of data in the Project Dashboard show information and properties pertaining to the project. You may sort projects by clicking the column heading. Sorting can be either ascending or descending. You may reduce the list of projects shown by entering a string to filter the information shown in the view.

To customize this view and change the columns that display and the column order, click the **Gear** icon. The standard columns are:

- **Name** (shown as a separate column in Flat View) - name of the project that is entered when the project is created. Projects with the same name may appear multiple times as different revisions or in different project groups
- **Group>Name>Revision** (3 columns condense to 1 in Hierarchical View) - tree structure for the project when viewing the dashboard using Hierarchical View. Click the expand or collapse icon on a Group or Project Name to show or hide a project or revision level. Hold the **Ctrl** button and click the expand or collapse icon on a Project Group to show or hide all projects and revisions levels in the group.
- **Project Action buttons** - 3 of 4 buttons allow a user whose role has appropriate permissions and/or assigned as a leader to **Edit** project properties, **Copy** a project revision or **Delete** the project revision. The fourth, **History** button allows all users to view the project history.



Project Option Buttons

    - **Edit** accesses the Edit Project form, which allows a user with appropriate permission to edit project name, user and leader assignment, status and configured user fields.
    - **Copy** - allows a user with permission to copy the project revision to a new revision.
    - **Delete** - allows a user with permission to delete a project revision and all sub-folders and files of the revision.
    - **History** - allows all users to view a projects change history.

- **Revision** (shown as a separate column in Flat View) - revision of the project that is entered when the project is created.
- **State** - current state of the project. The states may be **InActive**, **Active** or **Baselined**. Only projects in the **Active** state are enabled for file check in and check out.
- **Project Type** - identifies the type of project. Project type can be Capture Design, Footprint, General, or Schematic Symbol Library.
- **User May Add Folder** - shows whether users are allowed to check in new folders. If Unchecked, the user is unable to check in folders.
- **Assigned Leaders** - list of users assigned as a Project Leader for the project. Only users in this list are permitted to manage the project.
- **Assigned Users** - list of users or user groups assigned to work on the project. Only users on this list will be allowed to view and check out the project items.
- **Status** - status of the project. Values that represent the project status are dependent upon company guidelines.
- **User Fields** - additional fields configured from user fields for projects. User field columns are based on the Project Administrators setup in EDM Configuration for Customizable Fields for projects. These fields may be modified by an assigned leader in this Project View.
- **Vault** - name of the vault in which the project was created.
- **Project Group** (shown as a separate column in Flat View) - group entered or assigned when the project was created. Group is shown in the same column with Project Name and Revision in Hierarchical View.

4.3 View Project Details

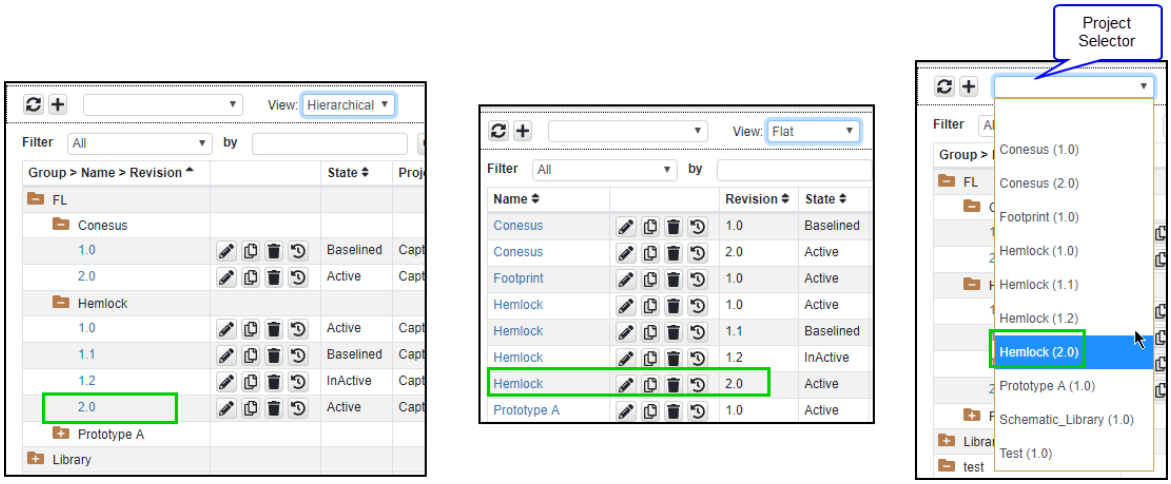
The Project Details page shows detailed information for a specific project. Each project type contains information and action buttons that are relevant for each. Team Libraries and Team Designs also has a separate view. Detailed information is provided for the following projects types, and team views:

- Capture Project View
- Footprint Project View
- Schematic Symbol Project View
- General Project View
- Team Design View
- Team Library View

Individual File and Folder Icons are used to indicate expanded/collapsed folders and various file types.

You can access the details of a project from the Project Dashboard using one of three methods:

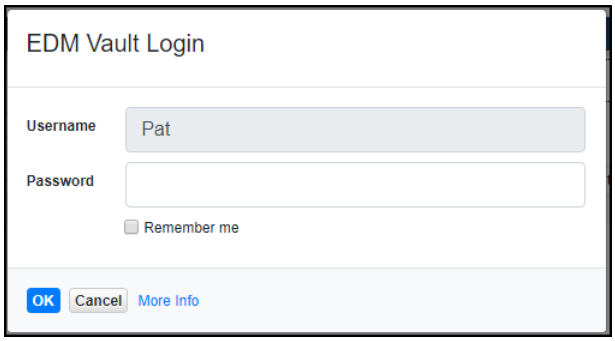
1. Select the project revision in the Hierarchical view
2. Select the project name corresponding to the desired revision in the Flat view
3. Select a project using the Project Selector drop-down list located in the upper left corner of the Hierarchical View or Flat View



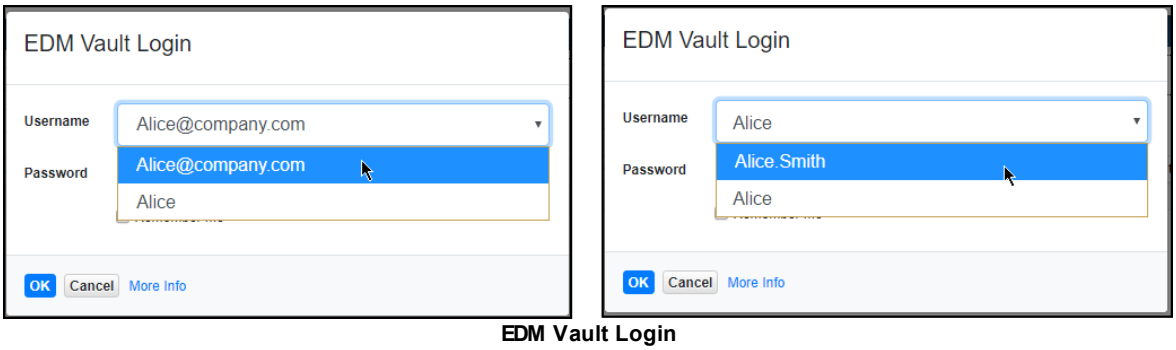
Three Methods to Select Project Details for a Project

When you select a project, you may be prompted to log into the EDM Vault, depending on your repository type.

The screen that follows shows the log in screen to the EDM Vault where you simply enter your credentials.



When your Windows user name is different from the EDM log in name, the screen may display a pick list for you to select the Windows user name. The following figure shows the EDM Vault Login screen for both methods of login.

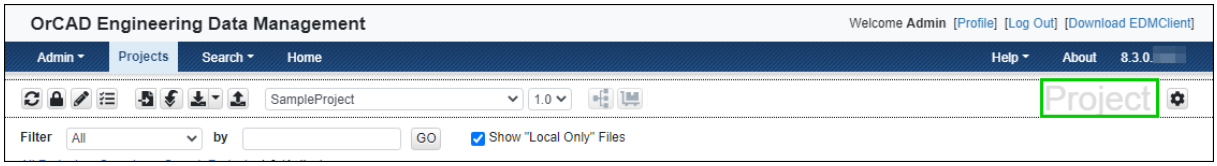


EDM Vault Login

4.3.1 Capture Project View

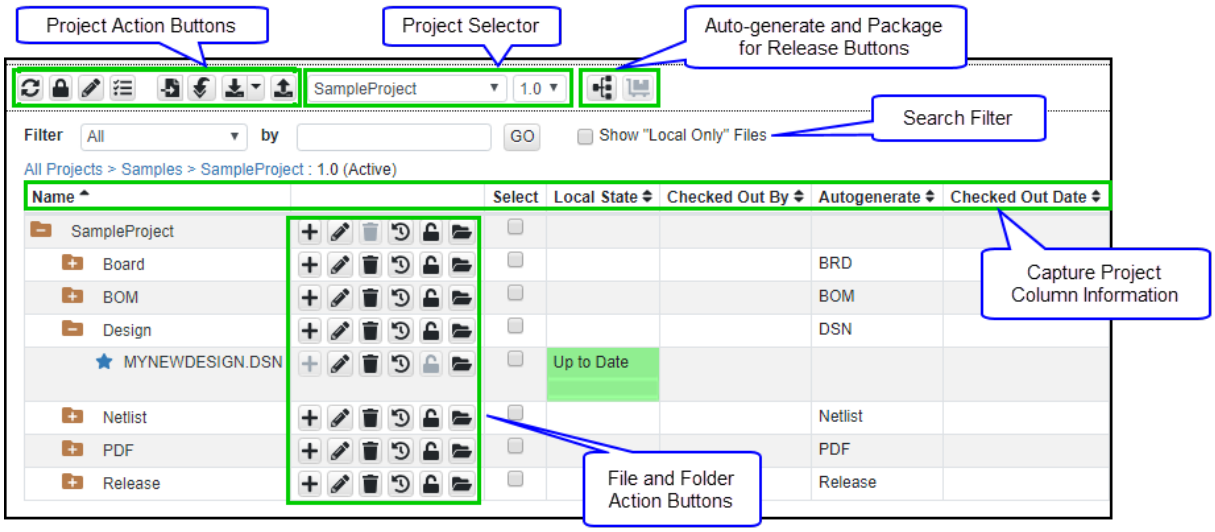
The **Capture Project View** shows detailed information for folder structure of files that are for an OrCAD Capture design and related files, PCB Editor board and related files, and any other types of files, including team libraries and team designs. Assigned leaders for capture projects will have see more buttons as enabled to manage the project. Assigned users whose role has been granted check in and check out permissions can check in and check out files.

A "Project" watermark is displayed on the upper right-hand corner of the Capture Project Detail View.



Capture Project Detail View

The organization of the Capture project is shown in the Capture Project Overview figure. Capture Project Action buttons are located at the top of the Capture project. File and Folder Action buttons for Capture project files are located inline with each file or folder. The **Project Selector** enables you to quickly navigate between projects without the need to open the project dashboard. The information in the project columns are defined in the Capture Project Column Information below.

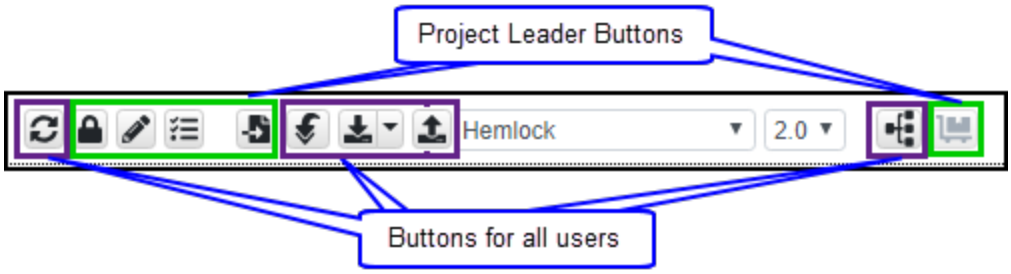


Capture Project Overview

Users may change the order or visibility of a column to customize their view.

Capture Project Action Buttons

The Capture Project action buttons appear in the upper left corner. The **Autogenerate** and **Package for Release** button appears to the right of the **Project Selector**. The actions enable you to do the following:



The bulleted list that follows describes each action button.

- **Refresh** button - refreshes the Project Details View. This button is enabled for all users.
- **Activate/Baseline** / button - toggles as you change from **Active** to **Baseline** state. When the state of the project is **Inactive** or **Baselined**, the button is used to activate/reactivate the project (). When the state of the project is **Active**, the button is used to baseline the project (). This button is enabled for Assigned Leaders whose role has been granted the Activate and Baseline Project permission.
- **Edit** button - accesses the Edit Project form. This button is enabled for Assigned Leaders whose role has been granted the Edit Project permission.
- **Mass Edit** button - This button allows you to delete a group of files or change properties for a group of selected files. This button is enabled for Assigned Leaders whose role has been granted the Edit Project permission.
- **Import** button - import design (.dsn) or board (.brd) into designated folders. This button is enabled for Assigned Leaders. The import schematic design feature is available when you open EDM from Capture. The import board feature is available from the EDM Desktop.
- **Full Project Download** button - downloads the entire project structure and files in the vault to your local workspace. This button is enabled for all assigned users, assigned leaders or users whose role has been granted View All Projects permission.
- **Check Out** button - check out items in the project. This button is enabled for all assigned users or leaders whose role has been granted the permission to perform check out and check in.
  - **Check Out options** button - presents options to Check Out (same as above) or Check Out and Download. These options are enabled for all assigned users or leaders whose role has been granted the permission to perform check out and check in.
- **Check In** button - check items into the project. This button is enabled for all assigned users or leaders whose role has been granted the permission to perform check out and check in.
- **Autogenerate** button:
  - Capture plugin: Auto-generate the selected Capture Design Related Files (Schematic **PDF**, **BOM**) for the local copy of the managed schematic design, without checking them in. This button will be disabled when there is no managed design.

- EDM Desktop application: Auto-generate the selected Board Related Files (Board **PDF, EDM Validation**) for the local copy of the managed board, without checking them in. This button will be disabled when there is no managed board.

- Package for Release button - allows an Assigned Leader whose role has been granted Package for Release permission to create a package that can be used for archive or release to manufacturing. This button is enabled only when EDM is opened with the desktop application.

The Search Filter allows you to display a subset of the files in the project by filtering on a data field. Click the **GO** button to apply your filter. Before or after clicking GO, toggle **Show "Local Only" Files** to show or hide the related files from the filter results.



Capture Project Search Filter

- **Field Filter** - select a field (e.g. Status) from the **Filter** list, then enter search parameter (e.g. 14) in the **by** text box.
- **Show "Local Only" Files** - select this box to show or hide local-only files in your filter. This applies to the current filtered results and you do not need to click the GO button to apply this option.

File and Folder Action Buttons

The file and folder action buttons perform actions on files and folders that are inline with the file or folder in the view. The buttons available for all users are: History, Cancel Check Out and Open. Assigned Leaders have three additional buttons to Add, Edit and Delete.



The bulleted list that follows describes each action button.

- **Add** button – allows an Assigned Leader to add a child folder. This button is disabled for Team folders or files. This button is only available for Assigned Leaders whose role has been granted the Edit Project permission. This button will not show for files or folder with Local Only state.

- **Edit** button – allows an Assigned Leader to edit folder and file properties. The edit folder form will display when this button is selected on a folder. The edit file properties form will display when this button is selected on a file. The edit team folder form will display when this button is selected on a team folder.This button is only available for Assigned Leaders whose role has been granted the Edit Project permission and is unavailable when files have the Local Only state.

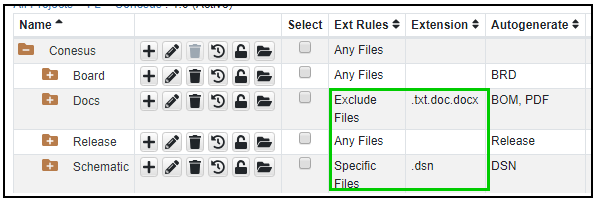
- **Delete** button – allows an Assigned Leaders whose role has been granted the Edit Project permission to delete the folder or file. The delete button is disabled for the project folder at the top level. This button is only available for Assigned Leaders. This button will not show for files with Local Only state. Deleting a file deletes it from the vault and your local workspace. Users who have a copy of the file in their local workspace need to refresh their view before the file is removed from their local workspace. Deleting a folder deletes the folder and its contents from the vault and your local workspace as well.
- **History** button – allows users to view the history of changes of an item. This feature is unavailable for files that have the Local Only state.
- **Cancel Check Out** button - allows users to remove the item from checked out state. When selected at the Project or Folder level, all files contained in sub-folders will be unlocked. This button is enabled only for Assigned Leaders, whose role has been granted with Cancel Project Checkouts, and users who have the item checked out. This button will not show for files with Local Only state.
- **Open** button - allows users to open the item using the program that has been associated to the item file type. For example, folder will be open using Windows Explorer, text files will be open with notepad. This button will not show for team folders.
- **Open Team folder** button - allows users to open team design folders that have been frozen.

Capture Project Column Information

The standard columns of information in Capture Project view contains the following:

- **Name** column – lists the names of items within the project. The icon in front of each name identifies the item type.
  - collapsed folder - select the name of the folder to view the contents of the folder.
  - expanded folder - select the name of the folder to hide the contents of the folder.
  - file
  - Team Folder - Team folder type will be identified in parenthesis next to the Team folder name. The Team folder types are Flat Team Design, Hierarchical Team Design and Team Library.
- **Select Checkboxes** - allows a user to select files to apply an action in the project. Select a checkbox and then click a Capture Project Action Button. *Note: Not all action buttons are applicable.
- **Local State** – shows the state of the file. Detailed information can be found in Local state Definitions. The local state helps you determine actions you can take when you need to make changes to a file or discard local changes when another user has check in changes.
- **Checked Out By** – identifies the user who currently has the file checked out.
- **Checked Out Date** – identifies the date and time stamp the user checked the file out.
- **Assigned Users** – lists users who have permission to check out the file.

- **Default Users** – contains a lists of users who will be automatically assigned when new files are checked into the folder. Field values for Default Users are only populated for folders.
- **Updated By** – identifies the user who last updated the file.
- **Last Check In Date** – date when file was last modified.
- **Version** - History version number associated with the Last Check In Date.
- These File Fields can be updated by a user when a file is checked in or by using the Edit File form.
  - **Status** – file field that shows the current status of the file, configurable in Customizable Fields as a pick list or free text. Folders do not have a status.
  - **Part Number** - file field that can be used to associated a part number to a given file, configurable in Customizable Fields as a pick list or free text. *Note: For Managed Design (.dsn) files, a Part Number can be added for the Core_Design and each variant.
  - **User Fields** - additional file fields, configurable in Customizable Fields as a pick list or free text.
- **Ext Rules** – shows the type of file extension rules configured for a folder by a project leader. The rule is applied when checking files into EDM. Files are configured by file extension. Rules that may be assigned to a folder are:
  - **Any Files** - allows all file extensions to be checked in.
  - **Specific Files** - allows only file extensions listed to be checked into the folder. All others will be ignored.
  - **Exclude Files** - allows all file extensions, excluding those listed, to be checked into the folder.



- **Extension** – file extensions correspond to the rule defined by the File Extension Rules for files that can be checked in the folder.
- **Autogenerate** - identifies the folders designated for files associated to the managed design and managed board where auto-generated files will be placed. Files auto-generated from the managed design are DSN, BOM, Netlist, PDF while files auto-generated from the managed board are BRD, PDF, and Release. The managed design file and the managed board are indicated by the blue star symbol. ★

Autogenerate Option

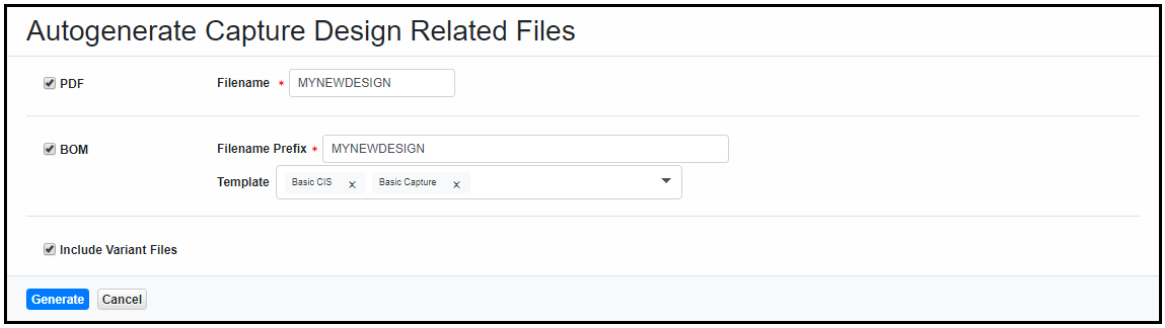
The autogenerate form opens when you click the **autogenerate** button.

- When open from the Capture plugin, you can **Autogenerate Capture Design Related Files**:

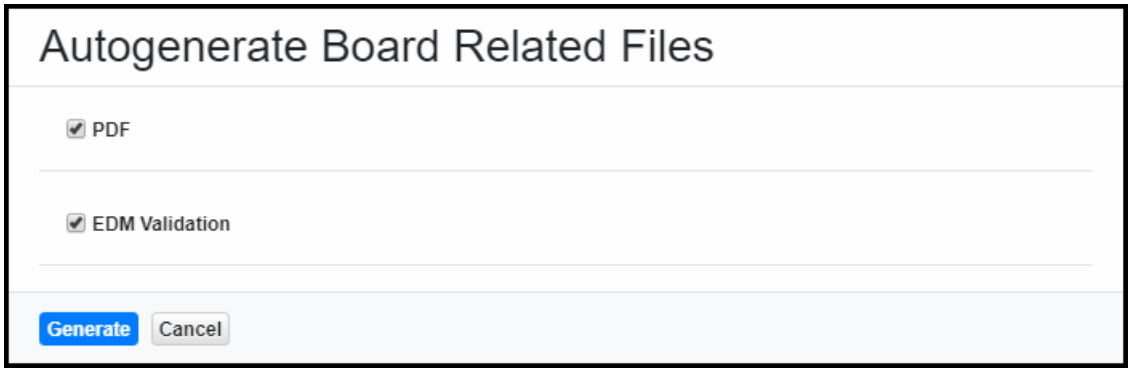
- **PDF** - You may change the PDF name generated using this method. The default shows the PDF name that is created when you use the autogenerate option during Check in of a Managed design.
- **BOM** - You may change the BOM **Filename Prefix** to use during BOM generation and select one or more BOM **Templates** to use. The default shows the BOM prefix that is used with the autogenerate option during Check in of a Managed design.
- **Include Variant Files** - For designs containing variants, selecting this option will also generate variants for your selected options.

- When open from the EDM Desktop application, you can **Autogenerate Board Related Files**:
  - **PDF** - There are no additional options to select for the managed board PDF.
  - **EDM Validation** - There are no additional options to select for the EDM Validation report.

Sample autogenerate forms from Capture Plug-in and EDM Desktop App are shown in the following figures.



Autogenerate Options from Capture Plug-in



Autogenerate Options from EDM Desktop App

4.3.1.1 Capture Designs in a Capture Project

Capture Design projects are one of the four types of projects that can be created in EDM. Capture Design projects track relationships between the managed schematic design (BOM, Netlist, PDF) and managed board (DB Doctor, EDM Validation, PDF). This allows associated files to be automatically generated when design and board files are checked in.

Design and board files may be imported into either a project template or a new project. When you create a Capture Design, you can select a project template to quickly create the project structure and properties.

Capture design projects include a Capture design file called the managed design, whether imported or auto-created during download. Only one design file can be designated as the Managed Capture Design. A team design may also be designated as the managed design. The managed design designation takes effect after the team design is frozen.

Files related to a Capture Design project are automatically generated when you Check In the managed design. The options for auto-generating files are presented when you select the managed design for check in. Auto-generating managed files upon check in keeps them in sync with the managed Capture design. The managed design is associated to related files by file names. If the file name of a managed design is changed in EDM after check in, the association between the managed design or board and associated files becomes out-of-sync. To update the file relationship, check out/check in the project files to regenerate the files with the updated managed name. To avoid or eliminate confusion, delete files that were checked in prior to the name change.

Capture Design projects include folder designations for auto-generated files. By default, folder designations are at the project folder level until you change the designation when you Add a Folder or Edit a Folder. The folder designation consist of the following:

- **DSN**

  The folder with this designation is used for the Managed Design. After Capture project is created, the project will not have any files. Before working on a project, you will need to download the project structure by clicking on the **Full Project Download** button. This creates the .DSN (managed design) and the .opj (Capture project file). The .opj file created by EDM contains the configured locations for the netlist and board files. To obtain an updated .opj file after changes are made to the project's folder designations, delete your local .opj file and perform a Full Project Download. The name of the managed design file is identified as a folder property. The managed design file is identified with a blue star (★) next to the filename. Check in of a managed design file from Capture enables the auto-generation functionality. Auto-generation of a BOM or PDF file creates files with the filenames configured in EDM. Changes to the BOM or PDF filenames must be made in the EDM project to retain the relationship of the managed design to these files. Checking in BOM or PDF with different filenames disassociates the relationship to the managed design.

- **BOM**

  The folder with this designation is used for the auto-generated BOM file based on the selected BOM Template(s). The BOM file that is autogenerated by default is given the same file name as the managed design. The file extension is either .BOM or .xlsx depending on the configured BOM Template. Changing the BOM file name disassociates the BOM from the managed design.

- **Netlist**

  The folder with this designation will be used for the auto-generated Netlist files. Three text files with your schematic design nets will be created and placed into this folder.

PDF

The folder with this designation will be used for the auto-generated PDF of the schematic. The PDF filename by default is given the same name as the managed design with a file extension of PDF. Changing the PDF file name will disassociate the PDF from the managed design. Auto-generating a PDF file of the schematic requires initial PDF Printer Setup.

BRD

The folder with this designation will be used for the Managed Board. The name of the managed board file is identified in a folder property. The file with the managed board file name is identified with a blue star (★) next to the filename. Check in of a managed board file from the EDM desktop application enables auto-generation functionality. Auto-generated reports (DB Doctor, EDM Validation) are placed into a Reports folder and the auto-generated PDF file is placed in the designated folder for the board file. These files are automatically created/updated when the board file is checked in or when manually running auto-generation.

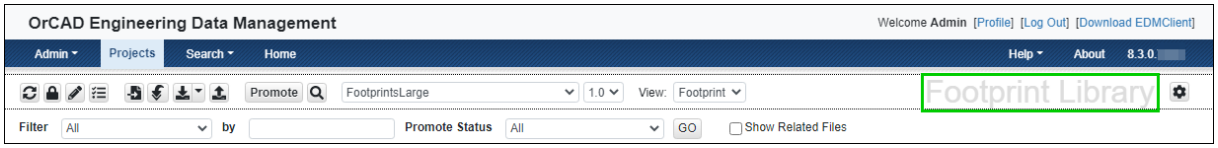
Release

The folder with this designation is used for files that are packaged for release to manufacturing. Selection of the Package for Release option allows you to select files for creation. The auto-generated files are compressed and saved in a zipped file that is saved in the project directory. Package for Release may include the auto-generated files: IPCs-2581, ODB++, Artwork, IPC 356, Fabmaster, Testprep, NC Route, NC Drill, and Pick and Place.

4.3.2 Footprint Project View

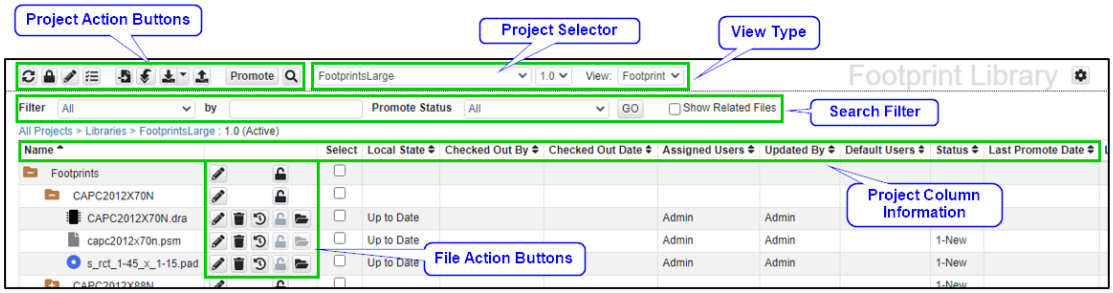
The Footprint Project View shows detailed information for footprints and related padstacks and step models.

A "Footprint Library" watermark is displayed on the upper right-hand corner of the Footprint Project Detail View.



Users assigned as a leader for footprint projects will have see more buttons as enabled to manage footprint project. Assigned users whose role has been granted check in and check out permissions can check in and check out footprint files. The purpose of the footprint project is controlling versions of files checked into this project. Placement of components for use in board files should use footprints from the promoted footprints location.

The general organization of the Footprint project is shown in the Footprint Project Overview figure. Project Action buttons are located at the top of the footprint project. File Action buttons for project files are located inline with each file or folder. The Project Selector enables you to quickly navigate between projects without the need to open the project dashboard. The information in the project columns are defined in the Footprint Project Column Information below.

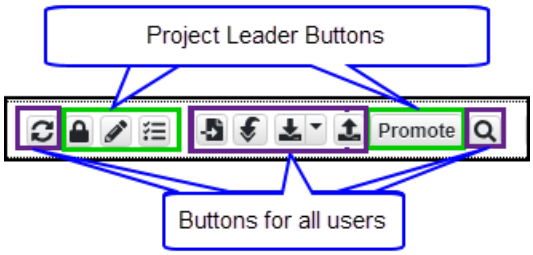


Footprint Project Overview

Users may change the order or visibility of a column to customize their view.

Project Action Buttons

The project action buttons appear in the upper left corner. The actions enable you to do the following:



- **Refresh** button - refreshes the Project Details View. This button is enabled for all users.
- **Activate/Baseline** / button - toggles as you change from **Active** to **Baseline** state. When the state of the project is Inactive or Baselined, the button will be used to activate/reactivate the project (). When the state of the project is **Active**, the button will be used to baseline the project (). This button is enabled for Assigned Leaders whose role has been granted the Activate and Baseline project permission.
- **Edit** button - accesses the Edit Project form. This button is enabled for Assigned Leaders whose role has been granted the Edit Project permission.
- **Mass Edit** button - This button allows you to delete a group of files or change properties for a group of selected files. This button is enabled for Assigned Leaders whose role has been granted the Edit Project permission.
- **Import** button - Import Footprints into existing folders. This button is enabled for Assigned Leaders.
- **Full Project Download** button - downloads the entire project structure including all files in the vault to your local workspace. This button is enabled for all assigned users, assigned leaders or users whose role has been granted View All Projects permission.
- **Check Out** button - check out selected items in the project. This button is enabled for all assigned users or leaders whose role has been granted the permission to perform check out and check in.

- **Check Out options** button - presents options to Check Out (same as above) or Check Out and Download. These options are enabled for all assigned users or leaders whose role has been granted the permission to perform check out and check in.

- **Check In** button - check items into the project. This button is enabled for all assigned users or leaders whose role has been granted the permission to perform check out and check in.
- **Promote** Promote button - copies the files associated to a selected footprint from the vault to the promoted footprint library locations specified for the footprint project. This button is enabled for Assigned Leaders whose role has been granted the permission to Promote Libraries.
- **View Promoted Library** button - displays the list of footprint files that have been promoted to the promoted library locations for the project.

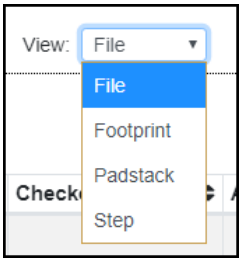
The Search Filter allows you to display a subset of the files in the project by filtering on a data field and/or Promote Status. Click the **GO** button to apply your filter. Before or after clicking GO, toggle **Show Related Files** to show or hide the related files from the filter results.



Footprint Project Search Filter

- **Field Filter** - select a field (e.g. Status) from the **Filter** list, then enter search parameter (e.g. 14) in the **by** text box.
- **Promote Status** - select a promote status (e.g. Not Promoted).
  - ***Note:** The Promote Status is determined by comparing the database Version of the latest copy of the file in EDM against the database Version of the file at the time it was last promoted. However, it **does not** physically open the promoted files or check for their existence in the configured promote path, but the promoted location should not be modified outside of EDM. The safest way to ensure this is to set the Library Promote Method to **EDM Web Application**.
- **Show Related Files** - select this box to show or hide associated files in your filter. This applies to the current filtered results and you do not need to click the GO button to apply this option.

Four different views are available from the View Control. You can toggle between the following view by selecting the View Control drop list menu at the top of footprint project.



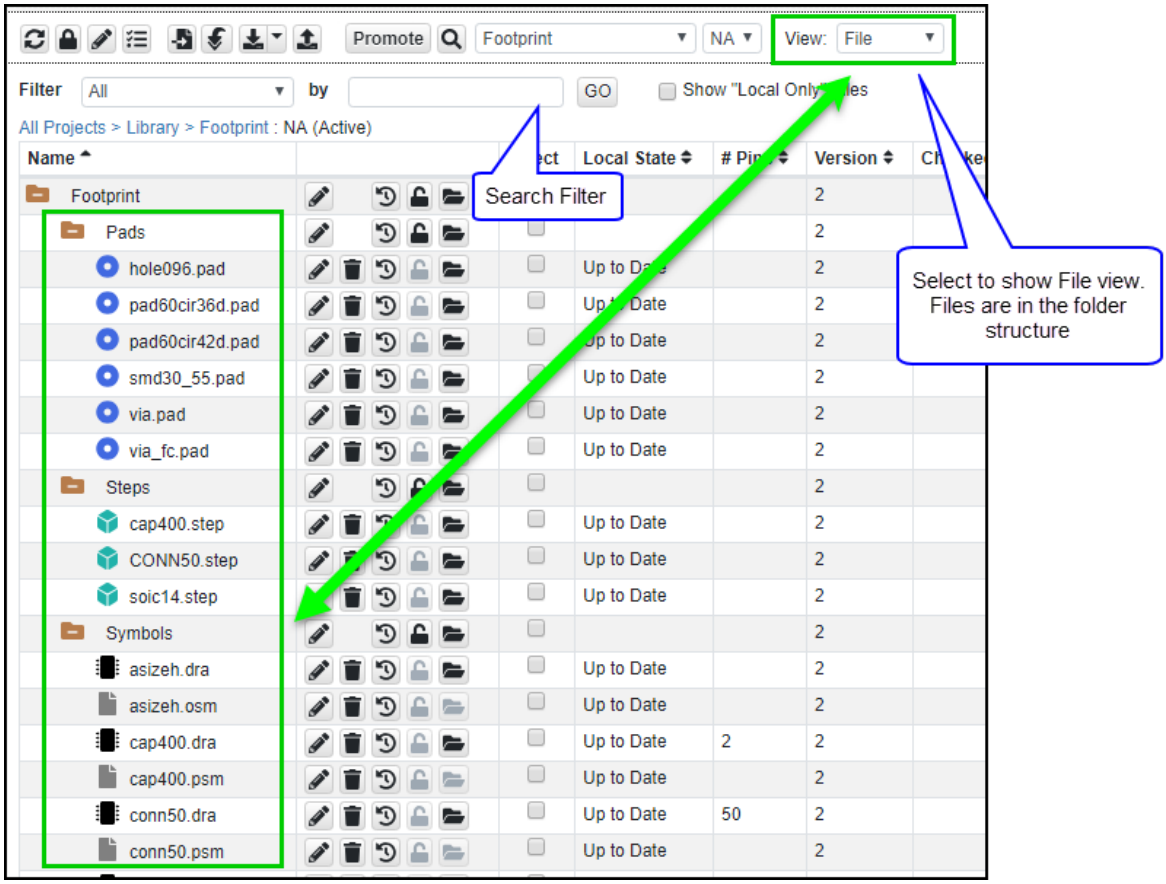
- File View - shows files in directory structure of the project. This matches files structure when you perform a full download to the project.
- Footprint View - shows files organized by footprints. Related padstacks and step files show with the footprint.

- Padstack View - shows files organized by padstacks. Related shape, flash and footprint files show with the padstacks.
- Step View - shows files organized by step models. Related footprint files show with the step models.

The list of promoted footprint and step files may be viewed by selecting the **View Promoted Library** button.

File View

The file view shows files in the structure that the project was created. This matches the organization typically used by PCB Editor. If you created your Footprint project with sub-folders and step models, the File view shows the three physical sub-folders of the footprint files as shown in the image below. A footprint project created without sub-folders will show all files under Footprint folder. The following figure shows an example with all physical sub-folders expanded.

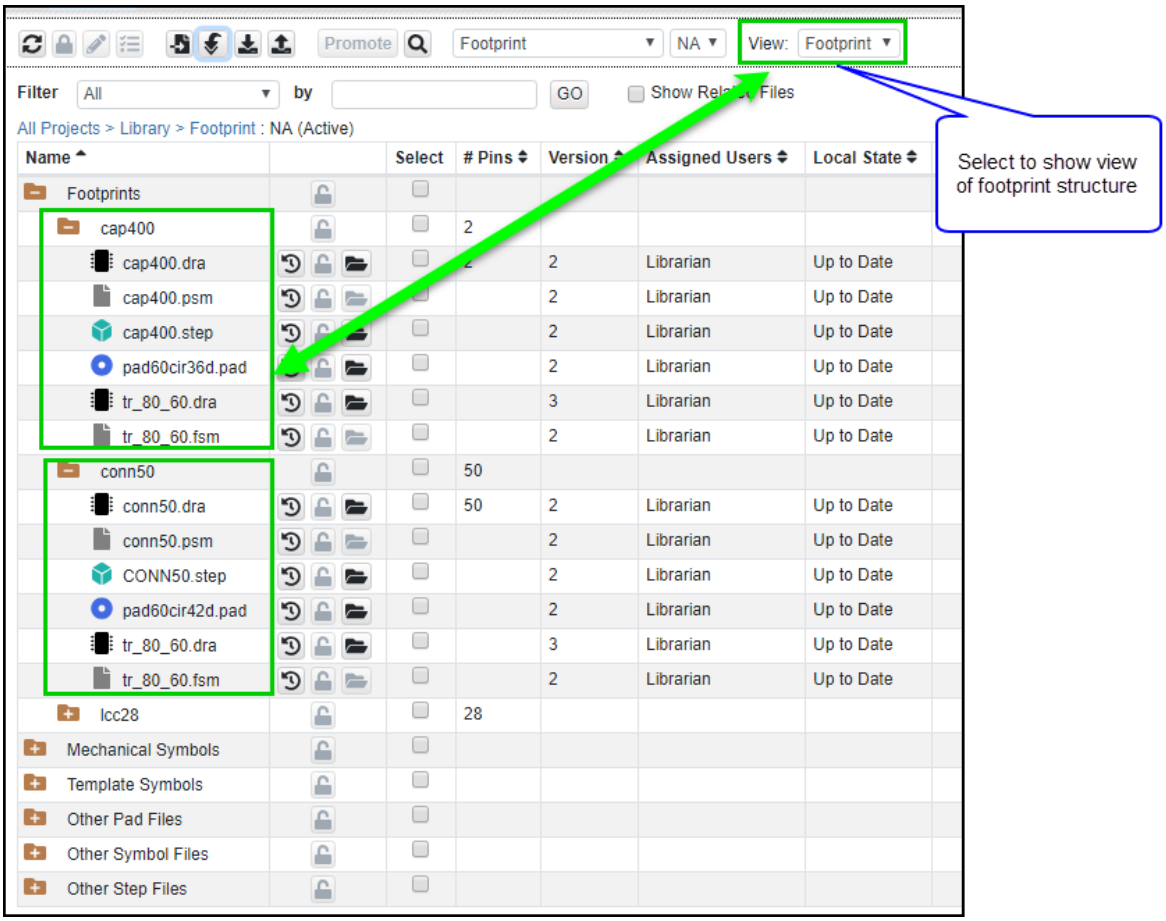


File View of Footprint Project

Footprint View

The footprint view lists the following six types of folders that can be expanded or collapsed by clicking the + and – symbols.

- **Footprints** – includes footprints (.dra and .psm) and associated step models (.step or .stp) and padstacks (.pad) with corresponding shape (.dra and .ssm) and flash (.dra and .fsm) files.
- **Mechanical Symbols** – includes mechanical symbols (.dra and .bsm) and associated step model (.step or .stp) and padstacks (.pad) with corresponding shape (.dra and .ssm) and flash (.dra and .fsm) files.
- **Template Symbols** - includes template (.dra and .osm) files.
- **Other Pad Files** – includes padstack (.pad) files and corresponding shape (.dra and .ssm) and flash (.dra and .fsm) files which are not associated to any footprints. This includes blind buried vias (BBVias).
- **Other Symbol Files** - includes shape (.dra and .ssm) and flash (.dra and .fsm) files not associated to any padstacks or compiled graphic (.psm) without corresponding footprint draw files.
- **Other Step Files** - includes step (.step or .stp) files not associated to any footprints.

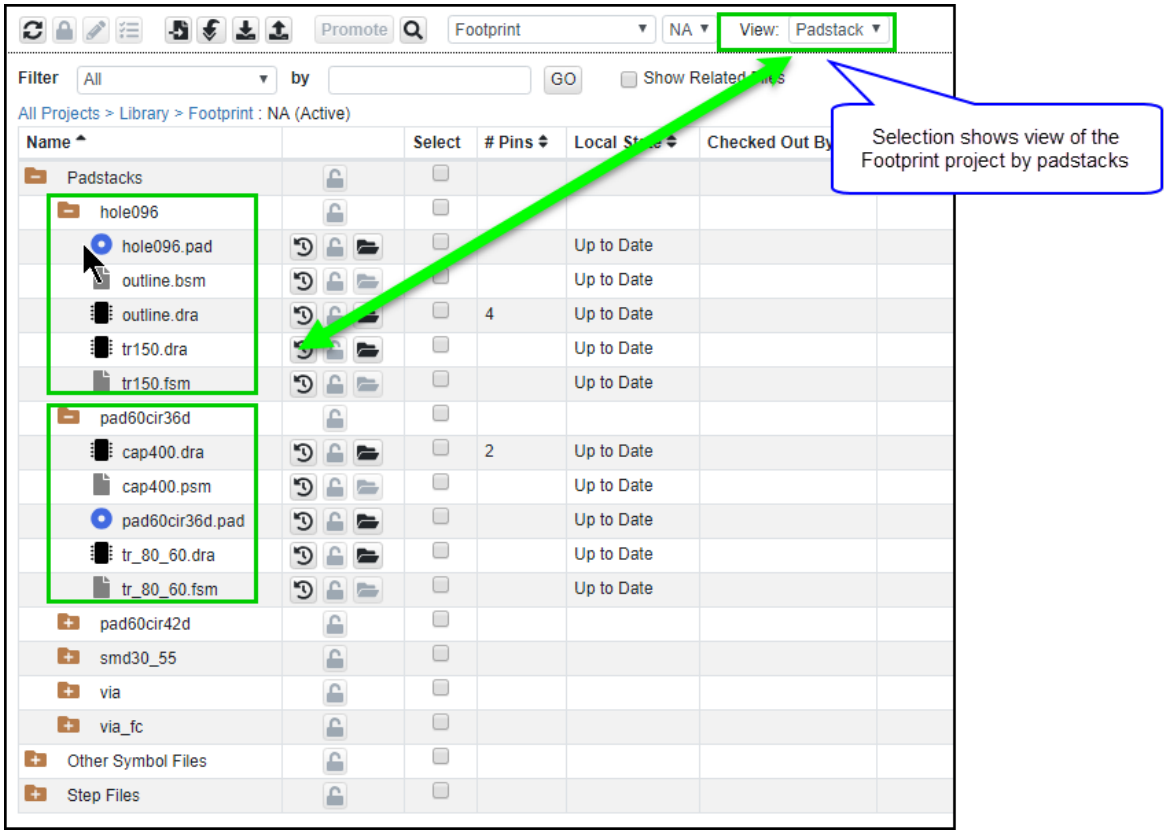


Footprint View of Footprint Project for General User

Padstack View

The Padstack view lists the following three types of folders that can be expanded or collapsed by clicking the + and – symbols.

- **Padstacks** – includes padstack (.pad) files with corresponding shape (.dra and .ssm) and flash (.dra and .fsm) files, and footprints (.dra and .psm) that use the padstack.
- **Other Symbol Files** - includes any symbol (.dra, .osm, .psm, .ssm) files that do not use any padstacks.
- **Step Files** - includes all step model (.step and .stp) files.

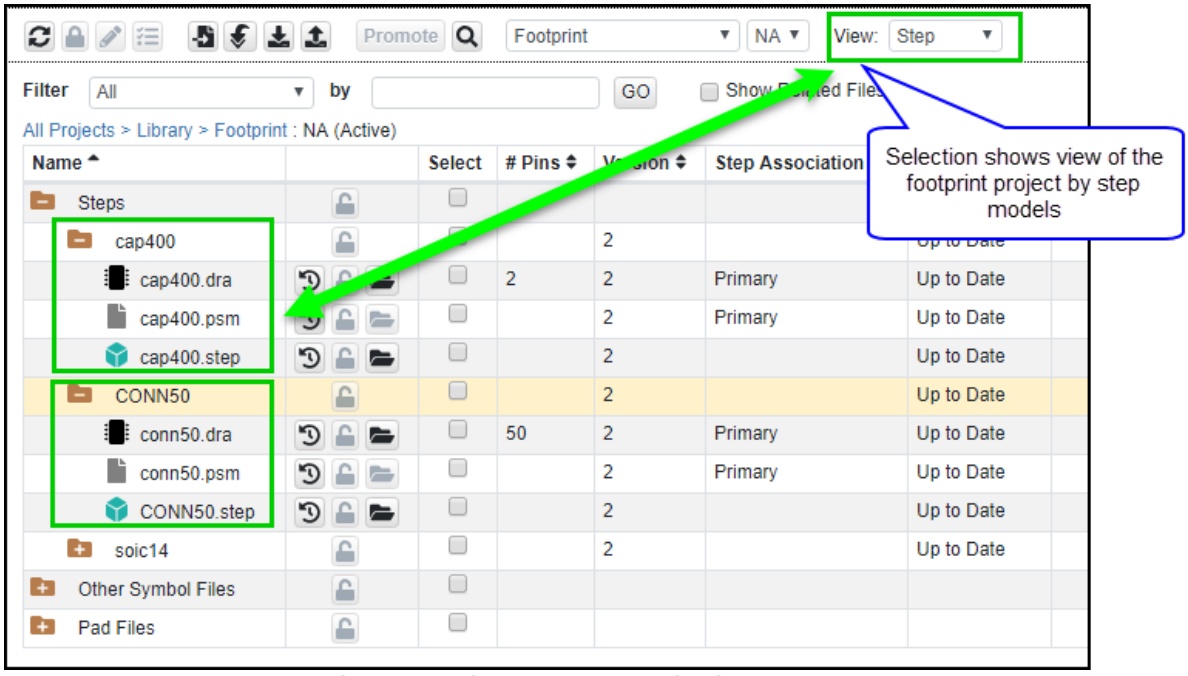


Padstack View of Footprint Project for General User

Step View

The Step view lists the following three types of folders that can be expanded or collapsed by clicking the + and – symbols.

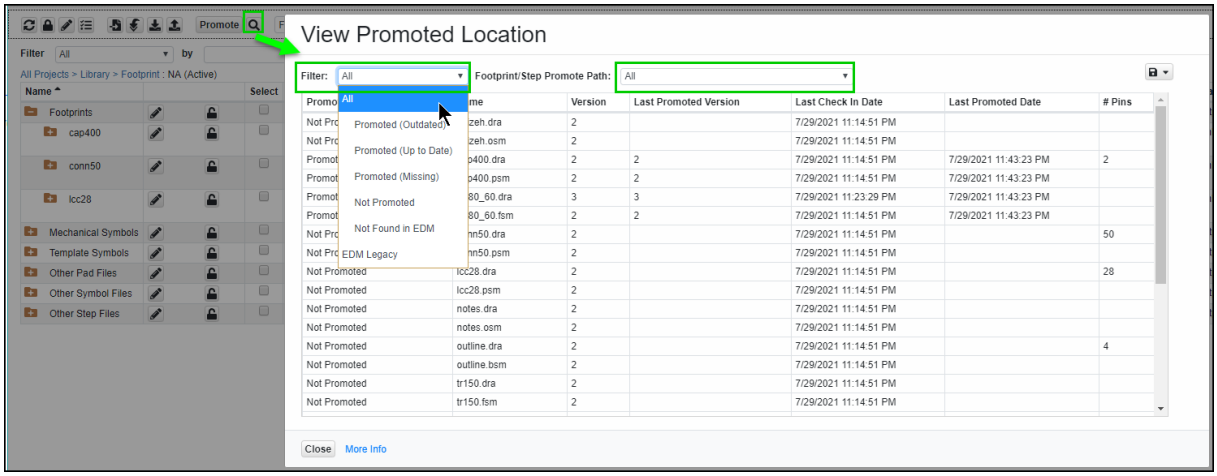
- **Steps** – includes step model (.step or .step) and footprint (.dra and .psm) files that use the step models.
- **Other Symbol Files** - includes any symbols (.dra, .osm, .psm, .ssm) files that do not use any padstacks.
- **Pad Files** - includes all padstack (.pad) files.



Step View of Footprint Project for General User

Viewing Promoted Footprints and Step Files

The **View Promoted Library** button in the Project Action Buttons opens a pop-up showing the promote status of your footprint project files.



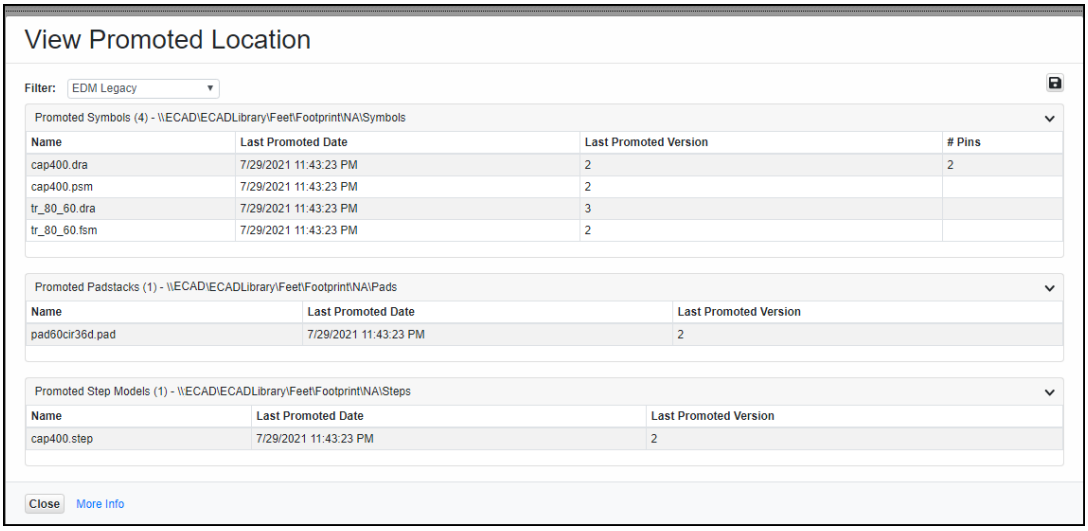
View List of Promoted Footprint and Step files

The Promoted Footprint View provides the full list of all footprints checked in and promoted to various locations, grouped by their Promote Status. There is a filter to narrow the list of files shown and an option to view each of the configured promoted locations.

***Note:** The Promote Status is determined by comparing the database Version of the latest copy of the file in EDM against the database Version of the file at the time it was last promoted. It also checks for the existence of the files in the promoted located. However, it **does not** physically open the promoted files, but the promoted location should not be modified outside of EDM. The safest way to ensure this is to set the Library Promote Method to **EDM Web Application**.

The filter options show the footprint files by the following:

- **All** - Shows all the footprint files in the project revision, regardless of promote status.
- **Promoted (Outdated)** - Promoted version is older than current version in project.
- **Promoted (Up to Date)** - Promoted versions match the current version in project.
- **Promoted (Missing)** - Promoted, but now the file is missing from promoted location.
- **Not Promoted** - Exists in project, but has not yet been promoted.
- **Not Found in EDM** - Exists in promoted location, but not in project.
- **EDM Legacy** - Provides footprints in groups that match the Promoted flyout from versions prior to 6.3.0.
  - **Promoted Symbols** - Shows symbol files found in the promoted location. Columns of data include Name (filename), Last Promoted Date, Last Promoted Version and Pin Count (for footprint .dra files only). Any manually added symbol files will show file name only.
  - **Promoted Padstacks** - Shows pad files found in the promoted location. Columns of data include Name (filename), Last Promoted Date, and Last Promoted Version. Any manually added pad files will show file name only.
  - **Promoted Step Models** - Shows step files found in the promoted location. Columns of data include Name (filename), Last Promoted Date, and Last Promoted Version. Any manually added step files will show file name only.

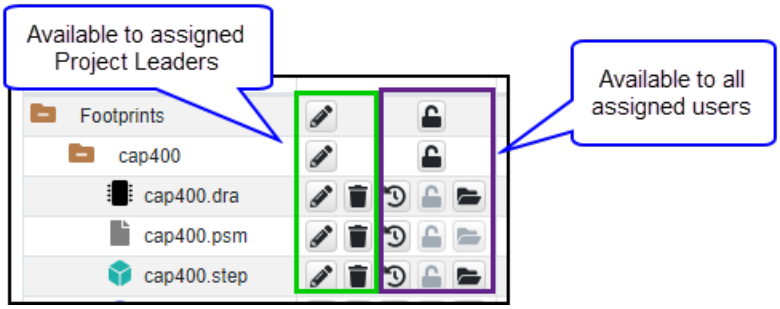


Sample EDM Legacy view

To export the list of all files found in the promoted location for off line view, click the **Export to File** button.

File Action Buttons

The file action buttons perform actions on the item that is inline with the buttons. The buttons available for all users are: History, Cancel Check Out and Open. Assigned leaders have two additional buttons to Edit and Delete items. Folders can't be deleted once created. You will need to delete the project.



Action Buttons

- **Edit** button - allows an Assigned Leader, whose role has been granted Edit Project, to edit either the selected folder (Edit Folder) or file properties (Edit File). Footprint folder properties that may be edited include: default users, file assignments and folder defaults at various levels. File properties that may be edited include: user assignments, status, and user field properties as shown in the following figure.
- **Delete** button - allows an Assigned Leader, whose role has been granted Edit Project, to delete a file. The delete button is disabled for folders. This button is only available for assigned leaders. This button will not show for files with **Local Only** state. To delete a folder, you must delete the project
- **History** button - allows users to view the history of changes on an item. This button will not show for files with **Local Only** state. The history can be exported to an Excel file or formatted as a CSV file.
- **Cancel Check Out** button - allows users to remove the item from checked out state. When selected at the Project or Folder level, all files contained in sub-folders will be unlocked. This button is enabled only for Assigned Leaders, whose role has been granted with Cancel Project Checkouts, and users who have the item checked out. When the check out of a .dra file is canceled, files associated to the .dra file (fsm, psm, ssm, osm, and bsm) are also automatically canceled. This button is unavailable for files with the **Local Only** state.
- **Open** button - allows users to open the item using the program associated to the item file type. For example, folder will be open using Windows Explorer, text files will be open with notepad. The open button is disabled for all .Xsm files.

Footprint Project Column Information

The columns of information in the detailed view shows properties for files and folders. You may customize this view and modify column order and visibility by clicking the **Gear** icon. The standard columns are:

- **Name** - name of the item within the project. The icon in front of each name identifies the item type, which includes:
  - collapsed folder - select the name of the folder to view the contents of the folder. Depressing the "**Ctrl**" button as you click will collapse all item at the folder.
  - expanded folder - select the name of the folder to hide the contents of the folder. Depressing the "**Ctrl**" button as you click will expand all item at the folder.

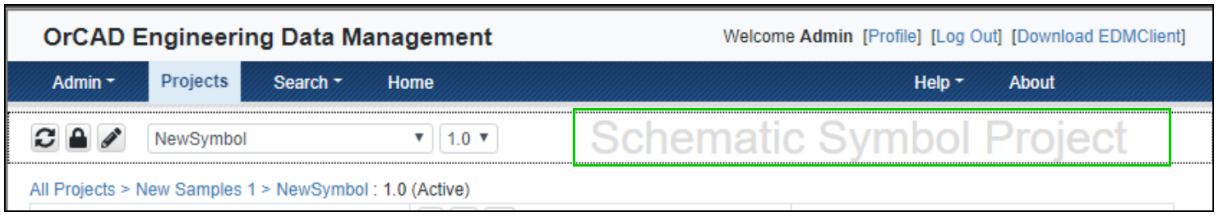
file

- **Select Checkboxes** - allows a user to select files to apply an action in the project. Select a checkbox and then click a Footprint Project Action Button. *Note: Not all action buttons are applicable.
- **Local State** - shows the state of the file. Detailed information can be found in Local state Definitions.
- **Checked Out By** - user that has the file checked out.
- **Checked Out Date** - date and time stamp when a user checked the file out.
- **Assigned Users -** contains list of users that has permission to check out the file.
- **Default Users** - contains a lists of users that will be automatically assigned when files are checked into the folder. Field values for Default Users are only populated for folders.
- **Relative Path** - shows folder and filename of the item.
- **Updated By -** user that last updated the file.
- **Last Check In Date** - date when symbol or page was last modified.
- **Version** - History version number associated with the Last Check In Date.
- **Pin Count** - number of pins that is associated with the footprint.
- These File Fields can be updated by a user when a file is checked in or by using the Edit File form.
  - **Status** – file field that shows the current status of the file, configurable in Customizable Fields as a pick list or free text. Folders do not have a status.
  - **User Fields** - additional file fields, configurable in Customizable Fields as a pick list or free text.
- **Last Promoted By** - identifies the user who most recently promoted the item.
- **Last Promote Date** - date that a footprint file has been promoted.
- **Last Promoted Version** - version of the footprint files that has been promoted.

4.3.3 Schematic Symbol Project View

The Schematic Symbol project view shows schematic symbol organized by created libraries.

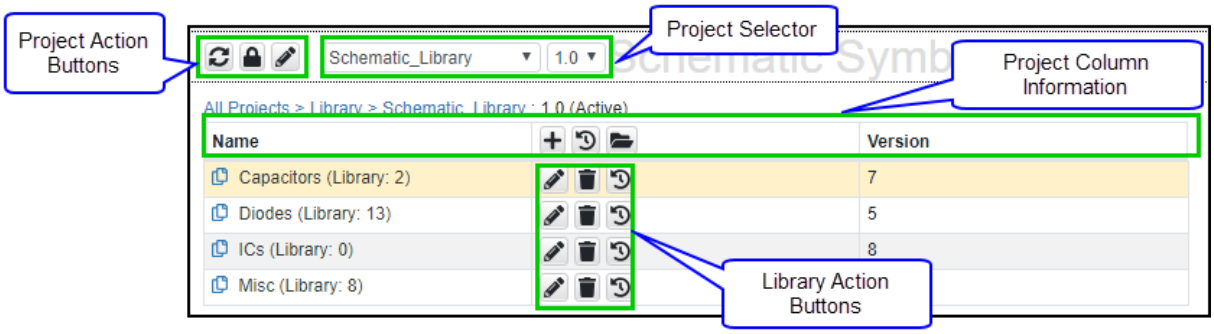
A "Schematic Symbol Project" watermark is displayed on the upper right hand corner of the Schematic Symbol Project Detail View.



Schematic Symbol Project Detail View

Users assigned as the project leader for Schematic Symbol projects will have see more buttons as enabled to manage library folders within the project. Assigned general users can check in and check out symbols. The purpose of the schematic symbol project is controlling versions of symbols checked into this project. Placement of symbols for use in designs should use symbols from the promoted symbols location.

Project Action buttons are located at the top of the schematic symbole project. Library Action buttons for library files are located inline with each library folder. If you are viewing a new Schematic Symbol project before library folders are created the page is blank. The Project Selector enables you to quickly navigate between projects without the need to open the project dashboard. The information in the project columns are defined in the Schematic Symbol Column Information below.



Schematic Symbol Project Layout

Project Action Buttons

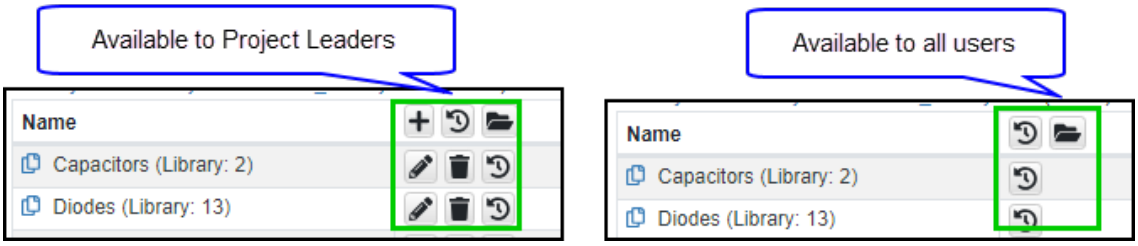
The project action buttons appear in the upper left corner. The actions enable you to do the following:



- **Refresh** button - refreshes the Project Details View. This button is enabled for all users.
- **Activate/Baseline** / button - toggles as you change from **Active** to **Baseline** state. When the state of the project is Inactive or Baselined, the button will be used to activate/reactivate the project ( ). When the state of the project is **Active**, the button will be used to baseline the project ( ). This button is enabled for Assigned Leaders whose role has been granted the Activate and Baseline Project permission.
- **Edit** button - accesses the Edit Project form. This button is enabled for Assigned Leaders whose role has been granted the Edit Project permission.

Library Action Buttons

The library action buttons perform actions on the library inline with the buttons. The button available for all users are: History and Open. Assigned leaders have three additional buttons to Add Library, and Edit and Delete items.



Action Buttons

- **Add** button – allows an Assigned Leader to add a library folder. This button is enabled for Assigned Leaders whose role has been granted the Edit Project permission.
- **Edit** button - allows an Assigned Leader to edit the selected library. Library folder properties and Schematic Symbol Library names may be modified. Schematic Library names across entire EDM must be unique. Symbol assignments may only be edit while in the Team Library Folder View. This button is enabled for Assigned Leaders whose role has been granted the Edit Project permission.
- **Delete** button - allows an Assigned Leader to delete a library. This button is enabled for Assigned Leaders whose role has been granted the Edit Project permission.
- **History** button - allows users to view the history of changes on the library. The history can be exported to an Excel file or formatted as a CSV file.
- **Open** button - allows users to open the directory of the project in the local workspace.

Schematic Symbol Column Information

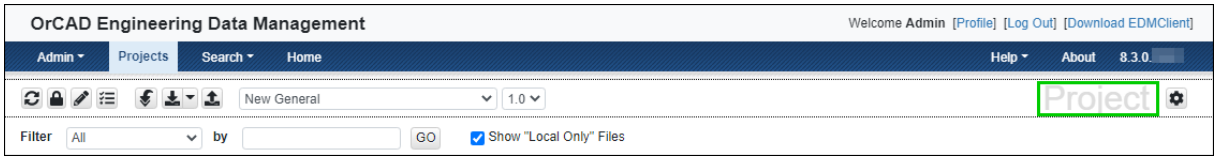
The columns of information in the detailed view shows properties for library folders. The columns are:

- **Name** - name of the library folder within the project shown with the number of symbols within each library. The icon in front of each name identifies the item type as a team folder. Click on the icon to open the Team Library Folder View.
- **Version** - shows the version number corresponding to the last action made to the library folder.

4.3.4 General Project View

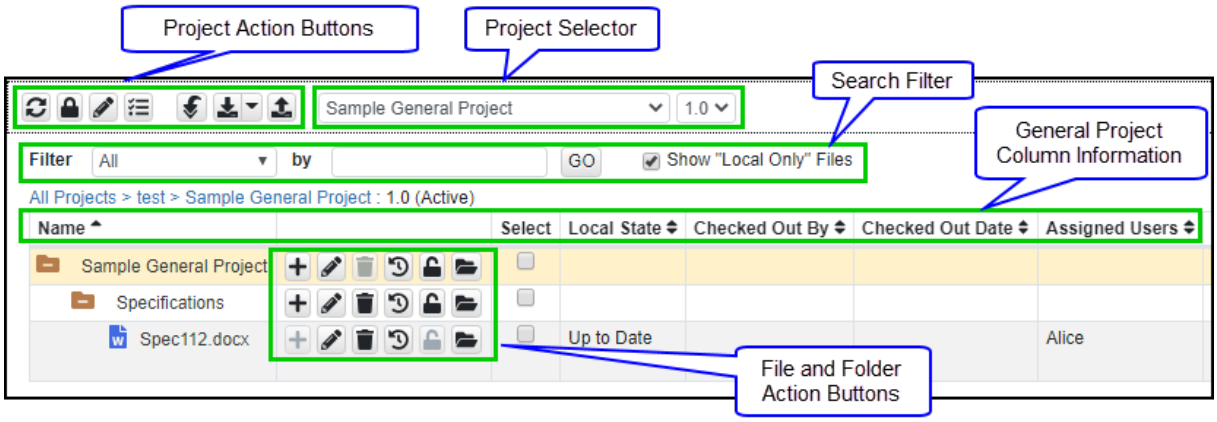
The General Project View shows detailed information for folder structure of files that may be used for any types of files, including team libraries and team designs. Besides team folders, no additional OrCAD Capture or PCB Editor functionality comes built in with the General Project. Users assigned as the project leader for general projects will have see more buttons as enabled to manage the project. Assigned general users can check in and check out files.

A "Project" watermark is displayed on the upper right-hand corner of the General Project Detail View.



General Project Detail View

The organization of the General project is shown in the General Project Overview figure. Project Action buttons are located at the top of the general project. File and Folder Action buttons for project files are located inline with each file or folder. The Project Selector enables you to quickly navigate between projects without the need to open the project dashboard. The information in the project columns are defined in the General Project Column Information below.

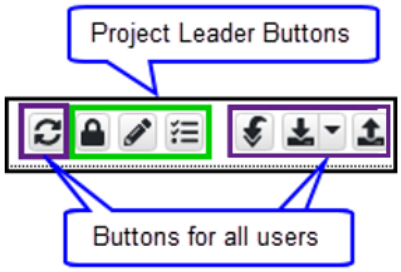


General Project Overview

Users may change the order or visibility of a column to customize their view.

General Project Action Buttons

The General Project action buttons appear in the upper left corner. The actions enable you to do the following:



The bulleted list that follows describes each action button.

- **Refresh** button - refreshes the Project Details View. This button is enabled for all users.
- **Activate/Baseline** / button - toggles as you change from **Active** to **Baseline** state. When the state of the project is **Inactive** or **Baselined**, the button is used to activate/reactivate the project (). When the state of the project is **Active**, the button is used to baseline the project (). This button is enabled for Assigned Leaders whose role has been granted the Activate and Baseline Project permission.

- **Edit** button - accesses the Edit Project form. This button is enabled for Assigned Leaders whose role has been granted the Edit Project permission.
- **Mass Edit** button - This button allows you to delete a group of files or change properties for a group of selected files. This button is enabled for Assigned Leaders whose role has been granted the Edit Project permission.
- **Full Project Download** button - downloads the entire project structure and files in the vault to your local workspace. This button is enabled for all assigned users, assigned leaders or users whose role has been granted View All Projects permission.
- **Check Out** button - check out items in the project. This button is enabled for all assigned users or leaders whose role has been granted the permission to perform check out and check in.
  - **Check Out options** button - presents options to Check Out (same as above) or Check Out and Download. These options are enabled for all assigned users or leaders whose role has been granted the permission to perform check out and check in.
- **Check In** button - check items into the project. This button is enabled for all assigned users or leaders whose role has been granted the permission to perform check out and check in.

The Search Filter allows you to display a subset of the files in the project by filtering on a data field. Click the **GO** button to apply your filter. Before or after clicking GO, toggle **Show "Local Only" Files** to show or hide the related files from the filter results.



General Project Search Filter

- **Field Filter** - select a field (e.g. Status) from the **Filter** list, then enter search parameter (e.g. 14) in the **by** text box.
- **Show "Local Only" Files** - select this box to show or hide local-only files in your filter. This applies to the current filtered results and you do not need to click the GO button to apply this option.

File and Folder Action Buttons

The file and folder action buttons perform actions on files and folders that are inline with the file or folder in the view. The buttons available for all users are: History, Cancel Check Out and Open. Assigned leaders have three additional buttons to Add, Edit and Delete.



The bulleted list that follows describes each action button.

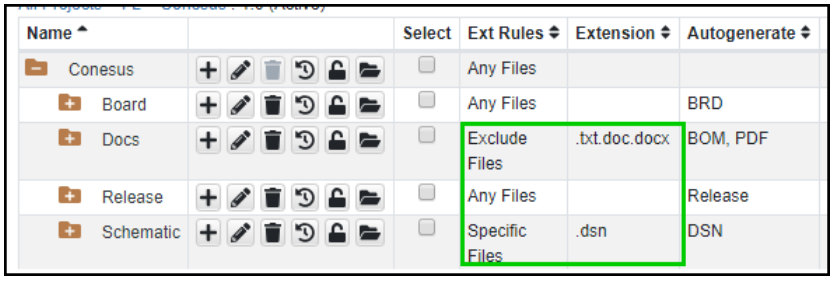
- **Add** button – allows an Assigned Leader to add a child folder. This button is disabled for Team folders or files. This button is only available for Assigned Leaders whose role has been granted the Edit Project permission. This button will not show for files or folder with Local Only state.
- **Edit** button – allows an Assigned Leader to edit folder and file properties. The edit folder form will display when this button is selected on a folder. The edit file properties form will display when this button is selected on a file. The edit team folder form will display when this button is selected on a team folder. This button is only available for Assigned Leaders whose role has been granted the Edit Project permission and is unavailable when files have the Local Only state.
- **Delete** button – allows an Assigned Leader to delete the folder or file. The delete button is disabled for the project folder at the top level. This button is only available for Assigned Leaders whose role has been granted the Edit Project permission. This button will not show for files with Local Only state. Deleting a file deletes it from the vault and your local workspace. Users who have a copy of the file in their local workspace need to refresh their view before the file is removed from their local workspace. Deleting a folder deletes the folder and its contents from the vault and your local workspace as well.
- **History** button – allows users to view the history of changes of an item. This feature is unavailable for files that have the Local Only state.
- **Cancel Check Out** button - allows users to remove the item from checked out state. When selected at the Project or Folder level, all files contained in sub-folders will be unlocked. This button is enabled only for Assigned Leaders whose role has been granted Cancel Project Checkouts and users who have the item checked out. This button will not show for files with Local Only state.
- **Open** button - allows users to open the item using the program that has been associated to the item file type. For example, folder will be open using Windows Explorer, text files will be open with notepad. This button will not show for team folders.
- **Open Team folder** button - allows users to open team design folders that have been frozen.

General Project Column Information

The standard columns of information in General Project view contains the following:

- **Name** column – lists the names of items within the project. The icon in front of each name identifies the item type.
  - collapsed folder - select the name of the folder to view the contents of the folder.
  - expanded folder - select the name of the folder to hide the contents of the folder.
  - file - refer to File and Folder Icons section on various file types
  - Team Folder - Team folder type will be identified in parenthesis next to the Team folder name. The Team folder types are Flat Team Design, Hierarchical Team Design and Team Library.

- **Select Checkboxes** - allows a user to select files to apply an action in the project. Select a checkbox and then click a General Project Action Button. *Note: Not all action buttons are applicable.
- **Local State** – shows the state of the file. Detailed information can be found in Local state Definitions.
- **Checked Out By** – identifies the user who currently has the file checked out.
- **Checked Out Date** – identifies the date and time stamp the user checked the file out.
- **Assigned Users** – lists users who have permission to check out the file.
- **Default Users** – contains a lists of users who will be automatically assigned when new files are checked into the folder. Field values for Default Users are only populated for folders.
- **Updated By** – identifies the user who last updated the file
- **Last Check In Date** – date when file was last modified.
- **Version** - History version number associated with the Last Check In Date.
- These File Fields can be updated by a user when a file is checked in or by using the Edit File form.
  - **Status** – file field that shows the current status of the file, configurable in Customizable Fields as a pick list or free text. Folders do not have a status.
  - **Part Number** - file field that can be used to associated a part number to a given file, configurable in Customizable Fields as a pick list or free text.
  - **User Fields** - additional file fields, configurable in Customizable Fields as a pick list or free text.
- **Ext Rules** – shows the type of file extension rules configured for a folder by a project leader. The rule is applied when checking files into EDM. Files are configured by file extension. Rules that may be assigned to a folder are:
  - **Any Files** - allows all file extensions to be checked in.
  - **Specific Files** - allows only file extensions listed to be checked into the folder. All others will be ignored. Files with unspecified extensions will show Local State as Local Only.
  - **Exclude Files** - allows all file extensions, excluding those listed, to be checked into the folder.

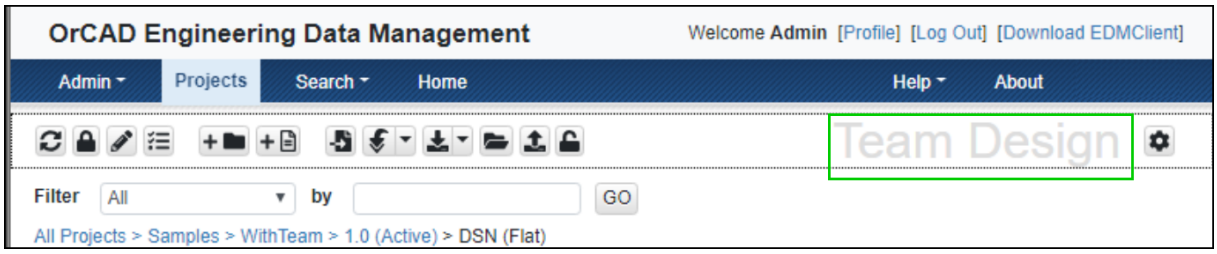


- **Extension** – file extensions correspond to the rule defined by the File Extension Rules for files that can be checked in the folder.

4.3.5 Team Design Detail View

A Team Design folder view shows information associated to pages for a design file that is either for a hierarchical or a flat design.

A "Team Design" watermark is displayed on the upper right hand corner of the Team Design Detail View.



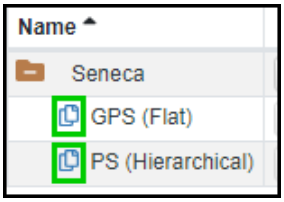
Team Design Details View

The pages for a hierarchical design are grouped by schematics that is part of the design file whereas pages for a flat design may appear in groups as defined by a project leader. The team design folder enables a team of users to work concurrently on different pages of a single OrCAD Capture design file. Refer to the example figures of Team Design Folder view of a Capture Design on the end of this section.

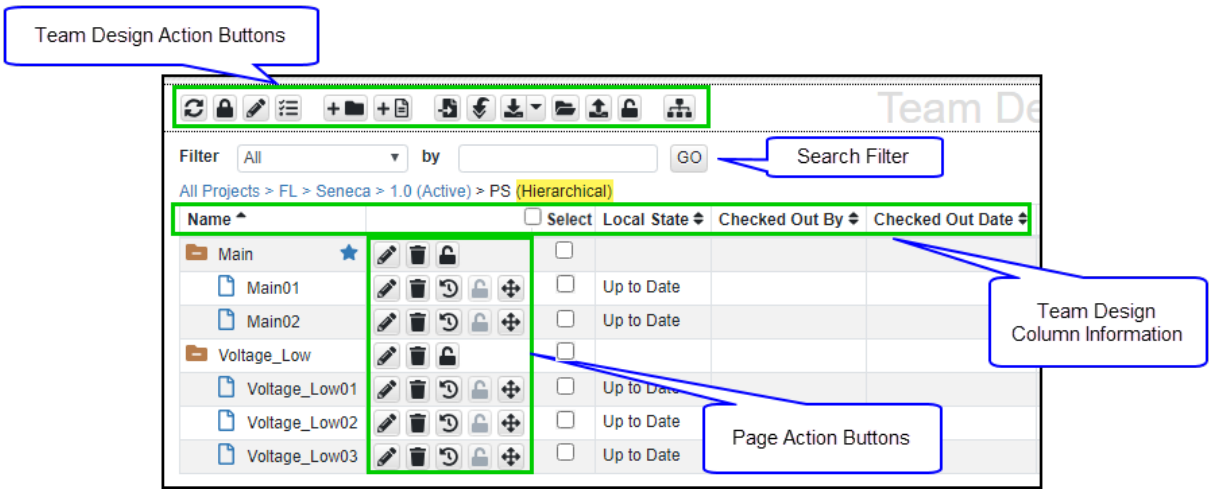
EDM enables Team Design by splitting a design to individual pages before import or check in and merging design pages when the design is downloaded or pages are checked out. To ensure reference designator can be managed properly, all team members working on a Team Design should always open EDM or enable RefDes when working on any portion of the design. A typical project work flow with EDM Team folders occurs when multiple users check out schematic pages each day as pages are being developed. To optimize synchronization with others, the schematic pages should be checked out and checked in daily.

Users assigned as the project leader for a project with the Team Design will have see more buttons as enabled to manage the team design. Assigned general users can check in and check out pages of the design.

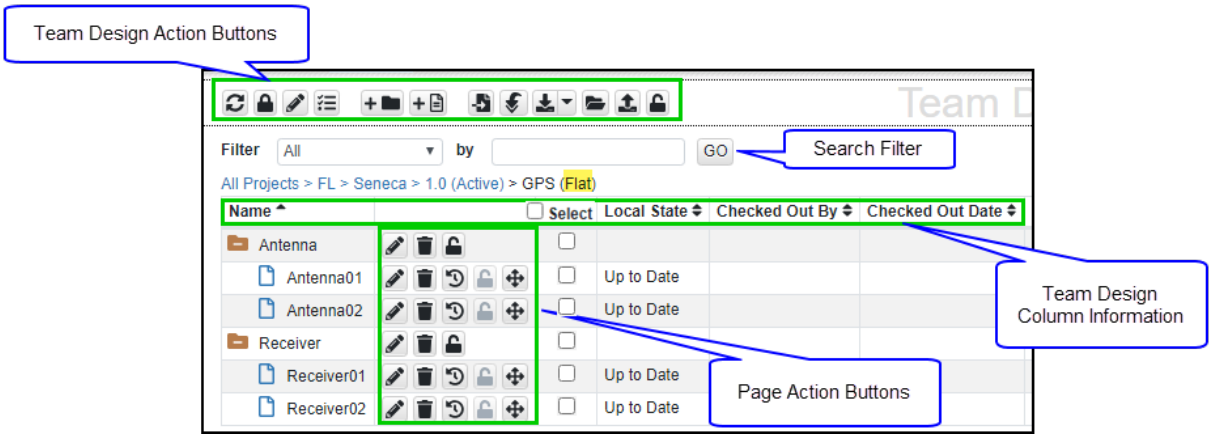
The Team folders are identified with the icon, in front of the folder name in the Project Details View for a Capture or General Project. You can open the Team Folder Details View by selecting the Team folder name or icon.



The general organization of the Team Design Folder view are shown the following two figures: Hierarchical Team Design Overview and Flat Team Design Overview. Team Design Action buttons are located at the top of the Team Design folder view. Page Action buttons for page files are located inline with each file or folder. The information in the Team Design columns are defined in the Team Design Column Information below.

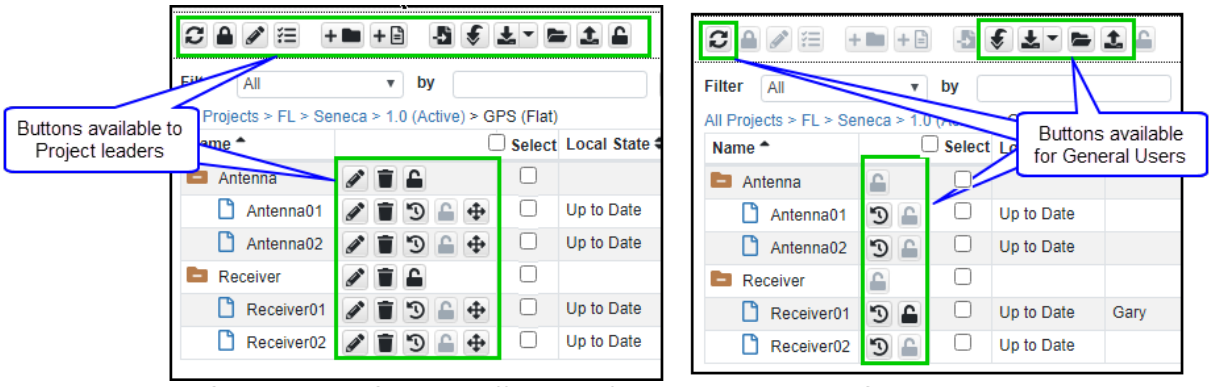


Hierarchical Team Design Overview



Flat Team Design Overview

The following figure shows the differences in view between a Assigned Leader and an Assigned User.

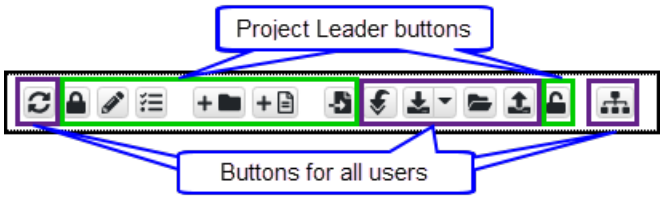


Sample Views Showing Differences for Project Leader and General User

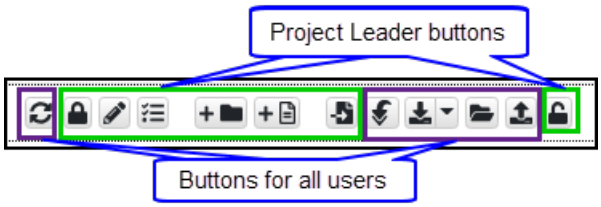
Users may change the order or visibility of a column to customize their view for Team Design. Additionally, users may use the filter to search for pages.

Team Design Action buttons

The Team Design action buttons appear in the upper left corner. The actions enable you to do the following:



Action Button for Hierarchical



Action buttons for Flat Designs

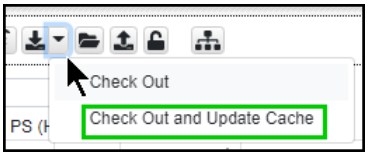
The bulleted list that follows describes each action button.

- **Refresh** button - refreshes the Team Design Details View.
- **Freeze** button - allows the Assigned Leader to freeze the Team Design to a single merged design file. This button is available for both Flat and Hierarchical Team Designs. Splitting the merge design to pages will require copying the project to a new revision.
- **Edit** button - allows the Assigned Leader whose role has been granted the Edit Project permission to edit user assignments across all groups or schematics.
- **Mass Edit** button - This button allows you to delete a group of files or change properties for a group of selected files. This button is enabled for Assigned Leaders whose role has been granted the Edit Project permission.
- Add **Group** or **Schematic** button - This allows the Assigned Leader whose role has been granted Edit Project permission to add a group with pages to the Flat team folder or add Schematics with pages to the Hierarchical team folder. This button is available for both Flat and Hierarchical Team Designs.
- Add **Page** button - This allows the Assigned Leader whose role has been granted Edit Project permission to add pages to an existing group (flat) or schematic (hierarchical). This button is available for both Flat and Hierarchical Team Design folders.
- **Import** button - This allows the Assigned Leader to import pages from an existing design. If a hierarchical design is imported to a flat team design, only the root schematic will be imported. If a flat design is imported to a hierarchical team design, the design will be imported as a schematic.

- **Full Download** button - download a copy of all pages in the Team design folder and merge pages to recreate your design file. The following options are available for Full Download, all of which require Capture Client plugin to merge the pages into a design:
  - **Full Download -** All items are merged to a single file. When a full download is performed to a team design that has been previously downloaded, pages with the same name will be updated. Any new pages will be added. Pages in your local design that haven't been check in are left untouched.
  - **Full Download with Update Cache** - With this option, in addition to the **full download, a forced update cache is performed** on the local workspace design file as well as on each downloaded page before merging into your local design file. If the pages in the Team Design were imported from a different design file, the symbols of the original pages may differ. The update cache option ensures symbols are updated to the latest library symbol before being merged. The Update Cache option takes longer for the download.
  - **Delete Local and Clean** - With this option, you local design is first deleted and then a **Full Download with Update Cache is performed**. If pages within the design have been renamed, a Full download will add pages with the new names to your local design containing the older page names. This option enables you to get a clean copy from the Vault. Any new pages that are in your local design and and haven't been checked in will be lost when using this option.



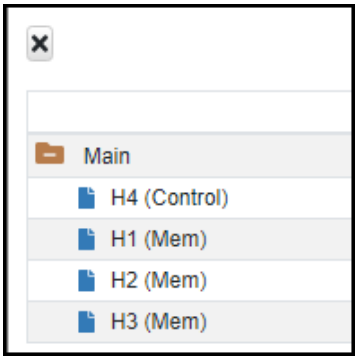
- **Check Out** button - check out pages selected in the team design folder. The following options are available for Check Out, all of which require Capture Client plugin to merge the pages into a design:
  - **Check Out -** A **Full Download** is automatically performed when you check pages out.
  - **Check Out and Update Cache** - A **Full Download with Update Cache** is automatically performed when you check pages out with this option.



- **Open Design** button - open the local merged design (in Capture, if you have configured .DSN files to be opened with Capture). This button will be disabled if you do not have a local merged design, in which case you will need to perform one of the **Full Download** or **Check Out** actions (with or without Update Cache) as described above.
- **Check In** button - check in items to update the vault with your local modifications.
- **Cancel Check Out** button - allows the Assigned Leader whose role has been granted Cancel Project Checkouts to cancel check out of all pages with a single action. All other users may only cancel check out his or her own checked out pages, using the Page Action Button for Cancel

Check out. The cancel check out action will restore the page to the latest saved version with the READ ONLY watermark on the page.

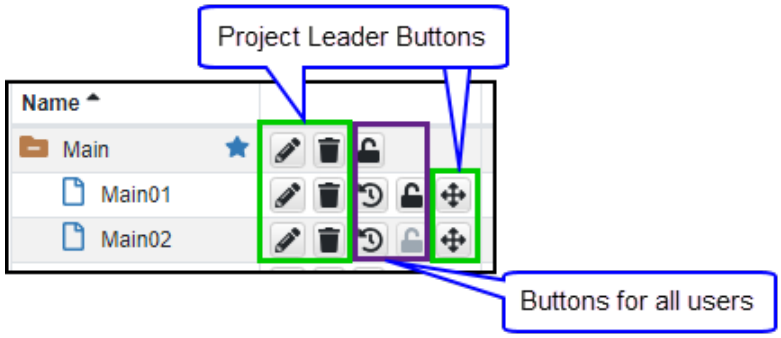
- **View Hierarchy** button - allows any user to display the hierarchy of the Hierarchical Team Design as shown in the following figure. This button is only available for a Hierarchical Team Design.



View Hierarchy

Page Action Button

The page action buttons perform actions on pages that are inline with the page, group or schematic. The buttons available for all users are: History and Cancel Check Out. Assigned Leaders have three additional buttons to Edit, Delete and Move Page.



Page Action Buttons

- **Edit** button - allows an Assigned Leader whose role has been granted the Edit Project permission to edit page and group/schematic properties for Team Designs. For group/schematics, the Edit Group form will display when this button is selected.
- Delete button - allows an Assigned Leader whose role has been granted the Edit Project permission to delete the page, group or schematic.
- History button - allows users to view the history of changes on an item. History is not enabled for group or schematic.
- Cancel Check Out button - allows users to remove the page or group of pages from checked out state. This button is enabled for Assigned Leaders whose role has been granted Cancel Project Checkouts and users who have the item checked out.
- Move button - allows the Assigned Leader whose role has been granted the Edit Project permission to move a page from one group to another group.

Team Design Column Information

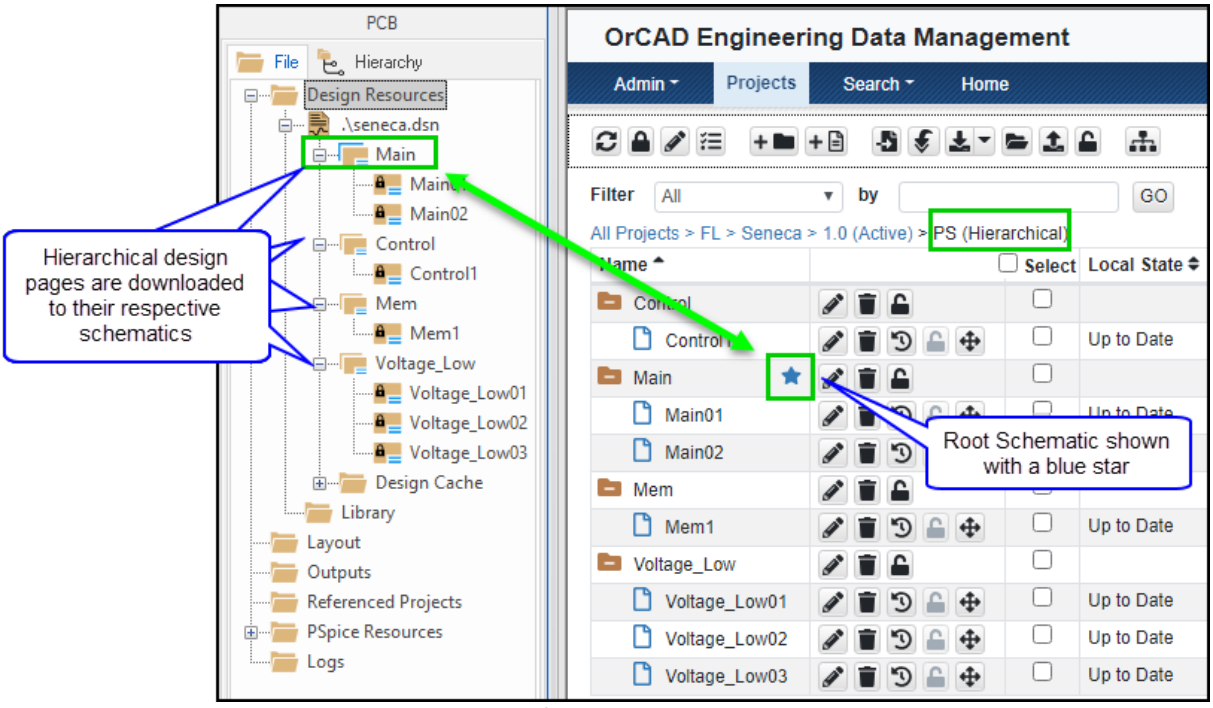
The standard columns of information in Team Design view contains the following:

- **Name** - name of item within the team design folder.The icon in front of each name identifies the item type. Types of items are as follows:
  - collapse group/schematic - select the name of the group/schematic to view the contents of the group/schematic.
  - expand group/schematic - select the name of the group/schematic to hide the contents of the group/schematic.
  - page within the group/schematic

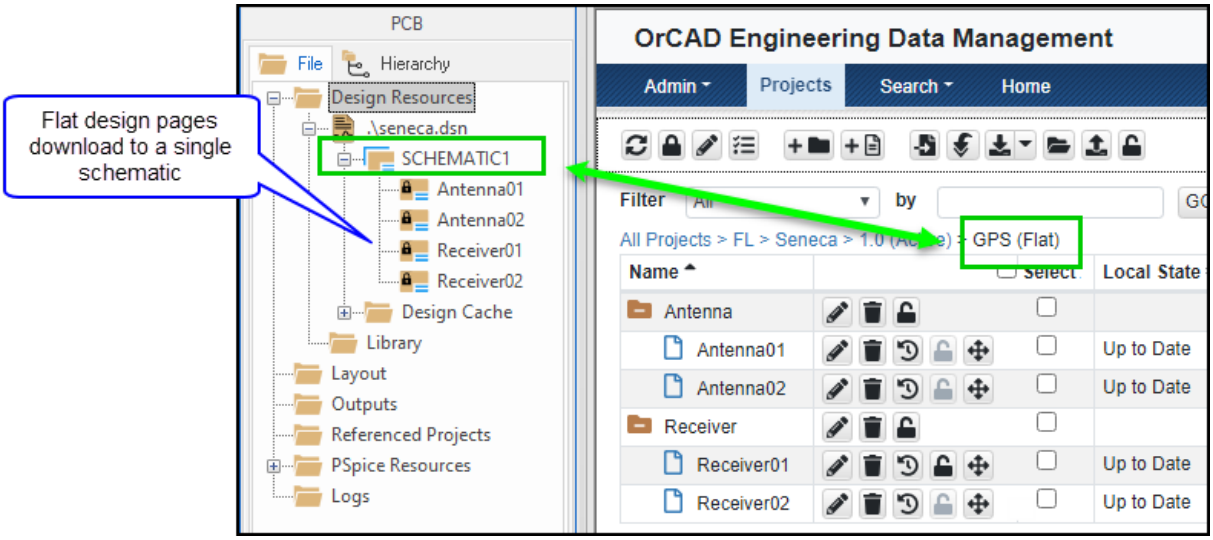
- **Check Out** checkboxes - Select the checkboxes next to the page or groups of page you want to check out. checkboxes for items that your are not assigned are disabled. Click the Check Out button at the top of the page to complete the check out action.
- **Local State -** shows the state of the page. Unlike files, EDM is unaware of changes made to pages in your local design file. There are only 3 valid local states in Team Designs. Design pages not yet checked are unavailable in the Team Design folder view and will not show as "Local only". Detailed information can be found in Local state Definitions. The Local State may have one of the following values:
  - **Up to Date** - the revision number of the checked out copy matches the revision in the vault, regardless whether you have made any changes to the checked out page.
  - **Out of date** - the revision number of the checked out copy is older than the revision in the vault.
  - **No Local Copy** - item is in the vault but has not been downloaded or checked out.
- **Checked Out By -** user who has the page checked out.
- **Checked Out Date**- date and time stamp when the user checked out the page.
- **Assigned Users** - user who has permission to work on the page.
- **Default Users** - contains a lists of users who will be automatically assigned when new pages are checked into the group or schematic. Field values for Default Users are only populated for group or schematic.
- **Updated By** - user who last updated the page.
- **Status** - status for the page within the team design folder. Schematics and Groups will not have a status. This field is typically updated by the user when pages are checked in. It may also be modified by an assigned leader when editing page or schematic/group.
- **User Fields** - additional fields configured from user fields. User field columns are based on the Project Administrators setup in EDM Configuration for Customizable Fields for team folder. These fields may be modified by an assigned leader when editing a page or schematic/group.
- **Version** - History version number associated with the Last Check In Date.
- **Last Check In Date** - date when a page was last modified and checked in.

Examples of Capture Design in Team Design Folder

The following two figures show a hierarchical and flat team designs next to their corresponding downloaded Capture design.



Hierarchical Capture Design Example

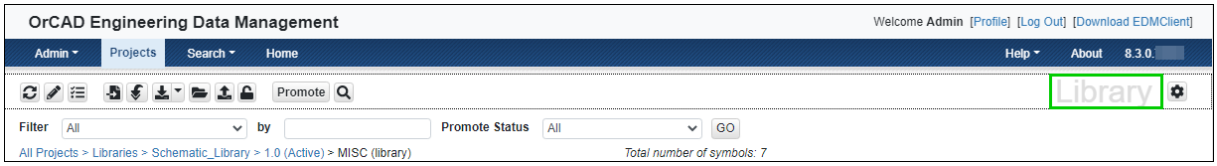


Flat Capture Design Example

4.3.6 Team Library View

Team Library folders can be created in Schematic Symbol, General, or Capture projects. A Team Library folder view shows information associated to individual symbols for an OLB library file.

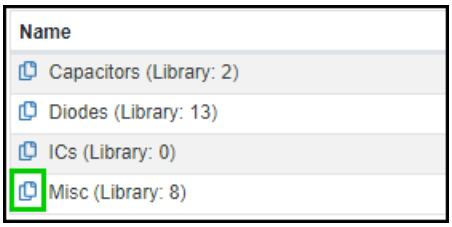
A "Library" watermark is displayed on the upper right hand corner of the Symbol Library Detail View.



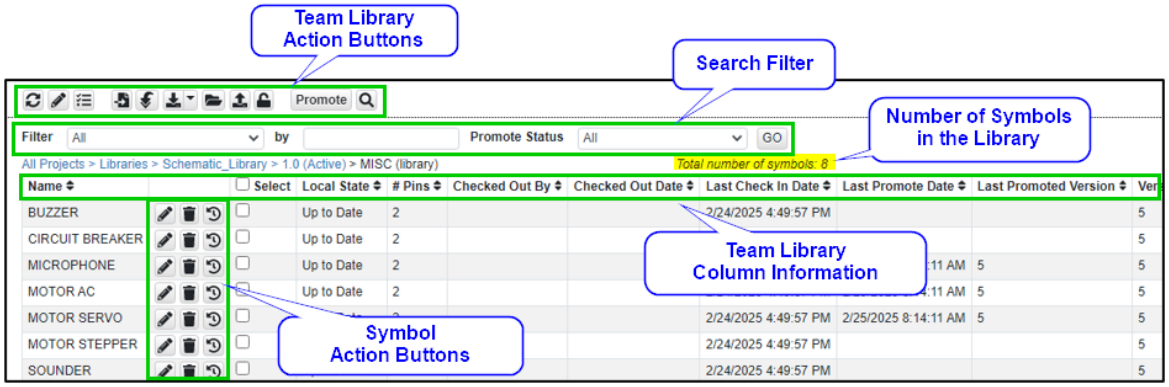
Team Library Detail View

The team library folder enables a team of users to work concurrently on symbols of a single OrCAD Capture library file. Users assigned as the project leader for a project with the Team Library will have see more buttons as enabled to manage the team library. Assigned general users can check in and check out symbols from the library.

The Team folders are identified with the Team icon, in front of the folder name in the Project Details View for a Capture, General Project or Schematic Symbol Project. You can open the Team Library Folder Details View by selecting the Team folder name or icon.



The general organization of the Team Library Folder view are shown in the Team Library Overview figure. Team Library Action buttons are located at the top of the Team Design folder view. Symbol Action buttons for page files are located inline with each file or folder. The information in the Team Design columns are defined in the Team Library Column Information below.



Team Library Overview

Users may change the order or visibility of a column to customize their view.

The Search Filter allows you to display a subset of the files in the project by filtering on a data field and/or Promote Status. Click the **GO** button to apply your filter.



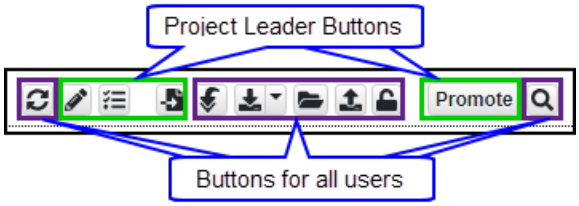
Team Library Search Filter

- **Field Filter** - select a field (e.g. Status) from the **Filter** list, then enter search parameter (e.g. 14) in the **by** text box.
- **Promote Status** - select a promote status (e.g. Not Promoted).
  - ***Note:** The Promote Status is determined by comparing the database Version of the latest copy of the file in EDM against the database Version of the file at the time it was last promoted. However, it **does not** physically open the promoted files or check for their existence in the configured promote path, but the promoted location should not be modified outside of EDM. The safest way to ensure this is to set the Library Promote Method to **EDM Web Application**.

The list of promoted symbols may be viewed by selecting the **View Promoted Library** button. Users may change the order or visibility of a column to customize their view for Team Design. Additionally, users may use the filter to search for symbols.

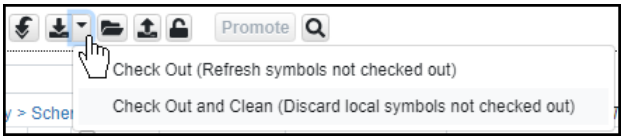
Team Library Action buttons

The Team Library action buttons appear in the upper left corner. The actions enable you to do the following:



The bulleted list that follows describes each action button.

- **Refresh** button - refreshes the Team Library Details View.
- **Edit** button - allows the assigned leader to edit user assignments across all symbols.
- **Mass Edit** button - This button allows you to delete a group of files or change properties for a group of selected files. This button is enabled for assigned leaders.
- **Import** button - allows the assigned leader to import symbols from a library file into the team library folder.
- **Full Download** button - downloads a copy of all symbols in the Team library folder. All symbols are merged to a single library file.
- **Check Out** button - check out symbols selected in the team library. All symbols are merged to a single library file. You have the option retain all symbols (Check Out) or retain only symbols checked out in your merged library file (Check out and Clean).



- **Open** button - open the merged library file in Capture if you have configured .OLB to be opened with Capture.

- **Check In** button - check in symbols to update the vault with your local modifications.
- **Cancel Check Out** button - allows assigned leaders to remove all selected symbols from checked out state. This button is enabled for assigned leaders and users who have the symbols checked out.
- **Promote** Promote button - copies the symbol from the vault to promoted library folder locations that are specified in Capture Configuration.
- **View Promoted Library** button - displays the list of symbols that have been promoted to the promoted library locations for team folders.

Symbol Action Button

The symbol action buttons perform actions on symbols that are inline with the symbol. Only the history button is available to all users. Assigned leaders have two additional buttons to Edit and Delete.



The bulleted list that follows describes each action button.

- **Edit** button - allows an assigned leader to edit the symbol properties for team libraries.
- **Delete** button - allows an assigned leader to delete the symbol.
- **History** button - allows users to view the history of changes on a symbol.

Team Library Column Information

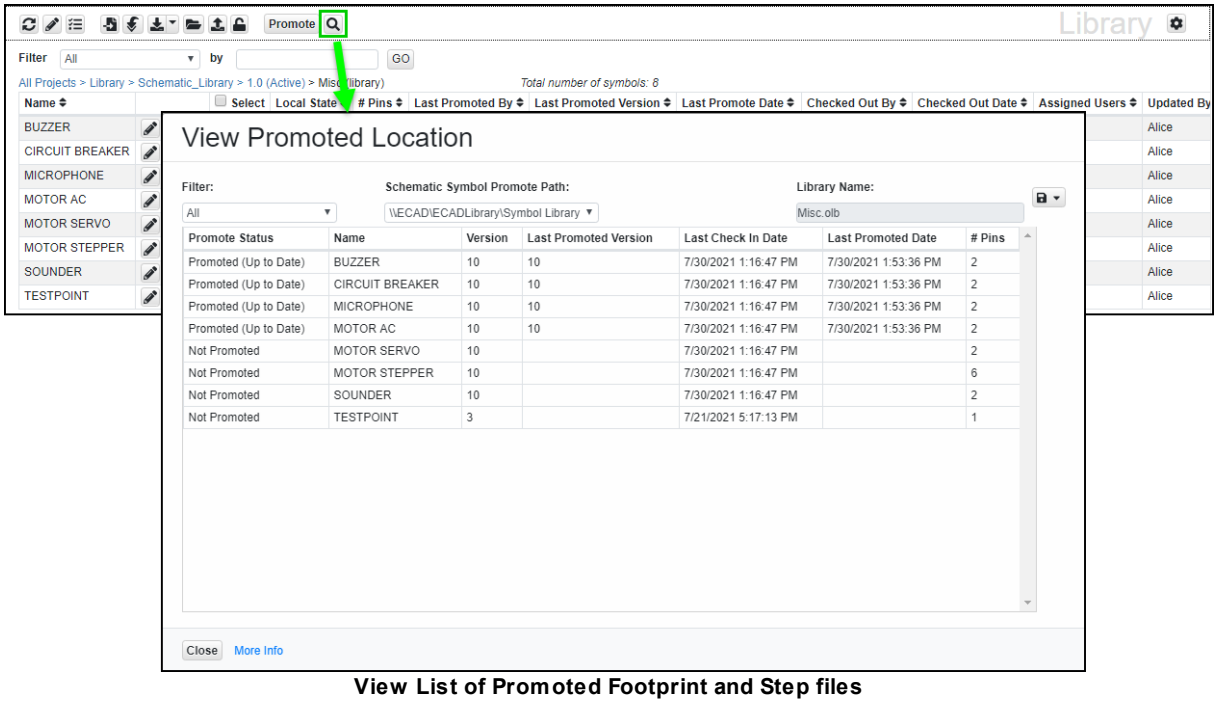
The standard columns of information in Team Library view contains the following:

- **Name** - name of symbol in the team library.
- **Select Checkboxes** - allows a user to select symbols to apply an action in the Team Library folder. Select a checkbox and then click a Team Library Action Button. *Note: Not all action buttons are applicable.
- **Local State** - shows the state of the Symbol. Unlike files, EDM is unaware of changes made to symbols in your local library file. There are only 3 valid local states in Team Libraries. Symbols not yet checked are unavailable in the Team Library folder view and will not show as "Local only". Detailed information can be found in Local state Definitions. The Local State may have one of the following values:
  - **Up to Date** - the revision number of the checked out copy matches the revision in the vault, regardless whether you have made any changes to the checked out symbol.
  - **Out of date** - the revision number of the checked out copy is older than the revision in the vault.
  - **No Local Copy** - item is in the vault but has not been downloaded or checked out.
- **Checked Out By** - user who has the symbol checked out.

- **Checked Out Date**- date and time stamp when the user checked out the symbol.
- **Assigned Users** - user who has permission to work on the symbol.
- **Updated By** - user who last updated the symbol.
- **Status** - status for the symbol within the team library folder. This field is typically updated by the user when a symbol is checked in. It may also be modified by an assigned leader when editing the symbol properties.
- **User Fields** - additional fields configured from user fields. User field columns are based on the Project Administrators setup in EDM Configuration for Customizable Fields for the Team Library. These fields may be modified by an assigned leader when editing the symbol properties.
- **Last Promote Date** - date that a symbol was most recently promoted.
- **Last Promoted Version** - version of the symbol that was most recently promoted.
- **Last Check In Date** - date when symbol was last modified and checked in
- **Pin Count** - number of pins that is associated with the symbol
- **Version** - History version number associated with the Last Check In Date.
- **Last Promoted By** - identifies the user who most recently promoted the symbol.

View Promoted Schematic Library Location

The **View Promoted Library** button in the Team Library Action Buttons opens a pop up showing a list of promoted symbols.



View List of Promoted Footprint and Step files

The Promoted Library Location View provides the full list of all symbols checked in and promoted to various locations. There is filter selection to narrow the list of files shown and the option to view each of the configured location of the promoted location.

The filter options show the symbols by the following:

- **All** - Shows all the symbols in the library, regardless of promote status.

- **Promoted (Outdated)** - Promoted version is older than current version in project.
- **Promoted (Up to Date)** - Promoted versions match the current version in project.
- **Promoted (Missing)** - Promoted, but now the file is missing from promoted location.
- **Not Promoted** - Exists in project, but has not yet been promoted.
- **Not Found in EDM** - Exists in promoted location, but not in project.
- **EDM Legacy** - Provides the same format information that matches the Promoted flyout from versions prior to 6.3.0.

4.4 Local State Definitions

The **Local State** column within the Project Details and Team Library/Design views shows the state of a file/item in EDM as compared to the corresponding file in your local workspace. The local state helps you determine actions you need to take when you need to make changes to a file or discard local changes when another user has check in changes.

Local State can have the following values:

Local State	Summary	Applies to:		
		Files	Team Design Pages	Team Library Symbols
Local Only / New	Item only exists in your local workspace, not vault. Files show Local Only, while Team Design Pages and Team Library Symbols show New.	x	Only shown for Check In	Only shown for Check In
Up to Date	Item in your local workspace is at the same version as the corresponding file in the vault.	x	x	x
Up to Date (Mod)	Up to Date item was locally modified.	x		
Out of Date	Item in your local workspace is of an earlier version than what is currently in the vault.	x	x	x
Out of Date (Mod)	Out of Date item was locally modified.	x		
No Local Copy	Item in the vault does not exist in your local workspace.	x	x	x
Not in Database	Item is not in your database, likely as a result of a check in or import issue.	x	x	x
Not in Repository	Item is not in your vault, likely as a result of a check in or import issue.	x	x	x
Local Case Mismatch	Local file name has a different capitalization than file name in the SVN vault repository.	x		
Multiple Files in Repository	Two or more files in the SVN repository with the same file name, but different capitalization.	x		

Unknown Unknown (Mod)	Unexpected state that may appear with an SVN vault, sometimes due to installing a trial version of VisualSVN Server that has since expired.	x	x	x

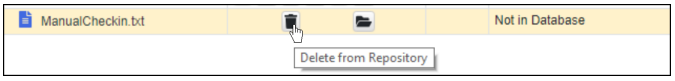
- **Local Only / New** - This state applies to files in a General, Capture, and Footprint project detail view. It does not apply to pages or symbols.
  - For files, when you create a new file locally and have not checked the file into the vault yet, you will see the Local State of **Local only**. You may check in any new files as long as you have been assigned to the project.
  - For Team Design or Library Symbol views, this state does not exist. Instead, new pages in a Team Design and new symbols in a Team Library will appear with a **New** state in the Check In Form.
- **Up to Date** - This state applies to files, pages, and symbols in all project detail views.
  - For files, if the copy of a file in your local workspace is at the same version as the corresponding file in the vault, the Local State will show that the file is **Up to Date**. Changes to the local file will change the state from **Up to Date** to **Up to Date (Mod)**. Files that have a local state of **Up to Date** and checked out by you will not show in the list of files in the Check In form since there are no updates.
  - For Team Design or Library Symbol views, the local state of a page or symbol shows as **Up to Date** if the item in your local workspace is at the same version as the vault regardless of change status. The Check In form for Team Designs and Symbols will show all pages as **New** or **Checked Out** by the user.
- **Up to Date (Mod)** - This state applies to files in a General, Capture, and Footprint project detail view. It does not apply to pages or symbols.
  - For files, when you modify your local copy of an **Up to Date** file, you will see the Local State change from **Up to Date** to **Up to Date (Mod)**. If you have checked out the file, you will be able to check the file into the vault. However, if you haven't checked out the file, you will not be able to check the file in.
  - For Team Design or Library Symbol views, this state does not exist because the change status of a page or symbol is not available. All **Up to Date** pages that have been checked out can be checked into the vault regardless whether they have been changed. Checked out pages in a Team Design or symbols in a Library will appear with a **Checked Out** state in the Check In Form.
- **Out of Date** - This state applies to files, pages, and symbols in all project detail views.
  - For files, if the copy of a file in your local workspace is of an earlier version than what is currently in the vault, the Local State will show the file as being **Out of Date**. To bring your local copy Up to Date, you can click **Full Project Download** for the out of date file. If you need to work on a file, you should check out the latest version first. Checking out an **Out of Date** file from a Capture or General project will automatically bring the latest version of the file to your local workspace. For Footprint projects, you will need to use the check out with the download option to automatically update your local workspace.

  - For Team Design or Library Symbol views, if the copy of a page or symbol in your local workspace is of an earlier version than what is currently in the vault, the Local State will show the file as being **Out of Date.** When you check out an **Out of Date** page or symbol, EDM will automatically bring the latest version of the page or symbol to your local workspace.
- **Out of Date (Mod)** - This state applies to files in a General, Capture, and Footprint project detail view. It does not apply to pages or symbols.
  - For files, when you modify a file that you have not checked out and someone else checks in a newer version, you will see an **Out of date (Mod)** state. When this happens, you should delete your out of date modified file and get the latest copy. This could also happen if an assigned leader cancels your check out and allows someone else to check in a newer version.
  - For Team Design or Library Symbol views, this state does not exist because the change status of a page or symbol is not available.
- **No Local Copy** - This state applies to files, pages, and symbols in all project detail views.
  - When the vault contains a file, page, or symbol that you haven't downloaded to your local workspace, the Local State will show them as **No Local Copy**. This state is typically seen in projects where you haven't performed a full download or on files where another user has checked in a new file, page, or symbol. A **Full Download** should change this state to **Up to Date**.
  - If you see a file with **Not in Database** state along with another file with **No Local Copy** state, most likely a file has been renamed. You should do a full download before taking any further action. The full download will update your local workspace to align the rename.

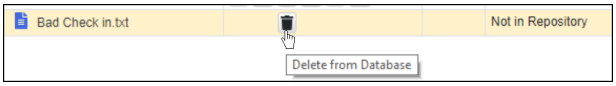
Additional Local States for File with Issues

The following local states may appear as a result of unexpected errors during import or check in operations. Such errors can be caused by network interruptions (to internet/intranet, VPN, EDM web app, etc.), Capture or PCB Editor crashes, or users making direct changes to the EDM vault or database via 3rd party software or command-line statements, bypassing the EDM user interface.

- **Not in Database** - When you see this state, you should immediately perform a **Full Download** to make sure your local workspace is updated to the latest revision in the vault. If you still see the **Not in Database** state, you should remove the file from EDM using the **Delete** button.



- **Not In Repository** - This state can result when a check in or import operation is interrupted by a user manually closing Capture or PCB Editor, or if Capture or PCB Editor crashes. The database may have been updated, but vault update did not complete. When you see this state, you should immediately perform a **Full Download** to make sure your local workspace is updated to the latest revision in the vault. If you still see the **Not In Repository** state, you should remove the file from EDM using the Delete button.

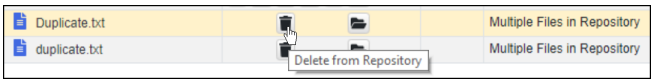


- **Local Case Mismatch** - This state shows when your file name has a different capitalization than file name in the SVN vault repository. (e.g. MyFile.txt vs. myfile.txt). This issue does not apply to

File System (FS) vault. To fix this, you can either fix the case of the file in your local workspace using the Fix Local Case button, or delete your local file and re-download the project.

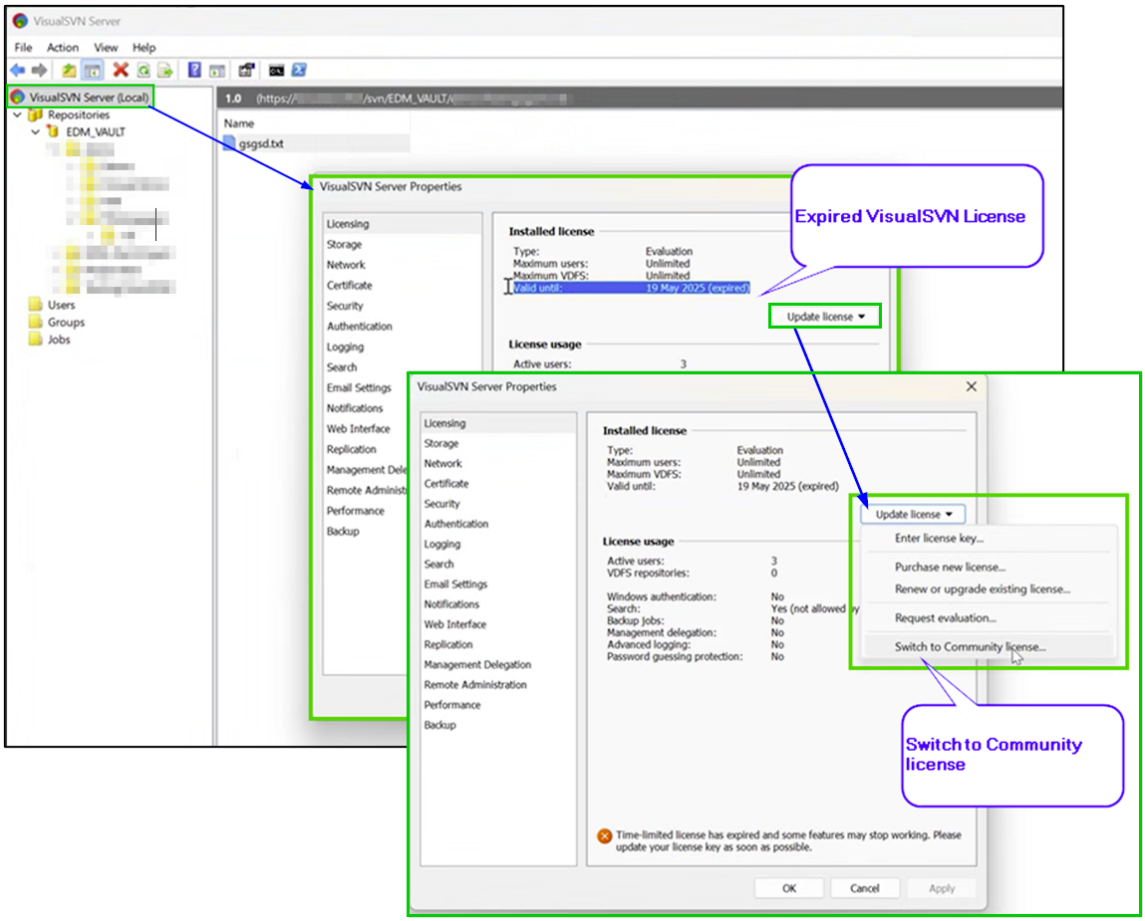


- **Multiple Files in Repository** - This state shows when there are two or more files in the SVN repository with the same filename but have different capitalization. This may occur when multiple people check in the same file with different cases. This issue does not apply to File System Vault. To fix this, you will need to remove the duplicate files until only one remains. You may encounter a **Local Case Mismatch** after that, which can be fixed with he Fix Local Case button as noted above.



- **Unknown or Unknown (Mod)** - This is an unexpected state that may appear with an SVN vault, sometimes due to installing a trial version of VisualSVN Server that has since expired. If you are no longer licensed to use a paid edition of VisualSVN Server, but you haven't switched your licensing back to a free (e.g. community) edition, all EDM check in or import operations will fail for all EDM users across all projects. These failures cause Unknown or Unknown (Mod) local states to appear.

  When this is the issue, the solution is to either renew or load a valid SVN license, or switch the licensing model to a free (e.g. community) edition. Open VisualSVN Server, right-click on the VisualSVN Server and go to **Properties.** Select **Update license**, then use the appropriate/desired option to renew or load a valid SVN license, or **Switch to Community license...** You may need to Restart the SVN Server for the changes to take effect.


VisualSVN Server
File Action View Help
VisualSVN Server (Local)
Repositories
EDM_VAULT
Users
Groups
Jobs
Name
qsgsd.txt
VisualSVN Server Properties
Licensing
Storage
Network
Certificate
Security
Authentication
Logging
Search
Email Settings
Notifications
Web Interface
Replication
Performance
Backup
Installed license
Type: Evaluation
Maximum users: Unlimited
Maximum VDFS: Unlimited
Valid until: 19 May 2025 (expired)
Update license
License usage
Active users: 3
Expired VisualSVN License
Enter license key...
Purchase new license...
Renew or upgrade existing license...
Request evaluation...
Switch to Community license...
Switch to Community license
Time-limited license has expired and some features may stop working. Please update your license key as soon as possible.
OK
Cancel
Apply

5 Manage Projects (Project Leader Tasks)

All tasks for managing and working with projects are performed from the Project Dashboard or one of the Project Detailed Views. These project tasks include: Project Set Up, Project Implementation, Freeze Team Design (optional for Team Designs), Layout, and Post Layout as outlined below.

1. At project initiation, users whose role has project leader permissions set up the project by doing the following:
   a. Create the EDM project.
   b. Create project structure by adding folders.
   c. For folders designated for Team Designs, add page for flat design, add schematics to hierarchical designs and/or import pages as the starting point for users.
   d. For folders designated for team libraries, import symbols as the starting point.
   e. Assign users to project, schematics and pages.
   f. Activate the project.
2. During implementation, assigned leaders will do the following:
   a. Track project by reviewing design file, page and symbol statuses.
   b. Make/modify assignment of users to design files, pages and symbols.
   c. For Team Design folders, integrate pages, manage reference designators and occurrence properties and check design progress throughout the project design phase.
3. For Team Designs at design freeze, assigned leaders will do the following:
   a. Run final cleanup to reference designators and occurrence properties.
   b. Perform final Netlist, DRC and other checks.
   c. Manage CIS Variants.
   d. Update entire design.
   e. Freeze design for PCB Layout.
4. During layout, assigned leaders will do the following:
   a. Track progress by reviewing board and other files.
   b. Make/modify assignment of users to files.
5. Post layout, assigned leaders will do the following:
   a. Perform back-annotation changes from layout.

The Team Design Library and Schematic Symbol Library differ from Team Designs. While assigned leaders may freeze a Team Design, libraries are not frozen as they are required for all design projects. As new symbols are added to the vault, they can be reviewed and then promoted once approved.

5.1 Project Structure and Options

EDM enables the management of designs that require multiple engineers to work in a team environment. All team members working on an EDM project should always open EDM or enable RefDes when working on any portion of the design. Daily check in of all files is encouraged to reduce risk of data loss and promote information sharing across team members.

The list below describes the flow of a typical EDM team design. Multiple users check out schematic pages daily as pages are being developed. The schematic pages can be checked out and checked in daily for synchronization with other users.

1. The first step is to create the project. assigned leaders will add folders to the project. The folders created dictate the type of project. Projects for a Team Design environment have options for flat, hierarchical, or library folders. Once the folder is created, the assigned leader will add Groups with pages to a flat design and may import pages from an existing design or for hierarchical designs, add pages or schematics. Projects Leaders will assign pages to users from the EDM user list. The assigned leader needs to Activate the project before the users begin to work on the projects.

2. Users who see pages assigned to them can check out their assigned pages. For pages that are not assigned to the user, they can check out the pages as Read Only. As everyone works on the project, check out of a complete set may help with synchronization of the project.

3. Users will work on their assigned pages during the day. When a user is at a good stopping point at the end of the day, the user should check in and cancel check out of all pages. Early sharing of work will help with on-going communications.

4. Assigned Leaders can bring the design together and annotate Ref Des at the project level. This can be done as many times as needed. Locking down reference designators for parts in a design that are firm will enable early layout for those parts of the design. The entire project should be checked out.

5. Assigned Leaders will check in the entire project if changes have been made.

6. When the design is at a point where the PCB layout can start, the PCB designer can check out the entire design as Read Only and start a preliminary placement while the engineers continue working on the design.

7. When the Engineers are done, the Assigned Leader will perform final Ref Des annotations, Netlist, CIS Variant management, DRC check and the reviews required by company procedures. Once this is done, the Assigned Leader will check in the entire design and freeze the project.

8. The full layout can be performed at this point on the frozen design. The PCB designer will check out the frozen design.

9. When layout is complete, the PCB designer will create a back annotation file, if changes were made that require back annotation.

10. The Assigned Leader can check out the frozen design, perform back annotation and check in the design to EDM.

The project will be complete at this time. If any changes need to be made to separate pages in the future, the frozen design can be copied to a new revision. The new revision will have all the pages separated and the entire process starts again.

For a Managed Capture Project, the Assigned Leader will create the Capture folder and select the files to be managed during check out and check in. File options are DSN, BOM, Netlist and PDF. The Assigned Leader will assign the files to users to work on. When the Capture design is Ready for Review, the Assigned Leader will Baseline the Project until it has been approved.

5.2 Add and Edit Projects

All projects are added from the Project Dashboard and only when using the Capture plug in or EDM Desktop applications. Users assigned roles with Leader permissions may add projects. Users whose role has granted project leader permissions are only able to edit a project when they are assigned to the project as a leader.

A project provides the structure in which folders and files can be managed and checked into the vault.

Projects can be created as one of the following types of projects:

- **Managed Capture** project
  - Standard folder, Team Design or Team Library folder may be added to the project
  - Check in and check out of any type of file.
  - The Capture project keeps track of the file relationships between the managed Capture design, the managed board, and associated files derived from the schematic design (BOM, Netlist, and PDF) and board (DB Doctor, EDM Validation, PDF).
  - The managed relationships allow files to be auto-generated when project files are checked in.
  - The Capture project provides package for release functionality.
- **General** project
  - Standard folder, Team Design or Team Library folder may be added to the project
  - Check in and check out of any type of file.
  - No managed design or board functionality provided.
- **Footprint** project
  - Sub-folders are added as part of project creation.
  - The Footprint project allows only import, check in and check out of footprints, shape, flash, templates, padstacks and step models.

  - Relationships of various footprint file types are extracted and recorded with Footprint project.
  - Footprints and step models may be promoted to a central library location.
- **Schematic Symbol** project
  - The Schematic Symbol project allows for libraries to be added as separate Team Library folders.
  - OrCAD Capture symbol libraries may be imported into library folders.
  - Symbols may be checked in and checked out individually or in bulk.
  - Symbols may be promoted to a central library location.

The EDM Vault must be configured before a project can be created. It is also highly recommended that the promoted library locations for schematic symbols and footprints are configured before project creation.

5.2.1 Add Project

During project creation, you will be required to specify the project name and type of project. To add a new project, click the **Add** button in the Project Dashboard.



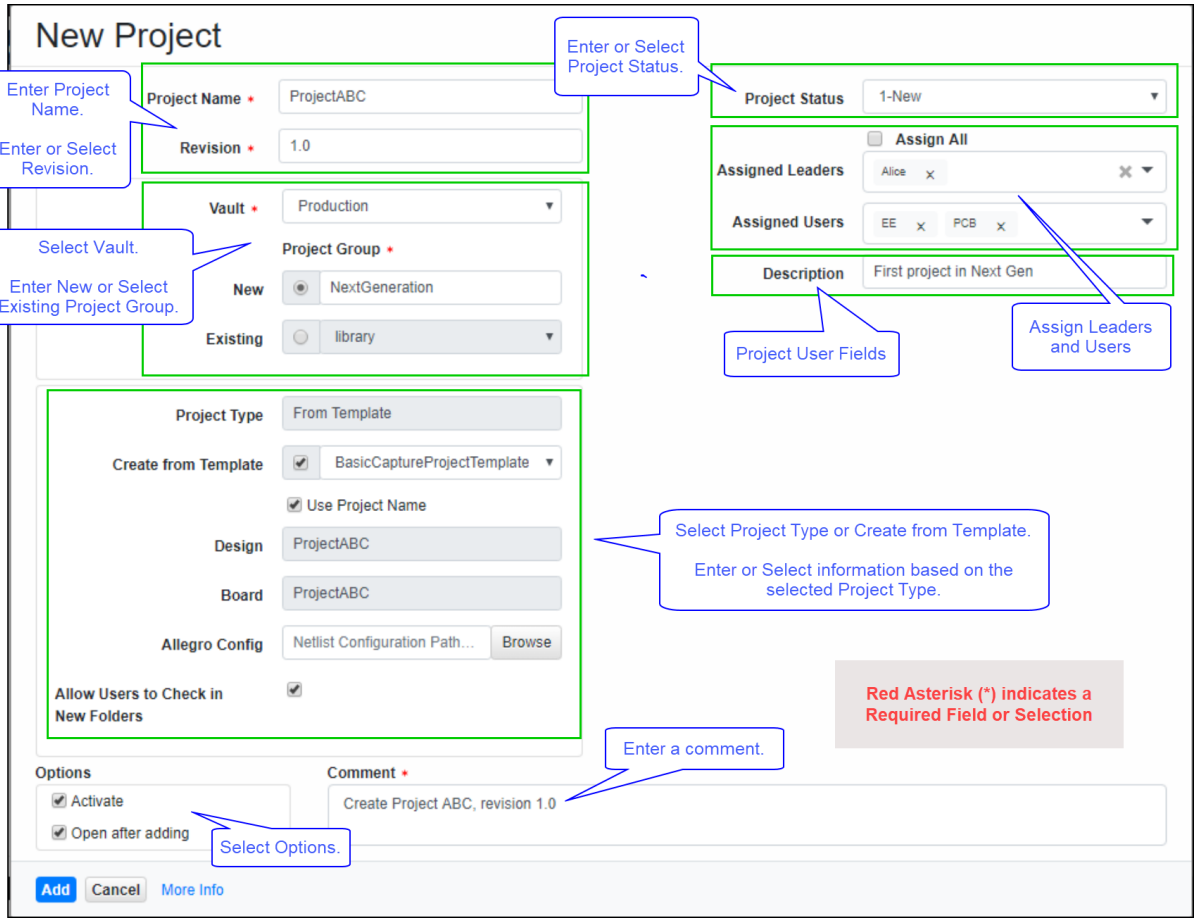
Add Project

A **New Project** Form appear where you may enter the following information.

- Project Name and Revision - Required information identifying project by name and revision
- Vault and Group for the new project - Required information in which to create the new project
- Project Types - Requirement information for the type of project to be created
  - General
  - Capture
  - Footprint
  - Schematic Symbol
- Project Status, user assignments and configured user fields - Optional status and user information that may be configured during project creation. Configured user fields may also be entered during project creation.
- Comment and Save Options

New Project Form

The **New Project** form opens as shown in the following sample of the **New Project** Form.



Sample New Project Form

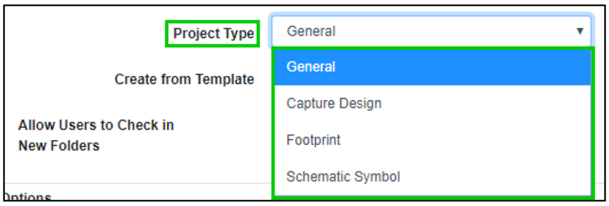
Project Name, Revision, Vault and Group

You will be required to enter the project group, name and revision for the new project. The names entered will be used as part of thre project structure and must conforms to Windows folder naming convention.

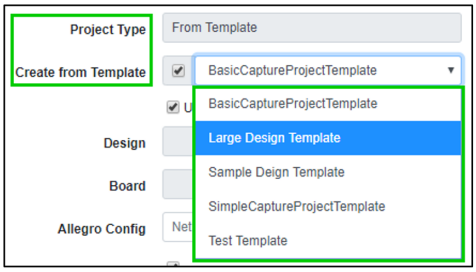
- **Project Name** - The project name must be a unique within a project group. Footprint and schematic library Project names must be unique across all other library projects within EDM. Project names may be changed after creation as long as the name is unique within a project group. Projects may be deleted and added with a new or the same name.
- **Revision** - The revision must be unique for the project. If specified as a pick list in Configuration, then select a value from the pick list. When copying a project revision, new revisions must be unique.
- **Vault** - The drop list for vault shows only the list of Vaults that has been assigned to you. If you do not see any vaults in the drop list, the vault may not configured or you may not be assigned to any vaults.
- **Project Group** - The project group allows you to group and view projects in a hierarchy. You may create separate groups for each project or assign projects to an existing group. When creating a new group, the group name must be unique.

Project Type (Capture, General, Footprint, or Schematic Symbol project)

You have the option to create one of four project types or create a project using an existing Project Template. You may select the Project Type drop list to select the type of project for creation. The Project Type section of the New Project Form updates based on the project type selection. To create a project from an existing template, select the **Create from Template** checkbox then select the template you wish to start with. When a project is added using a template, the folder structure using the define folder names in the template will be created automatically.

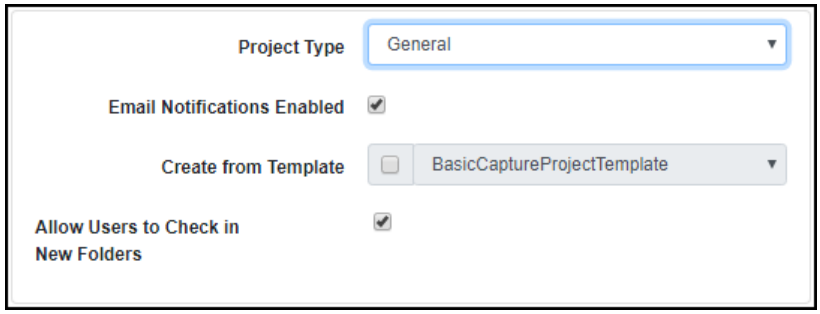


Choose Project Type



Choose Create from Project Template

1. **General Project** - General projects provide basic check in and check out functionality of all files. Folders and sub-folders may be added to general project after creation. Team Design and Team Library folders may also be added after creation. The general project has no additional built in functionality as with other project types.

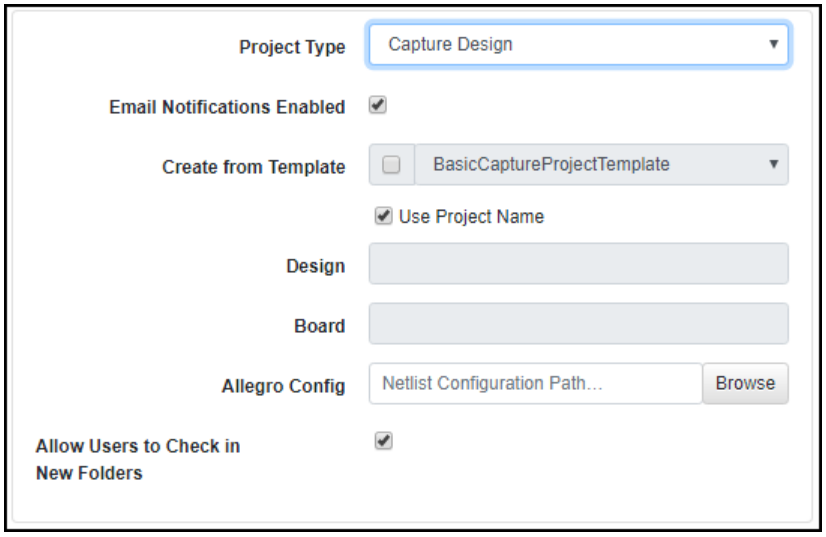


Options with General Project Type

Options when adding the General Project include:

- **Email Notifications Enabled** - This option allows you to enable/disable email notifications on a per-project basis. It will only be shown if you have Email Notifications enabled in the Admin Configuration.
- **Allow Users to Check in New Folders** - You may enforce the project structure or allow users to change the structure by checking in new folders. To enforce the structure, deselect the checkbox. When this option is not enabled, assigned leaders will need to create folders first before users may check in files to the folder.

2. **Capture Design Project** - Capture design projects enables management of Capture design files, PCB Editor board files and related files in addition to check in and check out functionality. After project creation, folder and sub-folder may be added with designations for managed Capture design and board file as well as auto-generated files.

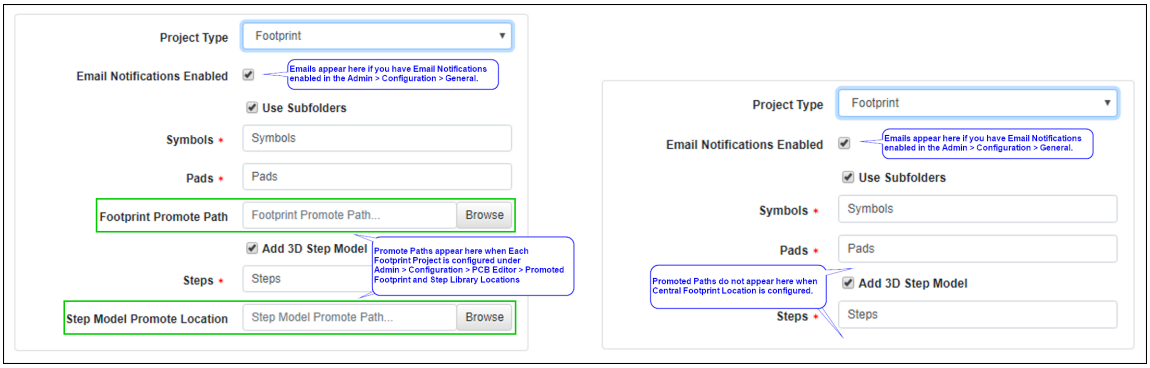


Options with Capture Project Type

Options when adding the Capture Design Project include:

- **Email Notifications Enabled** - This option allows you to enable/disable email notifications on a per-project basis. It will only be shown if you have Email Notifications enabled in the Admin Configuration.
- **Use Project Name** - This option enables you to change name your design file and board file name to be different from the the project name. This option is enabled by default. To enter different file names, deselect this checkbox.
- **Design** - This design file name field is not editable when the Use Project Name checkbox is selected. The design name is a required field when the "Use Project Name" checkbox is deselected. The default design file name is the project name. Modify the design name ensuring that it meets Capture standard design name.
- **Board** - This board file name field is not editable when the Use Project Name checkbox is selected. The board name is a required field when the "Use Project Name" checkbox is deselected. The default board file name is the project name. Modify the board file name ensuring that it meets PCB Editor standard for naming board files. Characters that may be used for PCB board files include a-z, 1-9 and _. Blank spaces are not permitted.
- **Allegro Config** - This option enables you to change the netlist configuration file that you want to use for the project. Leave this unchanged to use the default setting configured in Netlist Generation Configuration.
- **Allow Users to Check in New Folders** - You may enforce the project structure or allow users to change the structure by checking in new folders. To enforce the structure, deselect the checkbox. When this option is not enabled, assigned leaders will need to create folders first before users may check in files to the folder.

3. **Footprint Project** - The footprint project enables management of footprint and step files. Relationships are retrieved from files when imported or checked in and then provided in various views for easy management.



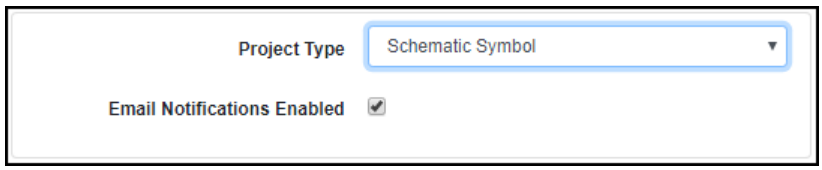
Options with Footprint Project Type

Options when adding the Footprint Project include:

- **Email Notifications Enabled** - This option allows you to enable/disable email notifications on a per-project basis. It will only be shown if you have Email Notifications enabled in the Admin Configuration.
- **Use Subfolders** - This option enables the storage of symbol and pad files in separate folder. This option is enabled by default. Deselect this checkbox to store all files in a single folder at the project level. This configuration option may not be changed after a project created.
- **Symbols** - This symbols folder name field is not editable when the Use Subfolders checkbox is deselected. The Symbols folder name is a required field when the "Use Subfolder" checkbox is selected. The default folder name is Symbols. Modify the symbols folder name ensuring that it conforms to Windows folder naming convention.
- **Pads** - This pads folder name field is not editable when the Use Subfolders checkbox is deselected. The Pads folder name is a required field when the "Use Subfolder" checkbox is selected. The default folder name is Pads. Modify the pads folder name ensuring that it conforms to Windows folder naming convention.
- **Footprint Promote Path** - This selection is available when the setting in Promote Footprint and Step Model Locations is configured for **Each Footprint Project**. UNC path is recommended for this setting. You will need to ensure that users defined into the Promote Method have access to the selected location. An error will appear if the define user for Promote Method does not have appropriate access to the location. Folders for Symbols and Pads in the promote location will match the configuration in the **Use Subfolders** settings within the New Project form. The **Promote Path/Location** for symbols and pads corresponds to paths set in PCB Editor for psmpath and padpath respectively.
- **Add 3D Step Model** - This option enables the addition of step files to be included in the project separate folder. This option is disabled by default. Select this checkbox to manage Step Models in your new Footprint project. The 3D step model option may be added after creation of the footprint project using Edit Project form. Once updated with this option, you may not remove this option.
- **Steps** - This pads folder name field is not editable when the **Add 3D Step Model** checkbox is deselected. The Steps folder name is a required field when the **Add 3D Step Model** checkbox is selected. The default folder name is Steps. Modify the steps folder name ensuring that it conforms to Windows folder naming convention.

- **Step Model Promote Location** - This selection is available when the **Add 3D Step Model** checkbox is selected and the setting in Promote Footprint and Step Model Locations is configured for each project. UNC path is recommended for this setting. You will need to ensure that users defined into the Promote Method have access to the selected location. An error will appear if the define user for Promote Method does not have appropriate access to the location. The folder for step models in the promote location will match the configuration in **Steps** settings within the New Project form.

4. **Schematic Symbol Project** - The schematic symbol project enables management of schematic symbols. After creation, each library may be added as a separate Team Library folder. Symbols may be imported or checked in and individually managed. No optional configurations are provided with this project type.



Options when adding the Footprint Project include:

- **Email Notifications Enabled** - This option allows you to enable/disable email notifications on a per-project basis. It will only be shown if you have Email Notifications enabled in the Admin Configuration.

Project Status and user assignments

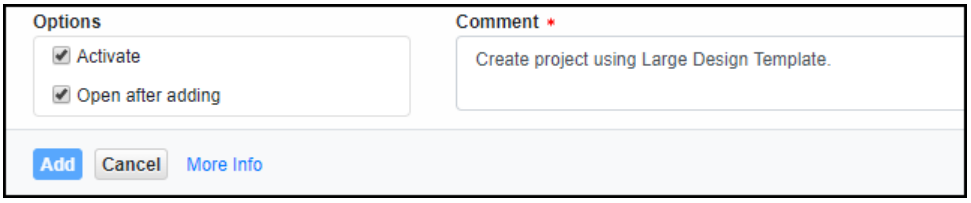
The project status, user assignments and configured user fields are optional information you may entered as you create the project. You may edit the project optional information using the Edit Project form.

Optional information includes the following:

- **Project Status** - status for the project to be created. If a default value was configured, it will be pre-populated in this form. If this field was configured as a pick list, you may choose a pick list value. Each company has different guidelines for project statuses.
- **Assigned Leaders** - only users whose role has been granted project leader permissions may be added as an assigned leader. Only assigned leaders are able to manage the project.
- **Assigned Users** - any user or user group may be assigned as users. Project users or user group members whose role has check in and check out permission are only able to check out and check in files, pages and symbols that they have been assigned. Users not assigned to the project cannot be added to assignment of items within the project. Configured project users will automatically be added as default users when new folders are added after project creation. Project users added using the Edit Project form will need to be applied to any existing project folders manually.
- **Additional configured User field** - fields shown in the New Project form is based on your specific configuration.

Comment and Save Options

Before the final action to add the project, you are required to enter a comment. The comment is a note reflecting your action. Comments are saved in the project history.



New Project Save Options

You have the following options with the **Add** button:

- **Activate** - activate the project during the creation of the project. Activated project are enabled for check in and check out of files.
- **Open after adding** - Open directly to the Project Details page after the project is created.
- **Comment** - Enter a comment associated with your action. The **Comment** field is required. Note contents are included in the History of the project.

After you enter all information for your project into the form, click the Add button to create the new project. Or to Cancel the New Project form click the Cancel button. New projects are included in the list of projects in the Project Dashboard.

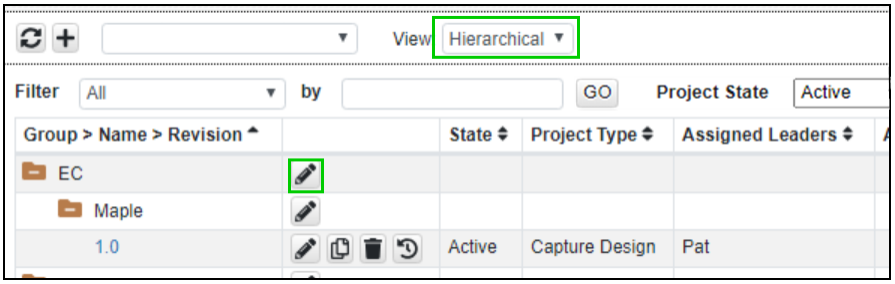
If you created a managed Capture Design and the managed design was not imported or added during project creation, you can add the managed project .opj file (Local version only) and blank .DSN file that controls the managed project structure by clicking the **Full Project Download** button. The initial download of an empty Capture project will auto-create the managed design and Capture project files.

5.2.2 Edit Project and Group

Project and Group names may be renamed. The Project or Group name can be modified from the Project Dashboard. You can edit properties for a Project Revision from the Project Dashboard view or the Project Details View.

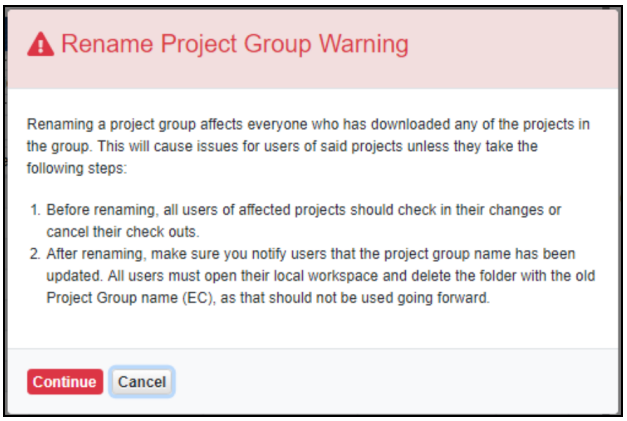
Rename Project Group

1. The **Project Dashboard** view must be configured for Hierarchy to rename the Project Group. Select the **Edit** button next to the project group name to open the Rename Group form. Keep in mind that changing a Group Name will require users to remove old downloads and re-download from projects with the new Group name.

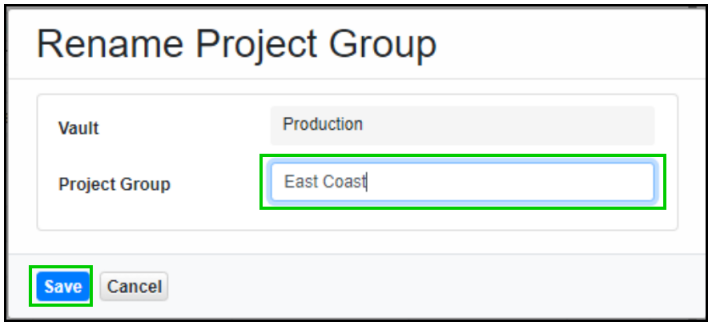


Select Edit to Rename Group

2. You will be notified if any files are checked out and will be unable to continue until updates are checked in or you cancel all check outs. A warning displays with suggestions on how to minimize impact on users assigned to projects within the group. Click **Continue** to proceed with renaming the group name.

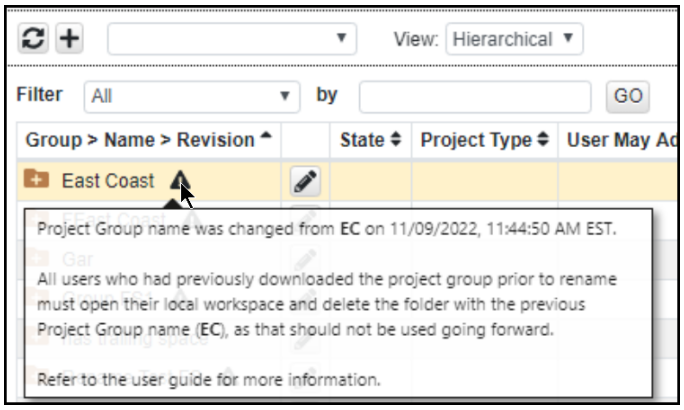


3. At the **Rename Project Group** form, type a new group name and click **Save**.



Rename Project Group

4. Click **Rename** at the confirmation pop up to finalize the rename.
5. A caution symbol shows on any group name that have been renamed. The caution remains for 30 days by default. The duration of the warning and can be changed from **Admin** > **General** > **Configuration**.

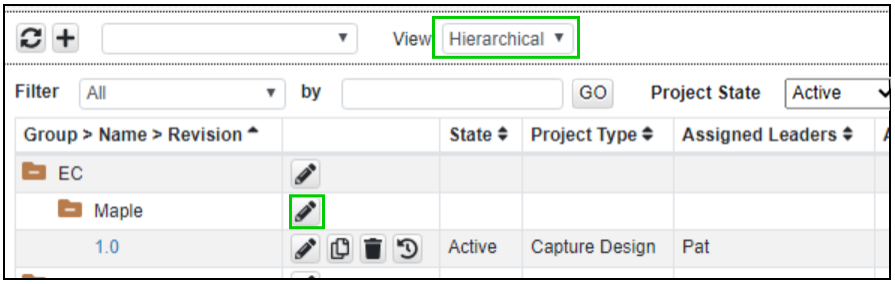


Caution for Renamed Project Group

Edit Project Name and Project Attributes for All Revisions

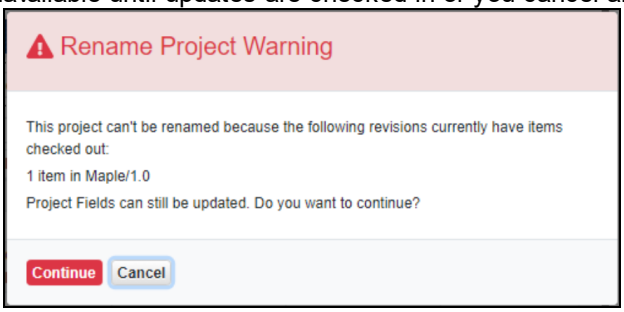
1. The **Project Dashboard** view must be configured for Hierarchy to rename the Project. Select the **Edit** button next to the project name to open the Edit Project form. Keep in mind that changing

a Project Name will require users to remove old downloads and re-download from any project revisions with the new Project name.

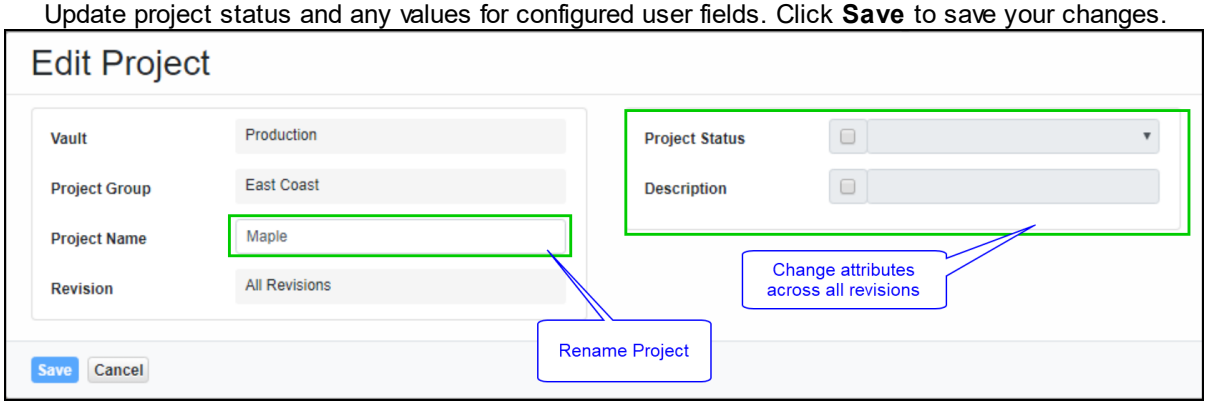


Select Edit to Rename Project

2. You will be notified if any files are checked out. You can continue to change other properties but Project Rename is unavailable until updates are checked in or you cancel all check outs.



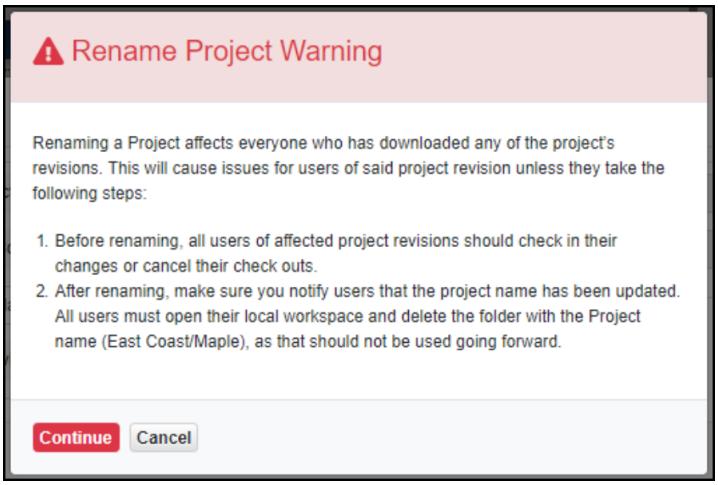
3. The **Edit Project** form displays. Enter a new name for your project if no files are checked out. Update project status and any values for configured user fields. Click **Save** to save your changes.



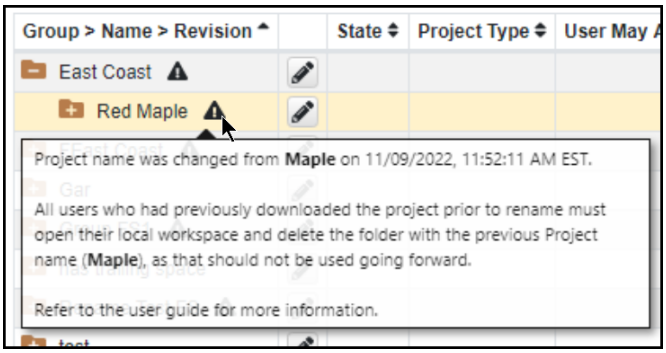
Edit Project Form for All Revisions

4. If you are only changing attribute values, the changes take effect immediately after clicking the **Save** button.

5. If renaming the project, a warning displays with suggestions on how to minimize impact on users assigned to project revisions within the project. Click **Continue** to finalize with renaming the project name.

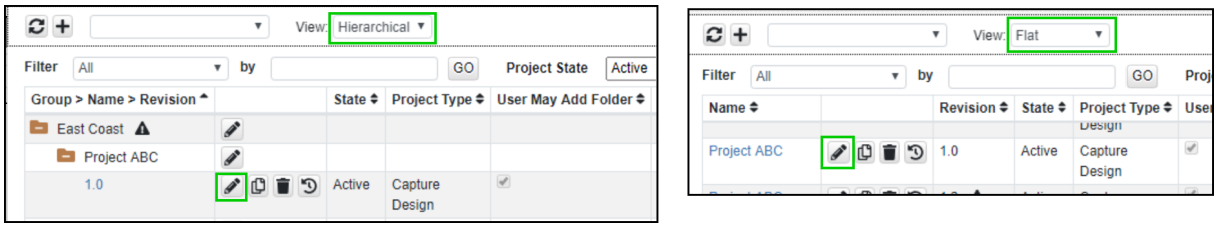


6. A caution symbol shows on any project name that have been renamed. The caution remains for 30 days by default. The duration of the warning and can be changed from **Admin** > **General** > **Configuration**.



Edit Project Revision Attributes

From the **Project Dashboard** view, select the **Edit** button next to the specific project revision to open the Edit Project Revision form.



Select Edit Project in Project Dashboard Shown for Hierarchical and Flat views

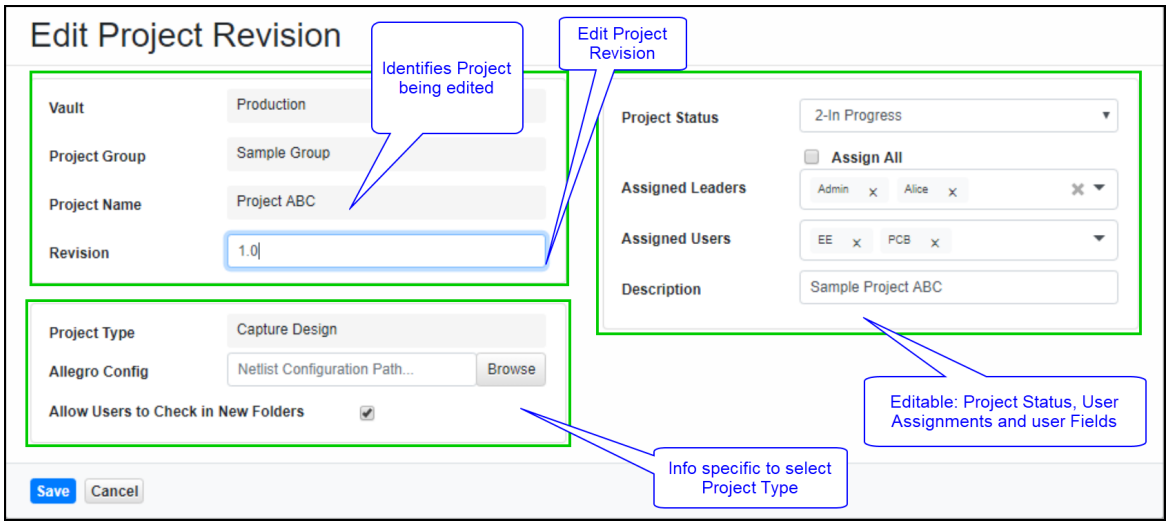
From the **Project Details** view, select the **Edit** button with the top project action buttons to open the Edit Project Revision form.



Select Edit Project in Project Details

The **Edit Project Revision** form shows the following information:

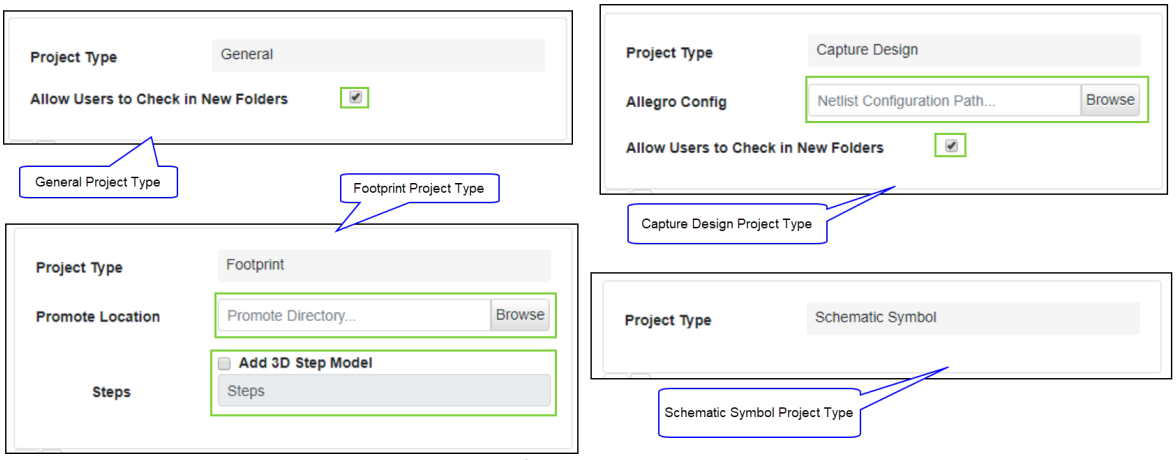
- Project identification information - name and revision of the selected project tor be edit. The Project Revision is the only editable attribute in the Project identification information.
- Project Type specific information - shows configurations that may be edited based on project type
- Project Status, user assignments and user field settings - All settings are editable.



Edit Project Form

Edit Project Type specific information

Information for each project type in the Edit Project Form is shown in the following figure.



Four Project Types Information in the Edit Project Form



Enable/Disable Email Notifications

Information that may be modified pertaining to each project type includes the following:

- **Allegro Config** - This Capture Design Project option enables you to change the netlist configuration file that you want to use for the project. Once changed, new auto-generated Netlist will use the new configured file.
- **Allow Users to Check in New Folders** - This option is available for edit for Capture and General projects. You may enforce the project structure or allow users to change the structure by checking in new folders. Once changed, check ins will restrict or allow (based on setting) check in of folders.
- **Email Notifications Enabled** - This option allows you to enable/disable email notifications on a per-project basis. It will only be shown if you have Email Notifications enabled in the Admin Configuration.
- **Footprint Promote Path** - This selection is available when the setting in Promote Footprint and Step Model Locations is configured for **Each Footprint Project**. UNC path is recommended for this setting. You will need to ensure that users defined into the Promote Method have access to the selected location. Once changed, you will need to re-promote all footprints that have been previously promoted. Ensure the Rules for Footprint promote are correctly configured when re-promoting footprints and step files.
- **Add 3D Step Model** - This Footprint option is available when Step model folders have not been configured. Enabling this option after project creation provides the same functionality when added during project creation.
- **Steps** - This pads folder name field is available and editable when editing the project to **Add 3D Step Model** functionality. The Steps folder name is a required field when the **Add 3D Step Model** checkbox is selected. The default folder name is Steps. Modify the steps folder name ensuring that it conforms to Windows folder naming convention.
- **Step Model Promote Location** - This selection is available when step model functionality has been enabled and the setting in Promote Footprint and Step Model Locations is configured for each project. UNC path is recommended for this setting. You will need to ensure that users defined into the Promote Method have access to the selected location. Once changed, you will need to re-promote all footprints that have been previously promoted. Ensure the Rules for Footprint promote are correctly configured when re-promoting footprints and step files.

Project Status, user assignments and user field settings

The project status, user assignments and configured user fields are editable in the **Edit Project** form.

- **Project Status** - Status for the project. If this field was configured as a pick list, you may choose a pick list value. Each company has different guidelines for project statuses.
- **Assigned Leaders** - only users whose role has been granted project leader permissions may be added as an assigned leader. Only assigned leaders are able to manage the project.
- **Assigned Users** - any user or user group may be assigned as users. Project users or user group members whose role has check in and check out permission are only able to check out and check in files, pages and symbols that they have been assigned. Users not assigned to the project cannot be added to assignment of items within the project. Configured project users will automatically be added as default users when new folders are added after project creation. Project users added using the Edit Project form will need to be applied to any existing project folders manually.
- **Additional configured User field** - fields shown in the New Project form is based on your specific configuration.

When changes are complete, click the Save button to save any changes or click the Cancel button to cancel the form.

5.3 User Assignments

Users may be assigned to a project as a project leader or as a project user. Assignments of files, pages or symbols for editing may be automatically assigned to only the user who checks in the item or to a specific default user list.

Assigned as Project Leader versus Project User

Users who are granted any of the permissions in the groups identified as **No Assignment Required** or **Requires Assignment as Leader** can be assigned as a leader to a project. When assigned to a project as a project leader, the user is able to perform the permissions granted within that project. But when assigned as a **User**, regardless of the assigned role in EDM, the user is only able to perform functions associated to a standard user for that project.

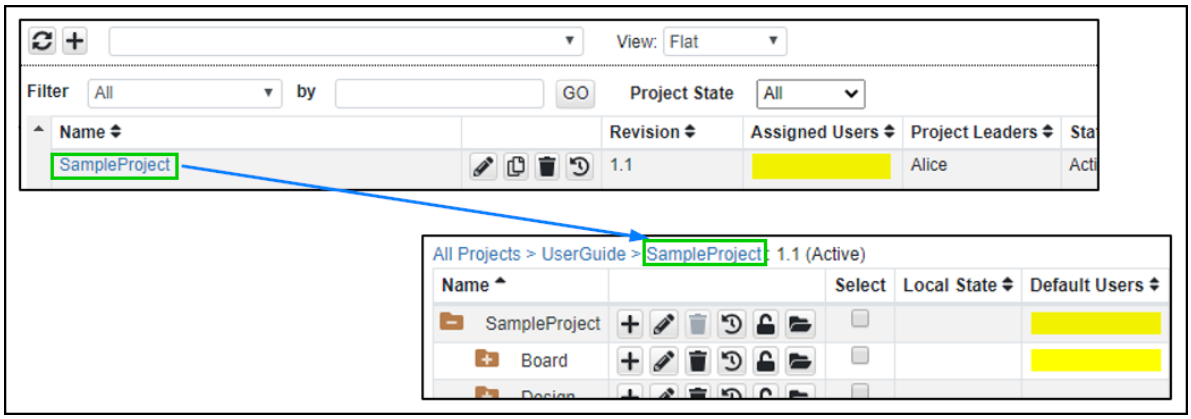
For example, the built-in user "Admin" can see the Admin menu items and all projects. If "Admin" is not assigned to a project either as a project leader or as a user, "Admin" can view project details but is unable to check out files for editing. When "Admin" is assigned to a project as a user, the "Admin" is able to check out only files that are explicitly assigned to "Admin" or a group that "Admin" belongs. "Admin" is not be able to perform project leader functions unless he reassigns himself as a project leader.

Default User Assignment

A *user* who is assigned to a project and given access to project folders is a *default user*. This contrasts to *assigned users* who are granted access to specific project files. Default users and assigned users are both field columns in the Project Dashboard and Project Details. New files added to a folder in which a user is already assigned as a default user will automatically have permission to check out and check in the newly checked in file.

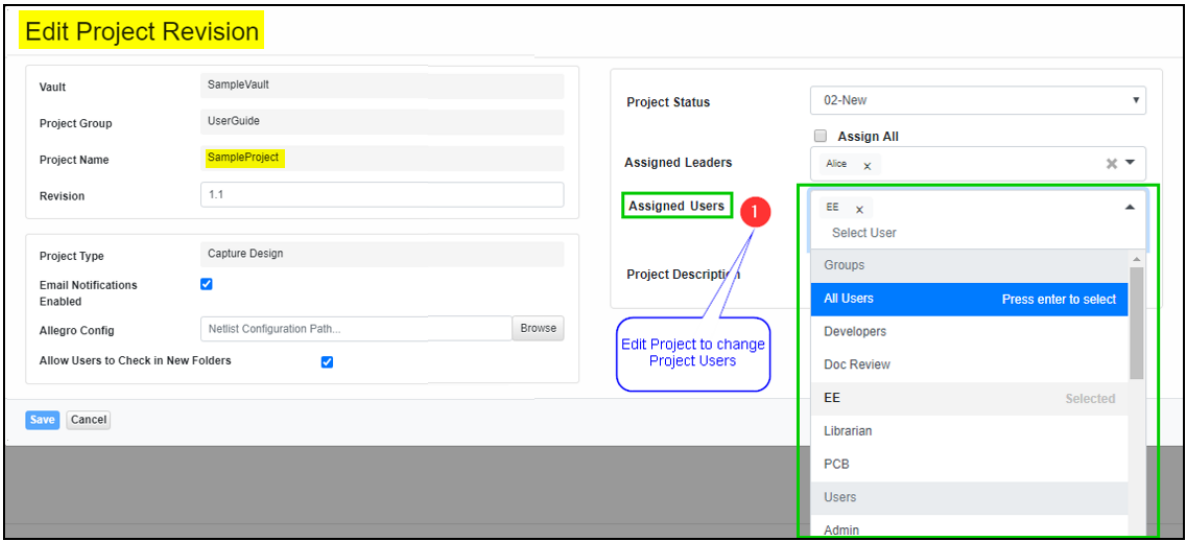
Individual users or user groups may be assigned to a project. All users added during project creation are included in the default user list as folders and files are added. Users added after a project or folder is created are not propagated to child folders and files automatically. The series of screen captures that follow provide an example of how to propagate user assignments.

For example, if you create a SampleProject without assigning users or groups to the project during project creation and then add sub-folders before you assign a user to the project, no one is assigned to the project except you, the project leader. This is shown in the screen capture that follows.

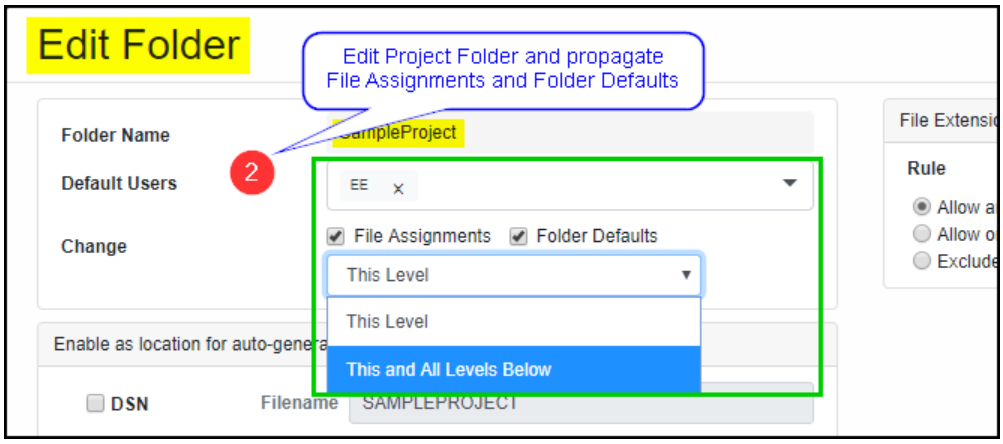


No Assignments at Project level

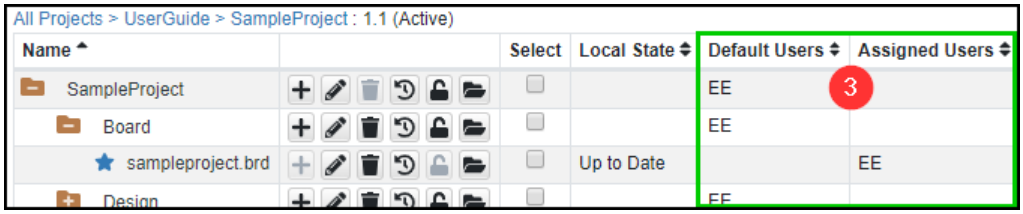
The next screen capture that follows shows the user group EE is added to the SampleProject. You need to add default assignments to the project folder and propagate the change before the changes become visible.



Edit Project > Project Users



Edit Project Folder > Add Default Users and Propagate



Updated Default Users and Assigned Users

5.4 Add, Edit, and Delete Folders

Folders help organize project files. Folders can be added to projects in the Project Details View or (optionally) during check in. Assigned leaders may add or edit project folders, manage folder contents, and add or edit schematic symbol library folders. Users assigned to a project folder may add a folder or sub-folder during check in if the **User May Add Folder/Allow Users to Check In New Folders** option is enabled.

A folder may be created for general use to store any file type. A folder may be configured as a Team Folder that provides shared file access to a DSN managed design.

Files that can be stored in folders include but are not limited to: designs, board files, Word/Excel documents, images, etc. A folder for a Capture Design project can be enabled as a location for auto-generated files that are associated to the schematic (DSN, BOM, Netlist, PDF); board (DB Doctor, EDM Validation, PDF) or Package for Release.

The Capture project tracks file relationships between the managed Capture design, the managed board, and the associated files derived from the schematic design (BOM, Netlist, and PDF) and board (DB Doctor, EDM Validation, PDF).

The **Team Folder** option can be enabled to manage a Flat Team Design, or Hierarchical Team Design or Team Library.

- Flat Team Design - allows management of a flat (single-schematic) design.
- Hierarchical Team Design - allows management of a hierarchical (multi-schematic) design.
- Team Library - allows management of schematic part (.OLB) libraries.

Assigned leaders may add folders from either the EDM Desktop application or from EDM in Capture. Assigned leaders may edit folders from any EDM user interface, which includes the EDM Desktop application, EDM in Capture, and from EDM when opened from a standard browser. However, the folder name may only be changed one of the two client plugins.

5.4.1 Add Folder

Folders may be added to Capture or General projects. Only Schematic Library Folders may be added to Schematic Symbol projects. Folders for Footprints projects are configured at the project level and may not be added separately. The section describes how to folders are added to the following project types:

- New Folder to Capture Project
- New Folder to General Project

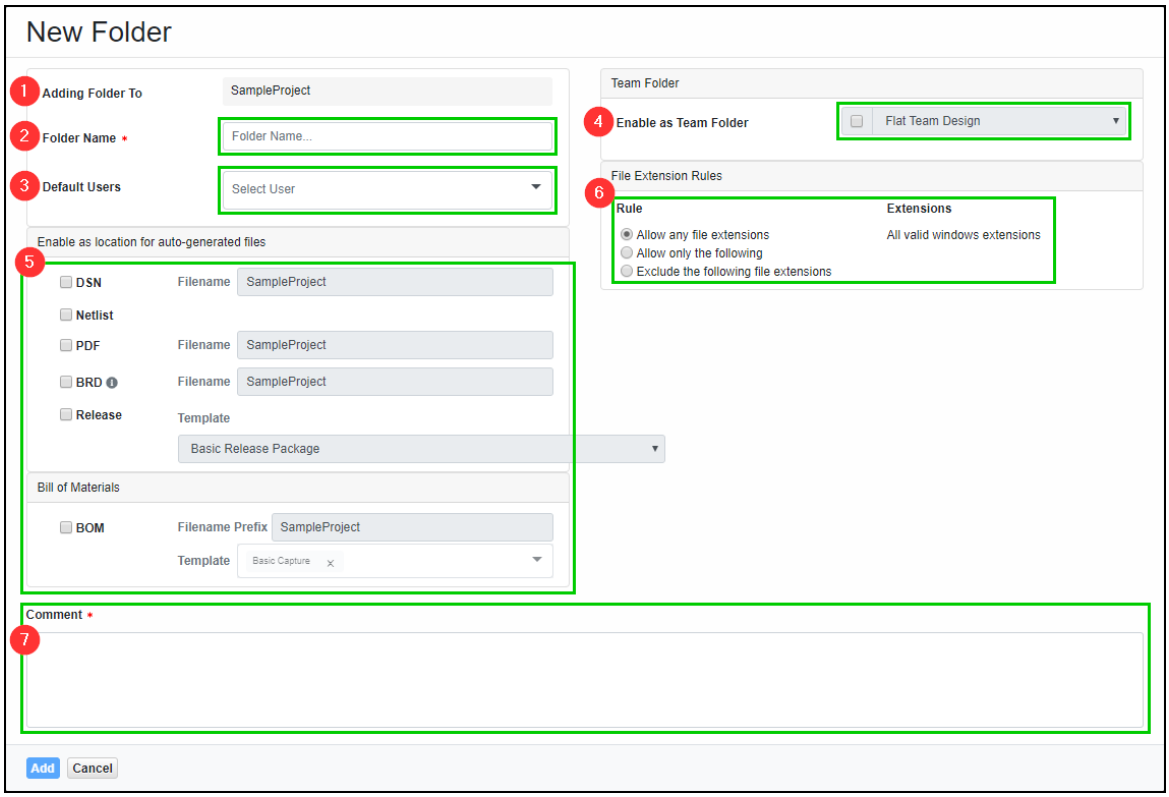
To add a folder or sub-folder within a project, click the **Add** button that is located next to a folder name of an open project. A **New Folder** form opens. The examples below detail the options available when adding project folders. After completing the **New Folder** form, click the Add button to create the folder with the selected options in the form. Or click the Cancel button to cancel and close the form.



Add Folder

New Folder to Capture Project

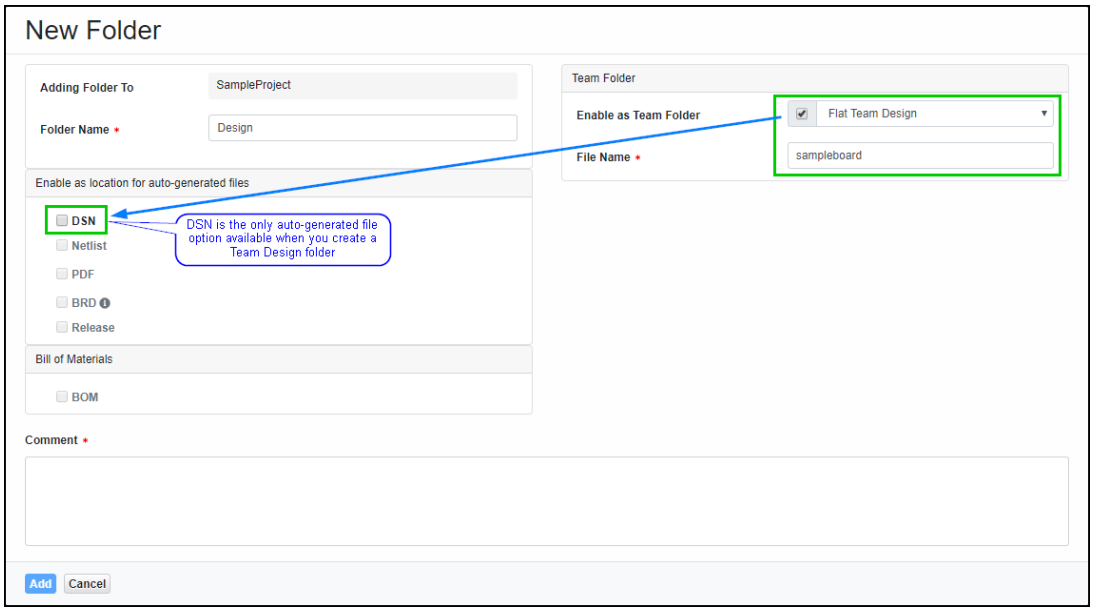
The **New Folder** form for a Capture Project allows the project leader to configure the folder options shown in the following figure. The image below outlines each field you can configure in green.



New Folder Form for Capture Project

The bulleted items that follow describe each field in the **New Folder** form. The Default Users and Comment fields are only available when you are adding a new folder to a project and do not apply when creating a new folder to a template.

- **(1) Adding folder to** – The field value that displays identifies the parent folder so you can ensure you are adding the new folder to the correct location. This field is uneditable.
- **(2) Folder Name*** – Enter a name for the folder you are creating. The name you enter must be a unique name within the project directory and conform to Windows naming conventions.
- **(3) Default Users** – This field identifies users who will be automatically assigned to new files within the folder. To display a pick-list, place your cursor within the field and click.
- **(4) Enable as Team Folder** – Select this checkbox to configure the folder as a Team Folder, then select Flat Team Design or Hierarchical Team Design or Team Library. The suggested file name of the Team Folder defaults to the project name. However, you may enter an alternate filename as shown in the figure below.



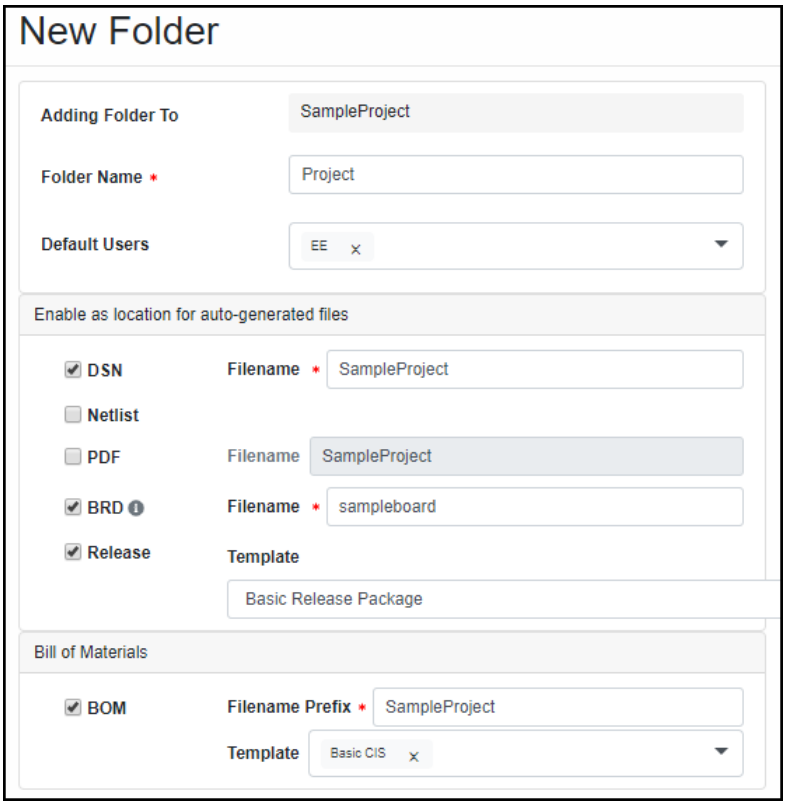
Add Folder form when the new folder is a Team Folder

- **(5) Enable as location for auto-generated files** – Select the checkbox(es) to indicate which automatically generated files you want to store in the folder you are creating.

  The name you enter for the board files (BRD) that are automatically generated when you check in your project files needs to remain in synch with the managed DSN file. When you select BRD ⓘ the filename entered will be applied to the brd file. Characters that may be used for the name may include a-z, 1-9 and _. Blank spaces are not permitted

  This feature is only available for managed Capture Designs that monitor the relationship between the managed Capture Design (DSN) and associated files (BOM, Netlist, PDF, BRD, and Release). This managed functionality updates when design and board files are checked in.

  When you are adding a folder to a project and you check either the DSN or BRD option, the form dynamically updates to allow you to accept the suggested filename for your DSN or BRD file, which defaults to match project name. You can apply this feature to your board file by clicking the ⓘ button. If desired however, you may type in an alternate filename for the managed design as shown in the following figure. The managed design file has a blue star (★) next to the filename within the project. If you are adding a folder to a template and you check either the DSN or BRD option, you may click the **Choose File** button and browse to select a DSN or BRD template.



Enter Managed Design Filename

When you are adding a folder to a project and select the **Release** option, the form dynamically changes to allow you to select a default Release Template for the project when you Package the project for Release.

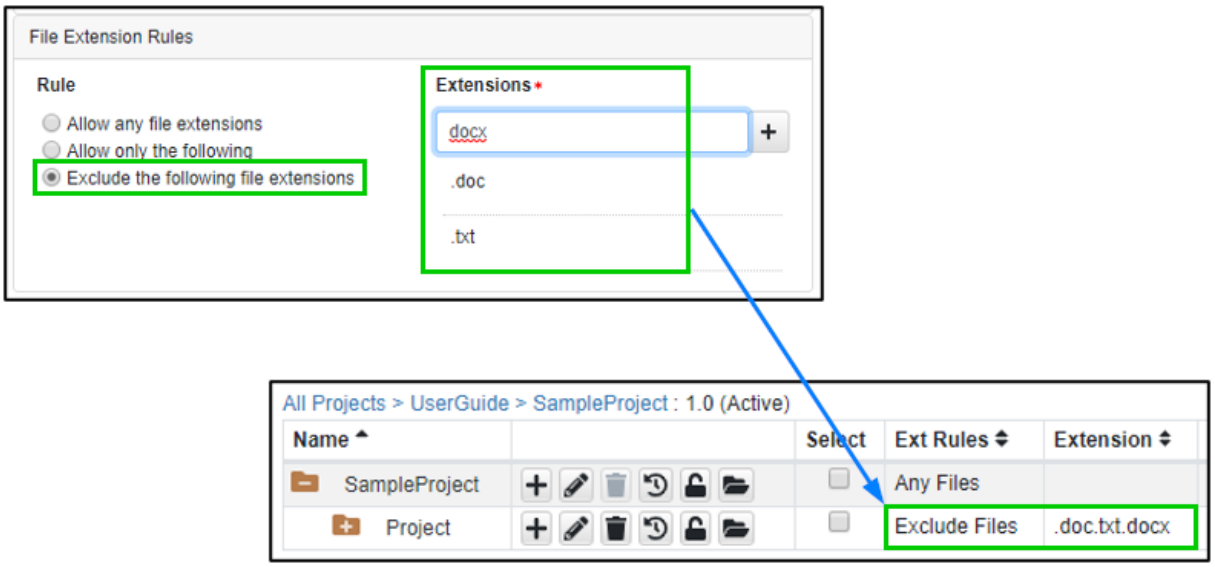
When you are adding a folder to a project and select the **BOM** option, you can use or update the **Filename Prefix** and select one or more BOM Templates to use for the auto-generated BOM(s). The BOM output may be either .BOM, .csv or .xlsx format, depending on the format of the BOM configuration.

- **Filename Prefix** - When a single BOM Template is selected, the Filename Prefix will be the BOM's filename (e.g. **<Filename Prefix>.<File Extension of selected BOM Template>**). When multiple BOM Templates are selected, the Filename Prefix will be the the prefix or root of each BOM's filename, followed by an underscore, and the BOM Template name in each case (e.g. **<Filename Prefix>_<Template>.<File Extension of selected BOM Template>**).
- **Template** - Select one or more BOM Templates to be assigned to the designated BOM folder that will be used during auto-generation when the managed design is checked in.

- **(6) File Extension Rules** – The rule you select controls file types that are allowed to be checked in for the given folder. File Extension Rules are disabled when the folder is enabled as a team folder. If you choose to **Allow only the following file**

extensions or **Exclude the following file extensions,** you need to enter each extension allowed or excluded into the File Extensions field and then click the **Add** button. To delete an extension from the list, hover over the extension and select the **Delete** button.

- Allow any file extensions - allows all file extensions to be checked in. This is the default option.
- Allow only the following file extensions - allows only file extensions listed to be checked into the folder. All others will be ignored. You must create the initial list to allow file extensions.
- Exclude the following extensions - allows all file extensions, excluding those listed, to be checked into the folder. You must create the initial list to exclude file extensions.

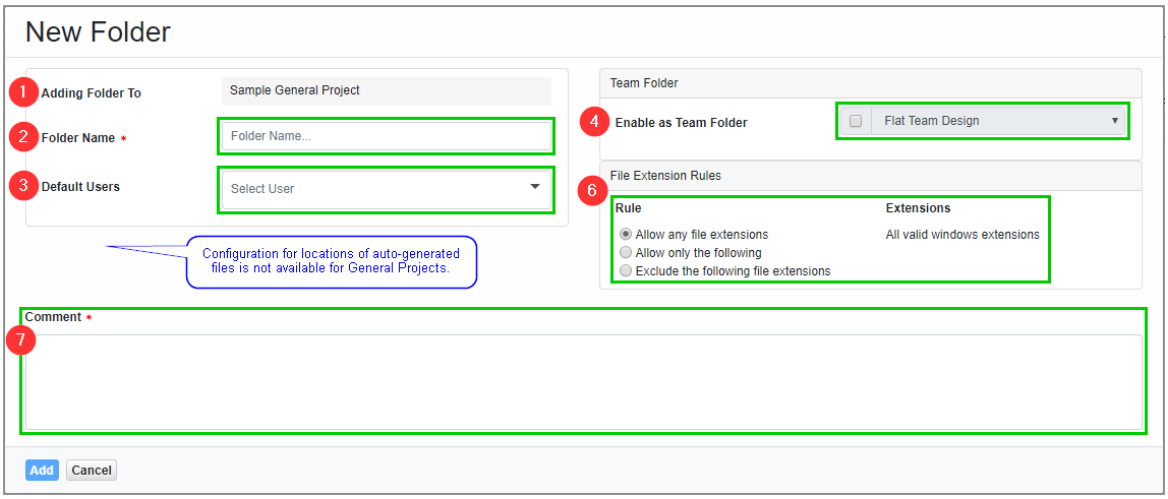


Note: File Extension Rules are disabled when the folder is enabled as a Team folder.

- **(7) Comment** – Enter a comment associated with your action. When you are adding a new folder to a project, the Comments field is required. Note contents are included in the History of the project. The Comments field is only available when you are adding a folder to a project.

New Folder to General Project

The **New Folder** form for a General Project allows the project leader to configure similar folder options to the Capture Project. The difference is that Auto-generated features are unavailable as shown in the following figure.



New Folder Form for General Projects

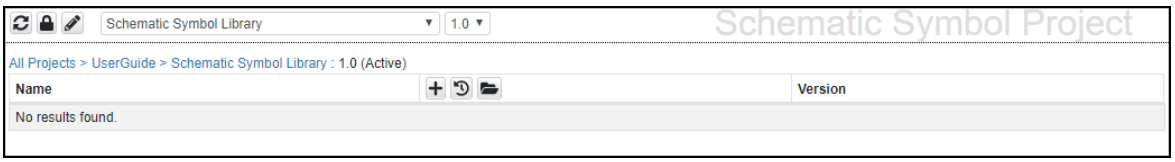
The bulleted items described in the New Form for Capture Project apply to the New Form for General Project as well. The Default Users and Comment fields are only available when you are adding a new folder to a project; not to a project template.

5.4.2 Add Schematic Symbol Library Folder

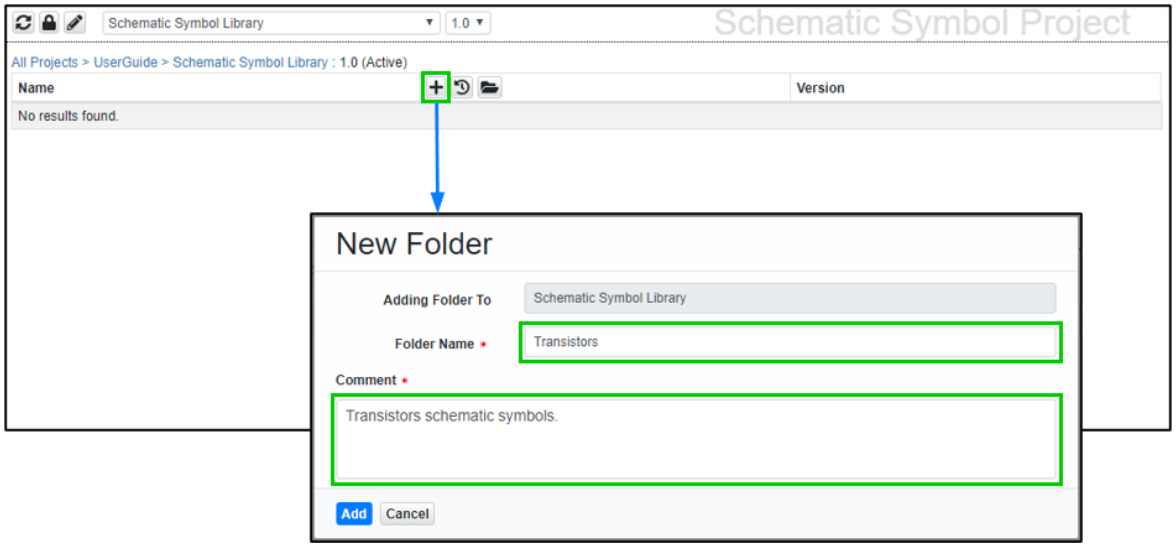
For Schematic Symbol Projects, only library folders may be added to the project.

To add a library folder to a schematic symbol project

1. Open your schematic symbol project from your project dashboard. The new schematic symbol project opens as a blank page.



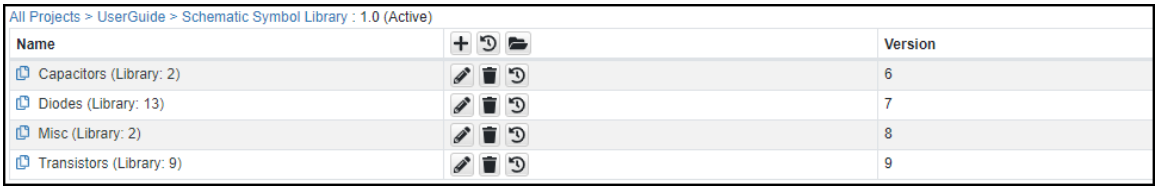
2. Click the Add Library Folder button to open the **New Folder** form.



Adding a new Schematic Symbol Library folder

3. Enter a new **Folder Name** and **Comment**. The red asterisk (*) indicates these are required fields.
4. Click the Add button to create the folder with the selected options in the form. The new library folder is added to the details page of the Schematic Symbols project. Or click the Cancel button to cancel and close the form.

The image below shows a Schematic Symbol project with several Schematic Symbol Library folders. Each folder shows the symbol count next to its name (e.g. **Capacitors (Library: 2)**).



User Created Schematic Symbol Library Folders within a Project

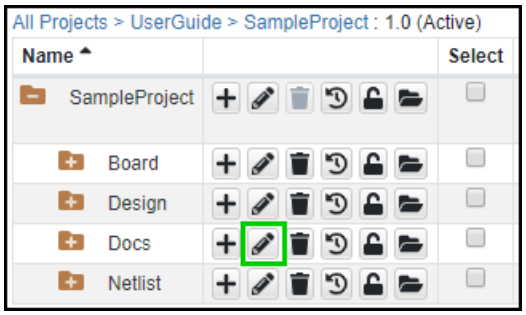
5.4.3 Edit Folder

Project folder properties that may be changed with the Edit Action depend on the type of project type. Only assigned leaders may edit folders. The Edit Folder Form allows you to change the folder name, properties based on the type of folder in which it is created. Folders name in Schematic Symbol and Footprint projects may not be changed. The following folder properties may be edited.

- Managed Capture projects
  - Capture Project folders
  - Team Design in a Capture Project
  - Team Library Folder
- General projects
  - General Project folder

- Team Design in a General Folder
- Team Library Folder
- Footprint projects
  - Footprint project folder
- Schematic Symbol Project
  - Symbol Library folder

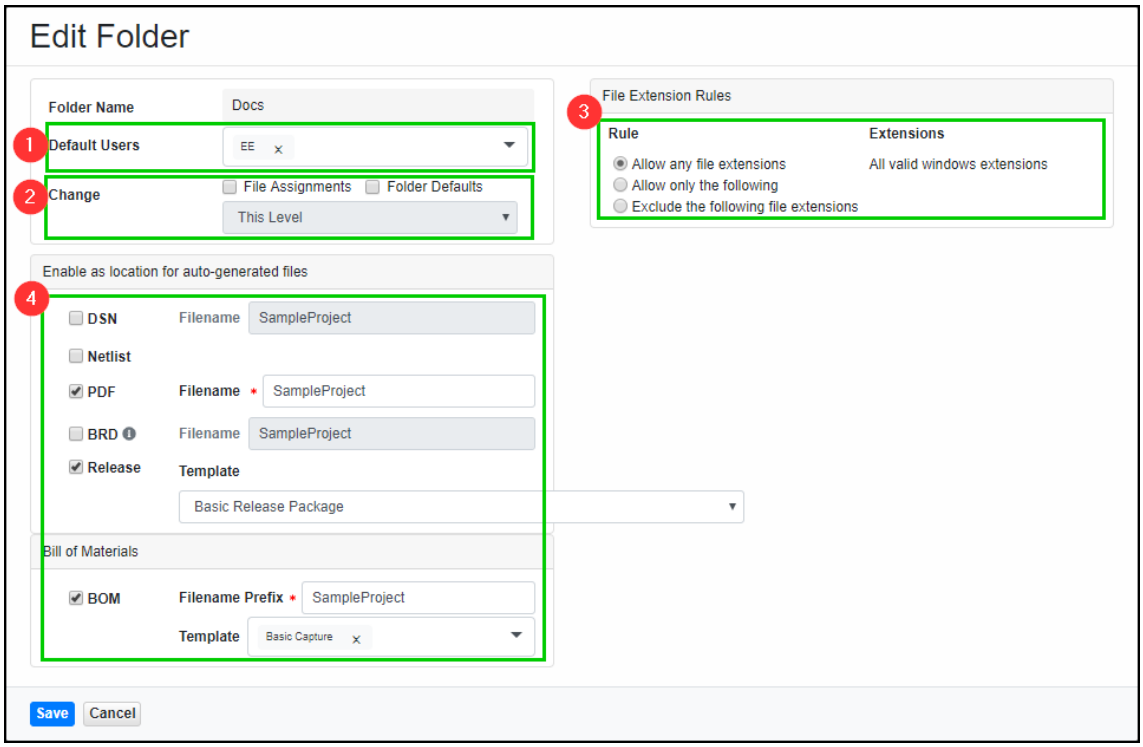
To edit a folder, click the **Edit** button that is located next to the folder name of an open project. The examples below detail the options available when editing project folders. After completing the **Edit Folder** form, click the Save button. Or click the Cancel button to cancel the folder update(s).



Edit Folder

Edit Capture Project Folder

The **Edit Folder** form for a Capture project allows the project leader to change the configuration shown in the following figure. The image below outlines each field you can edit in green. The bulleted list that follows describes each editable field by numbered call-out.



Edit Folder Form for Capture Projects

- **(1) Default Users** - This field shows the default assignments for all new files checked into the folder. To assign a username or work group to this field, place your cursor in the field and click to view a selection list of users and work groups.

- **(2) Change** - Allows you to customize the assignment of default users who are granted access to this folder and sub-folders.
  - **File Assignments** – Select this checkbox to update the assigned user list granted permission to files in folders selected in the drop down menu.
  - **Folder Defaults** – Select this checkbox to update default user assignment to folders selected in the drop down menu.
  - **This Level** – Select this drop-list menu item if you want to want to grant the default users access to files and folder(s) at the current level.
  - **This and All Levels Below** – Select this drop-list menu item if you want to grant default users access to files and/or folders at this level and all sub-folder levels.

- **(3) File Extension Rules** - allows control of file types in the given folder. Refer to File Extension Rules in Add Folder for an explanation of each rule.

- **(4) Enable as location for auto-generated files** (Managed projects) - Each checked file indicates the auto-generated files (DSN, BOM, Netlist, PDF, BRD, or Release files) to be stored in the folder you are editing. Select/deselect check marks to edit the file type placed in the folder when your managed design is checked in. Refer to Capture

Design Projects and Add Folder for additional information. Additional options are available with the following check boxes.

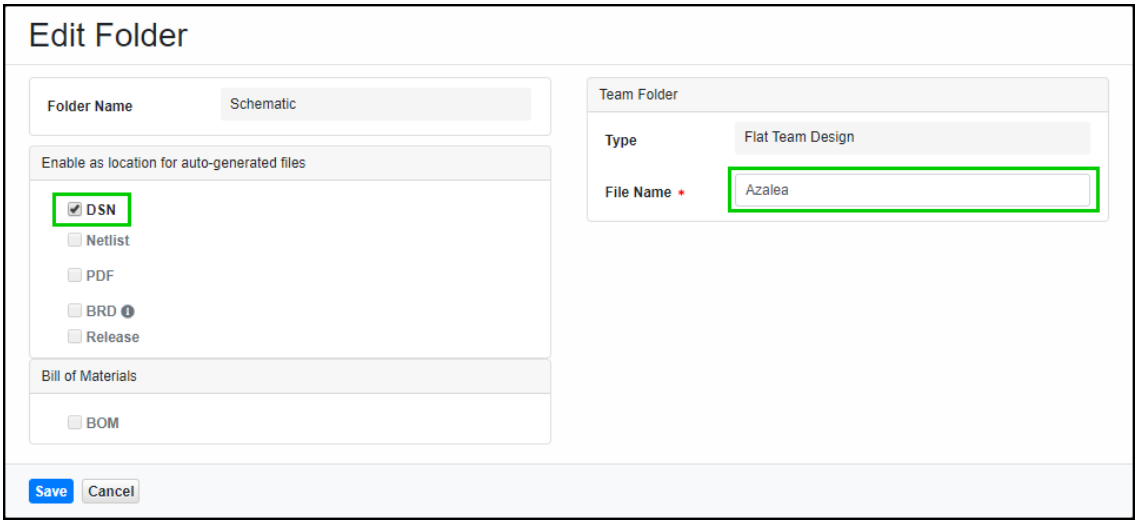
- **DSN** - If you want to edit the filename of the managed design, update the name of the DSN **Filename**.
- **BRD** - If you want to edit the filename of the managed board, update the name of the BRD **Filename**.
- **Release** - If you want to update the selected **Release Template** to be used for the Release Package, select the Template name from the drop-down list.
- **BOM** - If you want to update the selected **BOM Template(s)** to be used for the auto-generated BOM(s), select the BOM Template(s) from the drop-down list.

Edit Team Design Folder in Capture Project

The **Edit Folder form** for a **Team Design** belonging to a Capture Project allows an assigned leader to change the properties shown in green in the following figure. The properties of you may change include:

- **File name** of the Flat or Hierarchical Team Design as shown in the figure below.
- The option to enable the team design folder as the managed Capture Design. This option is available only when the Team Design Folder is created in a Capture Project.
- The type of team folder may not be changed once created.

Refer to Add Folder Section for more information on creation of Team Design or Team Library Folder



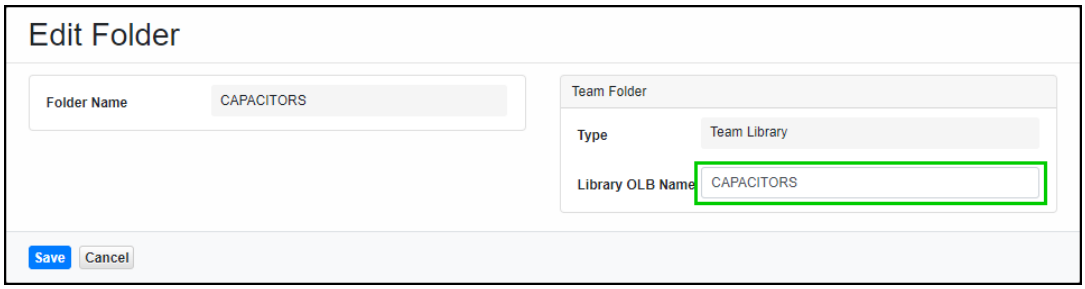
Edit Folder Form for A Team Design in a Capture Project

Edit Team Library Folder

The **Edit Folder form** for a **Team Library** allows a project leader to change the properties shown in green in the following figure. The properties of you may change include:

- **Library OLB name** of the merged library as shown in the figure below.
- The type of team folder may not be changed once created.

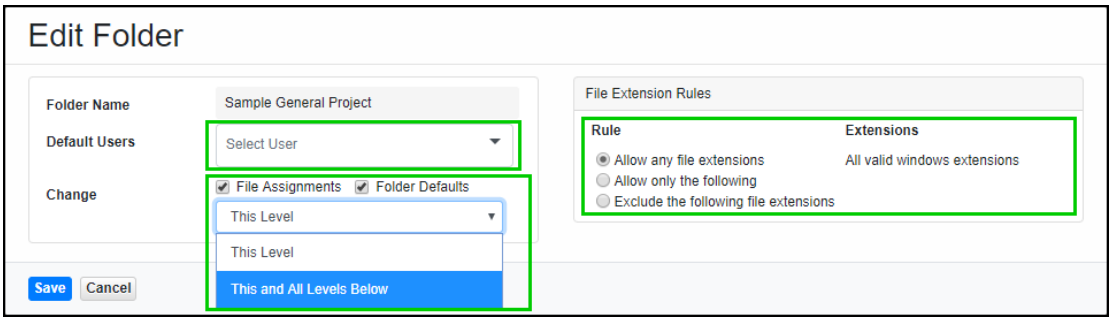
Refer to Add Folder Section for more information on creation of Team Design or Team Library Folder.



Edit Folder Form for a Team Library

Edit General Project Folder

The **Edit Folder** form for a General project allows the project leader to change Default Users, File Assignments, Folder Defaults (Default Users for the folder) and File Extension Rules as shown in the following figure. The image below highlights each field you can edit in green.



Edit Folder Form for a General Project

- **Default Users** - This field shows the default assignments for all new files checked into the folder. To assign a username or work group to this field, place your cursor in the field and click to view a selection list of users and work groups.

- **Change** - Allows you to customize the assignment of default users who are granted access to this folder and sub-folders.
  - **File Assignments** – Select this checkbox to update the assigned user list granted permission to files in folders selected in the drop down menu.
  - **Folder Defaults** – Select this checkbox to update default user assignment to folders selected in the drop down menu.
  - **This Level** – Select this drop-list menu item if you want to want to grant the default users access to files and folder(s) at the current level.
  - **This and All Levels Below** – Select this drop-list menu item if you want to grant default users access to files and/or folders at this level and all sub-folder levels.

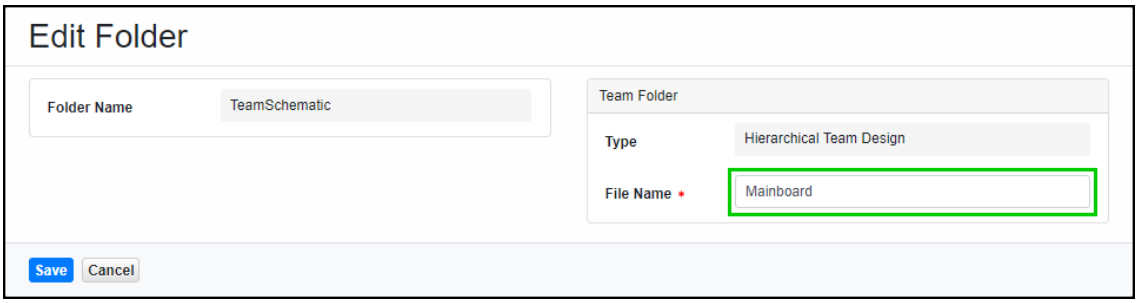
- **File Extension Rules** - allows control of file types in the given folder. Refer to File Extension Rules in Add Folder for an explanation of each rule.

Edit Team Design Folder in General Project

The **Edit Folder form** for a **Team Design** belonging to a General Project allows a project leader to change the properties shown in green in the following figure. The properties of you may change include:

- **File name** of the Flat or Hierarchical Team Design as shown in the figure below.
- The type of team folder may not be changed once created.

Refer to Add Folder Section for more information on creation of Team Design or Team Library Folder



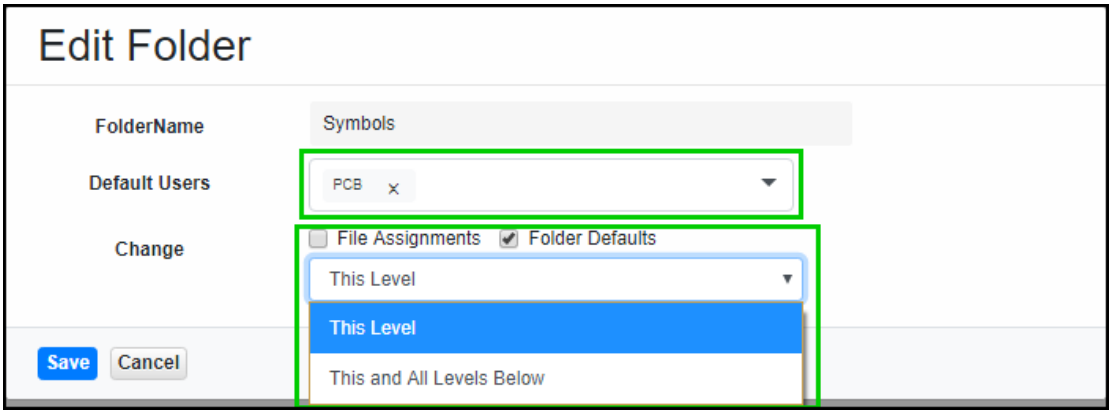
Edit Folder Form for Team Designs In a General Project

Edit Footprint Folder

The **Edit Folder** form for a Footprint project folder allows the project leader to change Default Users, File Assignments, and Folder Defaults (Default Users for the folder) as shown in the following figure.

- **Default Users** - This field shows the default assignments for all new files checked into the folder. To assign a username or work group to this field, place your cursor in the field and click to view a selection list of users and work groups.

- **Change** - Allows you to customize the assignment of default users who are granted access to this folder and sub-folders.
  - **File Assignments** – Select this checkbox to update the assigned user list granted permission to files in folders selected in the drop down menu.
  - **Folder Defaults** – Select this checkbox to update default user assignment to folders selected in the drop down menu.
  - **This Level** – Select this drop-list menu item if you want to want to grant the default users access to files and folder(s) at the current level.
  - **This and All Levels Below** – Select this drop-list menu item if you want to grant default users access to files and/or folders at this level and all sub-folder levels.



Edit Folder Form For Footprint Projects

Edit Schematic Symbol Library Folder

The **Edit Folder** form for a Schematic Symbol Folder allows the project leader to edit the name of the OLB library saved in a Schematic Symbol Library folder.



Edit Folder Form for Schematic Symbol Projects

5.4.4 Delete Folder and File

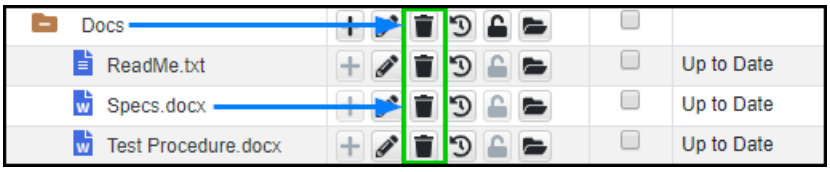
An assigned leader may delete folders or files from a project. Team folders may be deleted as well as individual pages, schematics/groups and symbols. Refer to Delete Schematic/Group and Page to remove pages from a Team Design. Refer to Delete Footprints and Step to remove files from Footprint projects. Refer to Delete Symbols to remove symbols from a library folder. Deleted files or folders may be retrieved from history at the folder or project level. You may delete a group of selected files using the **Mass Edit** button.

Note: Before deleting a folder or file, be sure to perform a **Full Download** first, which will ensure all files have up-to-date Local State values. If you are deleting a file because it has a Local State issue (e.g. Not in Database), it may resolve itself after performing a **Full Download**.

Delete Folder or File

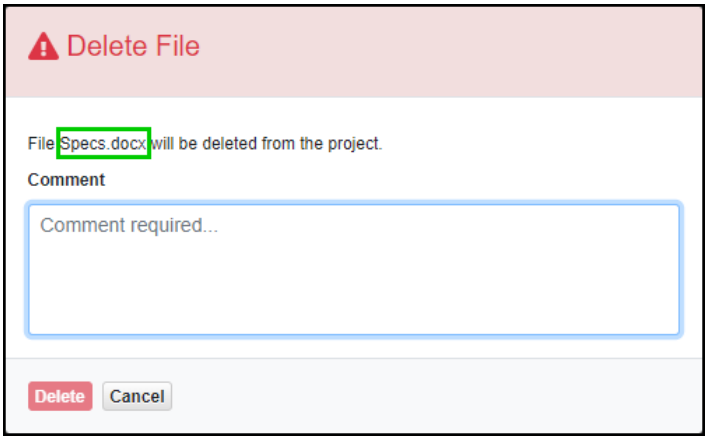
1. Click the **Delete** button located on the file or folder row that you want to delete, as shown in the following figure. Deleting a folder removes the child folders and files in that folder.The delete folder functionality is unavailable for Footprint projects.

   Note: Files that are checked out can not be deleted until checkout is canceled.

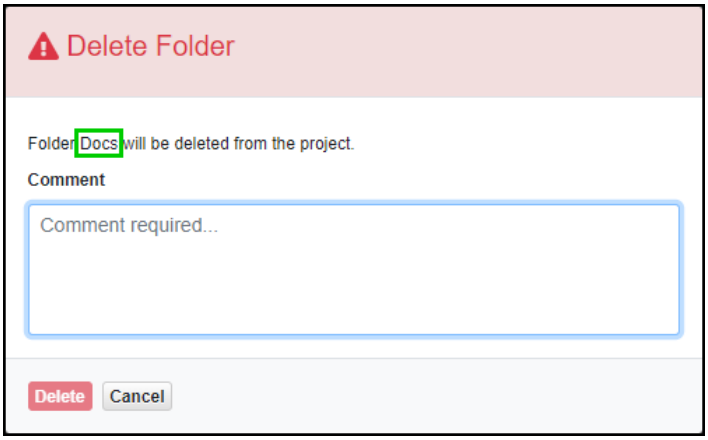


Delete File or Folder from a Project

2. After you click the **Delete** button, the respective *Delete File* or *Delete Folder* confirmation form appears. Enter a comment and then click the Delete button. Or click the Cancel button to cancel and close the form.



Delete File Confirmation



Delete Folder Confirmation

Note: Deleting a file deletes it from the vault and your local workspace. Users who have a copy of the file in their local workspace need to refresh their view before the file is removed from their local workspace. Deleting a folder deletes it from the vault and your local workspace as well. However, the local copy of the folder in other users' local workspace remains until it is manually removed.

If you try to delete a file that has been checked out, you will receive the following *Unable to Delete File* error. You will also see an error for delete a folder that has a checked out file.



Unable to Delete File Error

You may delete all files and folders from the Project Details View with the exception of the project folder itself. Refer to Delete Project Revision for removal of the project folder.



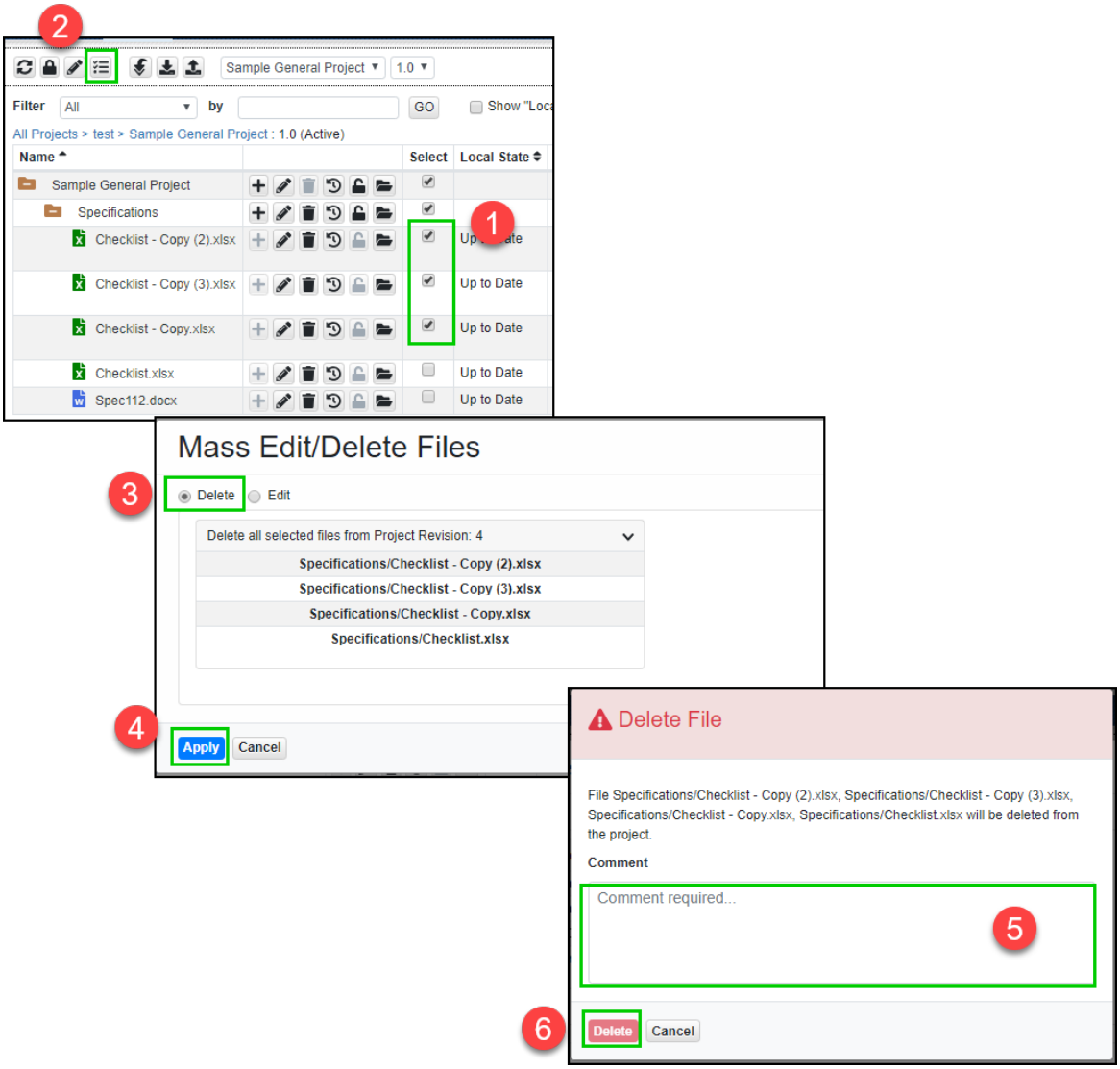
Delete Button for Project Folder Not Allowed from Project Details View

Delete a Group of Selected Files

The **Mass Edit/Delete Files** form allows you to delete selected files or change statuses and configured user fields on selected files.

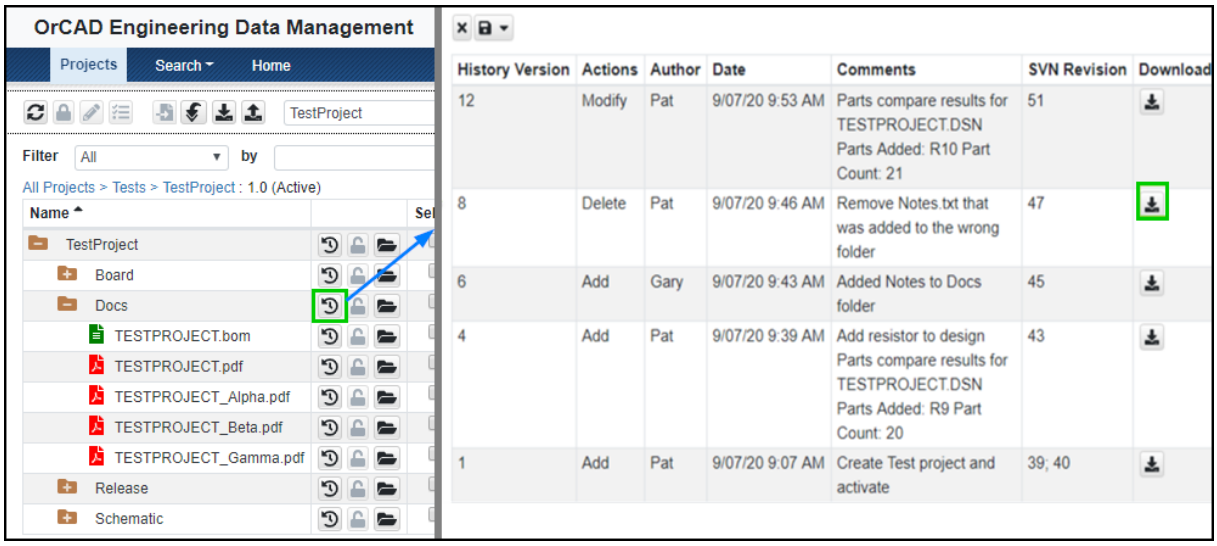
To delete several files at one time:

1. Select files that you delete using the check boxes in the **Select** column.
2. Select the **Mass Edit** button.
3. At the **Mass Edit/Delete Files** form, select the **Delete** radio button.
4. Review the list of files to be deleted and select the **Apply** button to bring up the **Delete File** pop up.
5. Enter a comment to be recorded in History.
6. Select the Delete Delete button to remove files from the project

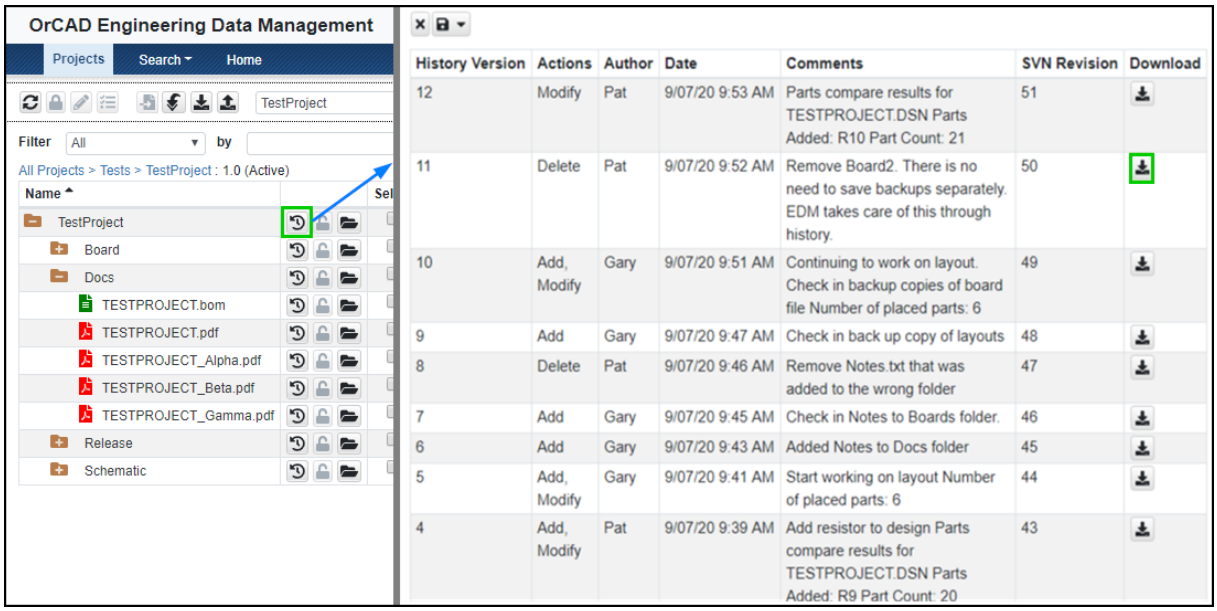


Download a deleted file from history

When a folder is deleted from a project, a record of the deletion is stored in the project history. To retrieve folders or files that have been deleted, you can open the history and download the version prior to the deletion of the item. You may retrieve the deleted file by selecting the folder or project history as shown in the following figures. If the folder has also been deleted, you will need to get it from Project History as shown in the *Retrieve Deleted Files from Project History* figure below.



Retrieve Deleted Files from Folder History



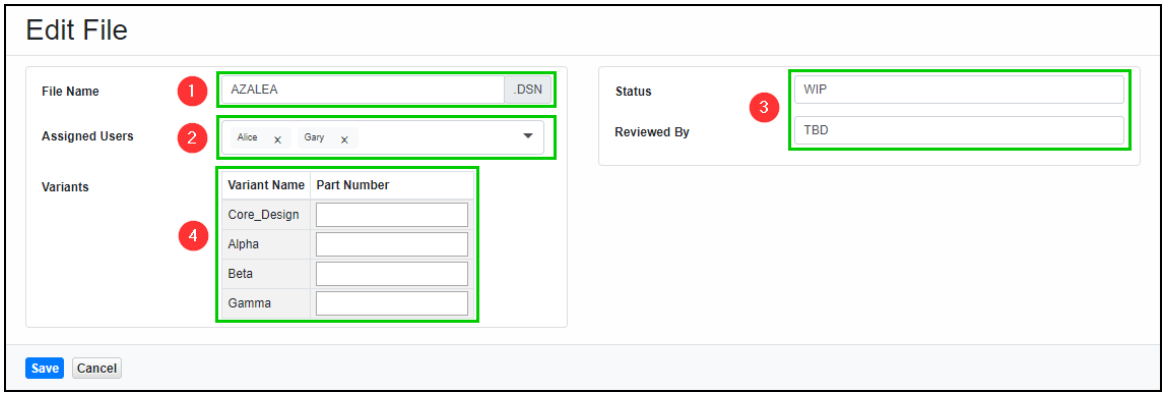
Retrieve Deleted Files from Project History

5.4.5 Edit File Properties

The **Edit** File Action button that is located next to the file name in the Project Details view opens a **Edit File** form. Assigned leaders can edit the file name, assigned users and file properties from the **Edit File** form. Additionally, when the file is a managed design or board file, the part number fields may also be edited. For Footprint projects, filenames may not be changed using the Edit File Form. You may edit properties on a group of selected files using the **Mass Edit** button. Changes to file name will be recorded in the history for the file.

Edit File Properties for Manage Design

The **Edit File** form allows the assigned leader to change the information identified in the green boxes shown in the following figure for a managed design.



Edit File Form for Managed Design

1. **File Name** - Update this if you want to rename the managed design (DSN) file.
2. **Assigned Users** - Allows you to specifically change the assignment to make this different from folder default.
3. **Status and User fields** - Update these fields as needed. The **Reviewed By** field in example above is a configured user field.
4. **Part Number / Variant Part Numbers** - The part number of your design file. If variants exist, they are read during check in of the managed design and all variants from the design will show in this list. When mapped for Titleblock updates, the variant part numbers will update the titleblock for each variant PDF.

Edit File Properties for Manage Board

The **Edit File** form allows the assigned leader to change the information identified in the green boxes shown in the following figure for a managed board file.



Edit File Form for Managed Board

1. **File Name** - Update this if you want to rename the managed board (BRD) file.
2. **Assigned Users** - Allows you to specifically change the assignment to make this different from folder default.
3. **Status and User fields** - Update these fields as needed. The **Reviewed By** field in example is a configured user field.

4. **Part Numbers** - The part number of your board file. If the part number field is mapped to update your PCB Titleblock will update the board file with this part number when you check the board file in.

Edit File Properties for All Other Files for Capture and General Projects

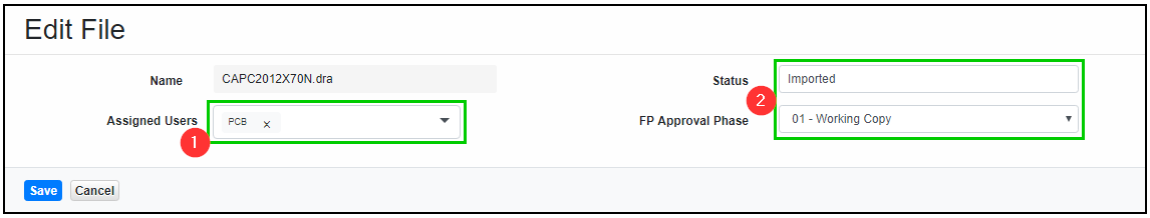
The **Edit File** form allows the assigned leader to change the information identified in the green boxes shown in the following figure for all other standard files.



1. **File Name** - Update this if you want to rename the file.
2. **Assigned Users** - Allows you to specifically change the assignment to make this different from folder default.
3. **Status and User fields** - Update these fields as needed. The **Reviewed By** field in example is a configured user field.

Edit File Properties for Footprint Projects

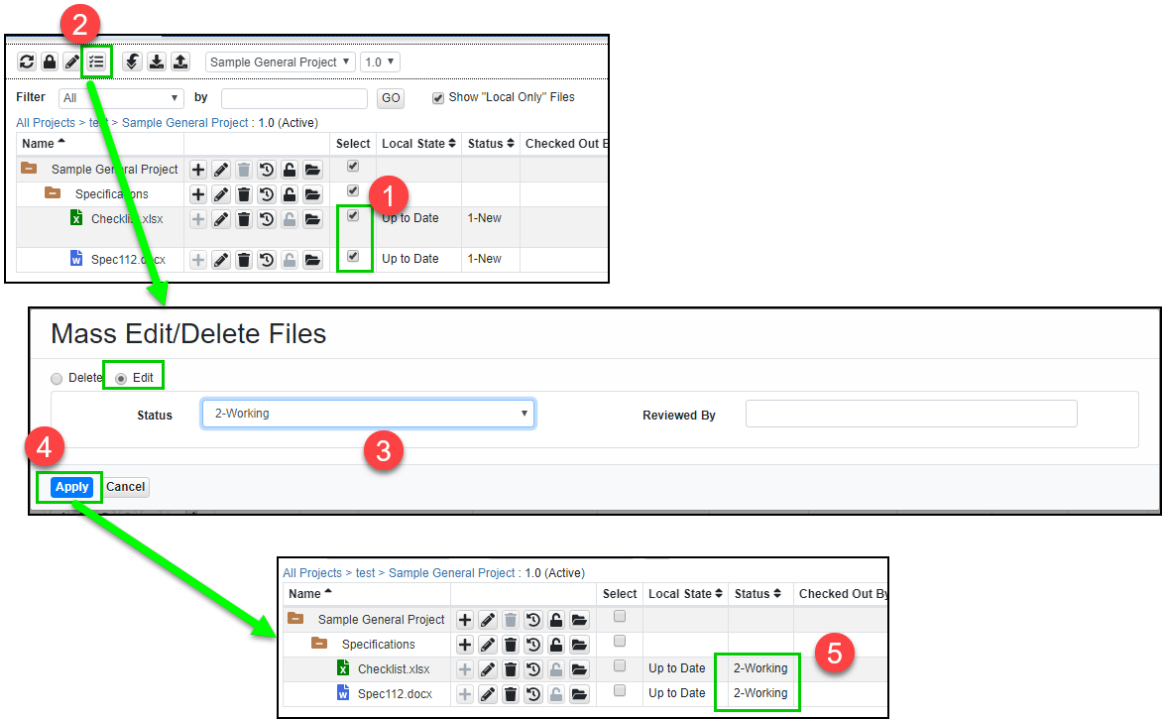
The **Edit File** form for a Footprint project allows for user assignment changes and status and user field updates.



1. **Assigned Users** - Allows you to specifically change the assignment to make this different from folder default.
2. **Status and User fields** - Update these fields as needed. The **FP Approval Phase** field in the example is a configured user field.

Edit Properties For A Group of Selected Files

The **Mass Edit/Delete Files** form allows you to delete selected files or change statuses and configured user fields on selected files.



Mass Edit Example

To change properties for several files at one time:

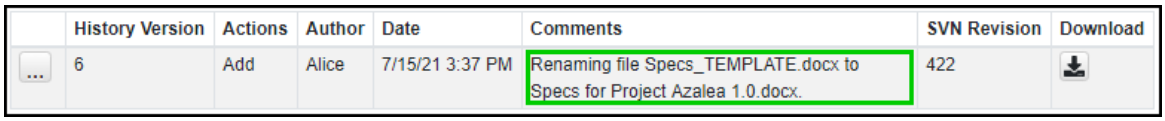
1. Select files that you want to edit properties using the check boxes in the **Select** column
2. Select the **Mass Edit** button.
3. At the **Mass Edit/Delete Files** form, make sure that you have the **Edit** radio button selected. Make on changes on the form.
4. Select the **Apply** button.
5. See file properties update.

History of File Name change

To verify the file name change in history, click the History button on the file's row.



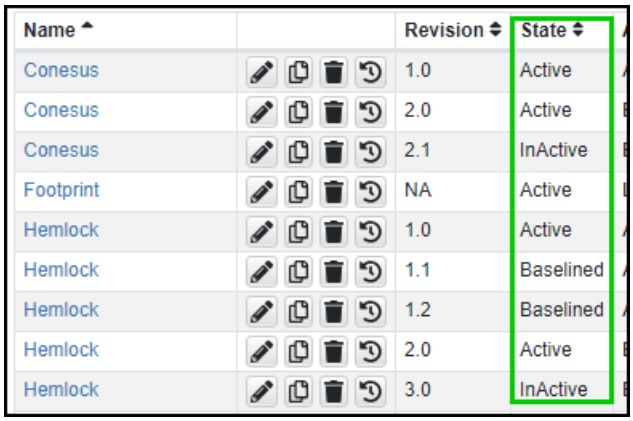
Click the History Button on the File's Row



Sample History Showing Filename Change

5.5 Activate/Baseline Project

The state of a project is shown in the Project Dashboard.



Name		Revision	State
Conesus		1.0	Active
Conesus		2.0	Active
Conesus		2.1	InActive
Footprint		NA	Active
Hemlock		1.0	Active
Hemlock		1.1	Baselined
Hemlock		1.2	Baselined
Hemlock		2.0	Active
Hemlock		3.0	InActive

Project States for Projects in Dashboard

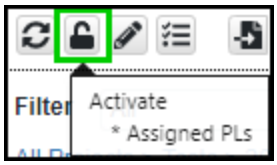
An EDM project may have one of three project states. Each time you change a project's state (Activate or Baseline), a confirmation pop-up form will appear. This allows you to change the project status, user assignments, and configured user fields before you change the project's state.

- **InActive** - default state when projects or project revisions are created. When a project is inactive, users are unable to download the project or check items into the vault. An **InActive** project can be activated.
- **Active** - check in and check out of items are enabled. Users can do a full project download or check out items, then check items back into the vault. An **Active** project can be baselined.
- **Baseline** - disables check out and check in. Users are able to download the project and view items from a history download. A **Baselined** project can be re-activated.

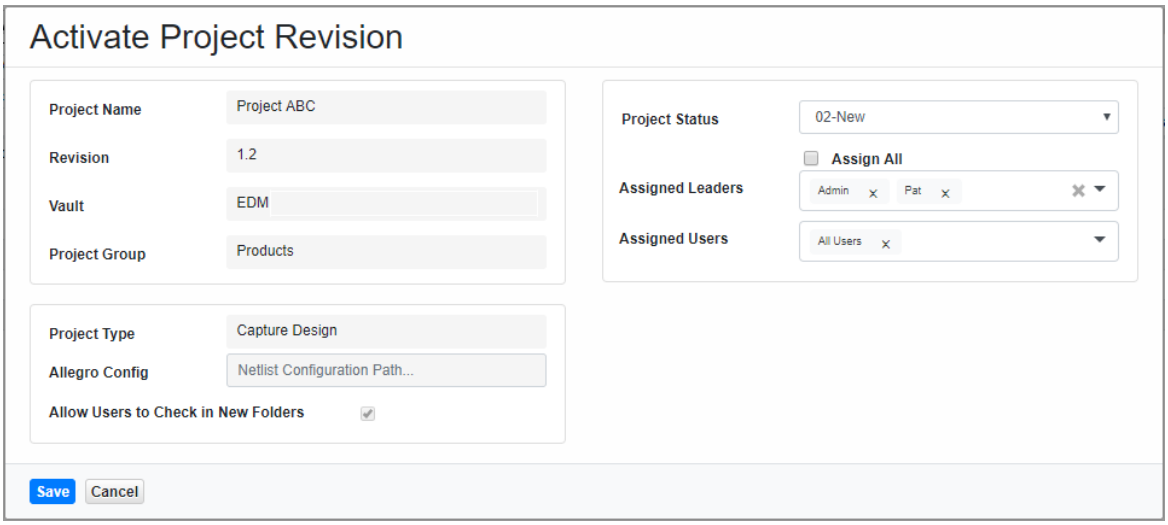
Activate Project

If not activated upon project creation, assigned leaders must Activate the project before users are able to begin working on the project. Once activated, users may begin checking items into the project.

To activate the project, click the **Activate** button in the Project Details View. If desired, update the project status, user assignments, and configured user fields. Click the Save button to activate the project, or click the Cancel button to close the **Activate Project** form.

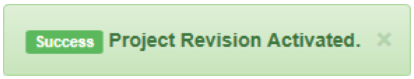


Activate Project Button

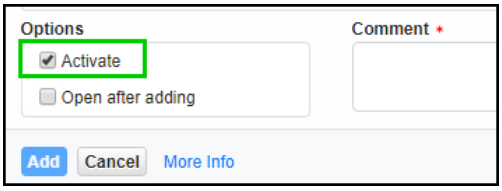


Activate Project Form

A project activation success message will display when completed. The state of the project revision in the Project Dashboard will be updated as well. You may dismiss the message manually by clicking on the X in the message or it will dismiss automatically after a few seconds.



Note: Prior to project activation, only the Edit Project and Activate buttons are enabled. Activation may occur from either the Inactive or Baseline state. Projects may also be activated on creation by selecting the **Activate** checkbox when adding a project or copying a project.



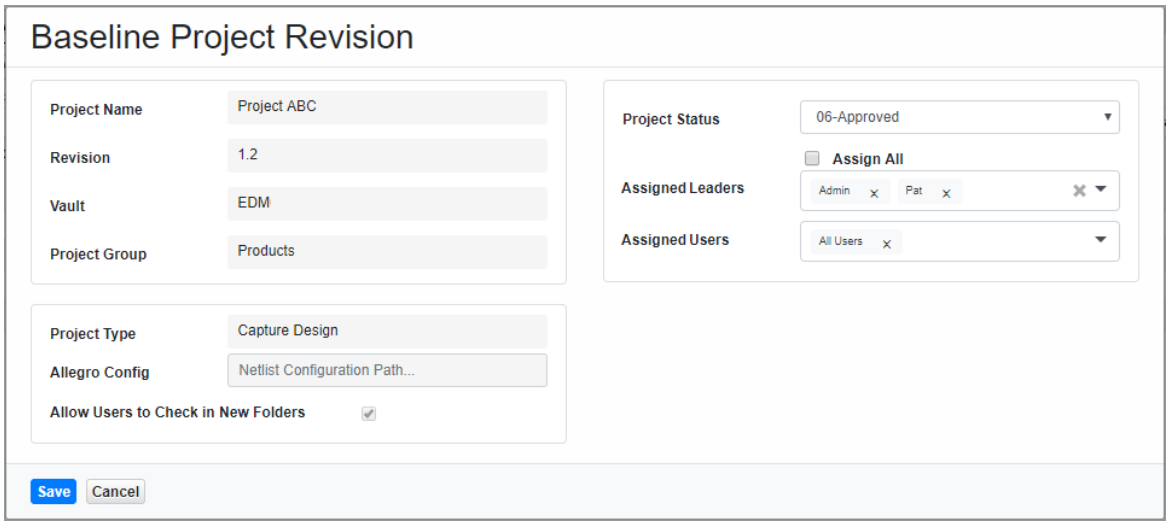
Activate On Add in New Project Form

Baseline Project

When the project is ready for approval or is put on hold, you should baseline the project so modifications are disabled. To baseline a project, click the Baseline button in the Project Details View. If desired, update the project status, user assignments, and configured user fields. Click the Save button to activate the project, or click the Cancel button to close the **Activate Project** form.

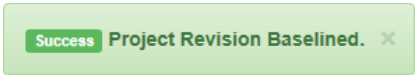


Baseline Project Button



Baseline Project Form

A success message will display when completed. The state of the project revision in the Project Dashboard will be updated as well.



After a project is Baselined, users are not able to check out or check in items to the project until it is Activated again.

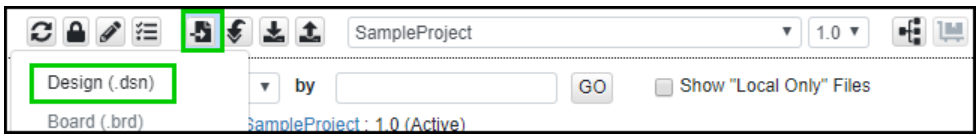
5.6 Import Design and Board Files

Existing design and board files can be imported into EDM projects. You can import a design file into EDM when you open EDM from Capture or Capture CIS. You can only import a board file, however, from the EDM Desktop application. Although a project may include both a managed DSN and BRD file, a project can only have one DSN and one BRD file. If the design you are importing includes variants and you open EDM from Capture CIS, you are prompted to assign part numbers to the core design and variants.

Import a Design (.dsn) into a project

1. Open the project for which you will import the design.

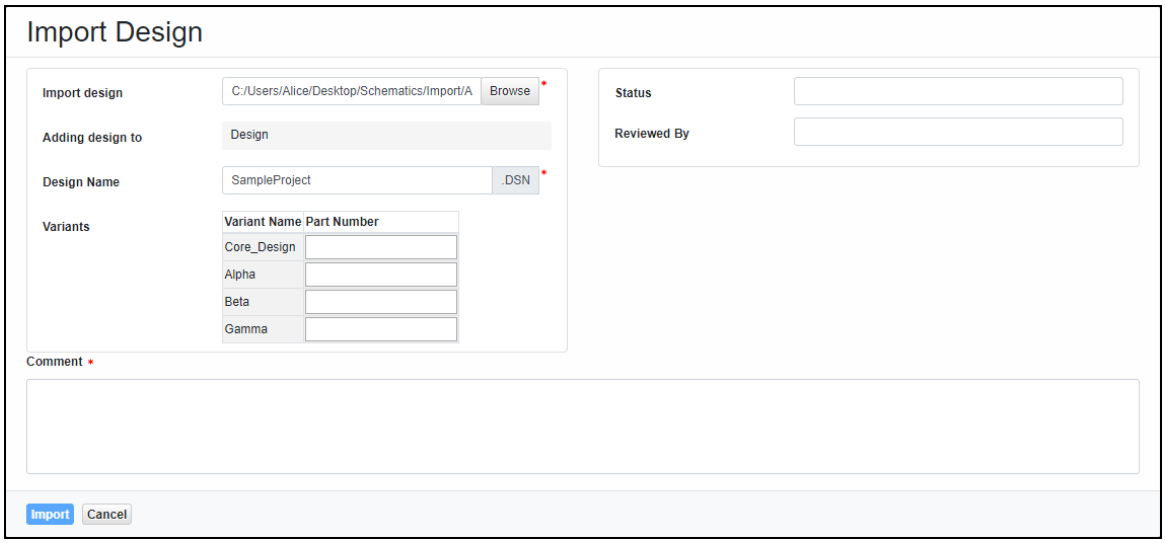
2. Click the **Import** button and choose **Design (.dsn)** from the Import drop-down list.



Import > Design (.dsn)

3. Browse to select the .DSN file you want to import and click Open to open the Import Design form.

   OrCAD Capture opens the selected .DSN file to determine whether the file includes variants. If variants are found, the Import Design form includes fields for you to enter part numbers to the core design and each variant.



Import Design Form

4. Enter values into the fields of the Import form. If you have an unversioned local file with the same name as the name of the file you are importing, a Import Feature Warning displays asking if you want to overwrite or keep both copies. A screen capture of this warning is provided at the end of this procedure.

   The bulleted list that follows describes the functionality of each field in the Import Design form.

   - **Import Design** - filename and path for the selected .DSN file for import is initially populated. If you want to select a different file, click the **Browse** button to open and select a new file. EDM opens the file to determine whether the design has variants.

   - **Adding design to** - indicates the folder within your project designated for the managed design. The design file will be imported to this folder. This field is not editable. If this doesn't look like the folder you designated as the DSN folder, click the Cancel button and review your projects and configuration.

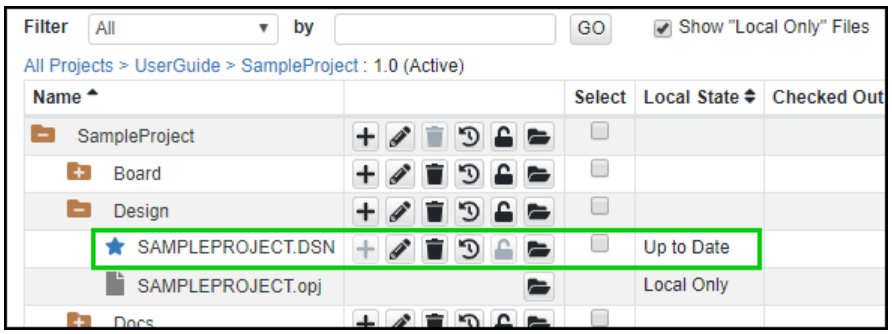
   - **Design Name** - file name that will be given to the file once it is imported into your project. The file name may be changed at this point and will update the managed design name.

   - **Part Number** - core design and variants that are found in the design file are listed. You may enter a part number for the core design and each variant. The part number can be used to update BOMs or title blocks when BOMs and PDFs are automatically generated.

- **Status** - status of the project. Values that represent the project status are dependent upon company guidelines.
- **User Fields** - additional fields configured from user fields for projects. In the sample figure above, the configured user field is Reviewed By.
- **Comment** - enter a comment regarding the import of the design. The red asterisk (*) indicates entry into the Comment field is required. The comment you enter is saved in the project history.

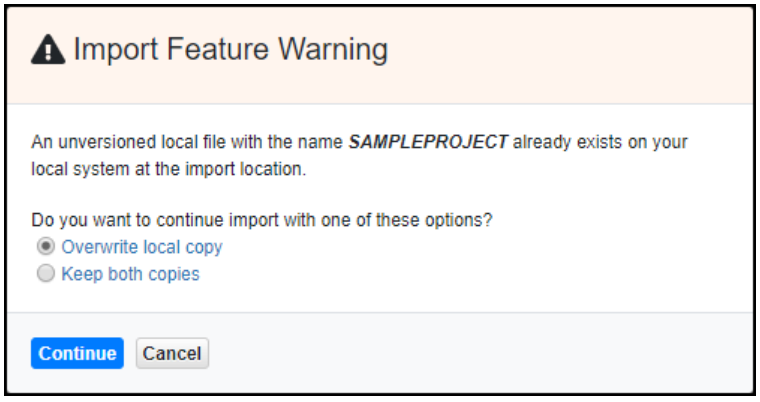
5. Click the Import button to import the design file. Or click Cancel to cancel the import. A success message displays when the importation process is completed.

The imported design is identified with a blue star (★) that indicates a managed design.



Blue Star Indicates a Managed Design

Note: When a design file is found in your local workspace that has the same name as the design you are importing, you will receive an *Import Feature Warning* message with options as shown in the following figure:



Choose one of the following options, then click Continue to proceed with the import and selected option.

- **Overwrite local copy** – replaces the design file in your local workspace with the imported file. Any changes made to your identified local design will be lost.

- **Keep both copies** – saves both copies in the project folder. One design file will be designated as the managed design while the second design file will be appended with a version number in brackets, e.g. *projectname***[1]**.dsn.

Import a Board (.brd) into a project

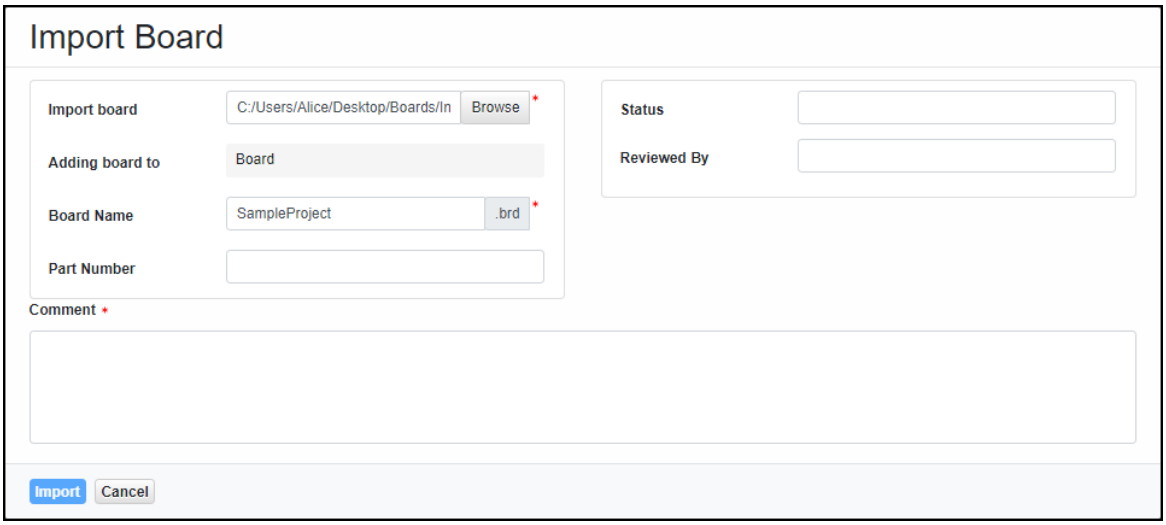
1. Open your project from the Project Details view. The board you are importing will be imported into this design.

2. Click the **Import** button and choose **Board (.brd)** from the Import drop-down list.



Import > Board (.brd)

3. Browse to select the .BRD file from the file selector you want to import and click Open to open the Import Board form.



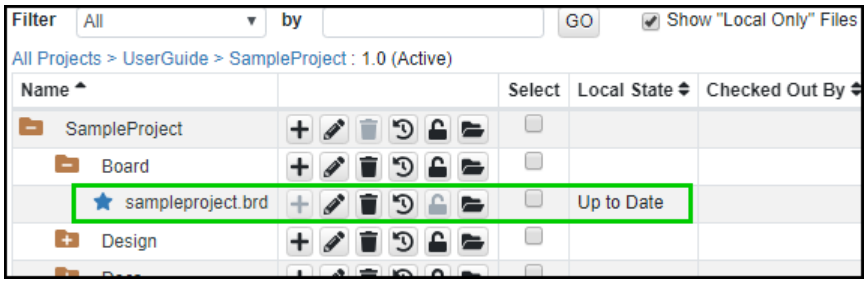
Import Board Form

.

4. Enter attribute values into the fields of the Import form. The red asterisk (*) indicates the Comment section requires input.

5. Click the Import button to import the board file. Or click Cancel to cancel the import. A success message displays when the importation process is completed.

The imported board file is identified with a blue star (★) that indicates a managed board.



Blue Star Indicates a Managed Board File

5.7 Manage Footprint Projects

Footprint projects can be used for footprints, mechanical symbols, padstacks, template symbols, and step models. When opened with the EDM Desktop application, footprint projects enable full footprint management and usage functionality.

Only assigned leaders will be able to perform footprint management tasks which includes the following:

- Import footprints and step models - Bring new files into the footprint project
- Promote footprints and step models - Copy files to a central location for use with PCB Editor to place components
- Delete footprints and/or associated files - Remove any files added by accident or no longer required
- Edit Folder properties and Edit File Properties - Change assignments, status and user field information

EDM uses the PCB Editor to determine and maintain file relationships. EDM will attempt to launch your PCB Editor when you work with footprints. You may be prompted by PCB Editor to select the product to launch before continuing. EDM establishes the connection to the PCB Editor after launch.

5.7.1 Import Footprints and Step Models

Footprint files can be brought into EDM by using import or check in functionality. EDM Desktop application will be required for any actions you take on the footprint projects. EDM connects to your PCB Editor during the import process to get obtain relationship information using built in PCB Editor functionality. Review troubleshooting information if EDM is unable to connect to PCB Editor.

Only assigned leaders may import library files. Any assigned user is able to check in new footprint files. Only assigned users may check out, modify and check in any assigned footprint files.

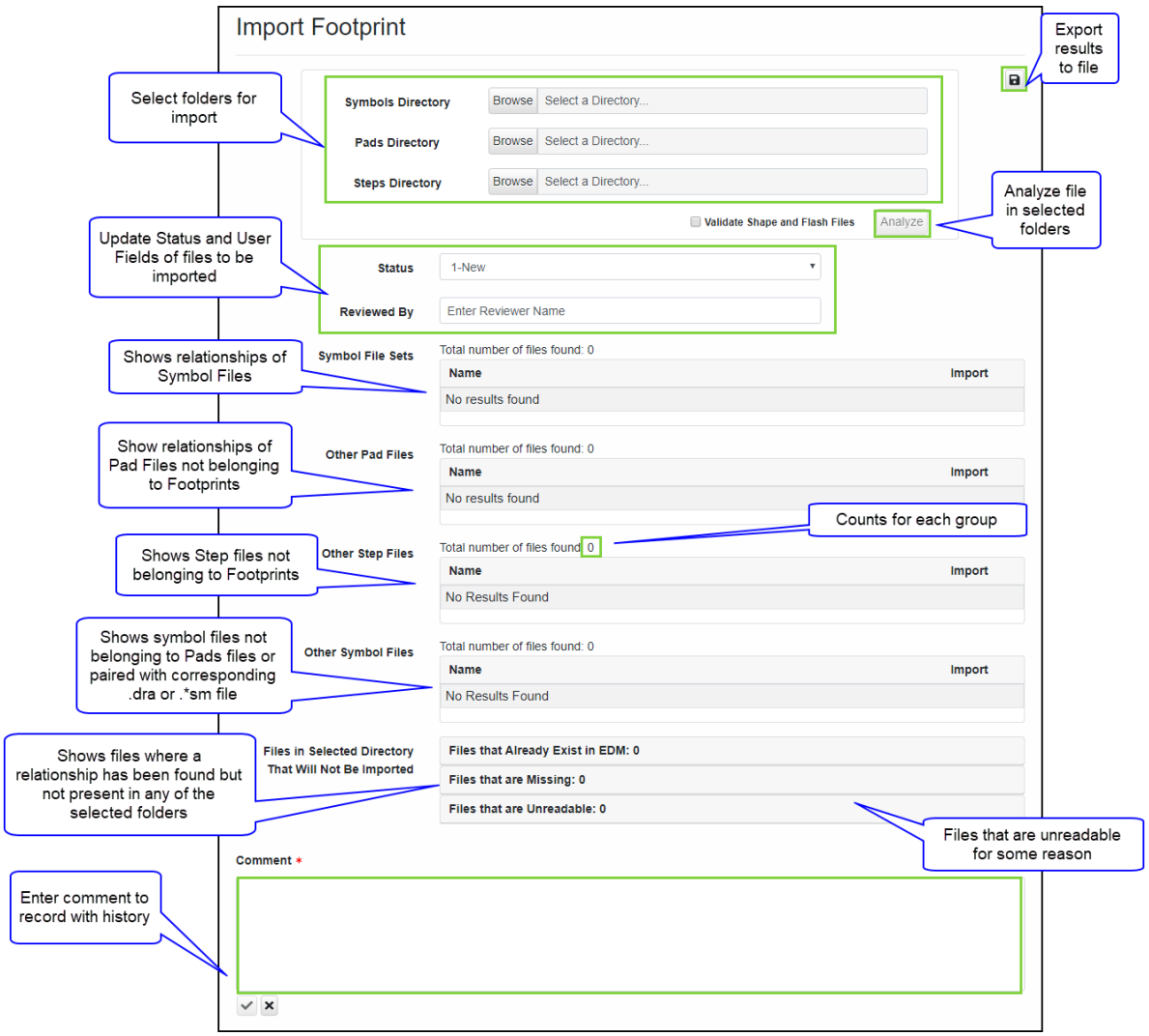
EDM uses the version ID from the .dra file which is normally in sync with the version ID in the .psm file when both are saved at the same time. If this is not the case for your footprints, you will need to run Sync DRA to PSM files before proceeding.

Select the **Import** button from the Footprint Project Action Buttons inn the Footprint Project View to open the **Import Footprint** Form.

Import Footprint Form

From the **Import Footprint** Form, use the following steps to import your footprints.

1. Select directories of various footprint files for import
2. Obtain relationships of all files in selected directories using Analyze
3. View relationships
4. Review files that will not be imported
5. Export relationships to a file for off-line analysis
6. Select files and Import

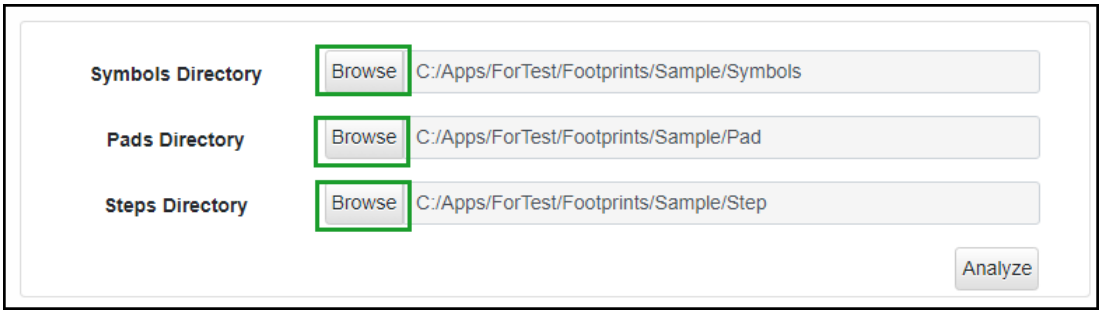


Import Footprint Library Form

Select directories

To select directories for each type of file for import, click on the Browse button and navigate to the folder containing the type of file selected. You may only select one of each type of folder. If you have files spread across multiple folders, you may repeat this process several times. EDM will only look for the

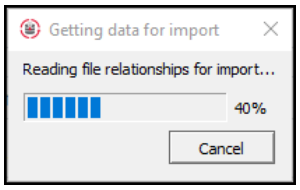
specified file types in each of the respective folders. For example, .pad files in a symbols directory will be ignored.



Each time you repeat this process, EDM will identify files already imported during when you Analyze the folders.

Analyze

You have to option to include validation of shape and flash draw files to ensure that they are of the correct format. After making your folder selections, click on the **Analyze** button. If **EDM** has not connected with PCB Editor, EDM will launch PCB Editor and attempt to connect. After connection, **EDM** will command PCB Editor to open each file in the selected folders to obtain file relationships. A progress pop up shows during this process.



If **EDM** fails to connect to **PCB Editor**, review troubleshooting section, EDM Fails to Connect to PCB, in the Appendix to resolve this issue.

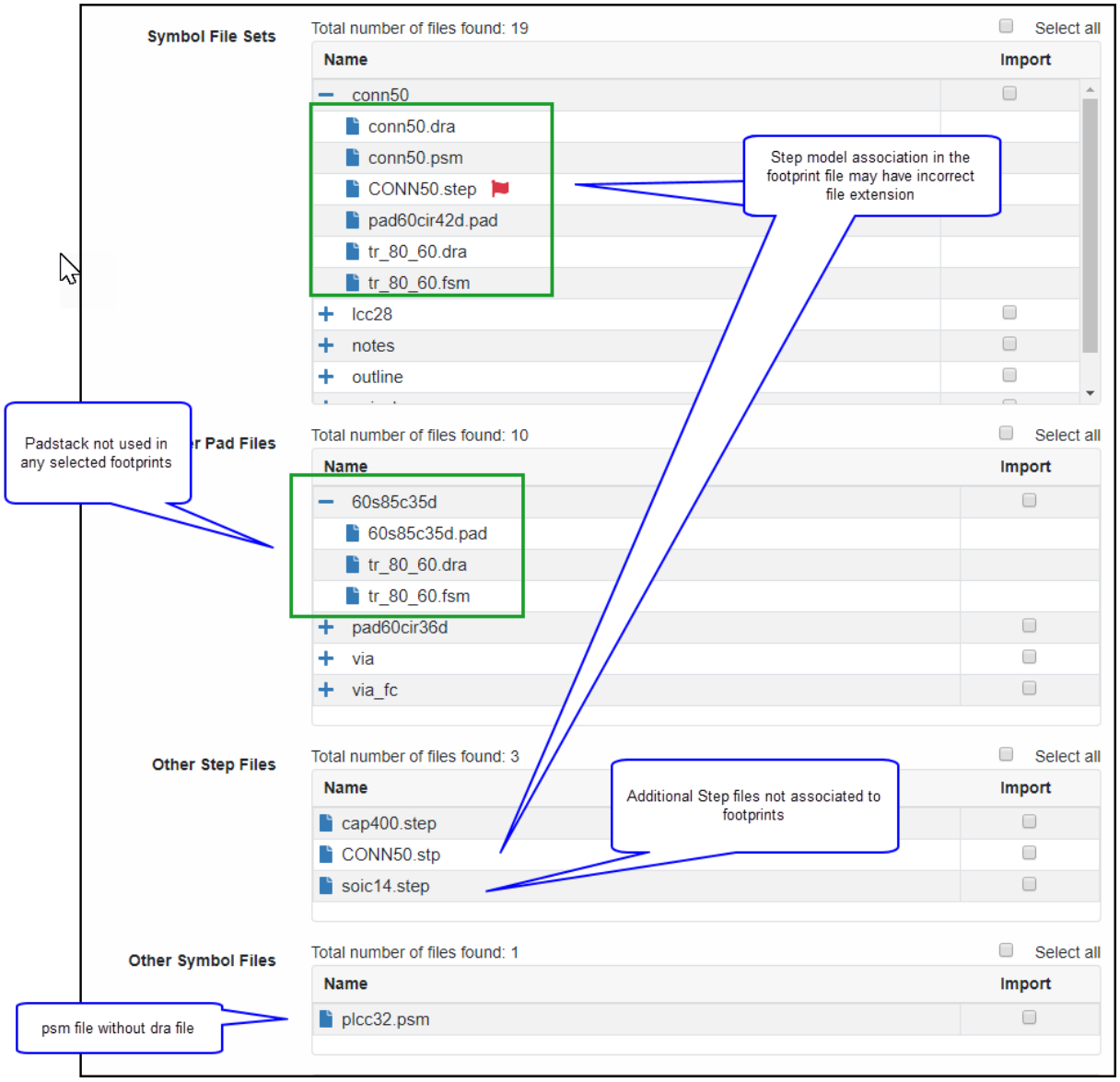
View Relationships

The Analysis of your footprint files are identified in the following four categories. Within each category, all related files analyzed are shown. Files not found in selected folders will be identified as missing. Files already in EDM and cannot be imported will also be identified.

- **Symbol File Sets** - shows footprints, templates, mechanical symbols sets with relationships found from files. Relationships include padstacks with corresponding shapes and flash and step model. Each footprint, template or mechanical symbol is shown collapsed. Expand each to review the full set of files.
- **Other Pad Files** - shows padstacks with corresponding shapes and flash not identified in Symbol File Sets. Each padstack is shown collapsed as separate lines. Expand each to review the full sets of files.
- **Other Step Files** - shows Step files not identified in Symbol file sets.
- **Other Symbol Files** - shows any symbol files found without relationships. This includes the following cases:
  - Both draw (.dra) and the corresponding compile graphic (.*sm) are not found in the import directories

- Shape and flash files are not identified as used in any padstacks

A sample analysis is shown in the following figure. All relationships found are identified in each category. This sample shows an example where a footprint file may have an association to a step model that is named different from the actual step model file name.



Sample Footprint Import Analysis

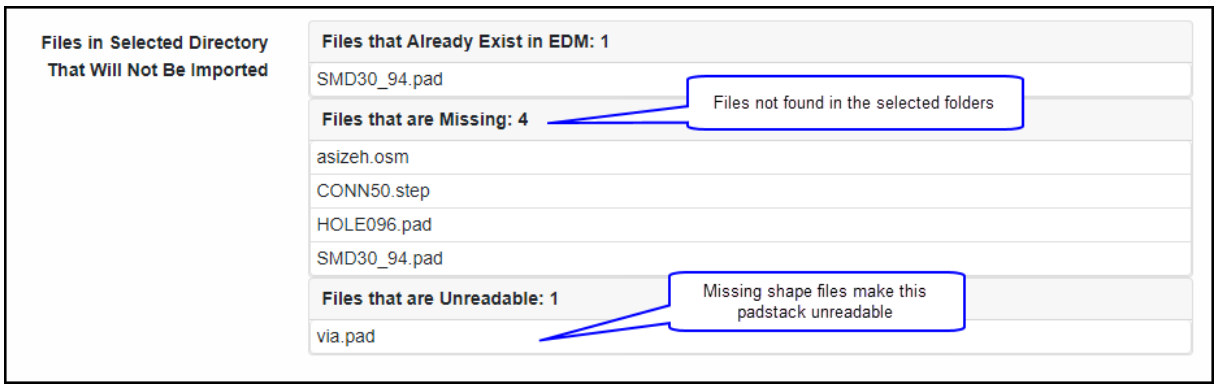
File That Will Not be Imported

Files that will not be imported into footprint project will belong to one of the following three categories.

- **Files that Already Exist in EDM** - If a file has been imported or checked in to EDM already, you may not use the import process to update the file. You will need to Check Out the file, modify it and then check it back into footprint project.

- **Files that are missing**: Shows a list of files where the analyze found relationships but cannot locate the file in the selected folder.
- **Files that are Unreadable**: Shows files that PCB Editor are unable to read. This could be files that are corrupted or pad files without the corresponding shape or flash files in the correct folder.

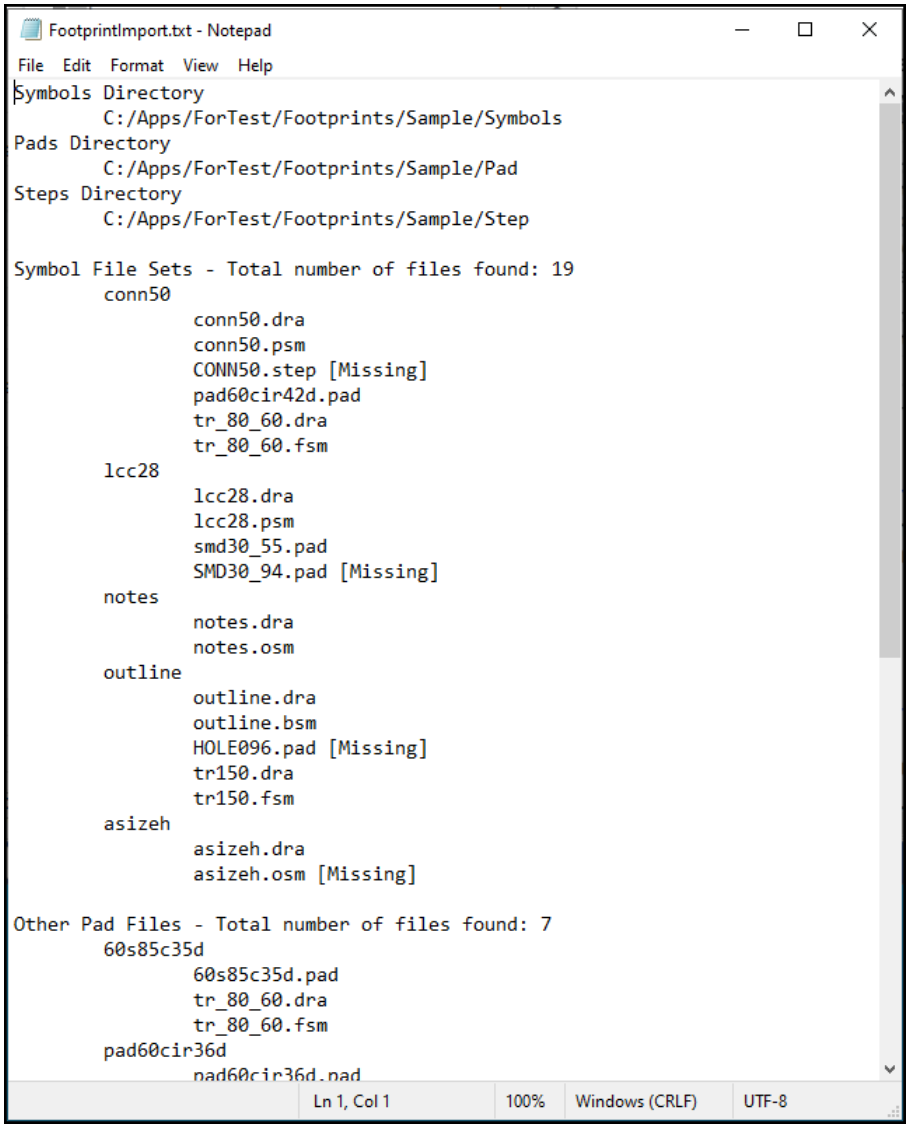
A sample of files that will not be imported is shown in the following figure.



Sample of Files That Will Not Be Imported

Export relationships

You may export the information in the Import Footprint Form to a File for offline review. Select the **Export to File** button to review the listing in notepad. An example of the export is shown in the following figure.



Sample Export of Import Footprint Form

Import Footprint Files

After review of the footprint import analysis, you may select files for import by individual files, sets or an entire group. Make your selection by selecting checkboxes for sets of files you want to import. Enter a comment for a history record of your import and select the **Import** ☑ Button.

You will receive an **Import Complete** confirmation at completion.

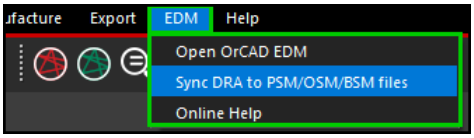


5.7.1.1 Sync DRA to PSM files

Before you begin to import or check in footprints, be sure all footprint files are at the same version (e.g. 24.1) as the version of PCB Editor that you are using. You must also set the version from EDM Desktop's **Settings > Set Cadence Version** menu**.**

Bulk conversion of all footprint files to the same PCB Editor format version can be achieved using *PCB Editor DBDoctor*. *DB Doctor* converts all .dra files independently from .psm files. If you use *DB Doctor* to convert PCB format version, you will also need to sync your .dra files to the .psm file.

To ensure reporting functionality for the manage board provides valid version ID information, the DRA and PSM files must be in sync. By default, the PSM file is automatically created when saving the DRA file in PCB Editor. However, if you have changed your settings from the default, your DRA files may be out of sync from the PSM files. Use of *PCB Editor DBDoctor* to uprev your footprint files will cause files to be out of sync. To sync these 2 files types in bulk, you can select the **Sync DRA to PSM/OSM/BSM files** menu option from the **PCB Editor** > **EDM** menu to select the directory location of your footprint files. Each DRA file in your selected folder will be automatically opened and saved. You will be prompted to verify if you want to continue and overwrite files with the updated version.



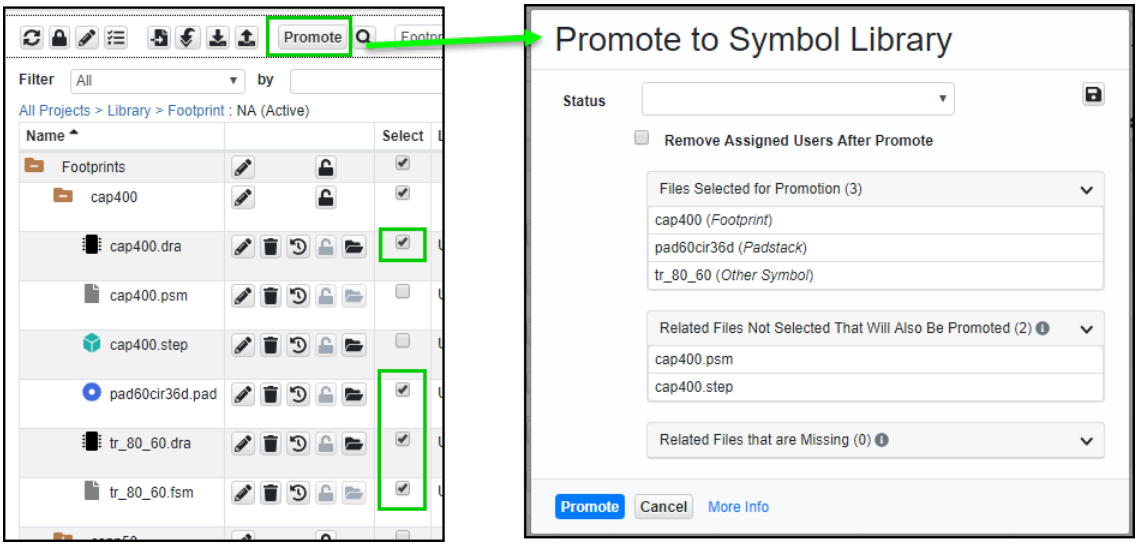
Uprev and Sync DRA files to PSM files

5.7.2 Promote Footprints and Step Models

Footprints imported or checked into the footprint projects should only be used on board files after they have been validated and promoted to a central location. When a footprint is promoted, all files related to the footprint are copied from the vault to a configured promoted location.

Only users with Promote Libraries permission may promote footprint and step model files from a footprint project. You may select file for promotion from any footprint project view.

To promote a footprint when using the Footprint View, select the checkbox next to the footprint you want to promote. When the File View or other view are used, you may select checkboxes next to the files you want to promote. Then select the **Promote** Promote button located in the Footprint Project Action Buttons. An error message shows if promoted location hasn't been configured or if there are any other errors encountered during promotion.



Sample Selection with Footprint Promote Summary Pop up

Promote to Symbol Library

The **Promote to Symbol Library** summary pop up shows your selected and related files to be promoted. The information in this window consists of the following categories. A count for the number of items in each category is shown along with the section title.

- **Files Selected for Promotion** - shows the items with types selected. Types match folders shown in the Footprint view: Footprint, Mechanical, Template, Padstack, Step and other Symbol. When both draw and compiled graphic files are selected, it is only shown once in this section. When only one of the draw or compiled graphic has been checked in and selected, the corresponding file will appear in the **Related Files that are Missing** section of this summary pop up.
- **Related Files Not Selected That Will Also Be Promoted** - shows all files by file name that have not been selected but will also be promoted. Files shown in this list will follow setting made in the Rules for Footprint Promote and Delete. The list will show all selected and related files when the rule is configured as Always promote selected and related files. The Sample Promote Footprint Pop Up figure shown above uses the rule to Only promote newer versions of selected and related files. As a result, tr_80_60.dra and tr_80_60.fsm are not shown in the pop up.
- **Related Files that are Missing** - shows all files where the Footprint project has an identified relationship but the file hasn't been checked in. Dependent on the missing file, you may encounter issues when attempting to place components onto a board. For example, if the .dra file for a footprint is available and the corresponding .psm is missing, the component cannot place on a board. However, if the .psm is available, you will not be able to edit the footprint even though you can place the component on a board.

You may export the lists shown in this window for offline review by selecting the **Export to File** button.

To promote files after reviewing the list shown:

1. Enter or select a Status for the files after promote. This will be updated only for files that are promoted (based on Rules for Footprint Promote and Delete in Configuration)

2. Select the the **Remove Assigned Users After Promote** checkbox if you no longer want users to modify the footprint after it has been promoted.
3. Click the **Promote** Promote button.

A confirmation shows after files have been promoted successfully. The Project Detail fields: Status, Last Promote Date, Last Promoted Version, and Last Promoted By for the selected symbols update to reflect your status selection as well as the current date and latest version of the checked in symbols.

The following figures show files in a footprint project after promotion. You can review all footprint files that have been promoted by using the Promote Status from the Search Filter of the Footprint Project View or by selecting the View Promoted Library button in the Footprint Project Action Buttons.

Name	Select	# Pins	Last Promote Date	Last Promoted Version	Last Promoted By	Status	Version
Footprints							
cap400		2				4-Production	
cap400.dra		2	7/31/2021 12:38:25 AM	2	Alice	4-Production	2
cap400.psm			7/31/2021 12:38:25 AM	2	Alice	4-Production	2
cap400.step			7/31/2021 12:38:25 AM	2	Alice	4-Production	2
pad60cir36d.pad			7/31/2021 12:38:25 AM	2	Alice	4-Production	2
tr_80_60.dra			7/31/2021 12:38:25 AM	4	Alice	4-Production	4
tr_80_60.fsm			7/31/2021 12:38:25 AM	4	Alice	4-Production	4

Example of Promoted Footprints in a Footprint Project

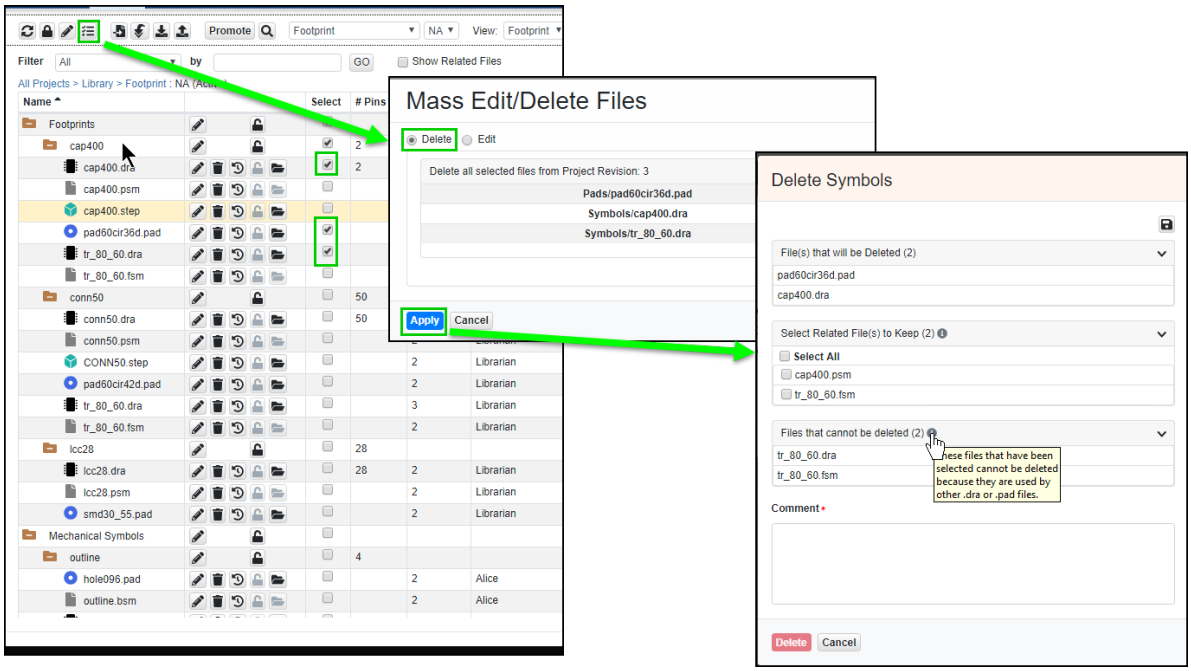
5.7.3 Delete Footprint and Step Model Files

When deleting footprint files, EDM verifies that files selected for deletion are no longer used in other footprints or padstacks before footprint files are deleted from the vault. Only assigned leaders to a project can delete footprint files. When files are deleted from the vault, they also be deleted from the promoted footprint location. The settings in Rules for Footprint Promote and Delete configures the option to include/exclude deletion from the promoted footprint location.

You can select multiple files for deletion or select the delete button to delete files one-at-a time. With either method, EDM will determine whether selected files may be deleted and show options in a **Delete Symbols** or **Delete File** pop up.

To delete multiple files, select the checkboxes next to the files you want to delete. Selecting the names for groups (e.g. footprint, padstack, etc) will select all files within the group. Then select the **Mass Edit** button located in the Footprint Project Action Buttons.

The Mass Edit/Delete Files form shows. Select the delete radio button and click apply to open the Delete Symbols pop up. The following figure illustrates the sequence to open the Delete Symbols pop up window.



Sample File Selections With Footprint Delete Pop Up

Delete Symbols Pop up

The **Delete Symbols** pop up window shows your selected and related files to be deleted. The window also shows files that cannot be deleted and a confirmation for files to keep. The information in this window consists of the following categories. A count for the number of items in each category is shown along with the section title.

- **File(s) that will be Deleted** - shows only filenames of items selected and can be deleted.
- **Select Related Files to Keep** - shows filenames of files that have not been selected for delete but are related to the selected files. Keeping these may affect editing of files or laying out components on boards. You may have reasons for keeping these files. You may select the checkboxes next to any of the files you want to retain.
- **Files that cannot be deleted** - shows files that have been selected but are still in use by other footprints or padstacks.

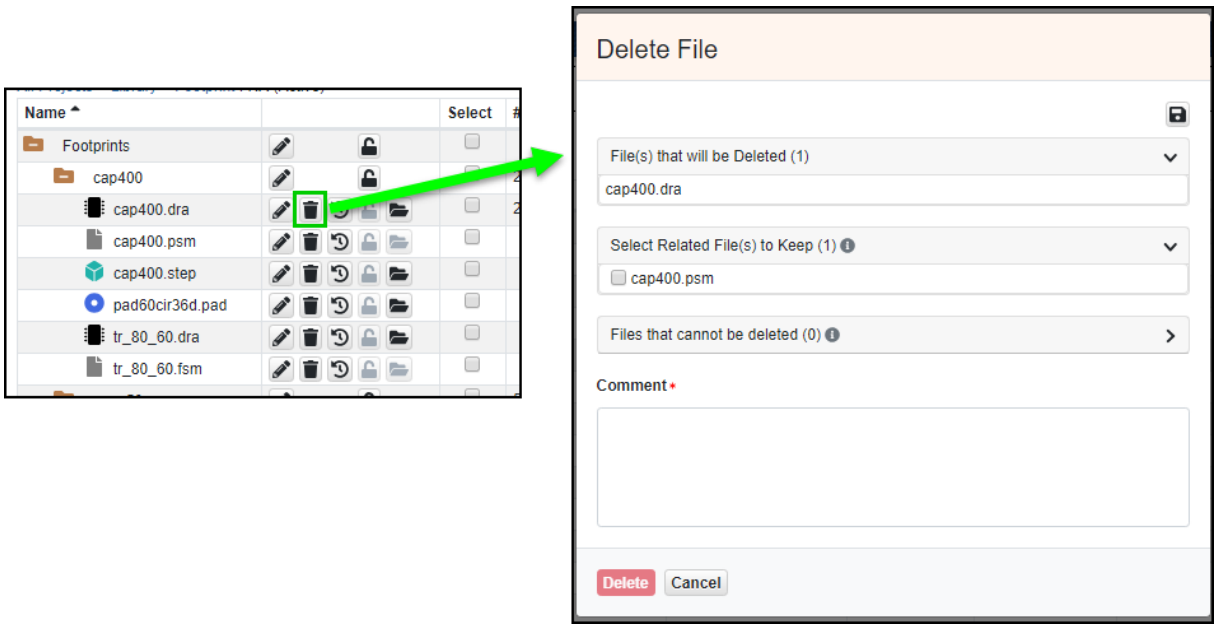
You may export the lists shown in this window for offline review by selecting the **Export to File** button.

To delete files after reviewing the list shown:

1. Review the list to make sure you have selected the correct files.
2. Select checkboxes for related files you want to keep
3. Enter a comment for a record in the history.
4. Click the **Delete** Delete button. Files in the promoted footprint location will be deleted also if configured in Rules for Footprint Promote and Delete.

Delete File Pop up

The **Delete File** pop up window shows the same information as the **Delete Symbols** pop up. The **Delete File** pop up opens immediately after selecting the **Delete** button next to a file. The following figure show a sample **Delete File** pop up.



After delete, the project view updates showing that deleted files are no longer in the view. You may select the **View Promoted Library** button in the Footprint Project Action Buttons to verify that the selected files have also been removed from the promoted location.

The deleted files may be recovered by reviewing the project history. Use the **History** button on the project folder to open the history flyout.

5.8 Manage Team Design Folders

Folders that are added as a Team folder allow multiple engineers to work on a single OrCAD Capture design file in a team environment.

Assigned leaders for a given project can manage the Team Design folder from the Team Folder Details View. When initially created, a new Team Folder is blank, without pages associated to the folder.

Assigned leader may perform any of the following tasks when managing a Team folder:

- Add schematics and blank pages to make user assignments
- Import pages from an existing design
- Edit schematics and pages to change status, properties and user assignments
- Move pages between groups/schematics
- Delete a schematic or group
- Manage Reference Designator consistency

- Freeze the design to maintain information that cannot be restore when after design pages are split and merge.

Team Design Folder functionality is only available when you open EDM from Capture.

5.8.1 Add New Pages to a Flat Design Folder

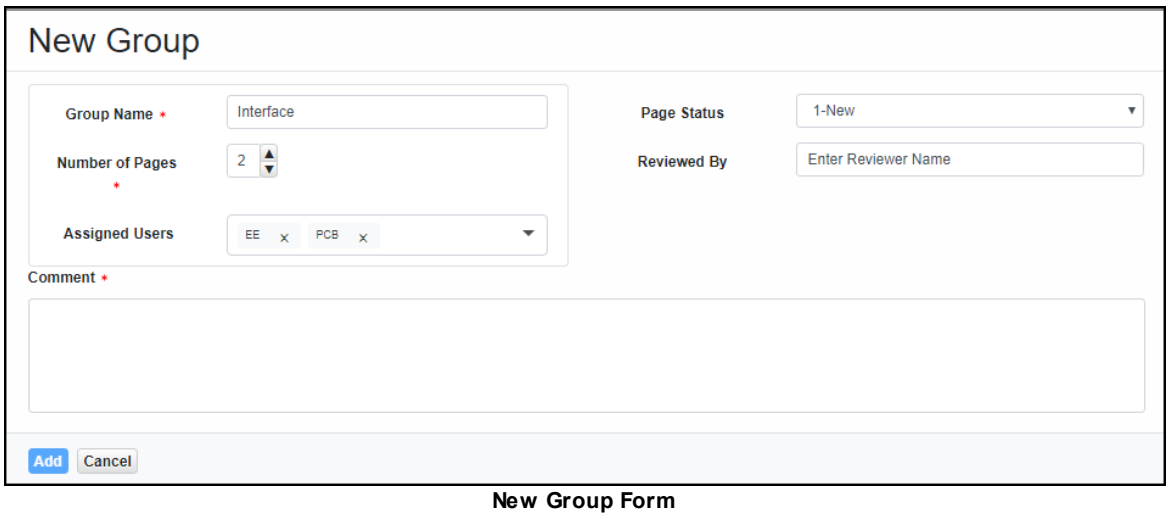
New pages or a group of pages can be added by an assigned leader or checked in by users. New pages in Team Design are created using settings in your design templates configured in OrCAD Capture. Creating blank pages with EDM enables the assigned leader to assign specific pages to users, as well as control the template in which the pages are created. Pages in a flat design after download merge to a single schematic that is opened in Capture. Within EDM Team Design view, a project leader may group pages to help with management of pages in a design.

New pages can only be added to groups in Team Design. When there are no pages in a Team Design, a group must be added as part of creating new pages. The Team Design Action Buttons located at the top of the Team Design View are used to create pages. There are three ways to add pages.

- Add new pages using Add Group
- Add a page to an existing group
- Add pages by editing a group

Add Group

The **Add Group** button opens the **New Group** form.



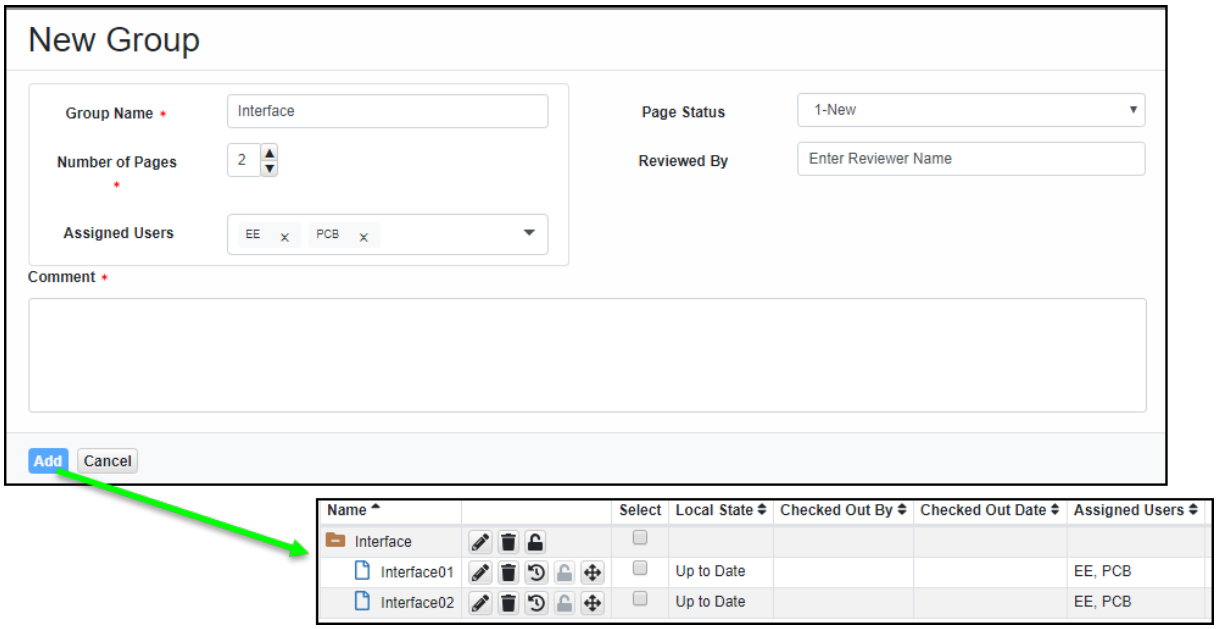
New Group Form

Fields available in the **New Group** form include:

- **Group Name** - name for the group of pages to be created. The group name must be unique.
- **Number of Pages** - number of blank pages to be created for this group. The page names will consist of the group name followed by auto-numbered pages. The group name and page number will be separated by a configured delimiter.

- **Assigned Users** - users or user groups assigned to work on the pages. The list of available users will be the list of users or user groups assigned to the project.
- **Page Status** - values for the status for the page to be created. If the status is configured as a pick list, you may select the value from the pick list. If a default status is configured, it pre-populates the form field.
- **User Fields -** Update these fields as needed. The **Reviewed By** field in example above is a configured user field.
- **Comment** - entered comment for the creation of the pages. Comments are saved in history for the project.

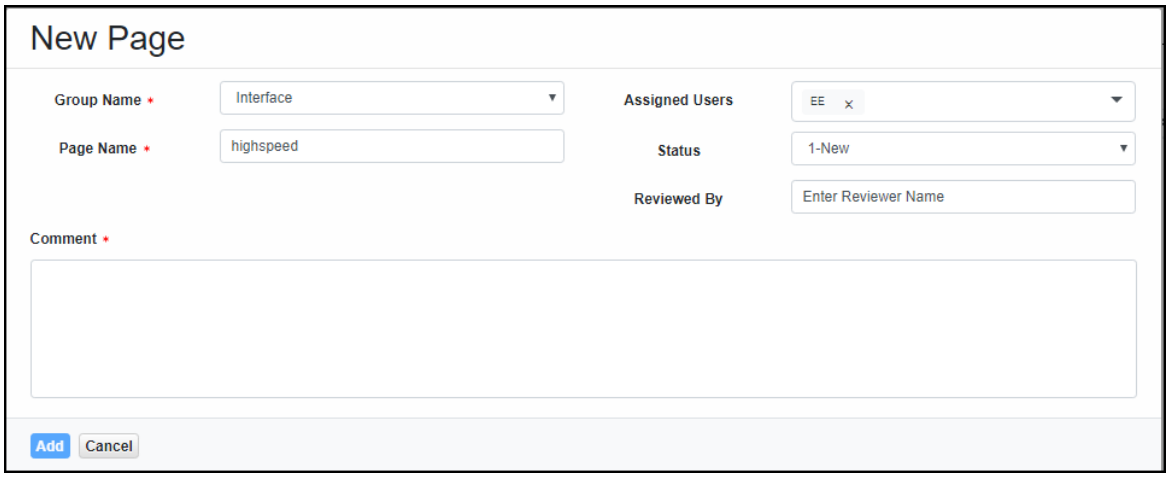
The following sample figure shows a Team Design with 2 new pages, Interface1 and Interface2 added to the Interface group.



Sample of Add Group

Add Page

To add a page to an existing group, click the **Add Page** button to open the **New Page** form.

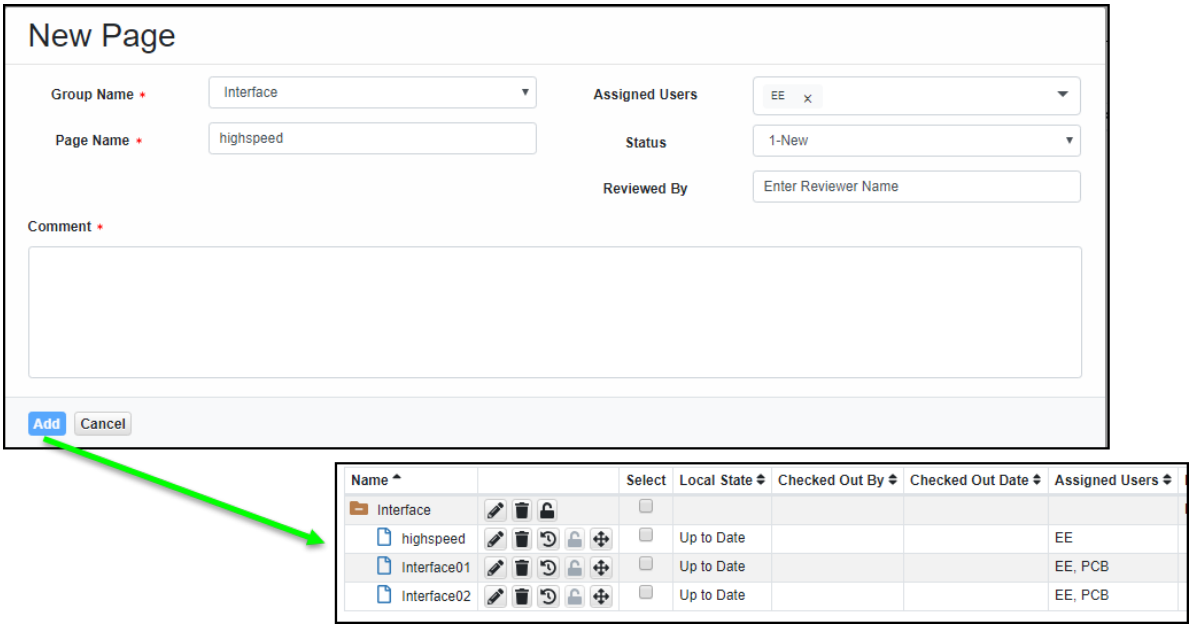


Add New Page to Group

Fields available in the **New Page** form include:

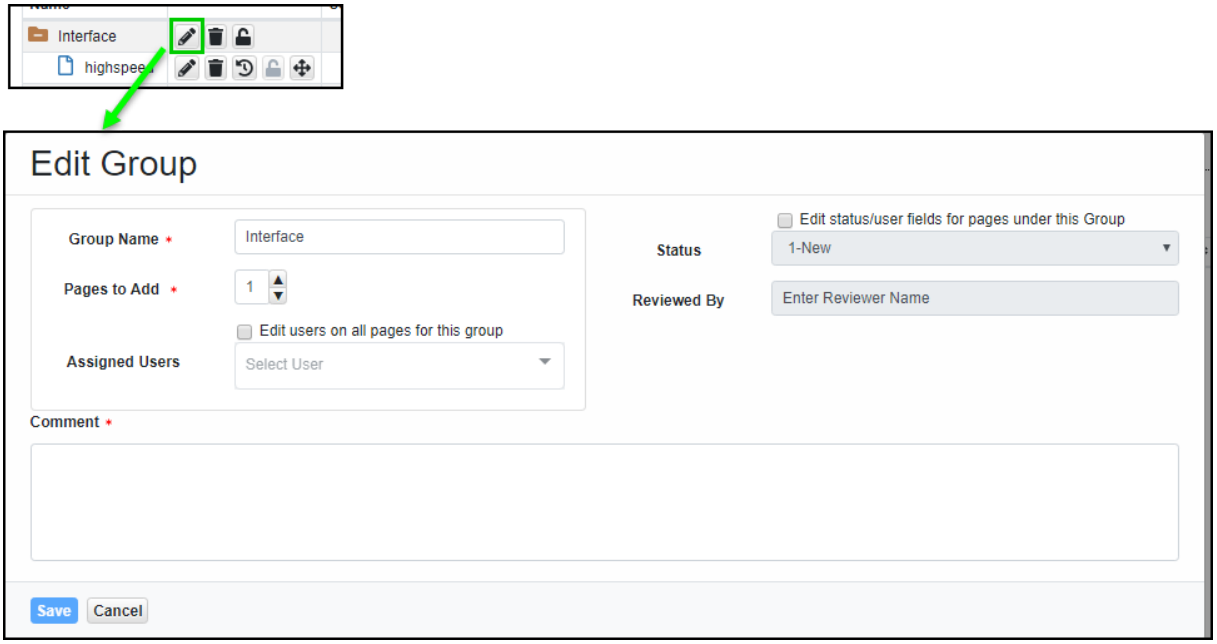
- **Group Name** - drop list containing all groups created for the Team Design project.
- **Page Name** - enter a page name using character set allowable by Capture for page names.
- **Assigned Users** - users or user groups assigned to work on the pages. The list of available users will be the list of users or user groups assigned to the project.
- **Status** - values for the status for the page to be created. If the status is configured as a pick list, you may select the value from the pick list. If a default status is configured, it pre-populates the form field.
- **User Fields -** Update these fields as needed. The **Reviewed By** field in example above is a configured user field.
- **Comment** - entered comment for the creation of the pages. Comments are saved in history for the project.

The following sample figure shows a new pages, highspeed added to the Interface group that has 2 pages.



Add Page By Editing Group

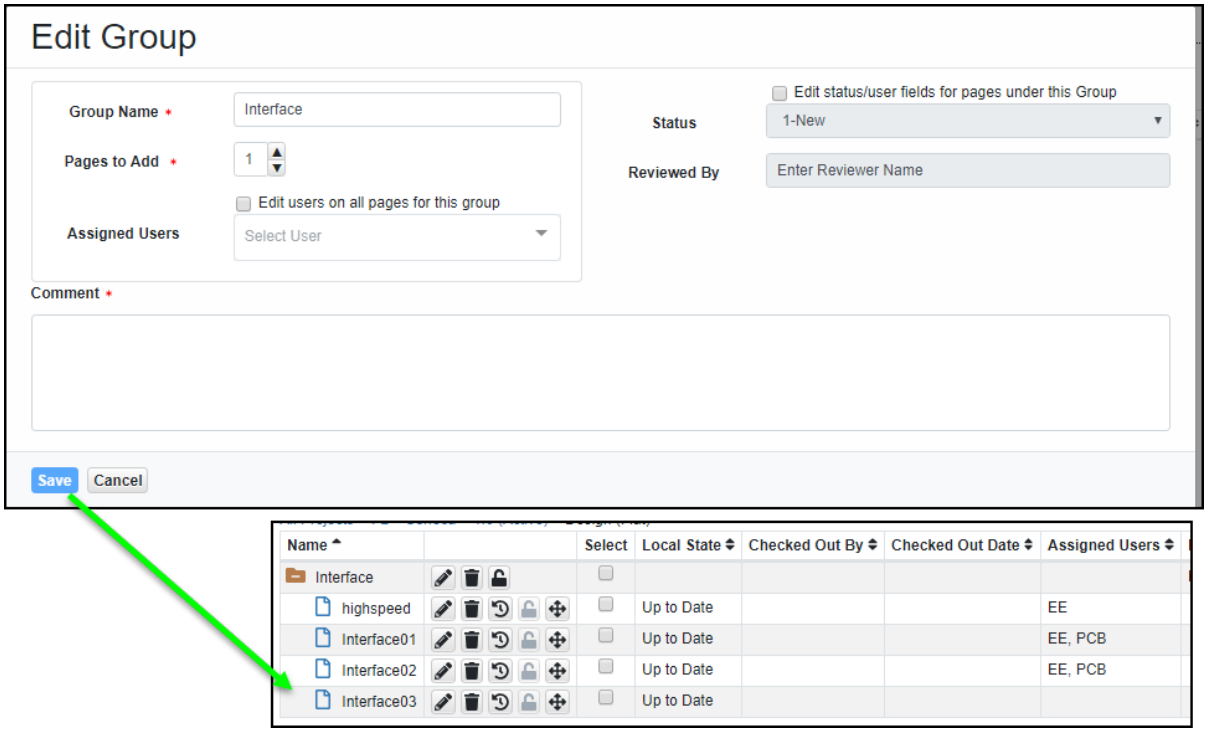
A new page may also be added by editing an existing group. Click on the **Edit** button for the group that you want to add a page. The **Edit** Group form appears.



A description of the fields available in the **Edit Group** form are provided in the section for Edit Schematic/Group and Page Properties. To add pages to the selected group, keep all fields the same and only change the **Pages to Add** field.

- **Pages to Add** - number of blank pages to be added for this group. The page names will consist of the group name (new group name if changed) followed by auto-numbered pages. The group name and page number will be separated by a configured delimiter.

The following sample figure shows an additional page added to the existing Interface group.



5.8.2 Add New Schematics and Pages to Hierarchical Design Folder

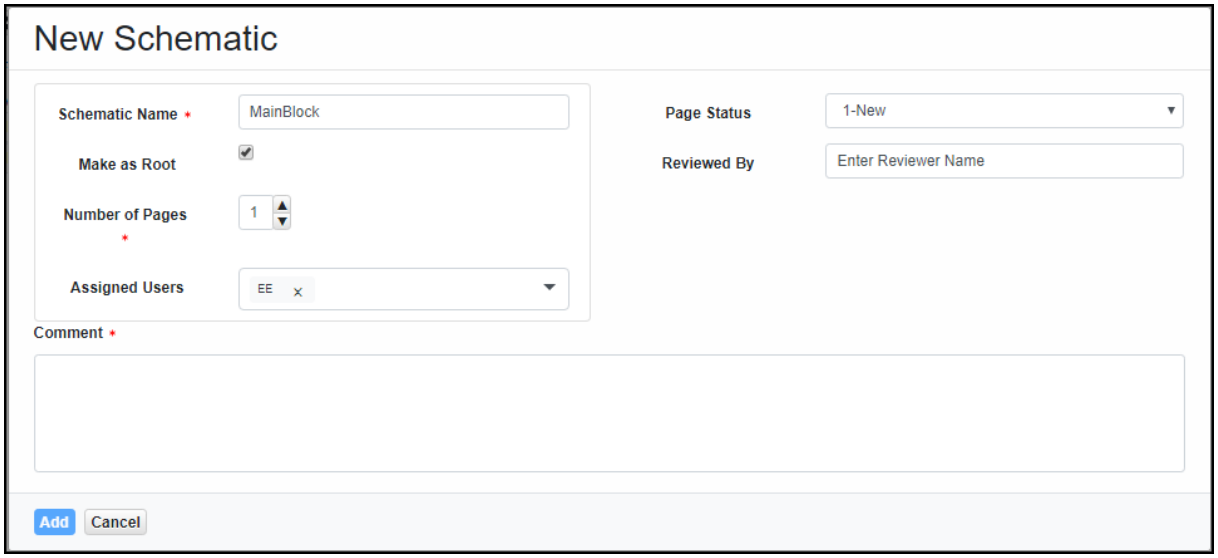
New schematics and pages can be added by an assigned leader or checked in by users. Schematics and pages added to the Team Design folder will download and merge to your design file. The hierarchy will show when you add hierarchical blocks to your design using the new schematics. Refer to hierarchical designs for instructions on how to ensure your hierarchical EDM projects maintain correct hierarchy after check out. The EDM Team Design view will mirror the hierarchical structure of the Capture design.

New pages can only be added to Schematics in Team Design. When there are no pages in a Team Design, a Schematic must be added as part of creating new pages. The Team Design Action Buttons located at the top of the Team Design View are used to create pages and schematics. There are three ways to add pages.

- Add new pages using Add Schematic
- Add a page to an existing schematic
- Add pages by editing a group

Add Schematic

The **Add Schematic** button opens the **New Schematic** form.

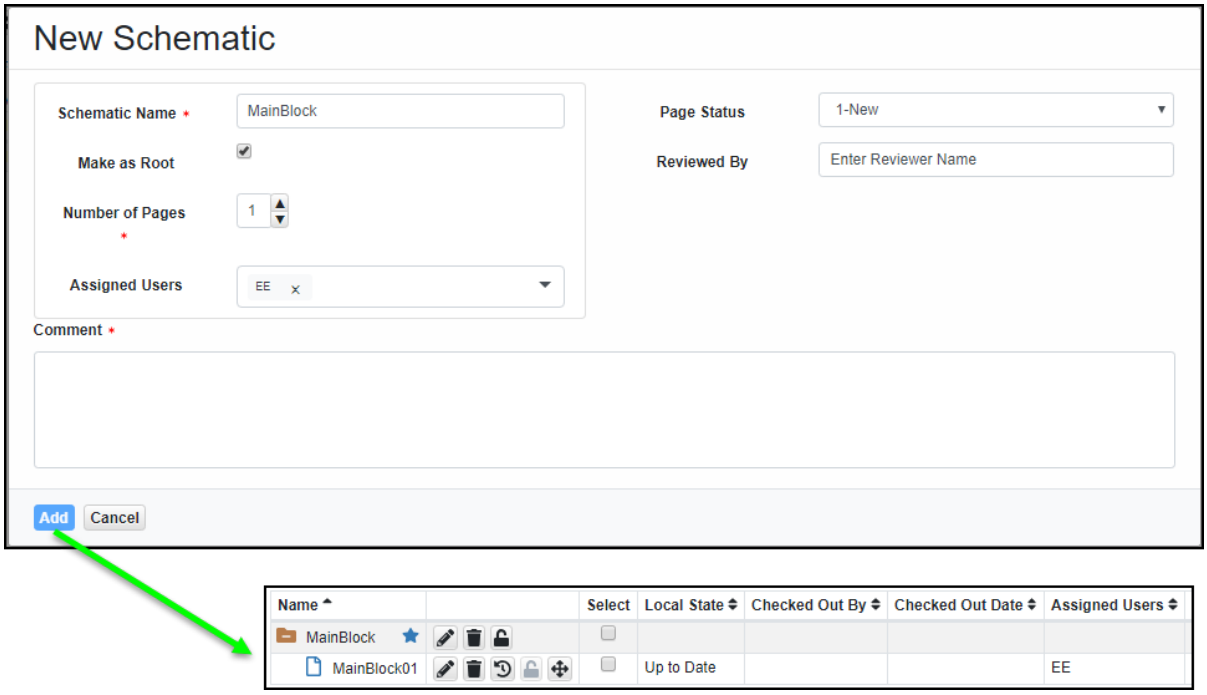


New Schematic Form

Fields available in the **New Schematic** form include:

- **Schematic Name** - name for the schematic that new pages are be created. The schematic name must be unique.
- **Make as Root** - selection to make the new schematic the Root schematic. The root is the schematic that is at the top of the hierarchy.
- **Number of Pages** - number of blank pages to create for this schematic. The page names will consist of the schematic name followed by auto-numbered pages. The schematic name and page number will be separated by a configured delimiter.
- **Assigned Users** - users or user groups assigned to work on the pages. The list of available users will be the list of users or user groups assigned to the project.
- **Page Status** - values for the status for the page to be created. If the status is configured as a pick list, you may select the value from the pick list. If a default status is configured, it pre-populates the form field.
- **User Fields -** Update these fields as needed. The **Reviewed By** field in example above is a configured user field.
- **Comment** - entered comment for the creation of the pages. Comments are saved in history for the project.

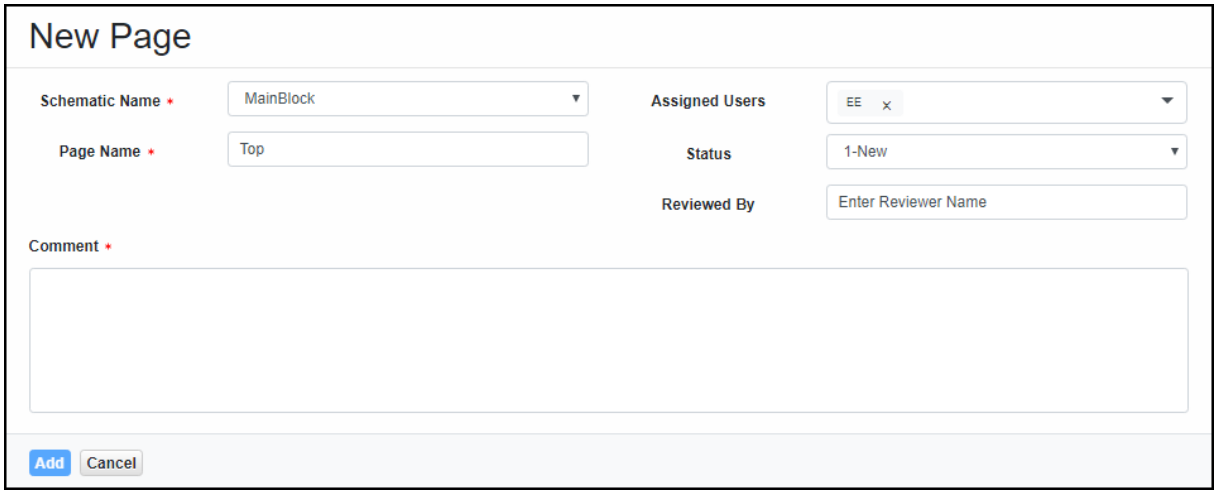
The following sample figure shows a Team Design with 1 new page, MainBlock1 added to the MainBlock schematic.



Example of Adding a Root Schematic

Add Page

To add a page to an existing group, click the **Add Page** button to open the **New Page** form.



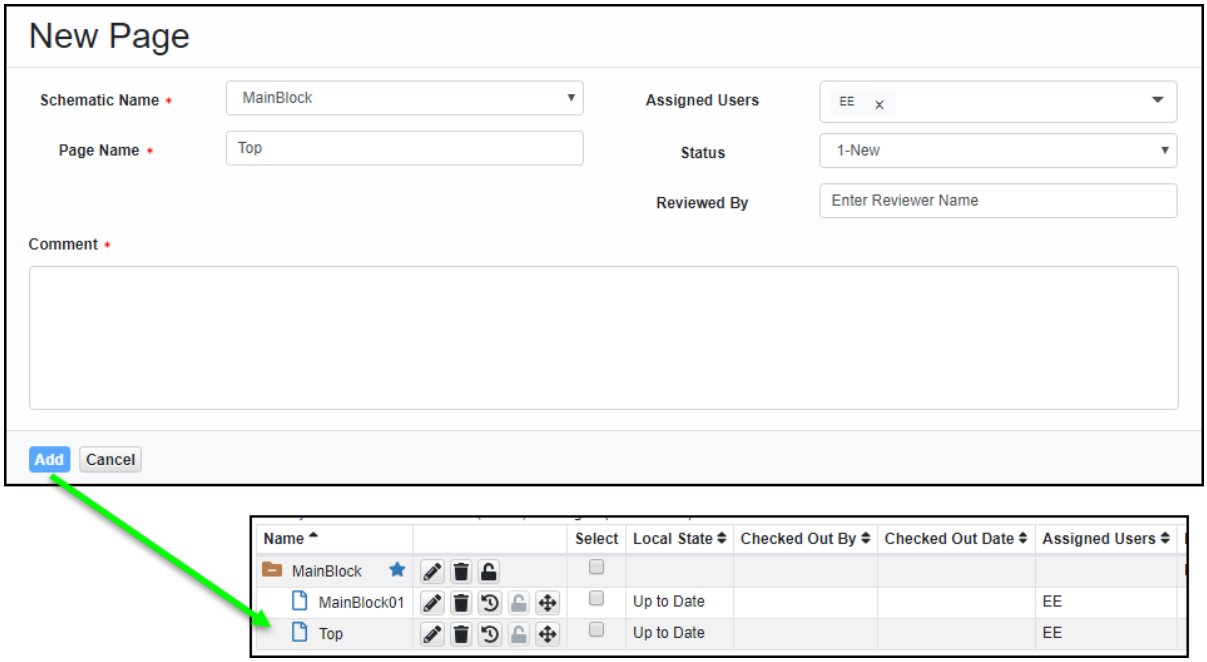
New Page Form for Schematic

Fields available in the **New Page** form include:

- **Schematic Name** - drop list containing all schematics in the Team Design.
- **Page Name** - enter a page name using character set allowable by Capture for page names.
- **Assigned Users** - users or user groups assigned to work on the pages. The list of available users will be the list of users or user groups assigned to the project.

- **Status** - values for the status for the page to be created. If the status is configured as a pick list, you may select the value from the pick list. If a default status is configured, it pre-populates the form field.
- **User Fields -** Update these fields as needed. The **Reviewed By** field in example above is a configured user field.
- **Comment** - entered comment for the creation of the pages. Comments are saved in history for the project.

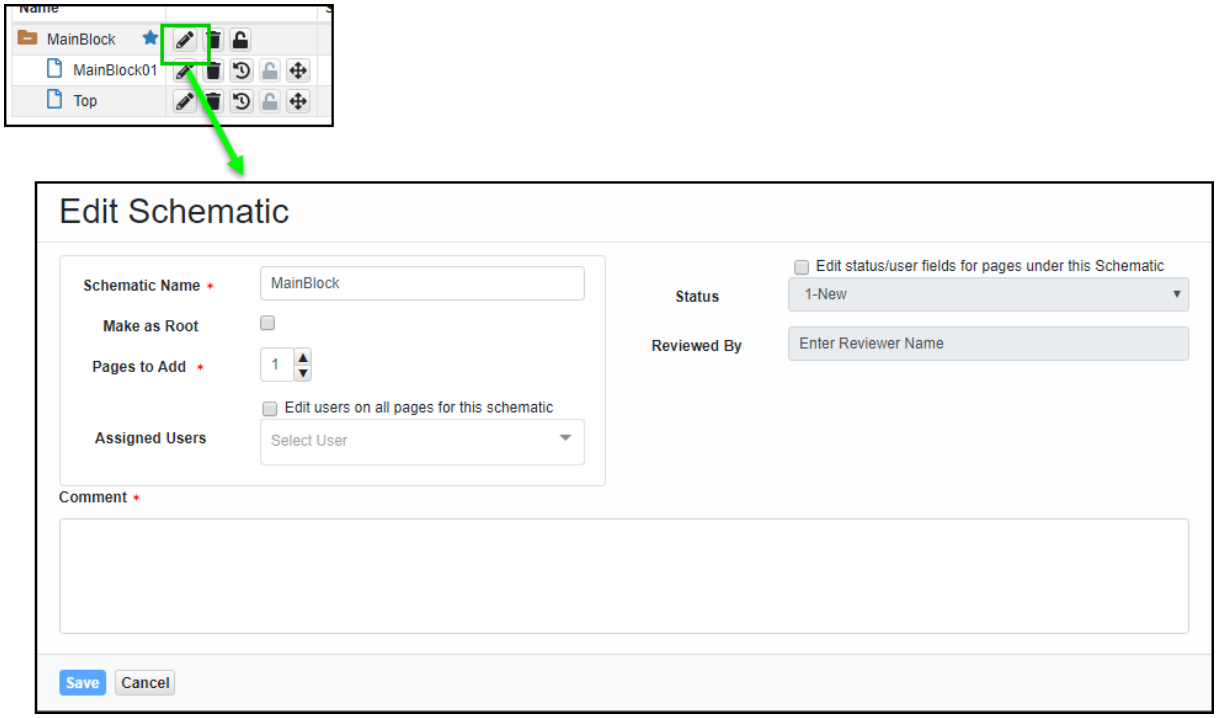
The following sample figure shows a new pages, Top added to the MainBlock schematic that has 1 page.



Example of Adding Page for Hierarchical Design

Add Page By Editing Schematic

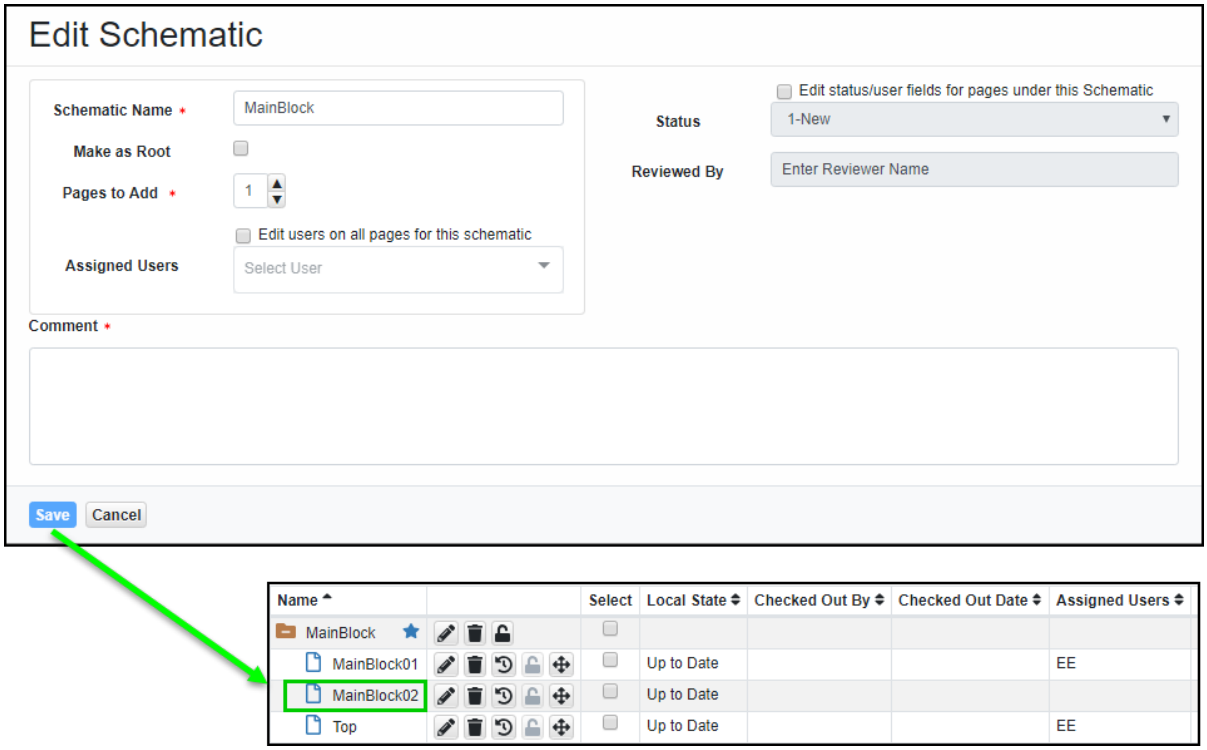
A new page may also be added by editing an existing schematic. Click on the **Edit** button for the schematic that you want to add a page. The **Edit Schematic** form appears.



A description of the fields available in the **Edit Schematic** form are provided in the section for Edit Schematic/Group and Page Properties. To add pages to the selected schematic, keep all fields the same and only change the Page to Add field.

- **Pages to Add** - number of blank pages to be added for this schematic. The page names will consist of the schematic name (new schematic name if changed) followed by auto-numbered pages. The schematic name and page number will be separated by a configured delimiter.

The following sample figure shows an additional page added to the existing MainBlock schematic.



Example of Adding A Page By Editing Schematic

5.8.3 Import Schematics and Pages

Schematic pages from an existing design can be imported into the team design folder. Both flat or hierarchical design files can be imported into corresponding team design folders.

For hierarchical team folders:

- The schematic names in the selected design file will import directly into the folder.
- All schematics and pages in the selected design file will display in the **Import Data** of the **Import Hierarchical Design** form.

For flat designs:

- All pages in the selected design file are imported to the selected group.
- Only the main schematic and pages in the selected design file will display in the **Import Data** of the **Import Flat Design** form

To Import a design

1. Click the **Import** button that is located with the Team Design Action Buttons in the upper left corner of the Team Design Details View.



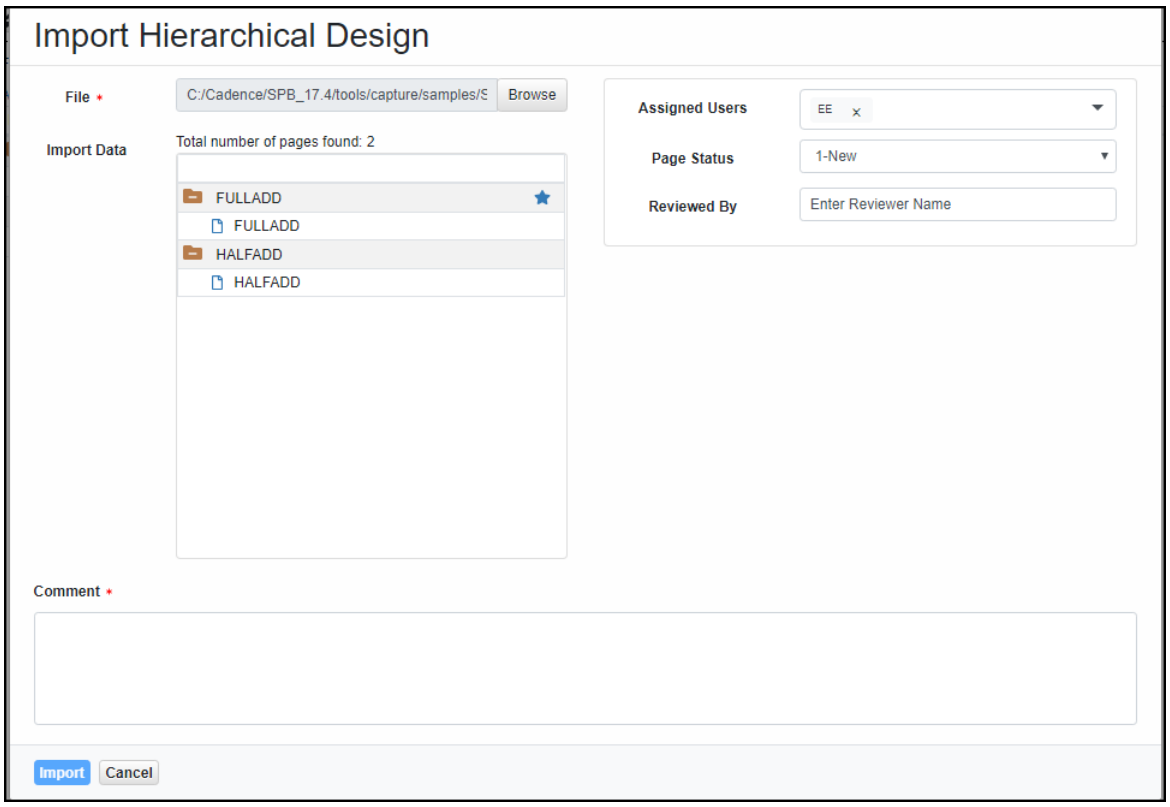
Project Leader and User Option Buttons For Hierarchical Designs



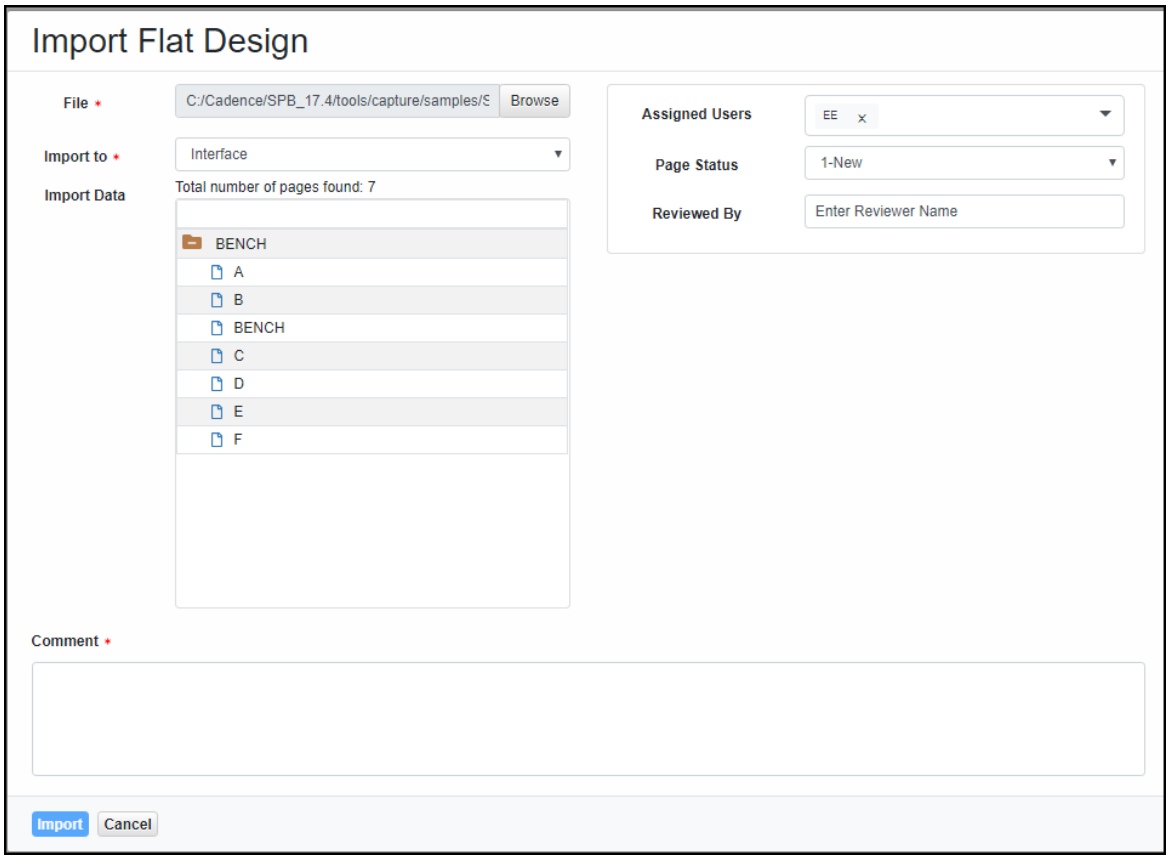
Project Leader and User Option Buttons For Flat Designs

The file selector display for you to select the DSN file for import.

2. Once the file has been selected, depending on whether you are importing to a Hierarchical Team Folder or Flat Team Folder, the corresponding **Import Hierarchical Design** or **Import Flat Design** form displays. The two screen captures that follow show an example of the Hierarchical and Flat Import Design forms:



Import Hierarchical Design Form



Import Flat Design Form

The page names of the selected .DSN file are shown in the **Import Data** section of the screen. If the file you selected is corrupted or is an incorrect format, the system will be unable to locate the page(s) and you will be unable to import the file.

3. Enter values into the form input fields. Both forms contain the following fields:
   - **File** - path and design file name to be imported. If this is not the file you want to import, click the **Browse** button to select another file.
   - **Import to** - required for flat designs only. You may select the group to import to. The schematic name from the original file will also be an import to option.
   - **Assigned Users** - users or user groups assigned to work on the pages. The list of available users will be the list of users and user groups assigned to the project.
   - **Page Status** - values for the status for the page to be created. If the status is configured as a pick list, you may select the value from the pick list. If a default status is configured, it pre-populates the form field.
   - **User Fields -** Update these fields as needed. The **Reviewed By** field in example above is a configured user field.
   - **Comment** - comment for the import of the pages. Comments are saved in history for the project.
4. When finished, click the **Import**  button.

Note: Pages in the import file that are the same as existing pages in the team design must be use the check out and check in process to update. You will receive *Unable to Import* messages when you try to import pages that exist in the team folder.

5.8.4 Edit Schematic/Group and Page properties

Within a Team Design folder, you may change the following:

- Page Names
- Move page between groups
- Page Assignments (by page, by schematic or all pages in a single action)
- Schematic Name
- Root Schematic (for Hierarchical designs only)
- Page status and user field values

There are three ways to start editing.

1. Using the **Edit** button that is located with the Team Design Action Buttons in the upper left corner of the Team Design Details View. This opens the **Edit Folder** form that enables you to change assignments of all pages within a team design only.
2. Using inline **Edit** button next to a page, schematic or group. The Edit button next to a page opens a Edit Page form. The Edit Button next to Schematic or group opens either the Edit Schematic or Edit Group form.
3. Using inline **Move** button to change the group or schematic that a page belongs.

Edit Team Folder Assignments for All Pages

To change assignment for all pages in the Team Design Folder:

1. Click the **Edit** button located with the Team Design Action Buttons.

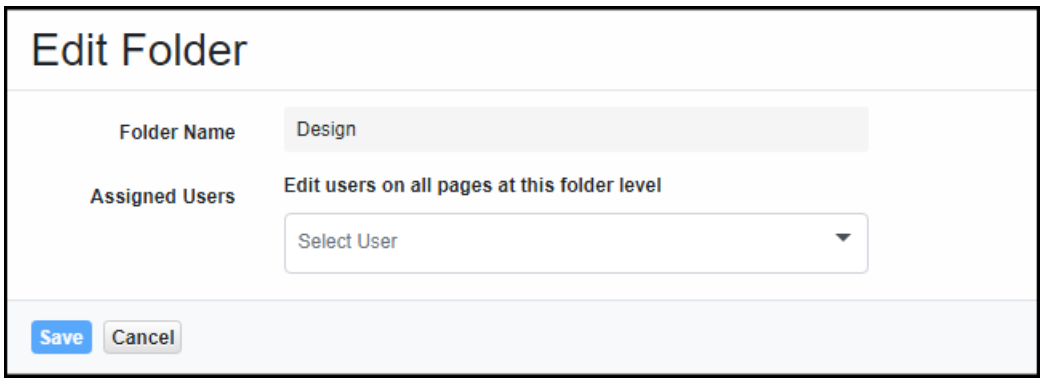


Project Leader and User Option Buttons For Hierarchical Designs



Project Leader and User Option Buttons For Flat Designs

The **Edit Folder** form opens.

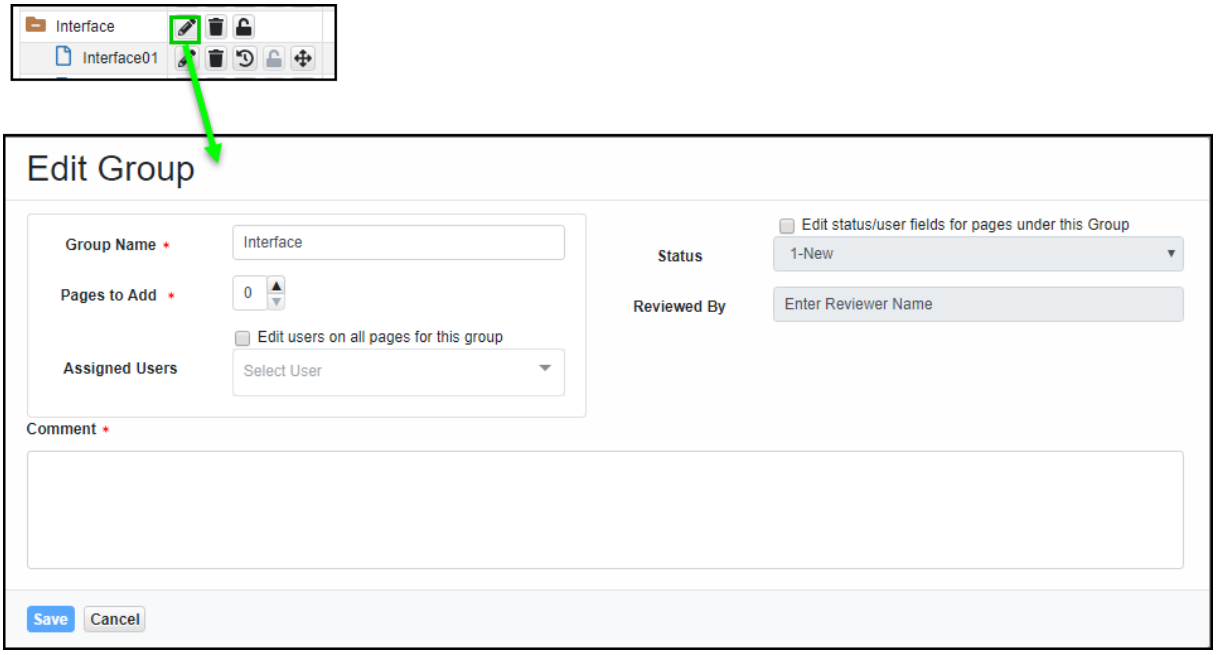


2. Select the user or group you want for the new assignment.

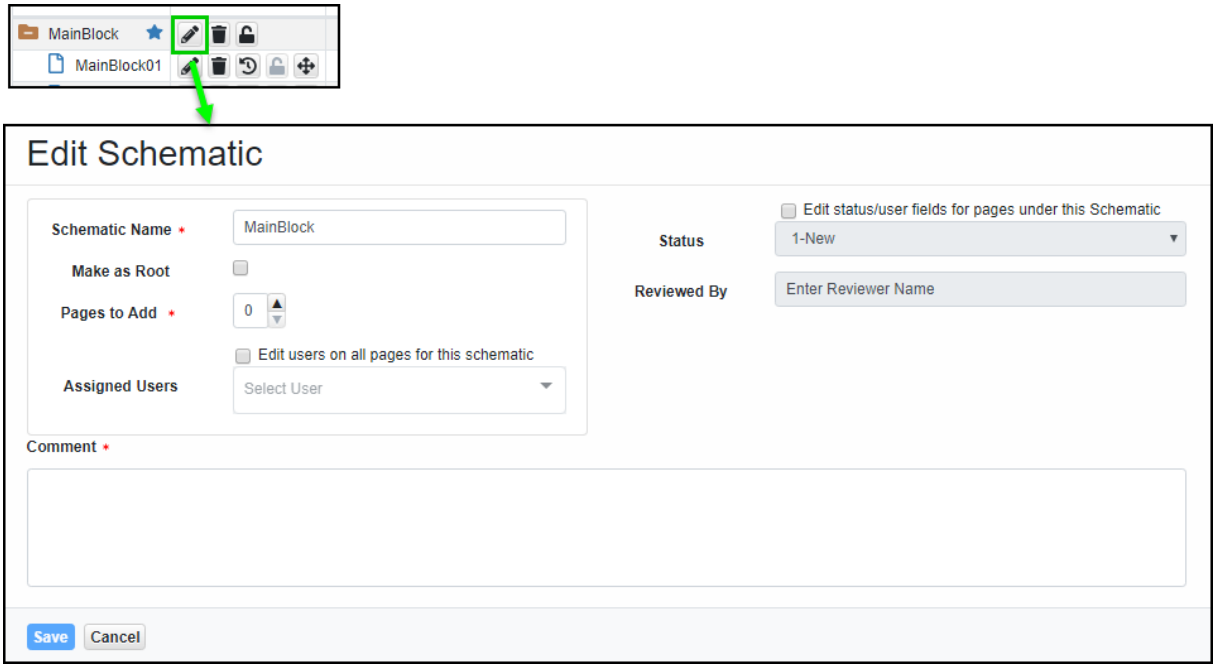
3. When complete, click the **Save** Save button, to accept the changes. Or to cancel your changes, click the **Cancel** Cancel button.

Edit Schematic/Group Properties

The **Edit Group** form provides input fields for you to edit the properties of a flat design. The **Edit Schematic** form provides input fields for you to edit the properties of a hierarchical design. The **Edit Group** and **Edit Schematic** forms are shown in the following two figures.



Edit Group Form



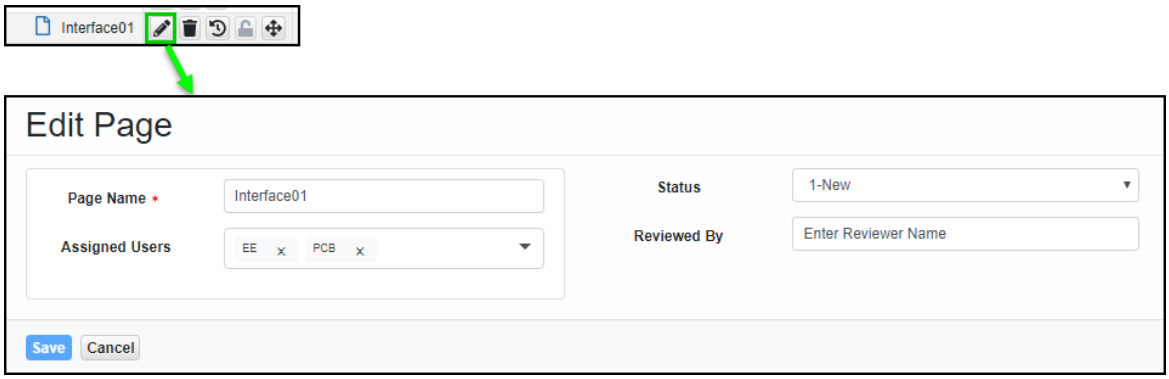
Edit Schematic Form

Fields that are available to edit in these two forms are identified in the following list.

- **Group Name** - Group name may be changed in the **Edit Group** form for a Flat Design. Keep this field the same to add a new page to the selected group. Changing the group name changes the name of the group for existing pages. Changes to group name have no affect on the downloaded design file.
- **Schematic Name** - Schematic name may be changed in the **Edit Schematic** form for a Hierarchical Design. Keep this field the same to add a new page to the selected schematic. Changing the schematic name will change the name of the schematic for existing pages.
  - **Note**: If you change a Schematic name, anyone who downloaded the team design prior to the name change and downloads the team design after will see both schematics. EDM will not remove any schematics or pages in a user's local design, since there is no way to differentiate new pages added (and awaiting check in) from pages that no longer belong in the design file.
- **Make as Root** - Selection to make the edited schematic the Root schematic for a Hierarchical Design. The root is the schematic that is at the top of the hierarchy. Changes to the Root will take effect when users perform a check out or a full download of the Team Design.
- **Pages to Add** - Number of blank pages to be added for this group or schematic. The page names will consist of the group or schematic name (new group/schematic name if changed) followed by auto-numbered pages. The group/schematic name and page number will be separated by a configured delimiter. Added pages will be merged to users' downloaded design when they perform a check out or a full download of the Team Design.
- **Assigned Users** - Assigned users may not be updated without selecting the checkbox for **Edit users on all pages for this schematic.** When the checkbox is selected, page assignments may be made and applies to all pages for the group or schematic. When the checkbox is not selected, new pages will be added with no user assignments. Click on the **Edit** button for the new page to change the user assignment of the new page different from the rest of the group.
- **Status and User Fields** - Values for the status and user fields for the added page. These may not be changed without selecting the checkbox for **Edit status/user fields for pages under this Schematic** or **Edit status/user fields for pages under this Group** checkbox**.** When the checkbox is selected, Status and user fields may be changed and applies for all pages. When the checkbox is not selected, new pages will be added without status or values to the user fields. Click on the **Edit** button for the new page to update status and user fields of the new page making it different from the rest of the group.
- **Comment** - entered comment for the creation of the pages. Comments are saved in history for the project.

Edit Page Properties

The **Edit Page** form allows you to change the page name, user assignments, status and user fields.



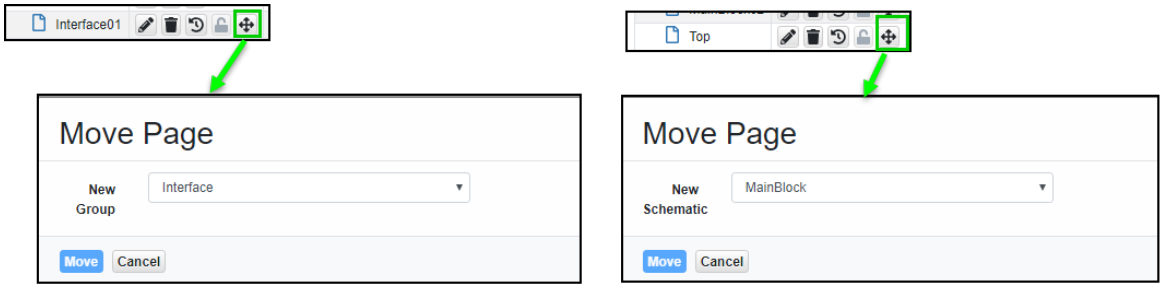
Sample Edit Page Form

The properties that can be changed are described below:

- **Name** - changes the page name. Changes to the page name is only made to the vault. Copies of pages in local workspaces will still have original name. Changing page names will require coordination outside of EDM to ensure that pages are not checked using the original name.
  - **Note**: If you change a page name, anyone who downloaded the team design prior to the name change and downloads the team design after will see both pages. EDM will not remove any pages in a user's local design, since there is no way to differentiate new pages added (and awaiting check in) from pages that no longer belong in the design file. When pages have been renamed by anyone, all other users should delete their local copy of the merged design to get a fresh copy of the design file.
- **Assigned Users** - changes the user or group assignment for the page.
- **Status** - updates the page status with a new status.
- **User fields** - updates configured user fields. The **Reviewed By** field in example above is a configured user field.

Move Page

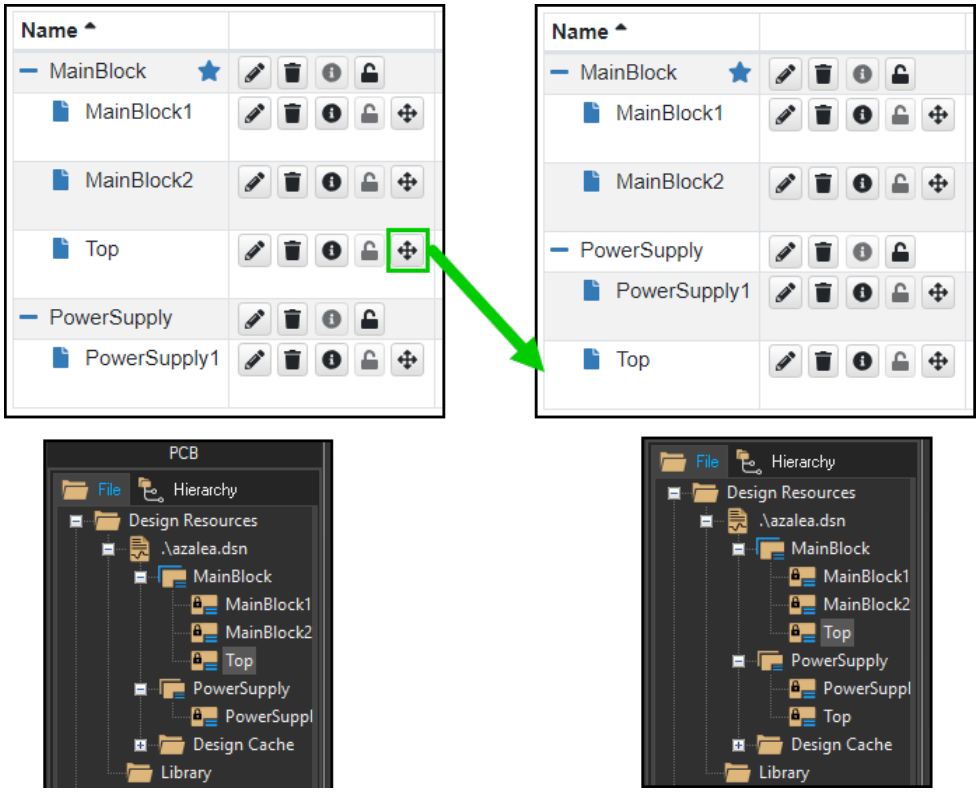
If a page was incorrectly placed into a group during check in you can move the page from one group to another. Click the **Move Page** button located next to the item you wish to move to open the Move Page pop-up.



Move Page Pop-Up for Flat and Hierarchical Designs

The drop-down list contains all groups or schematics in the design. Select the new group or schematic and click **Move**. If you move a page, anyone who downloaded the team design before the move and downloads the team design after will see the page in both Schematics. EDM will not remove any pages in a user's local design since there is no way to differentiate new pages added and awaiting checked in from pages that no longer belong in the design file.

The following figure shows an example of a page that is moved from one schematic to another.



Sample of Team Design and Local Design Before and After Move

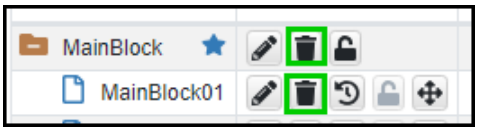
5.8.5 Delete Schematic/Group and Page

Design pages added to designs and checked in accidentally will need to be deleted from EDM to prevent them from being re-add during full download. Deleting pages or schematics only deletes it from the vault. EDM will not remove any schematics or pages in users' local designs since there is no way to differentiate new pages added and awaiting checked in from pages that no longer belong in the design file. The page or schematic in the merged design file of all users who have downloaded the design will remain. Users who have a copy of the page in their local workspace need to manually remove the page.

The inline **Delete** button next to a page, schematic or group allows you to remove a single or a group of pages.

To remove a page, schematic, or group:

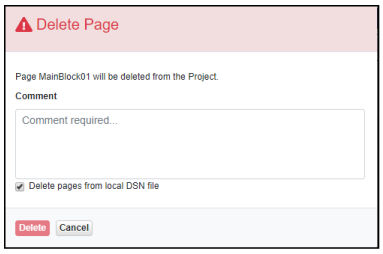
1. Click the **Delete** button that is located next to the page you want to remove. To remove a schematic or group, click the **Delete** button located next to the schematic or group you want to remove.



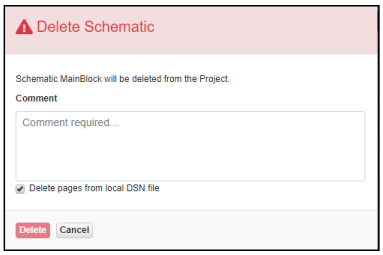
Delete Page or Schematic/Group from a Team Design Folder

Note: Pages that have been checked out can not be deleted until checkout has been canceled. A warning will show as shown in the figure at the bottom of this section.

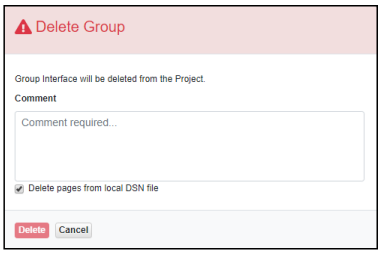
2. The respective **Delete Page**, **Schematic** or **Group** pop-up displays. Enter a comment that explains the need for the deletion. You need to enter your comments before you select the **Delete** Button. You may also remove the select item(s) from your local folder by selecting the **Delete pages from local DSN file** checkbox at the pop up.



Delete Page Pop Up

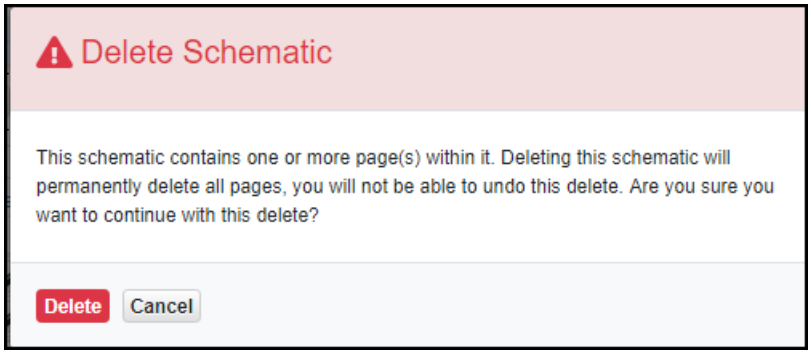


Delete Schematic Pop Up



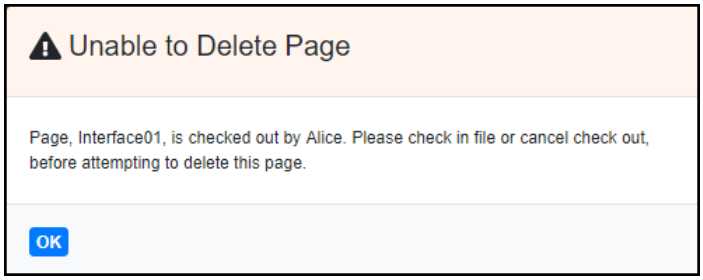
Delete Group Pop Up

3. When deleting a group or schematic, a confirmation appears. You must confirm the deletion of the before group or schematic before the deletion is completed.



Confirmation to Delete Schematic

If you try to delete a page that has been checked out, you will receive the following *Unable to Delete Page* error:



Delete Page Error

When pages are deleted, it is recorded in the folder and project history. You may obtain a copy of the deleted page by getting a copy of a deleted file from history.

5.8.6 Freeze Team Designs

Team Design functionality splits pages from a single design file to multiple pages for check in to the vault. During check out or full download, the pages are merged to your local design file. Each time EDM puts merges pages together from a Team Design into a single design file, a new file is created. Some information required for Capture CIS functionality such as back annotation or variant information is lost

when this is done. To ensure that these functions are not lost, you will need to freeze the design when the design is complete and ready for adding variants or for PCB layout. Freezing the design folder keeps all design pages in a single file.

The single design file enables Capture to back-annotate changes, such as component reference or pin swap, to the schematic that resulted from the PCB layout. Any required changes after this point will require checking out the frozen design file.

To freeze the design folder

1. Click the **Freeze** button with the Team Design Action Buttons of the Team Folder Details View. Ensure there are no checked out pages. A warning will shown when any pages are checked out.

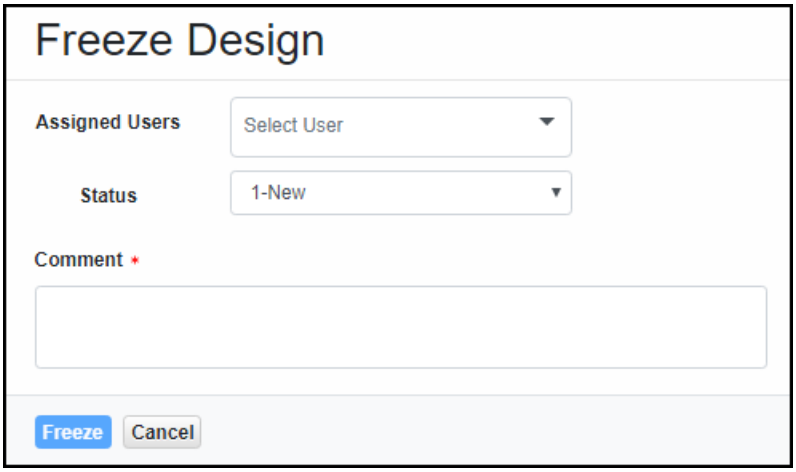


Project Leader and User Option Buttons For Hierarchical Designs



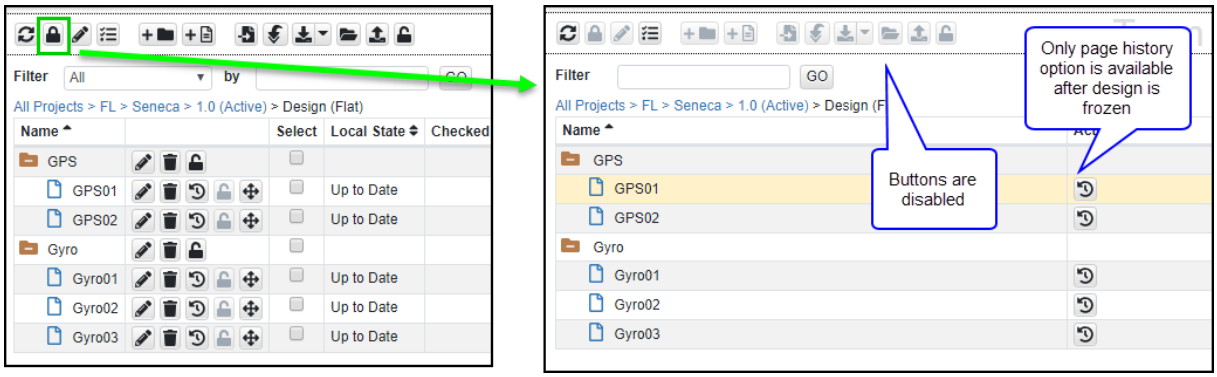
Project Leader and User Option Buttons For Flat Designs

2. The **Freeze Design** pop-up displays. Select the **Assigned users** or user group who will continue working on the design file. Update the status, and enter a comment in the **Comment** field.
3. Click the **Freeze** button to start the merge of the page to a frozen design. Or to cancel the command, click the **Cancel** button.



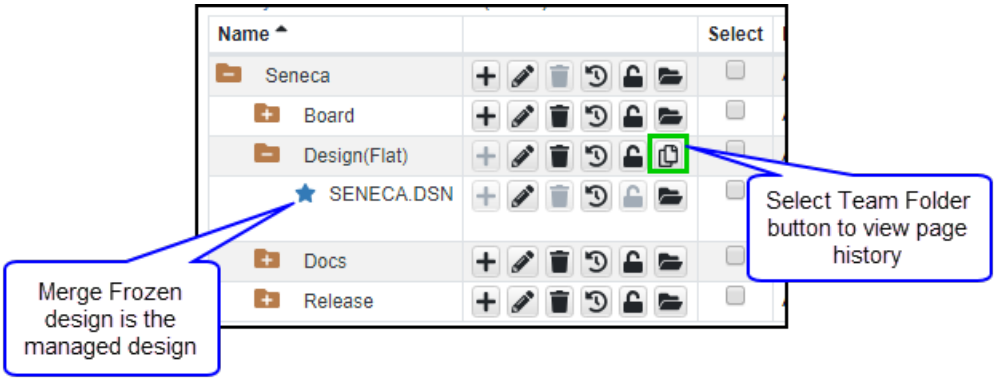
Freeze Design Pop-up

Note: After the design is frozen, you will be unable to check out the pages. User may only view history. Assigned users may only check out and check in the merged design file.



Frozen Folder Details

The merged design is visible from the folder level.



To split the design pages back out or **Unfreeze** the frozen design, you must copy the project to a new revision. You may also import the merge design into a new project.

5.9 Manage Team Library Folders

Library folders (aka Schematic Symbol or Team Library folders) enable multiple librarians and engineers to edit symbols within a library. You can use library folders to track changes to the symbol level regardless of whether you are using OrCAD Capture CIS. This feature supports librarians who manage symbol creation as well as engineers who create symbols.

Assigned leaders to a project containing a library folder or team library folder can manage a team library folder from the Team Folder Details View. The assigned leader may perform the following tasks:

- Import symbols from existing libraries
- Edit symbols to change status, properties, and user assignments
- Delete symbols or group
- Promote symbols to central library location

This functionality is only available when you open EDM from Capture. Management of library folders are unavailable when you open EDM from the Desktop Application.

Best Practices

If your company has librarians who support tasks related to symbol creation, a project should be used for all libraries. You will want to assign librarians as leaders and users for the project. General Users will use

the promoted symbols, but will be unable to see library projects if the librarian is the only project user and leader. If you want users to be able to create symbols and have the librarian review and bring them up to specification, you can add users to the library project, but do not assign them to symbols. This will enable users to see the project and check in new symbols.

Engineers who work for a company that has a librarian that manages symbol creation, can review the list of symbols available in the central library. If the desired symbol is not found and the engineer is unable to wait for the symbol to be built, the engineer can create an initial symbol for use in his/her designs and add it to the vault. The librarian can review the symbol and make modifications as needed to bring the symbol to corporate standards.

Engineers who work for a company without a librarian can search for symbols in Team Library or Schematic Symbol Library folders and reuse symbols created by other engineers. Or they can use an existing symbol as a starting point to create a new symbol.

5.9.1 Import Symbols

Symbols files can be brought into EDM by using import or check in functionality. Only assigned leaders may import library files. Any assigned user is able to check in new symbols. Only assigned users may check out, modify and check in assigned symbols. Only new symbols may be imported. Symbols already in the Team Library must be checked out before modification can be made.

To import symbols from a library file

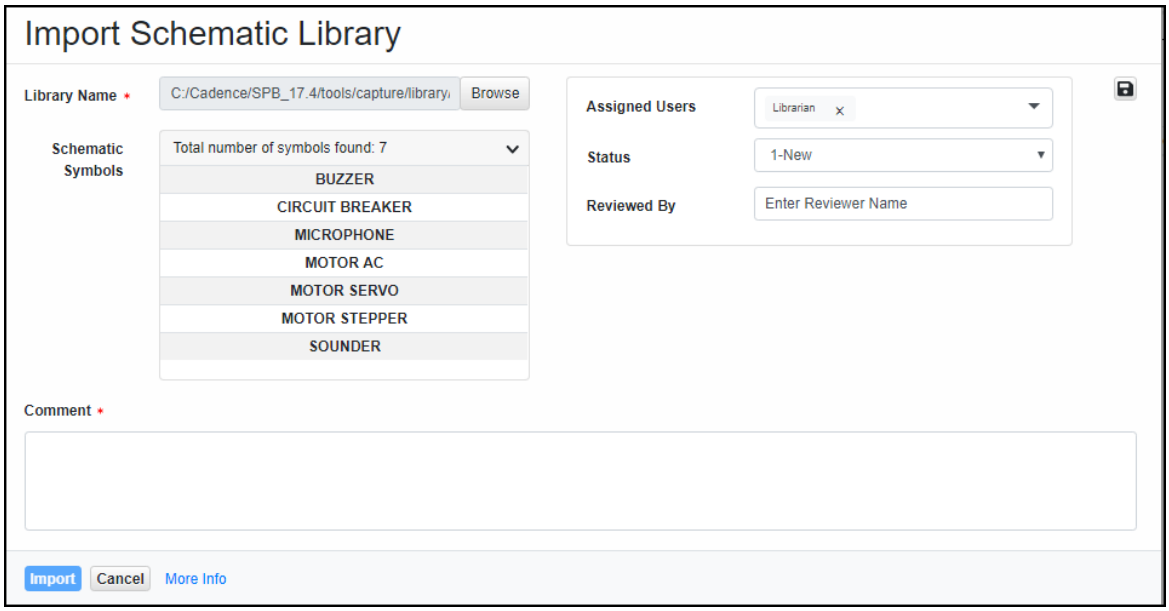
1. Click the **Import** button located with the Team Library Action buttons in the upper left corner of the Team Library Folder View.



Project Leader and User Action Buttons or Library Folders

The file selector opens.

2. Select your .OLB file for import. Once the file is selected, the Import Library form opens as shown in the following figure:

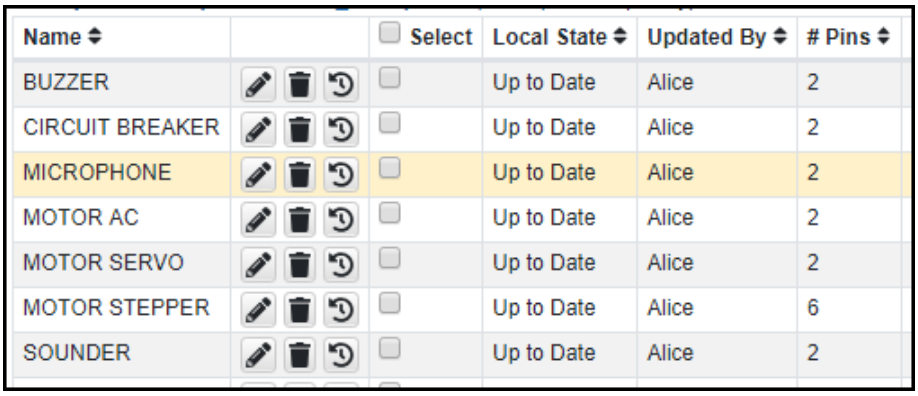


Sample Import Library Form

The symbol names from the selected .OLB file are listed in the **Import Data** section of the form. If the file you selected is corrupted or is an incorrect format, the system will be unable to locate the symbols and you will be unable to import the symbol file.

3. Enter field input into the form. Field names that have a red asterisk, such as the File and Comment fields are required fields. The form includes the following fields:
   - **File** - path and library file name to import. If this is not the file you wish to import, click the **Browse** button to select another file.
   - **Assigned Users** - select users or user groups to work on symbols. The list of available users or user groups is the list of users assigned to the project.
   - **Status** - entered values for the status for the symbol to import. If the status is configured as a pick list, you may select a value from the pick list. If a default status has been configured, it is pre-populated in the form.
   - **User Configured Fields** - If the assigned leader has configured user-defined fields for Schematic Symbols, those fields are also included in the Import Symbols form. The entry of information into these fields is optional. The **Reviewed By** field in example above is a configured user field.
   - **Comment** - comment for the import of the pages. Comments are saved in history for the project.
   - Select the **Export to File** button to review the list of symbols in a text file.
4. Click the **Import** button to initiate the symbol importation after you have finished specifying field input. Libraries with a large number of symbols take longer to import. If you do not want to import any symbols, click the **Cancel** button.

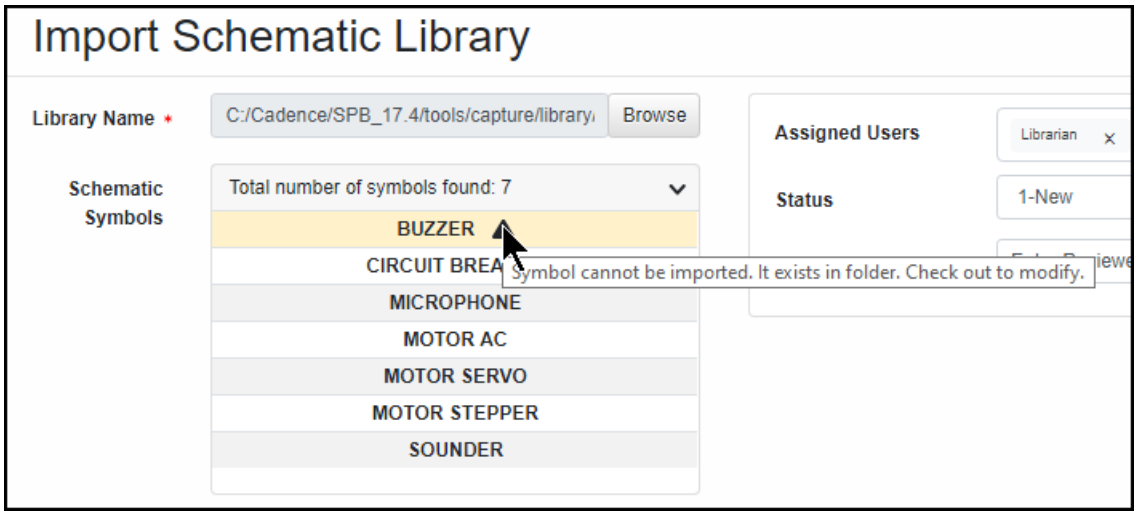
Once imported, the list of imported symbols are visible in the library folder as shown in the following figure. The property values as well as the number of pins associated to a symbol update during the import process.



Name		Select	Local State	Updated By	# Pins
BUZZER			Up to Date	Alice	2
CIRCUIT BREAKER			Up to Date	Alice	2
MICROPHONE			Up to Date	Alice	2
MOTOR AC			Up to Date	Alice	2
MOTOR SERVO			Up to Date	Alice	2
MOTOR STEPPER			Up to Date	Alice	6
SOUNDER			Up to Date	Alice	2

Imported Library

When you attempt to import symbols that exist in the team folder, a warning shows next to the symbol in the import list. You may continue to import the rest of the symbols that do not have warnings.



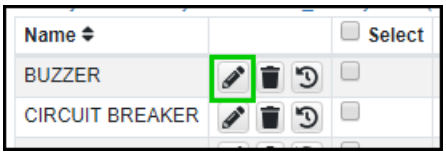
5.9.2 Edit Symbols

Properties for multiple symbols may be selected and edited as a group using the **Edit** button located with the Team Library Action Buttons or modified individually by using the inline **Edit** button next to symbols. Only assigned leaders may Edit Symbols.

Symbol properties may be changed individually using the inline **Edit** button next to symbols. The **Edit** button located with the Team Library Action Buttons allows you to change assignment for all symbols.

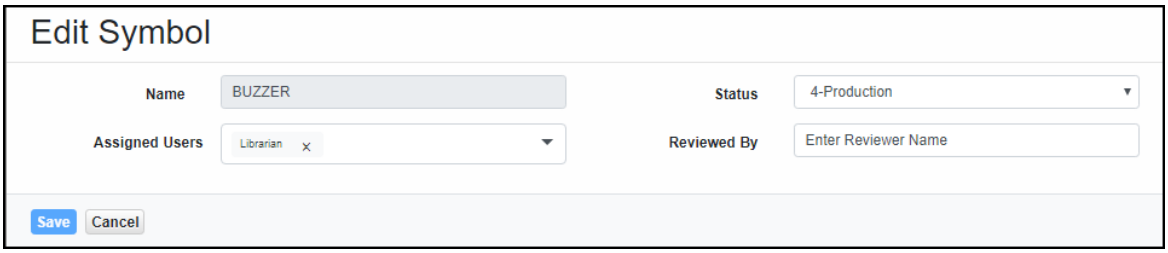
To change a symbol's properties

1. Click the **Edit** button located next to the symbol.



Edit Symbol

The **Edit Symbol** form displays. The properties that can be changed for the selected item display as shown in the following figure:



Properties of Symbols Available for Edit

2. Edit the following fields:
   - **Assigned Users** - change user assignment or user group assignment for the symbol.
   - **Status** - update symbol status with a new status.
   - **User fields** - update User Fields that are configured. The **Reviewed By** field in example above is a configured user field.
3. Click the **Save** Save button, to accept the changes. Or to cancel your changes, click the **Cancel** Cancel button.

To change symbol assignments for all symbols

1. Click the **Edit** button located located with the Team Library Action Buttons in the upper left corner of the Team Library Folder Details View.



The **Edit Folder** form for Symbol displays. You can change the library OLB name and the user assignment across all symbols within the Team Library folder. If you change the OLB name after you have promoted some symbols, you will need to manually remove the original OLB from the promote library location and re-promote all previously promoted symbols.

2. Select the user or group you would like all symbols to be assigned.
3. When complete, click the **Save** Save button, to accept the changes. To cancel your changes, click the **Cancel** Cancel button.



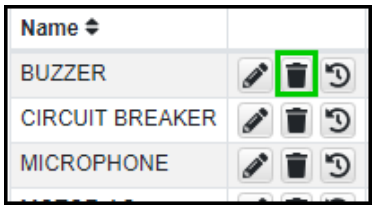
Edit Symbol Assignments and OLB name

5.9.3 Delete Symbols

Unlike pages and files, symbols are stored in the vault and a promoted library. When deleting symbols, you have the option to delete them from both locations or only from the vault. Only assigned leaders may delete symbols.

To remove a symbol

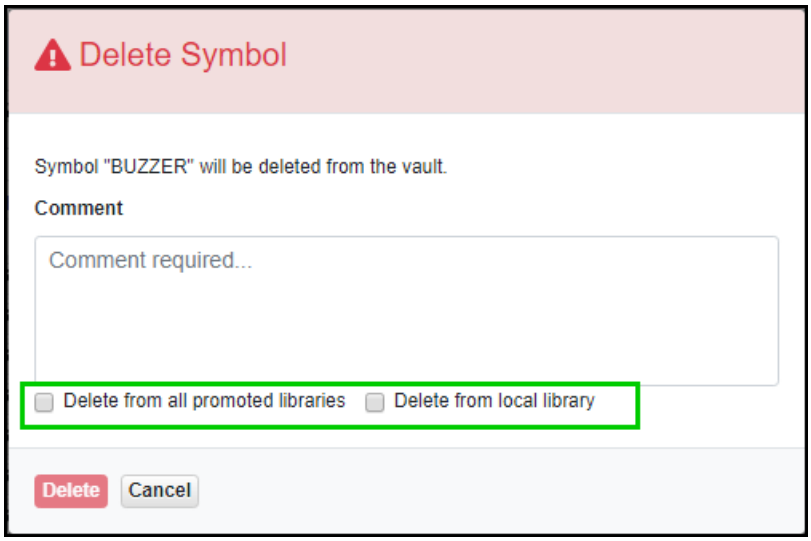
1. Click the **Delete** button located next to the symbol you want to remove. Symbols that are checked out cannot be deleted until checkout has been canceled.



Delete a Symbol from a Team Design Library Folder

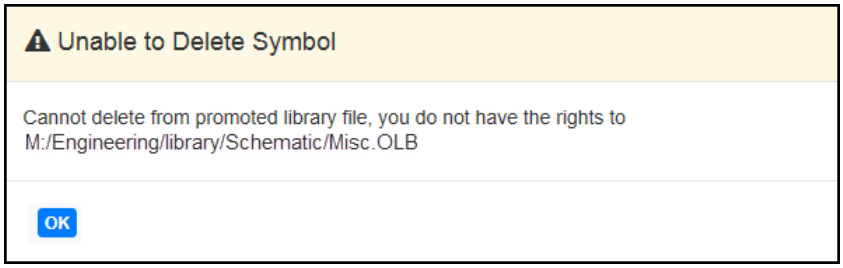
The **Delete Symbol** pop-up opens.

2. If you want to also delete the symbol from the corresponding promoted library and/or from your local library file, select the checkboxes for **Delete from all promoted libraries** and/or **Delete from local library.**
3. Enter a comment into the Comment field. This is a required field.
4. Click the **Delete** button or click the Cancel **button** to cancel the deletion.



Delete Symbol Pop-up

If you select the check-box to remove the corresponding symbol from the promoted library and you do not have permission to alter the promoted library, you will receive a **Unable to Delete Symbol** warning.



Permissions Warning

When symbols are deleted, it is recorded in the folder and project history. You may obtain a copy of the deleted symbol by downloading a copy of the deleted file from history.

5.9.4 Promote Symbols

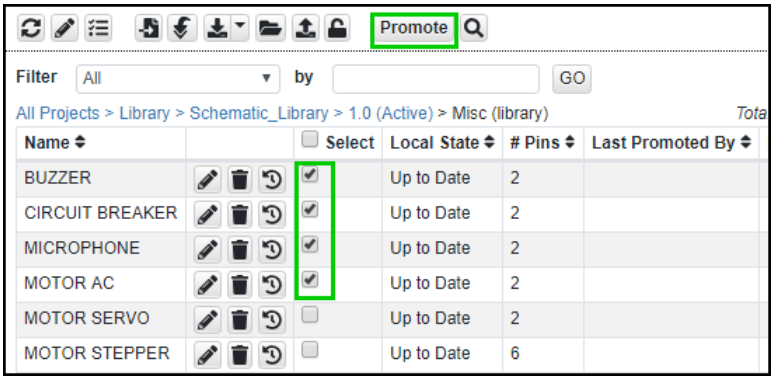
When individual symbols are promoted, the system copies the symbols from the client PC or EDM Web application to the promoted library location as specified in Admin > Configuration > Capture > Promoted Library. Only assigned leaders may promote symbols. Libraries in the promoted library location should never be manually edited or opened from the promote location. Opening the library file may lock the file from use by others.

To promote a symbol from a Team Library (Schematic Symbol) folder

Symbols may be promoted from a Team Library folder (also called Schematic Symbol Library folder). The steps that follow explain how to promote a symbol from a Team Library folder.

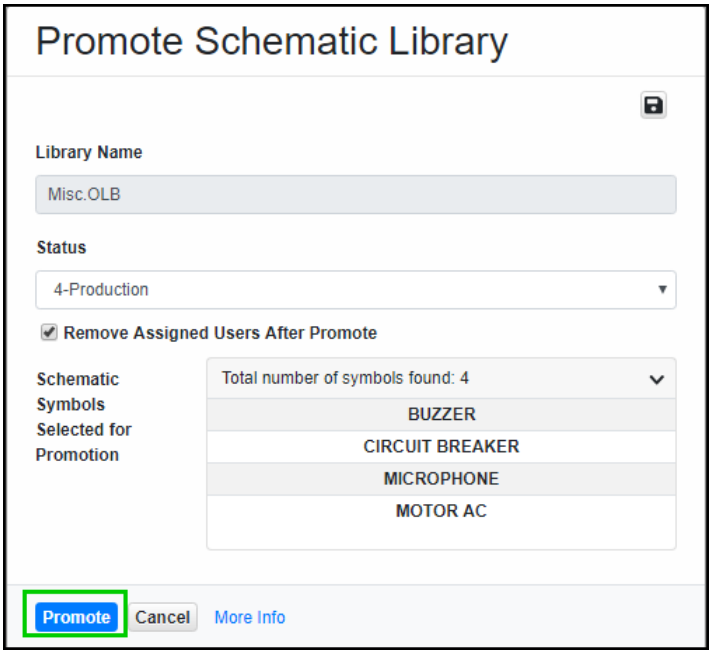
1. Select the symbols you want to promote by selecting checkbox in the select column in the symbol row as indicated in the image that follows.

2. Click the **Promote** Promote button located with the Team Library Action Buttons in the upper left corner of the Team Library Details View. See the figure below to see an example.



Promote Symbol

3. The Promote Schematic Library pop up opens showing the list of symbols selected.



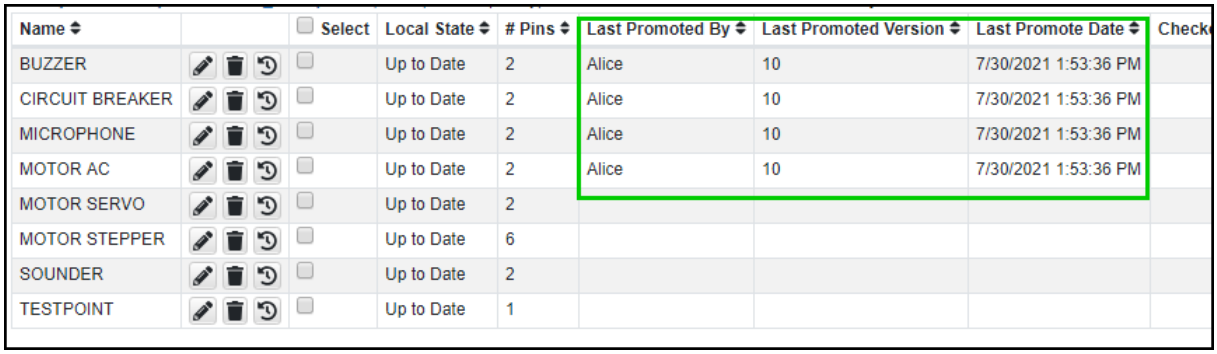
Promote Pop-up

4. Update the status if desired.
5. Place a check mark in the **Remove Assigned Users** checkbox if you want to remove assigned users. This enables you to prevent accidental check out and modification of the symbol.
6. Click the **Promote** Promote button to promote the symbols. Or, to dismiss the symbol promote, click the **Cancel** button.

The symbols are promoted to a library with the assigned name. The fields, **Status**, **Last Promote Date**, **Last Promote By** and **Last Promoted Version** for the selected symbols are updated to reflect your status selection, the current date, and latest version of the checked in symbols. If the library file does not exist, the program creates a new file. If one exists, the symbols are added to the existing file.

Note: If you do not have **Full control** permission on the library, you receive an error message notifying you.

The following figures show symbols in a Team Library after promote. You may select the **View Promoted Library** button in the Library Project Action Buttons to review all symbols that have been promoted.



Name		Select	Local State	# Pins	Last Promoted By	Last Promoted Version	Last Promote Date	Check
BUZZER			Up to Date	2	Alice	10	7/30/2021 1:53:36 PM	
CIRCUIT BREAKER			Up to Date	2	Alice	10	7/30/2021 1:53:36 PM	
MICROPHONE			Up to Date	2	Alice	10	7/30/2021 1:53:36 PM	
MOTOR AC			Up to Date	2	Alice	10	7/30/2021 1:53:36 PM	
MOTOR SERVO			Up to Date	2				
MOTOR STEPPER			Up to Date	6				
SOUNDER			Up to Date	2				
TESTPOINT			Up to Date	1				

Example of Promoted Footprints in a Footprint Project

5.10 Delete Project Revision

To delete a project revision

1. Access the Project Dashboard.

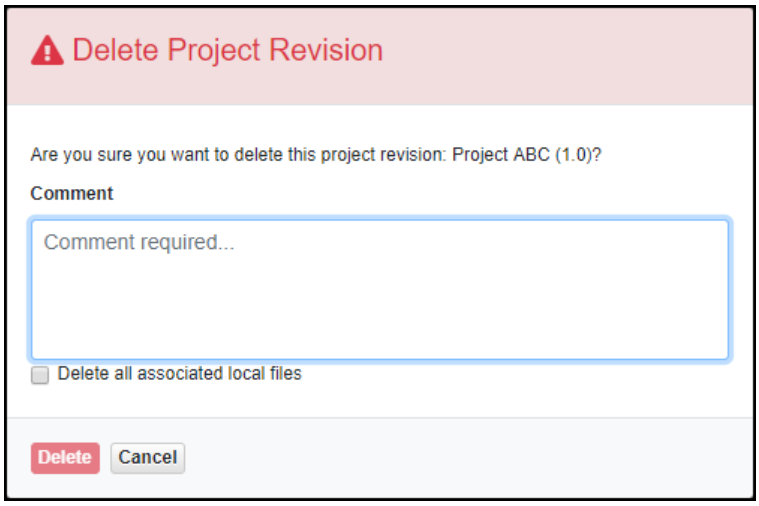
2. Click the **Delete** button that is located next to the project revision you wish to delete. This opens the **Delete Project Revision** form.



Name		Revision	State
Project ABC		1.0	Active
Project ABC		1.1	Active
Project ABC		1.2	Active

Delete Project Revision from Project List

Caution: **the project name as well as all project files and sub-folders will be deleted from the vault. The deleted folders and files are unrecoverable.**



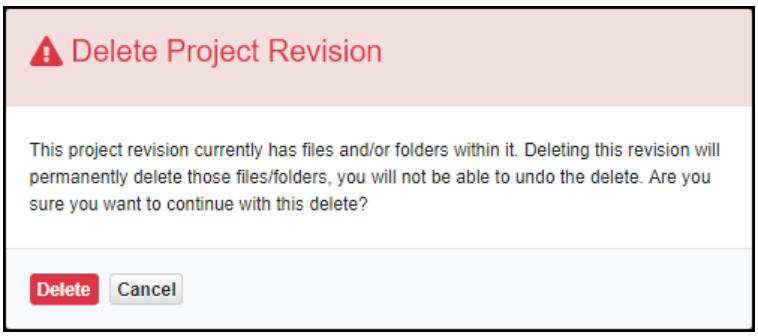
Delete Project Revision Pop-up Form

3. Enter a **Comment**. This is required and the Delete button will be disabled until a comment is entered.

4. Optionally select **Delete all associated local files** to also remove the project revision files and folders from your local workspace.

5. Click the Delete button. Or cancel the deletion by clicking Cancel. Canceling will close the window and return to the Project Dashboard.

6. The following **Delete Project Revision** confirmation opens to confirm the deletion. ***__This is your last chance to cancel the deletion__.*** Click the Delete button. Or cancel the deletion by clicking Cancel.

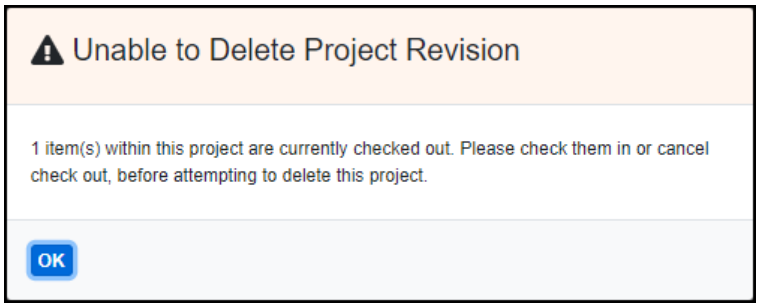


Delete Project Confirmation

Caution: **the project name as well as all project files and sub-folders will be deleted from the vault. The deleted folders and files are unrecoverable.**

Note: To remove all revisions of a project, repeat the delete process for each revision. A project is unable to be deleted when any of the project files are checked out.

If you attempt to delete a project that has a file(s) checked out, the following Unable to Delete Project Revision pop-up window will appear. You will need to cancel check out on all files before you are able to remove the entire project.



Unable to Delete Project Revision

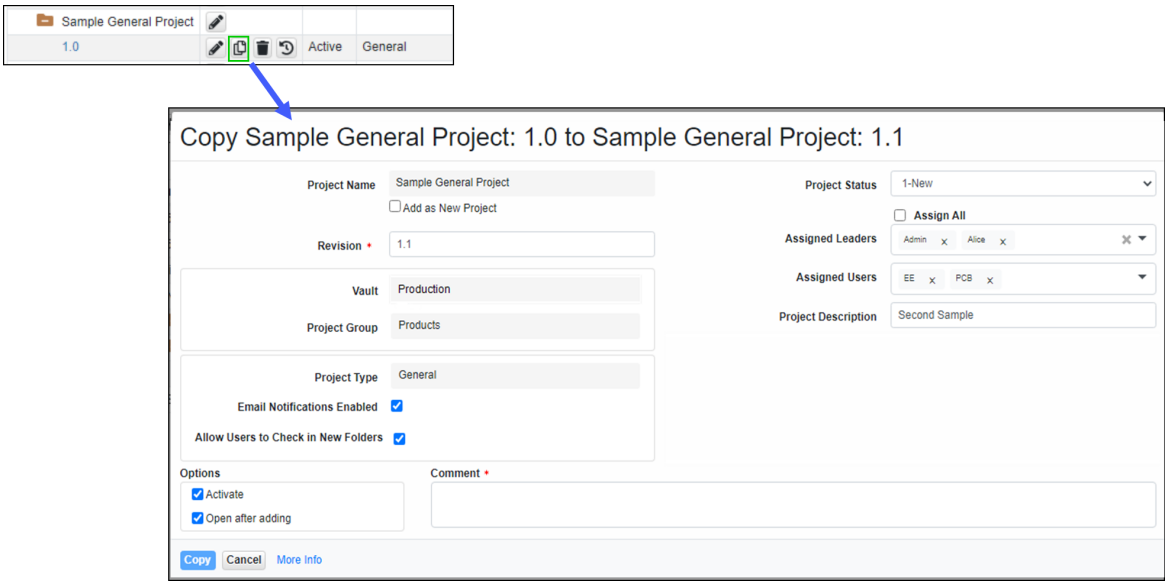
5.11 Copy Project to New Revision/Name/Group

You can copy an existing project revision to a new revision and then use the copy to mark a point in your project, mark the beginning of the layout, each prototype, etc. You can also copy the existing project revision to a new project name and revision. The **Copy Project** form is available when copying a project revision using the EDM Desktop Application or the Capture Plug-in. When copying a project revision that contains Frozen Team Design folders, use EDM in Capture to ensure all options are available in the **Copy Project** form.

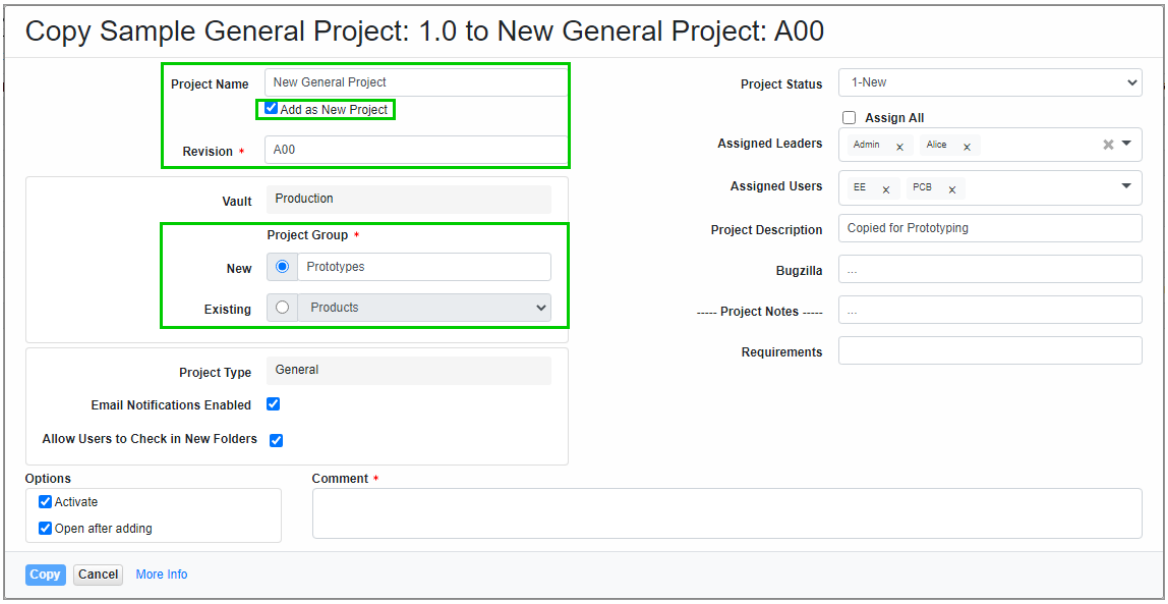
Open the Project Dashboard and locate the project revision you want to copy. Select the **Copy** button to open the **Copy Project** form. After completing the **Copy Project** form, click the Copy button to create a copy of the project with the selected options in the form, or click the Cancel button to cancel and close the form. The new revision will contain copies of the selected files and folders. User assignments and statuses for items within the project will also be copied.

Copy a Project to New Revision/Name/Group

The project name and revision are shown in the title of the **Copy Project** form.



Copy Project Form - Existing Project Name



Copy Project Form - New Project Name

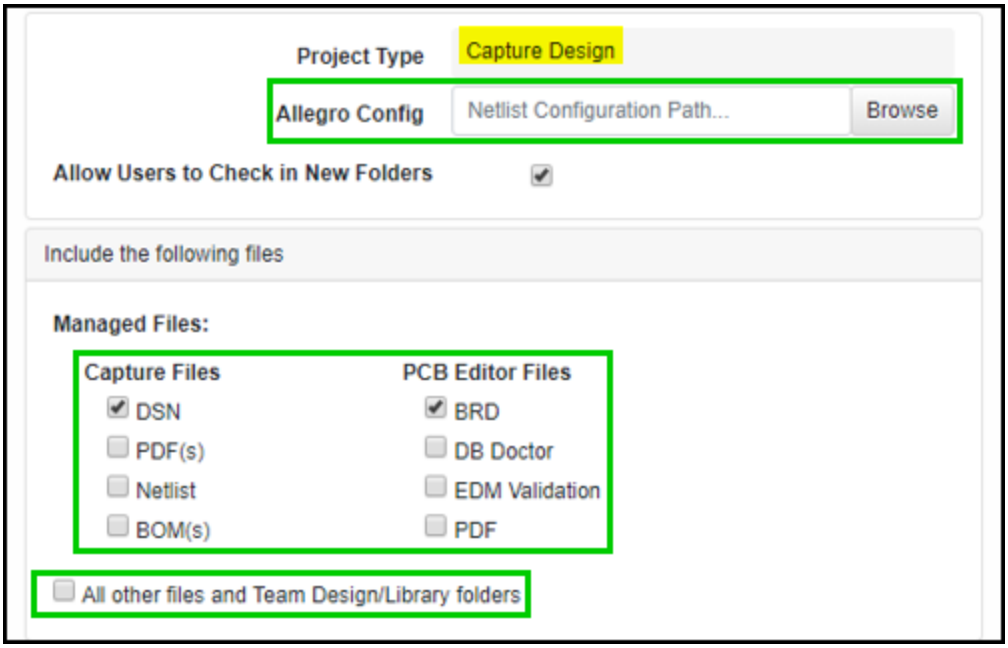
The form initially populates with information from the selected revision. The form consists of the following:

- **Project Name** and **Project Group** remain unchanged by default. Click the **Add as New Project** check box to enable input for the Project Name and Project Group. Similar to the Add Project form, you may enter a new Project Name and use a new or existing Project Group when this option is selected.
- **Vault** and **Project Type** are copied based on the project being copied and **cannot be changed** in this form.

- **Revision** - Enter or select the new **Revision** for your copy. It must be unique to the project name (e.g. ProjectA and ProjectB can both have a revision 1.0). If specified as a pick list in Configuration, then select a value from the pick list.
- **Allow Users to Check in New Folders**
- **Email Notifications Enabled** - This option allows you to enable/disable email notifications on a per-project basis. It will only be shown if you have Email Notifications enabled in the Admin Configuration.
- **Project Status** - status for the project. If this field was configured as a pick list, you may choose a pick list value. The default value auto-populates. You mat change the status
- **Assigned Leaders** - only users whose role has been granted project leader permissions may be added as an assigned leader. The assigned leader list is copied in the form and you may change it as desired before copying the project.
- **Assigned Users** - any user or user group may be assigned as users. The list of assigned users is added to the new copy at the project level. User assignments on folders and files will be copied. Changing folder and file assignment may be made after the project is copied.
- **Additional Configured User Fields** - Project user fields shown in the **Copy Project** form based on your specific configuration. In this example, **Project Description** (user_field_01) is enabled.
- **Options**:
  - **Activate** - activate the project when the project is copied. Projects must be activated before users can check files in and out, but this can be done during or after copying a project.
  - **Open after adding** - Open directly to the Project Details page after the project is copied.
- **Comment -** Enter a comment associated with your action. The **Comment** field is required. Note contents are included in the History of the project.

Copy a Project Revision with a Managed Capture Design

Additional options are available when copying a Capture Design project, regardless when the project has a Frozen Team Design.

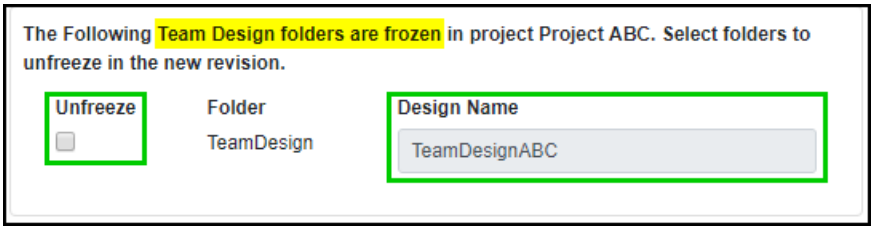


Additional Options when Copying a Capture Design Project

- **Allegro Config** - This option enables you to change the netlist configuration file that you want to use for the copy of the project. Leave this unchanged to use the default setting configured in Netlist Generation Configuration.
- **Include the following files**: options in this section allow you to select which managed files to copy and whether to copy additional files and Team Design/Library folders. All folders, except Team Design/Library folders, will be copied regardless of the options selected in this section; this includes the designated folders for each type of managed file (DSN, BRD, BOM, etc.)
  - **Managed Files > Capture Files**:
    - **DSN** - Copy the managed **Capture Design (.DSN)** file. The designated DSN folder will be copied whether you select this option or not.
    - **PDF(s)** - Copy the **PDF files** associated with the managed Capture Design. The designated **PDF folder** will be copied whether you select this option or not.
    - **Netlist** - Copy the **Netlist files** files associated with the managed Capture Design. The designated **Netlist folder** will be copied whether you select this option or not.
    - **BOM(s)** - Copy the **BOM files** associated with the managed Capture Design. The designated **BOM folder** will be copied whether you select this option or not.
  - **Managed Files > PCB Editor Files**:
    - **BRD** - Copy the managed **Board (.BRD)** file. The designated **BRD folder** will be copied whether you select this option or not.
    - **DB Doctor** - Copy the **DB Doctor files** associated with the managed **Board (.BRD)** file. The **Reports sub-folder** under the designated **BRD folder** will be copied whether you select this option or not.
    - **EDM Validation** - Copy the **EDM Validation file** associated with the managed **Board (.BRD)** file. The **Reports sub-folder** under the designated **BRD folder** will be copied whether you select this option or not.
    - **PDF** - Copy the **PDF file** associated with the managed **Board (.BRD)** file. The designated **BRD folder** in which the PDF is located will be copied whether you select this option or not.
  - **All other files and Team Design/Library folders** - Copy **all other files** not specified in the list of Managed Files above, and all **Team Design and Team Library folders**.

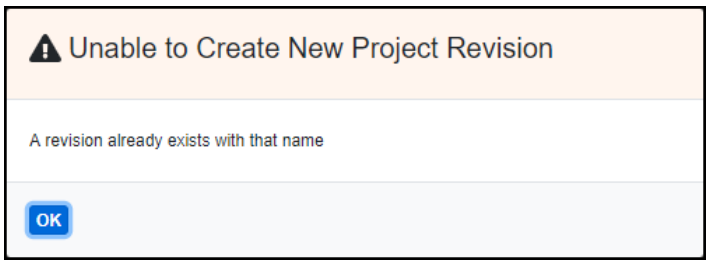
Copy a Project Revision with a Frozen Team Design

If the project revision you are copying has a Frozen Team Design, the **Copy Project** form has an additional section that allows you to unfreeze the design when copying, if desired. If you opened EDM from Capture, you may unfreeze the design by selecting the **Unfreeze** checkbox. Leaving the checkbox unchecked keeps the design as a single merged file in the copy. The unfreeze option is disabled when you copy the project revision from the EDM Desktop application.



Additional Options when Copying a Project with a Frozen Team Design

If you entered a revision that already exists, the following *Unable to Create New Project Revision* error occurs. Click OK to close the error and return to the **Copy Project** form. From there, you can update the Revision and Copy again, or click the Cancel button to cancel and close the form.



Unable to Create New Project Revision Error

5.12 Package For Release

When you have completed a development phase, an assigned project leader can use EDM Desktop to create a package for release. This package may be used as an archive, for internal review, or it can be delivered to a contract manufacturer.

Note: This feature is available from the EDM Desktop application (EDM.exe) and is not available when you open EDM from Capture or a supported browser.

1. Create a Release Package to generate the release .zip package and check it into the designated Release folder.

2. Review Release Package Contents to verify the expected outputs and any error/warning files.

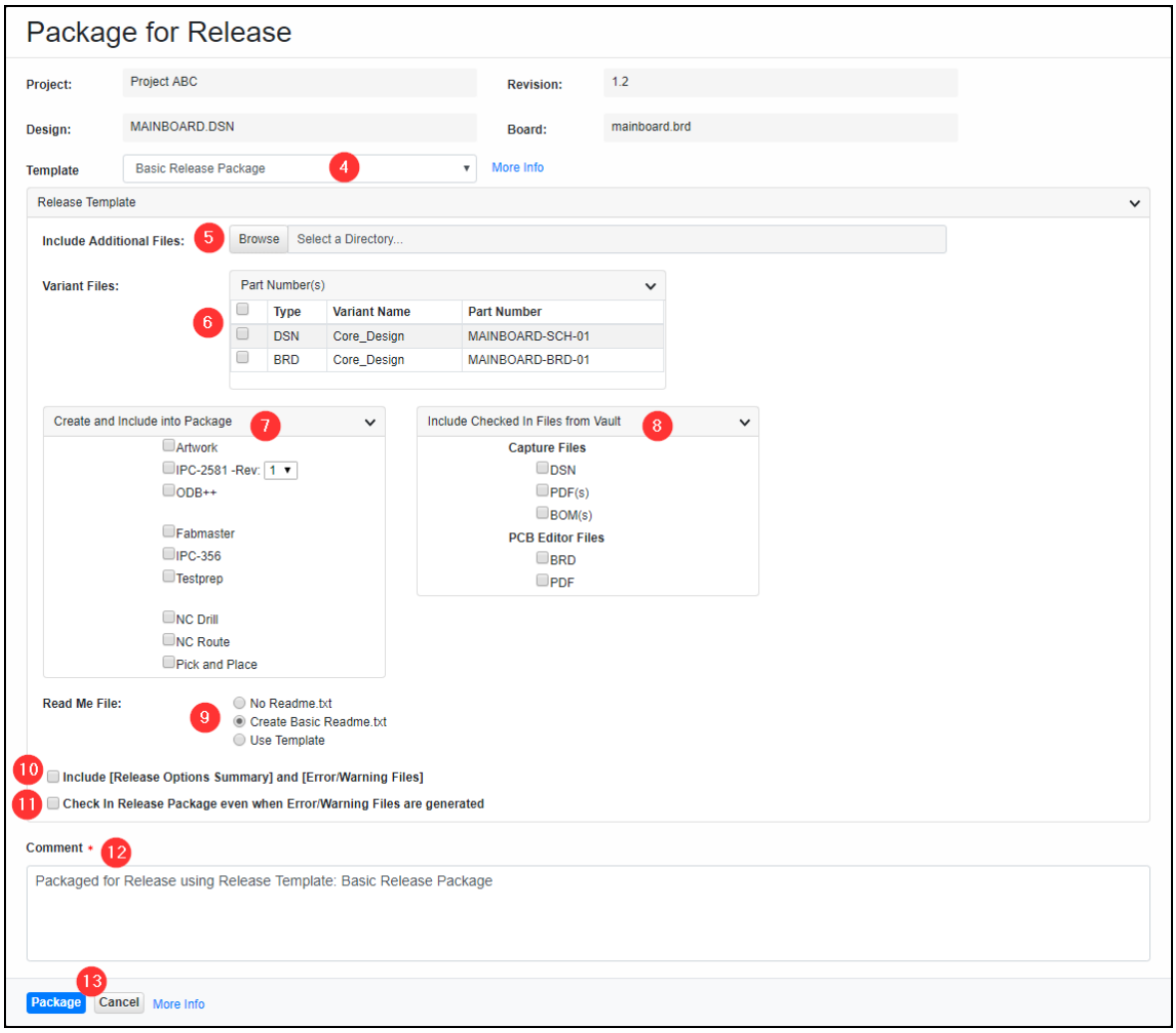
5.12.1 Create a Release Package

To create a Package for Release

1. Launch the EDM Desktop application and log in as an assigned project leader for the desired project revision.
2. Open the project revision to be packaged.
3. Click the **Package for Release** button as shown below. This will open the **Package for Release** form as shown.



Package for Release Button



Package for Release Form

4. The (Release) **Template** assigned to the project's designated Release folder is selected and loaded by default. You may select a different template if desired. In either case, the associated template options are loaded and can be updated on-the-fly in the form.

5. Select the **Browse** button next to **Include Additional Files** if you plan on including additional files/folders during the package process that do not exist in EDM. Navigate to the folder path you'd like to include with the release package .zip file , then click **Select Folder**. The selected folder, all its sub-folders, and all files will be included at the root of the release package .zip file, maintaining the original structure of the selected folder. Additionally, all the files and folders listed in the selected folder will be included in the readme.txt file that is (optionally) included with the release package .zip file.

6. The initial **Variant Files** selected will be based on the Release Template's Variant Files option for **Core Design with All Variants** or **Core Design**. Select the desired **DSN and**

BRD Variant Files by clicking the All/None checkbox and/or individual checkboxes for each variant file.

7. Select or deselect options to **Create and Include in Package**. Selected files will be created during packaging and included in the release package. For the **IPC-2581** option, select the **Rev** from the drop-down list (values **1, A, B**).

   - **Artwork** – Outputs for artwork are specified in the board's Film Domain Settings, under the Artwork Control Form Film Setup. If not found, all films will be included.

   - **IPC-2581** – Exports physical design data from BRD that are defined in the IPC2581 Layer Mapping Editor dialog box and stored in the board's IPC2581_EXPORT_OPTIONS design property. Creates file: <design name>_ipc2581.xml. For more information, refer to the Allegro PCB Editor User Guide section under "IPC 2581."

   - **ODB++** – Generates a .TGZ file from BRD. Requires software to be downloaded and installed before file generation.

   - **Fabmaster** – Exports fabmaster output from BRD as txt file and creates the cad file named: <design_name>.cad. No configuration settings are required with the selection of this option.

   - **IPC 356** – Exports physical design data as .TXT files generated from BRD. For more information, refer to the Allegro PCB Editor User Guide section under "IPC-D-356."

   - **Testprep** – Generates the Quick Report/Testprep Report that is based on TESTPREP availability in the BRD file. Due to the high density of design elements on PCBs, it is recommended that you run test prep to optimize testing. For more information, refer to the Allegro PCB Editor User Guide section under "PCB Editor and APD: Test Preparation."

   - **NC Drill** – Accesses data in your BRD and nc_param.txt file and creates customized drill output files. For more information, refer to the Allegro PCB Editor User Guide section under "NC Drill Legends" and "NC Parameters File."

   - **NC Route** – Accesses data in your BRD and nc_param.txt file to generate ROU files in Excellon Format. For more information, refer to the Allegro PCB Editor User Guide section under "Running NC Route."

   - **Pick and Place** – Exports the positions and orientations of placed components and creates a generic Quick Report/ASCII report: pcp_rep.rpt. External parameter files and BRD settings are not required.

8. Select or deselect options to **Include Checked In Files from Vault**. The **latest vaulted/EDM version** of selected files will be included in the release package. These files **will not be created** during packaging.
   a. **Capture Files** - The selected files below will be added to the **schematic files** folder in the release package .zip file.
      - **DSN** – the project's managed design (.DSN) file
      - **PDF(s)** – the project's managed/auto-generated PDF file(s) for the selected design variants
      - **BOM(s)** – the project's managed/auto-generated BOM file(s) for the selected design variants

   b. **PCB Editor Files** - The selected files below will be added to the **allegro files** folder in the release package .zip file.
      - **BRD** – the project's managed board (.BRD) file
      - **PDF** – the project's managed/auto-generated PDF file for the selected design variants

9. Select the desired option for the **Read Me File**.
   a. **No Readme.txt** - This will not create a Readme.txt file when packaging for release.
   b. **Create Basic Readme.txt** – This will create the **Basic Readme.txt** file, listing all folders and files contained in the release package.zip file. Refer to this example.
   c. **Use Template** – This will use the Read Me file stored in the selected Release Template. The file name will appear, but will only be configurable when Managing Release Templates.

10. Select or deselect whether to **Include [Release Options Summary] and [Error/Warning Files]** in the release package .zip file. This option determines whether the same **logs** folder used in the project's designated Release folder will be included in the release package .zip file. This includes the more detailed individual .log files for the errors/warnings as described in the Error/Warning Files below.

    Note: The Release Options Summary and Error/Warning log files will always be checked into the **logs** folder under the project's designated Release folder, but the detailed individual error/warning log files **will not be available** in EDM if **Include [Release Options Summary] and [Error/Warning Files]** is unchecked. However, the user's local **%AppData%\..\Local\Temp\EDM** folder will contain (only) the most recent individual error/warning log files.

    a. **Release Options Summary** - This is a text **.log** file summary of selected options from the Package for Release form. Refer to this example.
    b. **Error/Warning Files** - This is a text **.txt** file that contains a summary of any errors or warnings that were encountered during packaging and references the names of more detailed individual .log files to review for more details.. If no such errors/warnings occur, the file will not be created. Refer to this example.

11. Select or deselect whether to **Check In Release Package even when Error/Warning Files are generated**. This option determines whether the release package .zip file will

be checked in if any errors/warnings are generated. If no such errors/warnings occur, the release package will be checked in upon completion.

a. When selected, the release package will be checked in even if errors/warnings were generated during packaging.
b. When deselected, the release package will not be checked in and will remain Local Only if any errors/warnings were generated during packaging.

12. Add or update the **Comment** that will be associated with the project history of the release .zip file. It will default to the comment from the selected Release Template and can be updated as desired.

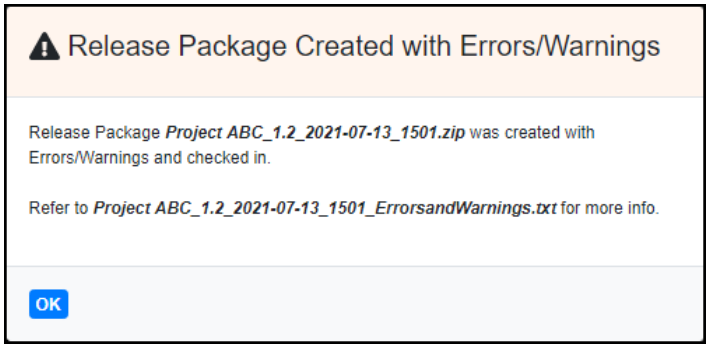
13. Click the Package button to Package for Release using the selected options in the current form. This will create a release .zip file with the format **<ProjectName>_<ProjectRevision>_<YYYY-MM-DD_HHmm>.zip**. e.g. **Bypass_03_2020-09-10_0756.zip** was packaged for released on September 10, 2020 @ 7:56AM. Upon completion, a pop-up window will appear. Review the message and click **OK** to close the window.

a. **Release Package Created** - This shows the packaged file name and indicates that no errors or warnings occurred. The packaged .zip file is checked into the designated Release folder.



Release Package Created

b. **Release Package Created with Errors/Warnings** - This shows the packaged file name and references an errors/warning .txt file to review for more info.



Release Package Created with Errors/Warnings

5.12.2 Review Release Package Contents

After you've Created a Release Package, the outputs can be reviewed and verified. The release package will contain the appropriate files based on the selected options in the Package for Release form.

The designated Release folder for the project revision will contain these items that are automatically checked into the vault during release.

1. **Release Package** - This is the **.zip** file that contains the files included with the package. The file name is in this format **<ProjectName>_<ProjectRevision>_<YYYY-MM-DD_HHmm>.zip**.
   e.g. **Project ABC_1.2_2021-07-13_1509.zip** was created with options shown in the example Release Options Summary below.



Release Packages in the Project's Release Folder

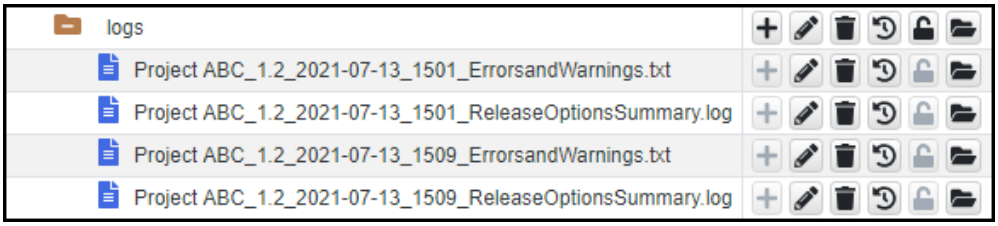
If the **Create Basic Readme.txt** or **Use Template** option is selected from the Read Me File section in the Package for Release form, a .txt file will be created with options shown in the example Release Options Summary. It will be included in the release package, listing all folders and files contained in the .zip file.

```
FILES INCLUDED IN Project ABC_1.2_2021-07-13_1509.zip

allegro files/mainboard.brd
allegro files/mainboard.pdf
artwork/CircuitSpace.art
logs/ipc356_out.log
logs/ncdrill.log
logs/photoplot.log
logs/Project ABC_1.2_2021-07-13_1509_ErrorsandWarnings.txt
logs/Project ABC_1.2_2021-07-13_1509_ReleaseOptionsSummary.log
schematic files/MAINBOARD.DSN
schematic files/MAINBOARD.pdf
schematic files/MAINBOARD.xlsx
mainboard.cad
mainboard.cvg
mainboard.ipc
mainboard_odb.tgz
pcp_rep.rpt
readme.txt
testprep.rpt
```

Create Basic Readme.txt with Include [Release Options Summary] and [Error/Warning Files]

2. **Release logs Folder** - This is a **logs** sub-folder in the Release folder that contains information about the release .zip packages. These files can also be included in a **logs** folder in the release package when Include [Release Options Summary] and [Error/Warning Files] is selected.



Release logs Folder

a. **Release Options Summary** - This is a text **.log** file that contains a summary of the options selected in the Package for Release form. Its date/time stamp should correspond with that of the release .zip package itself, which is Project ABC_1.2_2021-07-13_1509.zip in this example.

```
-----------------------------------------------------------------------------------------------
EDM Package for Release Initiated
        Start Date/Time: 07/13/2021 (MM/DD/YYYY) 15:09:51 (HH:MM:SS)
        Initiated by: Alice
-----------------------------------------------------------------------------------------------
Project Name: Project ABC
Revision Name: 1.2
-----------------------------------------------
Design: MAINBOARD.DSN
Board: mainboard.brd
-----------------------------------------------
Template: Basic Release Package
-----------------------------------------------
Include Additional Files: None
-----------------------------------------------
Variant Files:
        Type    Variant Name            Part Number
        DSN             Core_Design             MAINBOARD-SCH-01
        BRD             Core_Design             MAINBOARD-BRD-01
-----------------------------------------------
Create and Include into Package:
        [x] Artwork
        [x] IPC-2581 - Rev: 1
        [x] ODB++

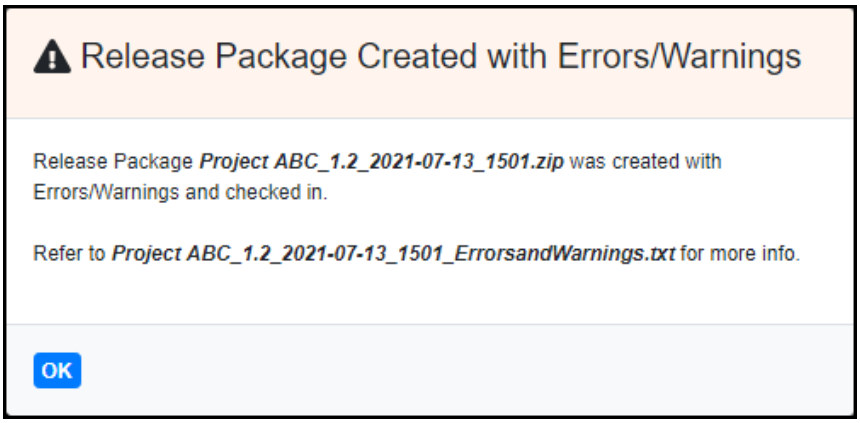
        [x] Fabmaster
        [x] IPC-356
        [x] Testprep

        [x] NC Drill
        [x] NC Route
        [x] Pick and Place
-----------------------------------------------
Include Checked In Files from Vault:
        Capture Files:
                [x] DSN
                [x] PDF(s)
                [x] BOM(s)
        PCB Editor Files:
                [x] BRD(s)
                [x] PDF
-----------------------------------------------
Read Me File: Create Basic Readme.txt
-----------------------------------------------
Include [Release Options Summary] and [Error/Warning Files]: [x]
Check In Release Package even when Error/Warning Files are generated: [x]
-----------------------------------------------
Comment: Packaged for Release using Release Template: Basic Release Package
-----------------------------------------------


-----------------------------------------------------------------------------------------------
EDM Package for Release Completed:
        End Date/Time: 07/13/2021 (MM/DD/YYYY) 15:10:04 (HH:MM:SS)
-----------------------------------------------------------------------------------------------
```

b. **Error/Warning Files** - This is a text **.txt** file that contains a summary of any errors or warnings that were encountered during packaging and references the names of more detailed individual .log files to review for more details.. If no such errors/warnings occur, the file will not be created.

In this example, a release package was created with Errors/Warnings.



Release Package Created with Errors/Warnings

After extracting the release package (**Project ABC_1.2_2021-07-13_1509.zip**), a **logs** folder shows the **Release Options Summary** log, **Errors and Warnings** log, and associated log files (**ncdrill.log, ipc356_out.log**, and **photoplot.log**) for the warnings and errors encountered because Include [Release Options Summary] and [Error/Warning Files]) was selected during release.



Extracted Release Package Contents Showing Additional Error/Warning Logs

The Errors and Warnings summary log file (**Project ABC_1.2_2021-07-13_1509_ErrorsandWarnings.txt**) shows there were Warnings in the ARTWORK **photoplot.log**, IPC356 **ipc356_out.log**, and Errors in the NCDRILL **ncdrill.log** during the most recent package for release.

Note: These detailed individual error/warning log files **will not be available** in EDM if **Include [Release Options Summary] and [Error/Warning Files]** is unchecked. However, the user's local **%AppData%\..\Local\Temp\EDM** (e.g. C:\Users\AliceEngineer\AppData\Local\Temp\EDM) folder will contain the most recent individual error/warning log files. It will not contain any previous error/warning log files as they are overwritten each time.

```
ARTWORK;Warnings;photoplot.log
IPC356;Warnings;ipc356_out.log
NCDRILL;Errors;ncdrill.log
```

Contents of Errors and Warnings Log

Reviewing the individual error/warning log files (**ncdrill.log, ipc356_out.log**, and **photoplot.log** in this case) referenced in the Errors and Warnings log file above will help discover the root cause for the errors and warnings encountered.

```
(------------------------------------------------------------------)
(                                                                  )
(    NC DRILL Log                                                  )
(                                                                  )
(    Drawing          : mainboard                                  )
(    Software Version : 17.4S017                                   )
(    Date/Time        : Tue Jul 13 15:09:53 2021                   )
(                                                                  )
(------------------------------------------------------------------)



 Processing NC Parameters file 'nc_param.txt' ...

 WARNING(SPMHMF-325): No NC Parameters file found under the search path defined
in environment variable 'ncdpath' ... using defaults.

 NC PARAMETERS
 -------------

 FORMAT                2.3
 MACHINE-OFFSET        x:0.00000       y:0.00000   (inch)
 FEEDRATE              1
 COORDINATES           ABSOLUTE
 OUTPUT-UNITS          ENGLISH
 TOOL-ORDER            INCREASING
 REPEAT-CODES          YES
 SUPPRESS-LEAD-ZEROES  NO
 SUPPRESS-TRAIL-ZEROES NO
 SUPPRESS-EQUAL        NO
 TOOL-SELECT           NO
 HEADER                none
 LEADER                12
 CODE                  ASCII
 SEPARATE              NO
 SEPARATE-ROUTING      NO
 OPTIMIZE_DRILLING     NO
 ENHANCED_EXCELLON     NO
 SCALE                 1.000000

 WARNING(SPMHMF-363): Units system is being switched when generating drill output file data.


 Drill files being output to directory 'C:/Users/AliceEngineer/AppData/Local/Temp/EDM' ...
 WARNING(SPMHMF-346): No holes were found in this drawing.
```

Log File Referenced in Errors and Warnings Summary: ncdrill.log (truncated)

```
(------------------------------------------------------------------------)
(                                                                        )
(    IPC-D-356                                                           )
(                                                                        )
(    Software Version : 17.4S017                                         )
(    Date/Time        : Tue Jul 13 15:09:53 2021                         )
(                                                                        )
(------------------------------------------------------------------------)


Warning: Integer value 1044682 truncated in columns 36-41  on line 153.


-----------------------------------------
SUMMARY:
  Errors:       0
  Warnings:     1

Info: Most likely cause of "Integer value" errors are:
      - (Most likely) Coordinates in the database exceed IPC's coordinate system.
        Try setting the database origin to center the design to reduce the large
        coordinate values.
      - Large pad sizes. Reduce any large pad sizes.
```

Log File Referenced in Errors and Warnings Summary: ipc356_out.log (truncated)

```
(--------------------------------------------------------------------)
(                                                                    )
(    ARTWORK LOG                                                     )
(                                                                    )
(    Drawing          : mainboard.brd                                )
(    Software Version : 17.4S017                                     )
(    Date/Time        : Tue Jul 13 15:09:52 2021                     )
(                                                                    )
(--------------------------------------------------------------------)



 ****************************************
 WARNING: DRC OUT OF DATE on this layout.
 ****************************************

 ---- Photoplot outline rectangle not found; using drawing extents.


 Can't open parameter file ... using default values:
     DEVICE-TYPE        GERBER6X00
     G-CODES            NO
     STATIONS-PER-WHEEL 24
     FILM-SIZE          24000 16000
     COORDINATES        ABSOLUTE
     OUTPUT-UNITS       INCHES
     FORMAT             5.3
     ABORT-ON-ERROR     abort film & continue
     SUPPRESS-LEAD-ZEROES     YES
     SUPPRESS-TRAIL-ZEROES    NO
     OPTIMIZE           YES
     SUPPRESS-EQUAL     YES
     SCALE              1.0

 WARNING: Generating english output data from a metric database
 may cause slight roundoff error of coordinates in output file.

================================================================
     PROCESSING FILM < CircuitSpace >
```

Log File Referenced in Errors and Warnings Summary: photoplot.log (truncated)

6 Project, Folder and File History

As you perform actions pertaining to items in the vault, many of the input forms include the comments field. The comments you enter are included in the project history. Actions that require the comments field include:

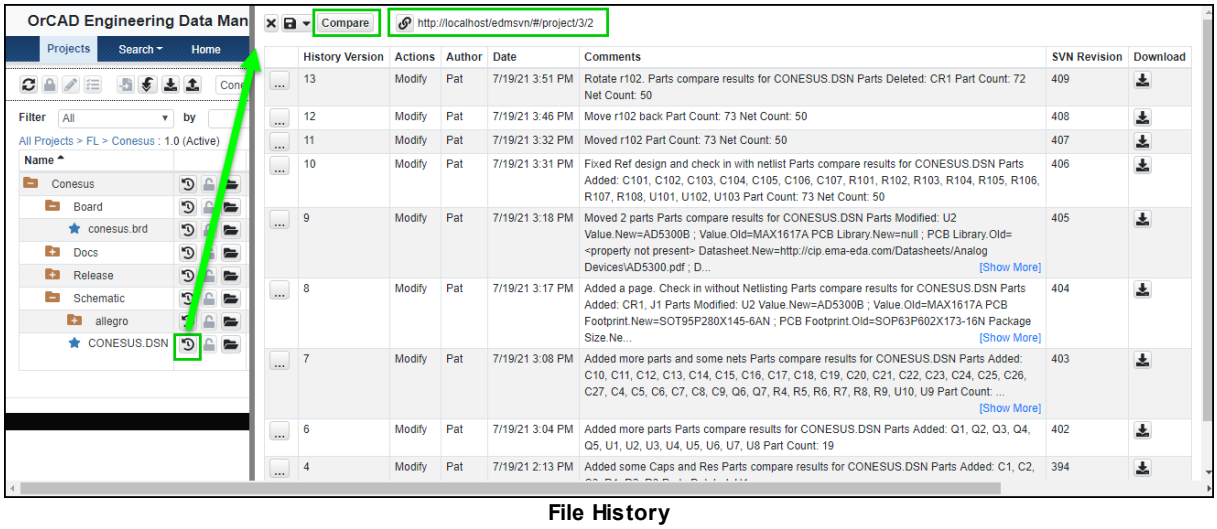
- Add and delete folders, schematics/groups and pages
- Import pages and symbols
- Delete symbols
- Check in of items to the project

The history information for a project or folder shows all history of check in and check out activities on all it's files and sub-folders. A direct link to a file within a project is provided in the history information for a file. History may be downloaded when using a web browser, EDM Desktop application or the Capture plug-in.

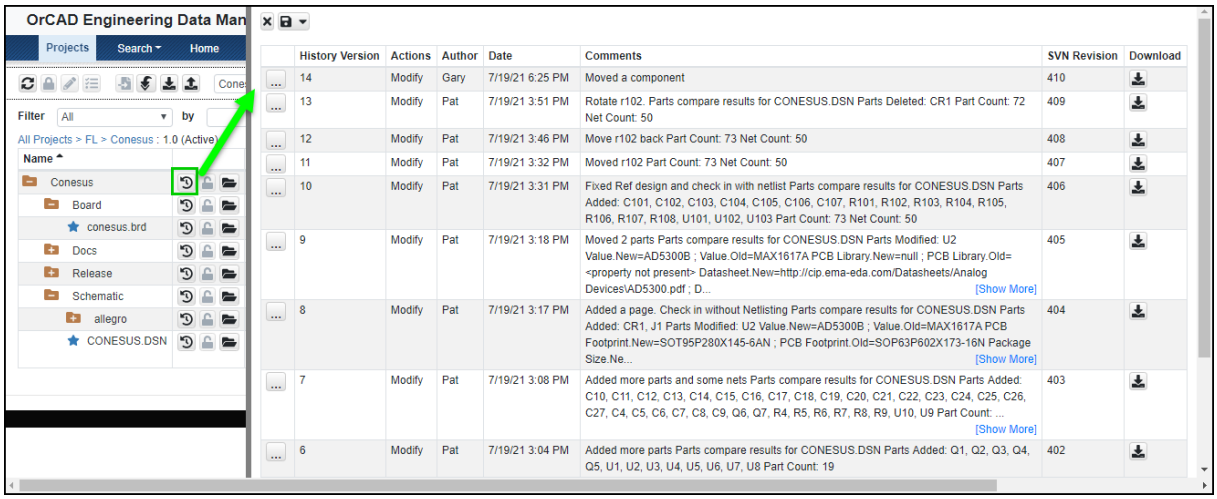
EDM must be opened inside Capture to obtain history version of team designs and libraries. View and download history from the project or folder level to recover deleted files, pages or schematic symbols.

To view the history of any item in a project, click the **History** button located next to the item name. The following two figures show examples of the history of a file and a folder respectively. The columns of information provide are the same for both. The file history enables you to review file differences by comparing two versions certain file types as well as obtain a link to file within the project.

You may copy the file link by selecting the **copy link** button then open the file from the link by pasting the link on your Home page.



File History



Folder History

The columns of data in the history provide the following information:

- **Ellipses Button -** detailed history log for the selected version. Select the button of the version that you want more information to open the History Version Log or Team Design History.
- **History Version** - internal vault version reference number used to keep track of change events on the project.
- **Actions** - all actions associated with the update event. Actions that can be performed on items include Add, Delete and Modify within a project. If a single check in modifies one file and adds another, both Modify and Add actions will be shown.
- **Author** - user that made the update to the vault.
- **Date** - date and time stamp of when the update was made.
- **Comments** - comments entered by the user as well as through the use of auto comments for Capture files and auto comments for Flat Team Designs. Select the **[Show More]** link to see the full comment when a long comment has been added.
- **SVN Revision** - This column is shown only when your EDM set up uses SVN server as the Vault. The corresponding SVN revision is provided in case it is required for debug.
- **Download** - allows the user to download a zip file containing a snapshot of the associated vault version using the Download button. Select the location of your download file from the file selector that displays. The zip file of your history snapshot is placed into your selected folder.

Snapshot zip file formats for Vaults using SVN Server are downloaded in the following format:

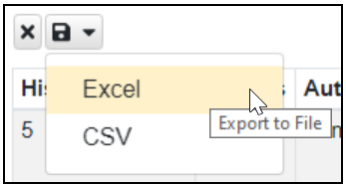
- Project: <ProjectName>_r<SVN Revision>_<MMDDYYYY>.zip
- Folder: <ProjectName>_<FolderName>_r<SVNRevision>_<MMDDYYYY>.zip
- File: <ProjectName>_<FileName>_r<SVNRevision>_<MMDDYYYY>.zip
- Team Design Pages: <ProjectName>_<FolderName>_r<SVNRevision>_<MMDDYYYY>.zip

Snapshot zip file formats for Vaults using File System are downloaded in the following format:

- Project: <ProjectName>_r<History Version>_<MMDDYYYY>.zip
- Folder: <ProjectName>_<FolderName>_r<History Version>_<MMDDYYYY>.zip

- File: <ProjectName>_<FileName>_r<History Version>_<MMDDYYYY>.zip
- Team Design Pages: <ProjectName>_<FolderName>_r<History Version>_<MMDDYYYY>.zip

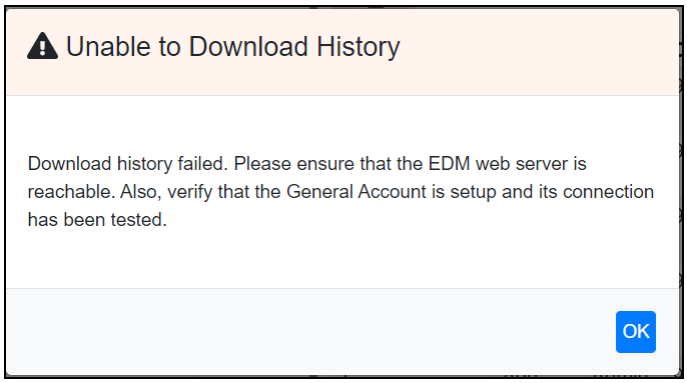
If you would like to export a listing of the history into Excel or CSV format, select the drop-down option with the **Export to File** button and select your choice of format for export.



Historical information export is available for history after EDM version 5.0. The default filename format of the exported file is the following format:

- Project: <ProjectName>_History_<YYYY-MM-DD-mm-ss>.xlsx (or .csv)
- Folder: <FolderName>_History_<YYYY-MM-DD-mm-ss>.xlsx (or .csv)
- File: <FileName>_History_<YYYY-MM-DD-mm-ss>.xlsx (or .csv)

Note: Downloading of history when using a web browser to open EDM requires the General Account to be set up with valid credentials. If you received the following *Unable to Download History* error, notify your Administrator to setup the General Account in the **Admin** menu, under **Vault Setup.**

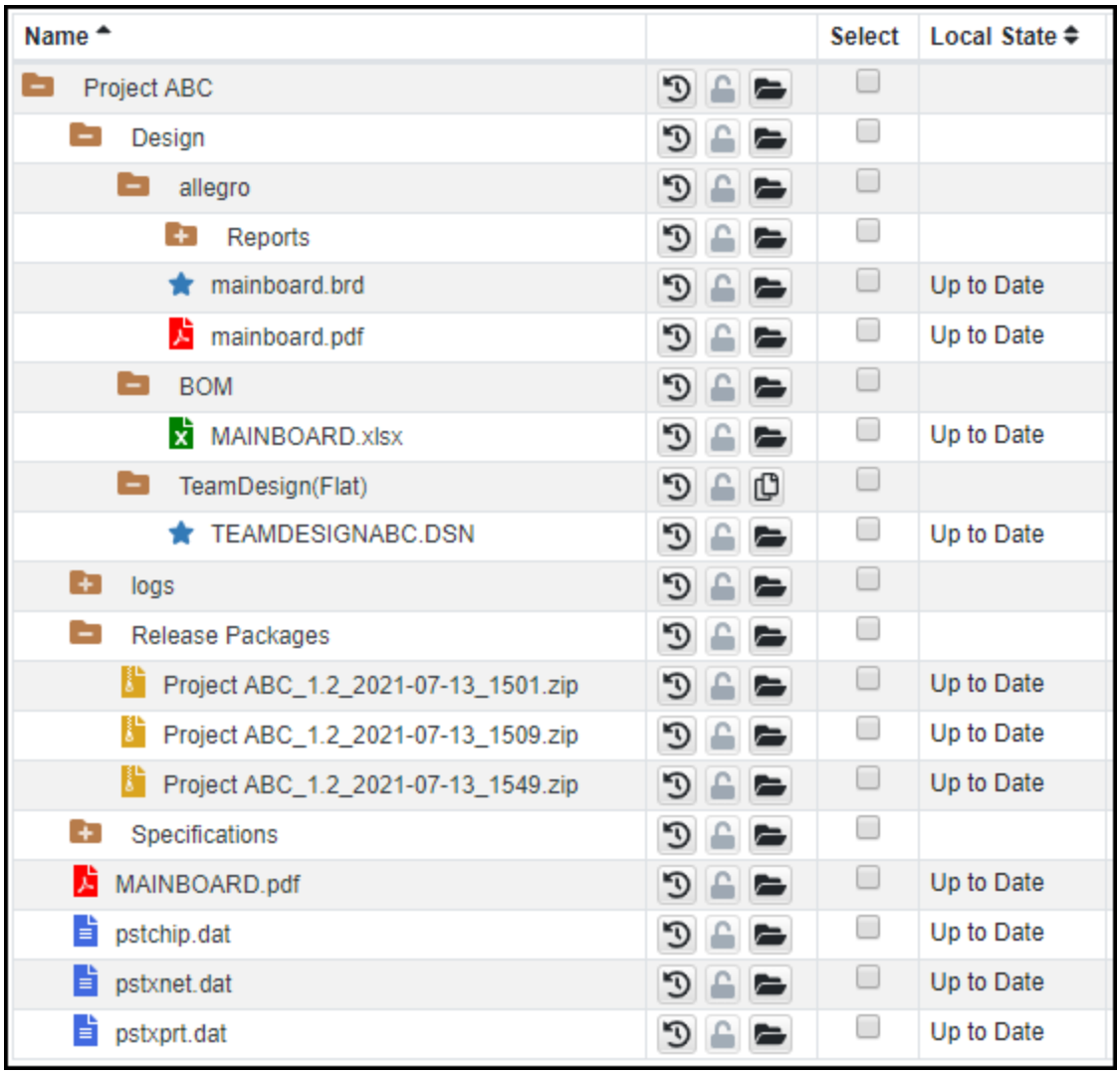


Unable to Download History Error

6.1 File and Folder Icons

The File and Folder Icons allow users to more quickly identify items in a given project. This is often helpful when looking for a file to check out, and it can be used in conjunction with the filter and column sorting functions.

The following figure shows an example project with a mix of expanded/collapsed folders, managed board and design, frozen Team Design folder, and various other file types.



Example Project with Various File/Folder Icons

EDM loads folder and file icons based on the table below.

Icon	Folders/Pages
	Folder Collapsed (+)
	Folder Expanded (-)
	Team Design (Unfrozen)/Team Library Folder
	Team Design Folder (Frozen)
	Team Design Page
Icon	**Files (Extensions)**
	EDM Managed Board (.brd)
	EDM Managed Design (.dsn)
	MS Excel (.xls/.xlsx)
	Text BOM/CSV (.bom/.csv)
	MS Word (.doc/.docx)
	Text/Config/etc (.txt/.ini/.dbc/.xml/.html/.config/.json/.c/.h/.dat/.lic/.log/.rtf)
	MS PowerPoint (.ppt/.pptx)
	Portable Document Format (.pdf)
	Archive (.7z/.rar/.zip)
	Image (.bmp/.dng/.gif/.jpg/.jpeg/.png/.tif/.tiff)
	Audio (.aac/.aif/.mp3/.ogg/.wav/.wma)
	Video (.avchd/.avi/.mkv/.mov/.mp4/.wmv)
	All Other Files
	Footprint Draw File (.dra)
	Footprint Compiled File (.psm/.bsm/.fsm/.osm/.ssm)
	Footprint Padstack (.pad)
	Footprint STEP Model (.step/.stp)

File and Folder Icon Table

6.2 History Version Log

Selecting the **ellipsis** button opens the log of the history information and listing of files pertaining to the check in of a specific version.



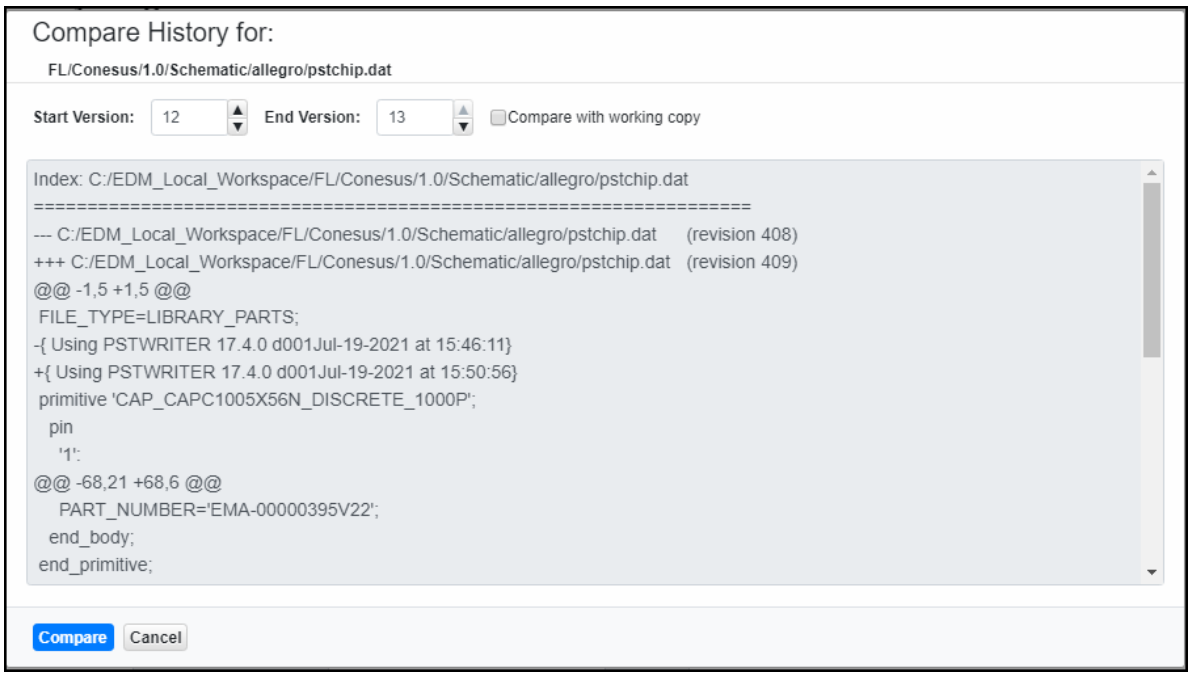
The History Log provides the following information:

1. **Version** - Selected version (EDM project history version number). Select a version number to view detailed log on a different version.
2. Listing of changed files with the following information:
   a. **Compare** - The compare **File** button is active when file differences can be compared. Supported file formats are text files and Capture design (DSN) files. Select the **File** button to view file differences between 2 versions of a file.
   b. **Path** - List of all files associated with the selected version. Review Team Design History for page and symbol path formats.
   c. **Action** - Shows the action taken on each file for the selected version. Actions include: Add, Missing (Footprint projects only), Modify, and Delete. Footprint projects will include an action of Missing when an import or check in is missing one or more related files.
3. **Comment** - Comment that was entered by the user who performed the action or that was auto commented.
4. History information for the selected version:
   a. **Version Create Date** - the date in which the selected version was created.
   b. **Version Author** - the user who performed the action(s) for the selected version.
   c. **History Version** - the selected Version (EDM project history version number).

d. **SVN Revision** - the corresponding SVN revision for the selected version, which may be useful for troubleshooting purposes. This only applies when Repository Server Type is **SVN**, not **FileSystem**.

6.3 File Differences

You may compare 2 versions of a text or OrCAD Capture design file with EDM. The compare functionality is available from History. When you select the compare button from the file history, the following history compare options appear. You may also select the file button from History Version Log to open the history compare options.



Compare Text Files

To compare 2 versions of a text file:

1. Select 2 different versions number that you want to compare.
2. Select the **Compare** button to start the compare.
3. Files with contents in text format will be compared showing any differences between 2 file versions.

Lines starting with a Minus (-) character shows that texts in the Start Version are no longer in the End Version.

Lines staring with a Plus (+) character shows that texts in the End Version but are not in the Start Version.

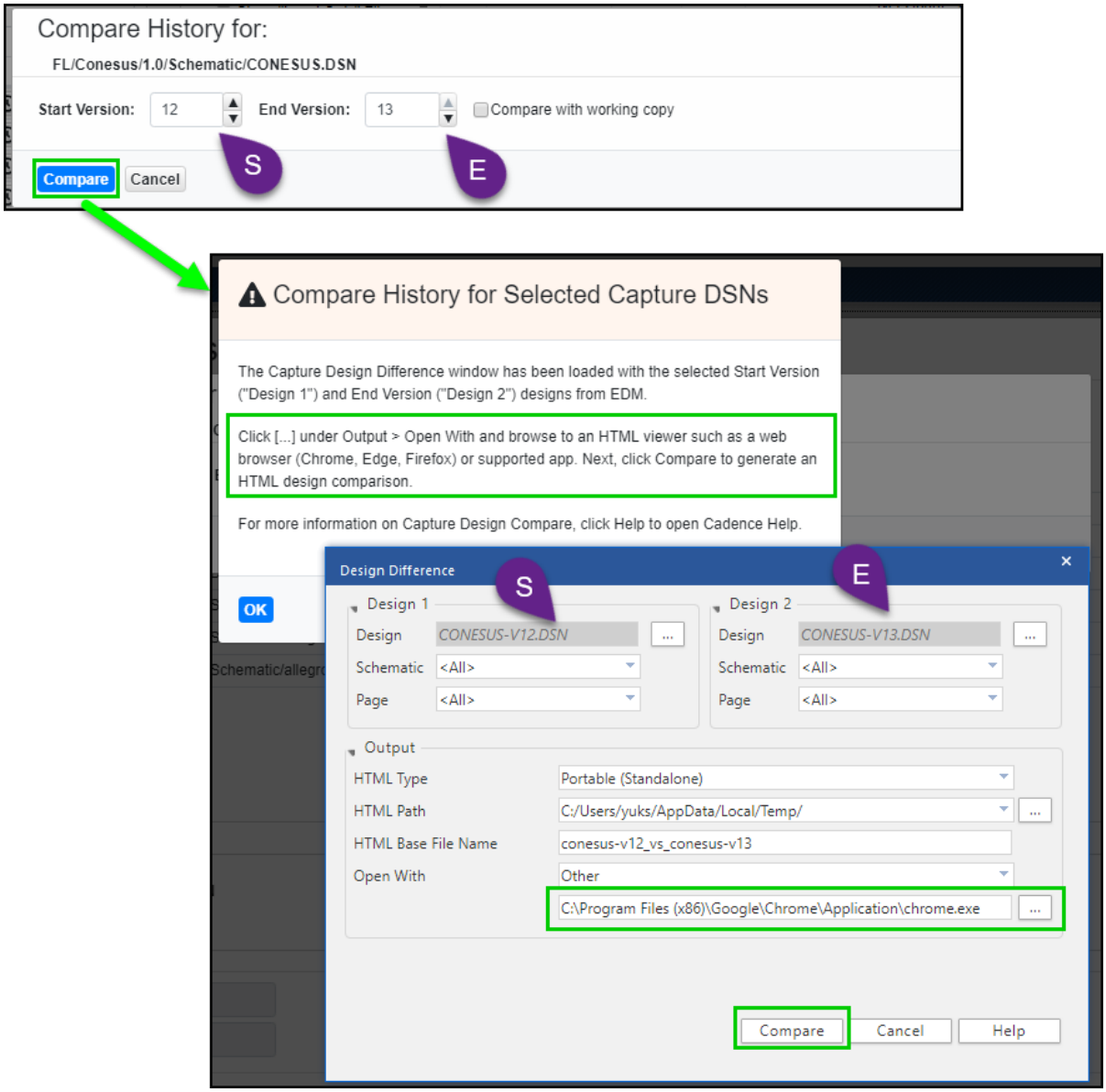
You may compare a version in EDM to your working copy in your local workspace by selecting the Compare with working copy checkbox.

Compare Capture Design Files

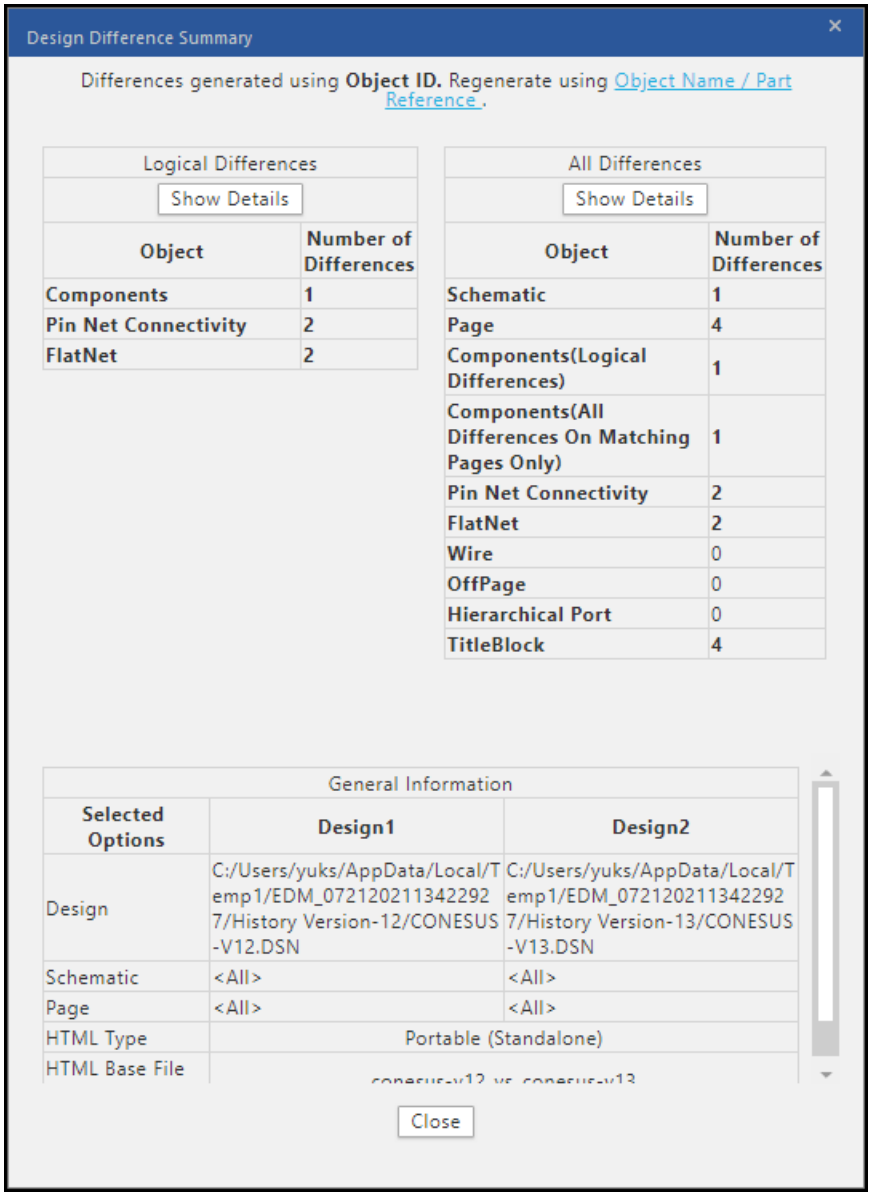
The built-in Capture design compare functionality is used to compare two versions of a Capture design file.

To compare two versions of a design file:

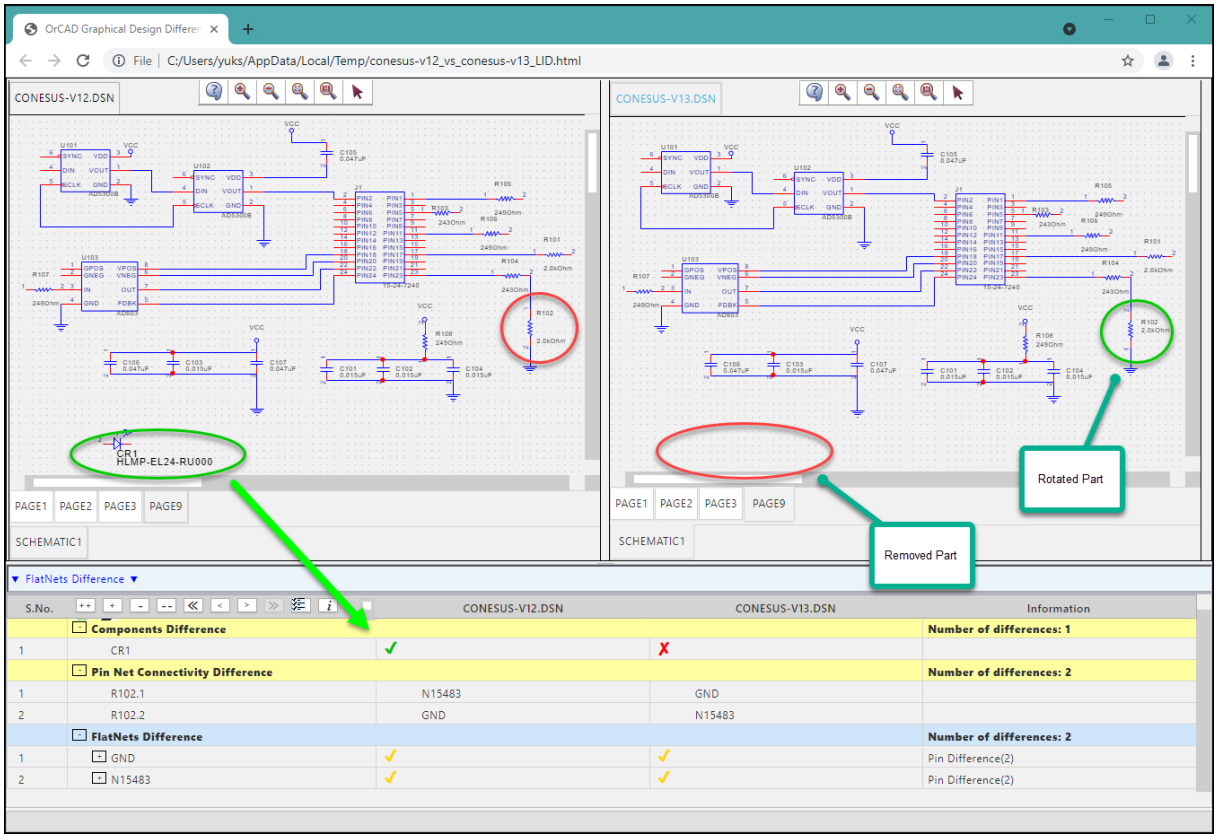
1. Select the 2 different versions number (Start and End) that you want to compare.
2. Click the **Compare** button to launch the Capture Design Difference feature.
3. An instruction and Capture Design Difference dialogs appears. The 2 versions of your design is auto-populated in the Design Difference dialog. Configured your browser executable location if not already configured.
4. Select the **Compare** button to start the compare. Select the **Help** button to review more information on design compare.



5. A summary is provided for logical and all differences. Select the **Show Details** button corresponding to the type of difference information you want to review. The **Show Details** button is disable when no differences are found.

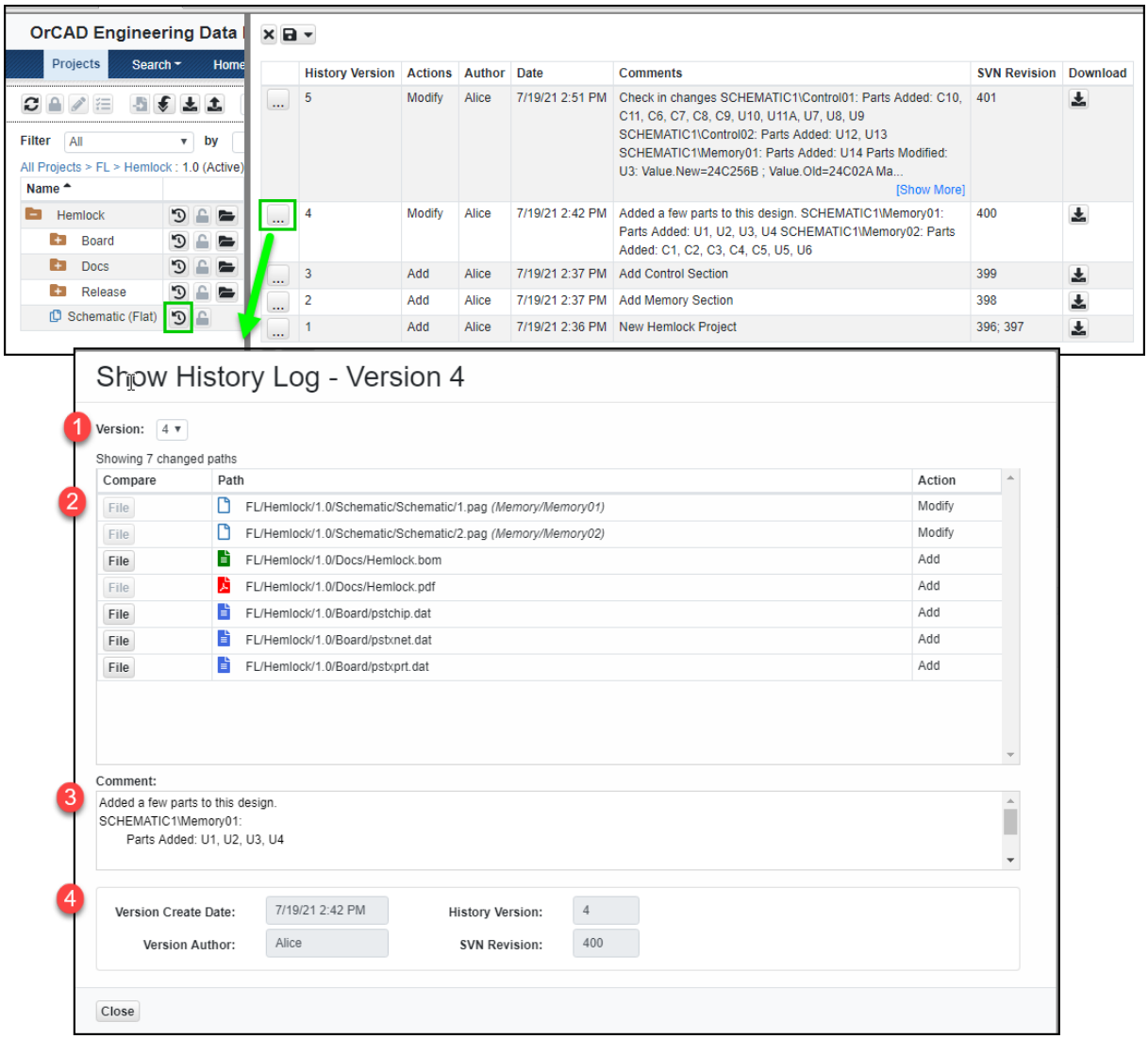


The following example shows logical differences where 1 part is removed and another part has been rotated.



6.4 Team Design History

Selecting the **ellipses** button next to a team design or library folder opens the log of the history information and listing of symbols or files and pages pertaining to the check in of a specific version.



The History Log provides the following information:

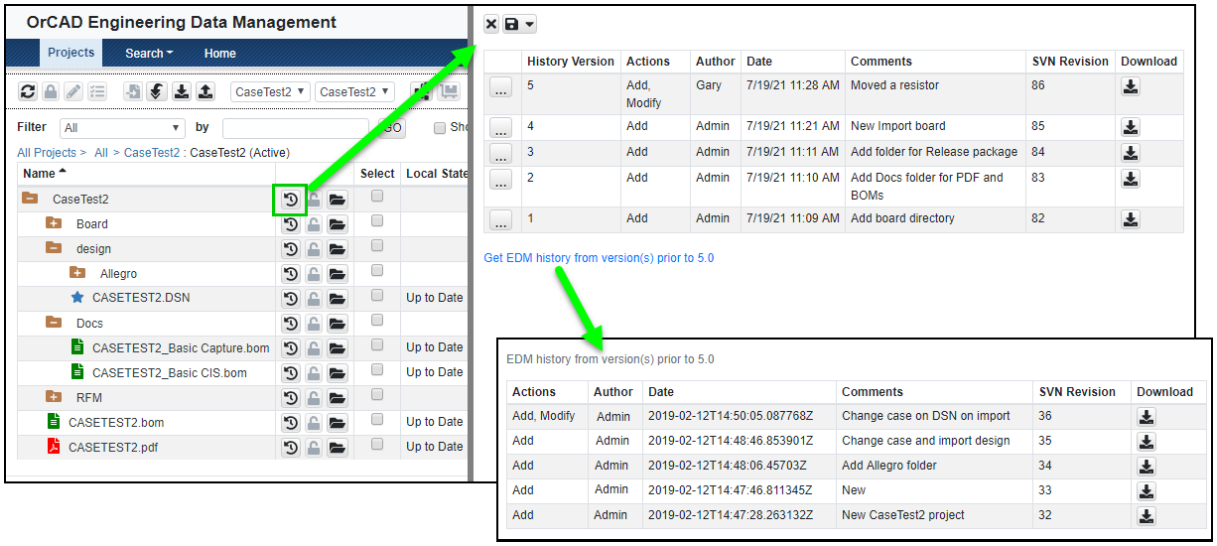
1. **Version** - Selected version. You may change version number to view detail log on a different version
2. Listing of files checked in with the following information:
   a. **Compare** - The **File** button shows for files that differences can be provided. File formats that are supported are text files and Capture design (DSN) files. Select the **File** button to view file differences between 2 version of a file. Comparison of 2 pages or 2 symbols are not supported.
   b. **Path** - List of all symbols or files and pages associated with the check in. Page and symbol names are shown in parentheses next to the path.
   c. **Action** - Shows the action taken on the file as part of the check in. Actions include: Add, Modify and Delete.
3. **Comment** - Comment entered by the user who performed the check in.
4. History information on the check in -
   a. Version Create Date - the date in which this version was checked in.
   b. Version Author - the user who performed the check in.

c. History Version - the EDM project history version number of the check in

d. SVN Revision - the corresponding SVN revision for the check in. This is provided in case it is required for debug.

6.5 History From Versions Prior to 5.0

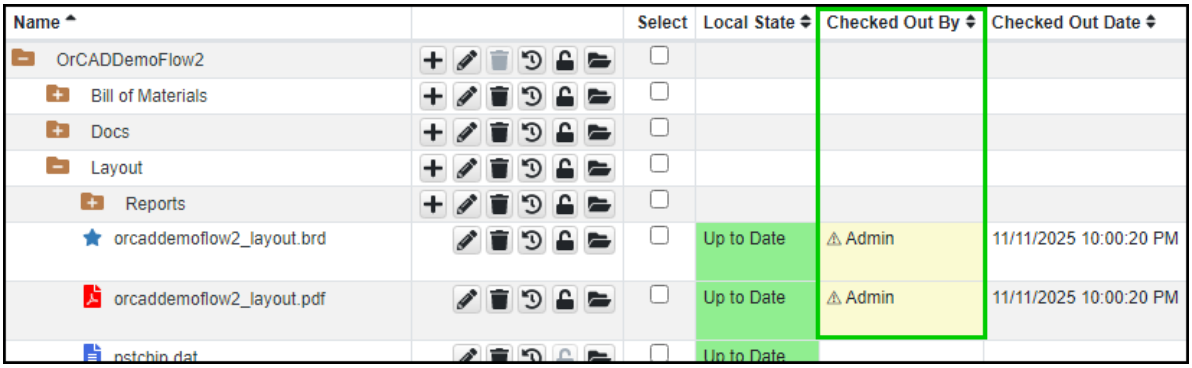
History prior to version 5.0 of EDM for Vaults using SVN was saved in a different form. If you have historical files from versions of EDM prior to 5.0, you will see a link to **Get EDM history from version(s) prior to 5.0**. Click on the link to see more history, as shown in the following figure. Vaults using File System will not show this link.



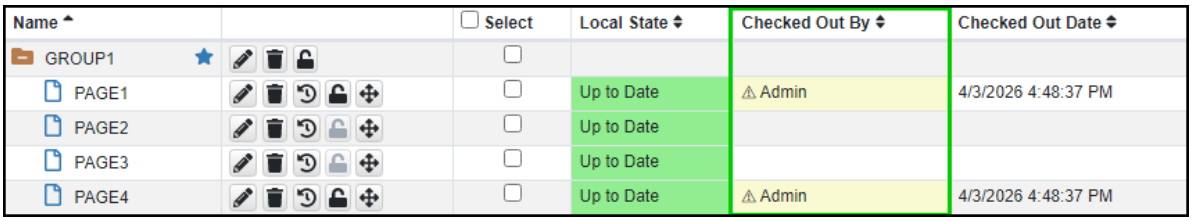
7 Check Out

Project items must be checked out before work on the files begins. You may check out any project item you are assigned. Project items that can be checked out include files, Team Design pages, footprint files, and Schematic/Team Library symbols.

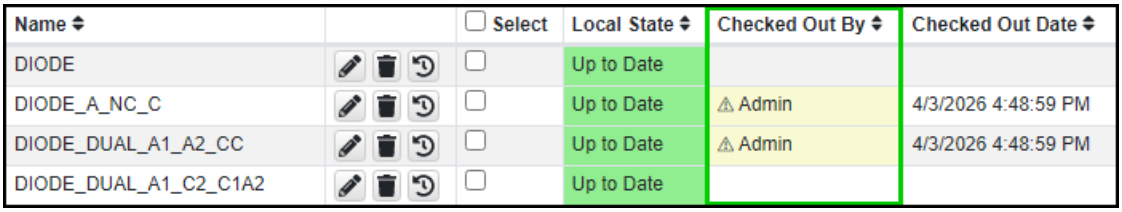
After items are checked out, the user who checked them out has exclusive editing permission until they are checked in or the checkouts are canceled. Other users can open the files, but should not modify them without checking them out. The **Checked Out By** field highlights the user name when the item is checked out by a user other than the one viewing the project.



Files Currently Checked Out By Another User



Team Design Pages Currently Checked Out By Another User



Team Library Symbols Currently Checked Out By Another User

After selecting the desired project from the list in the Project Dashboard, log into the vault (if prompted). You may download a full copy of the project or check out specific items from the Project Details view. Downloading a copy of the project does not allow you to check in changes made to files. When you want to make changes and check them in, you must first check out the desired files.

You need to be in the Team Folder Details view before you can check out team folder items such as Team Design pages and Schematic/Team Library symbols.

Check out can be performed by users assigned to the items or by assigned leaders for the project.

- Download and Check Out are not available when EDM is opened in a web browser.
- Download and Check Out are allowed from the Capture plug-in and Desktop application, both of which will show details for **Local State**.
- Download and Check Out of Team Design or Team Library folders are not available from the Desktop Application or a web browser.

Refer to the Appendix for details regarding Warnings and Errors a user may encounter during Check Out.

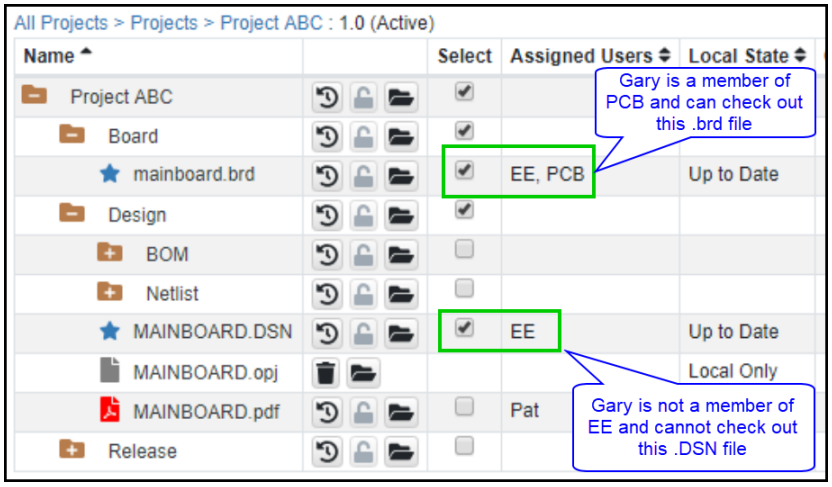
7.1 Check Out Files

To obtain a copy of all files in the project, click the **Full Project Download** button located in the Project Action Buttons in the upper left corner of the Capture project view or General project view. This downloads all files without checking any files out. The **Select** column lets you select the item(s) you want to Check Out.

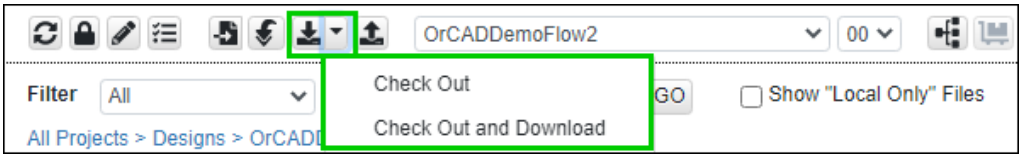
Locate your file by using the filter options if needed. Select the checkbox in the **Select** column for the file(s) you want check out, then click the **Check Out** button to check them out. The check out will not download the files, so if you haven't yet downloaded them, the Local State will still show No Local Copy after you checked the files out in this scenario.

If you'd like to download the files during check out, click the **Check Out** drop arrow located in the Project Action Buttons in the upper left corner of the Capture project view or General project view and select **Check Out and Download.** Alternatively, selecting **Check Out** is the same as clicking the **Check Out** button described above.

Refer to section on Check out Manage Capture file when checking out managed design and board files or files related to the managed design in a Capture Design Project.



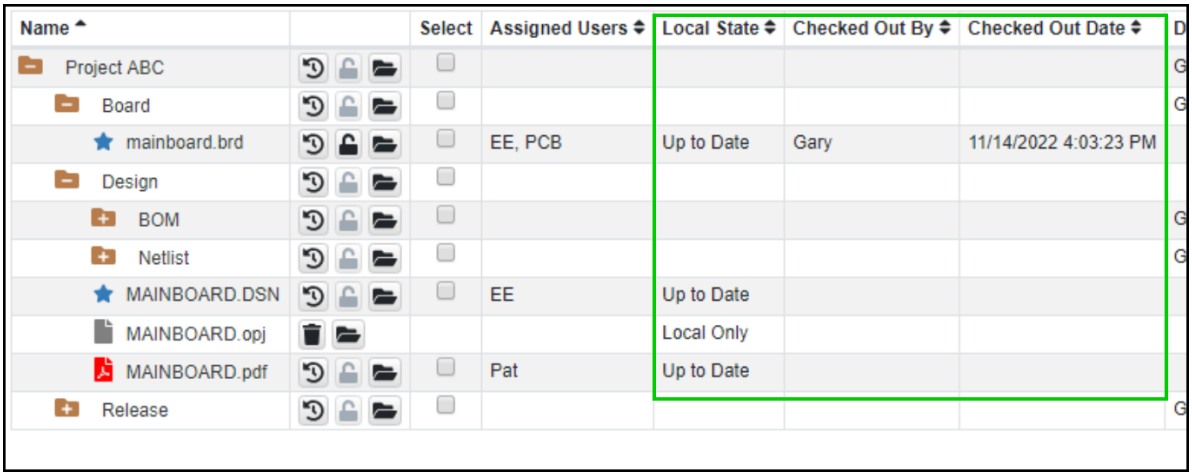
Check Out Selection



Check Out Options

Local Workspace and Check Out Status of files

After you perform download or check out, the **Local State**, **Checked Out By,** and **Checked Out Date** fields update to reflect the local workspace state and Vault state. The figure that follows shows Gary checked out the **mainboard.brd** and the board is "**Up to Date**."



Files Currently Checked Out

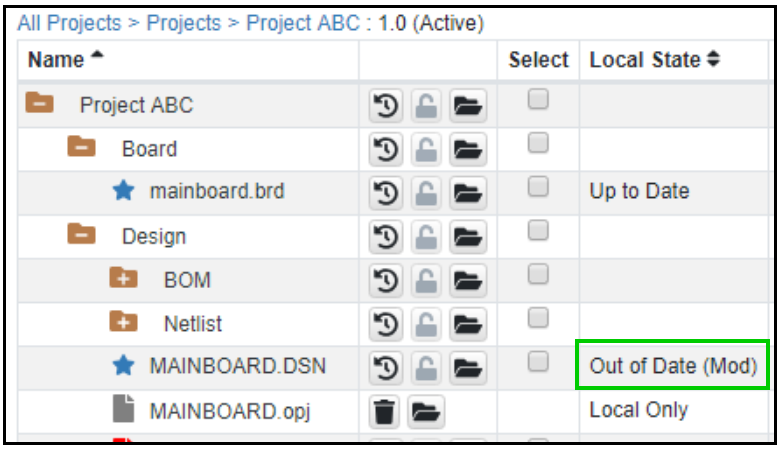
The following columns update when a user check items in and out of the vault:

- **Checked Out By** - shows the user who has the file checked out. If you need to work on a file that is currently checked out by another user, you will need to contact that user or an assigned leader to Cancel the Check Out. The user name in this field will be highlighted as shown above when the item is checked out by a user other than the one viewing the project.
- **Checked Out Date** - shows the date when the file was checked out.
- **Local State** - represents the state of your files in the local workspace. Folders will not have any Local State. Detailed information can be found in Local State Definitions.

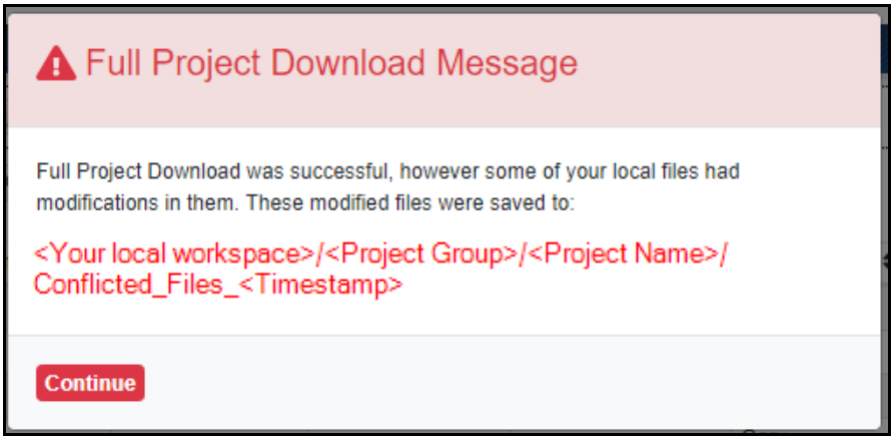
After your files are checked out, you may open a file directly from EDM by selecting the **Open** button located next to the item you want to open.

Note: You may receive an error when a project file is opened during a Full Project Download or Check Out and Download process, if EDM is unsuccessful updating all files in your local workspace.

If you have edited a file before checking it out, and it has since been modified and checked in by another user, the Local State for your file will show that it is **Out of Date (Mod)** as shown in the following figure. When you try to download or check out from a project and you have any **Out of Date (Mod)** files, a conflict will occur. The conflict is due to a version mismatch between your local workspace and the EDM vault. EDM will save a copy of your file to another location and update your local workspace to the latest version.



Out of Date (Mod) Local State

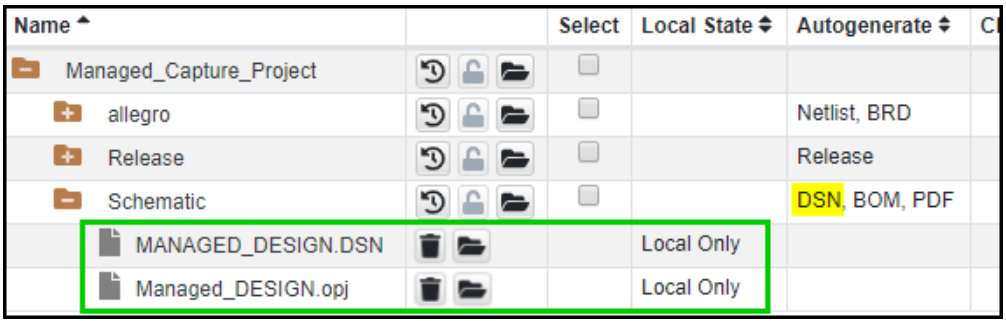


Conflicted Files After Download

7.2 Check Out Managed Capture Files

When a project is created as a Capture Design project, EDM handles the check out process of the managed Capture design and related files differently from standard files that are not part of a managed project. Capture projects in the Project Details View may have folders that are designated for the auto-generated files, DSN, BOM, Netlist, and PDF. These files are listed in the **Autogenerate** column.

When a Capture project is first created, the project will not have any files. Before working on a project, you will need to download the project structure by clicking on the **Full Project Download** button. When you bring the full project to your local workspace, it downloads Team Design pages and Team Library/Schematic Symbol library symbols if they are part of the project. If a managed design has not been imported or added during project creation, the initial download of an empty Capture project will auto-create the .DSN (managed design) and .opj (Capture project file). Both of these files will have the state of **Local Only** since they were created new and did not come from the vault, as shown in the figure below.

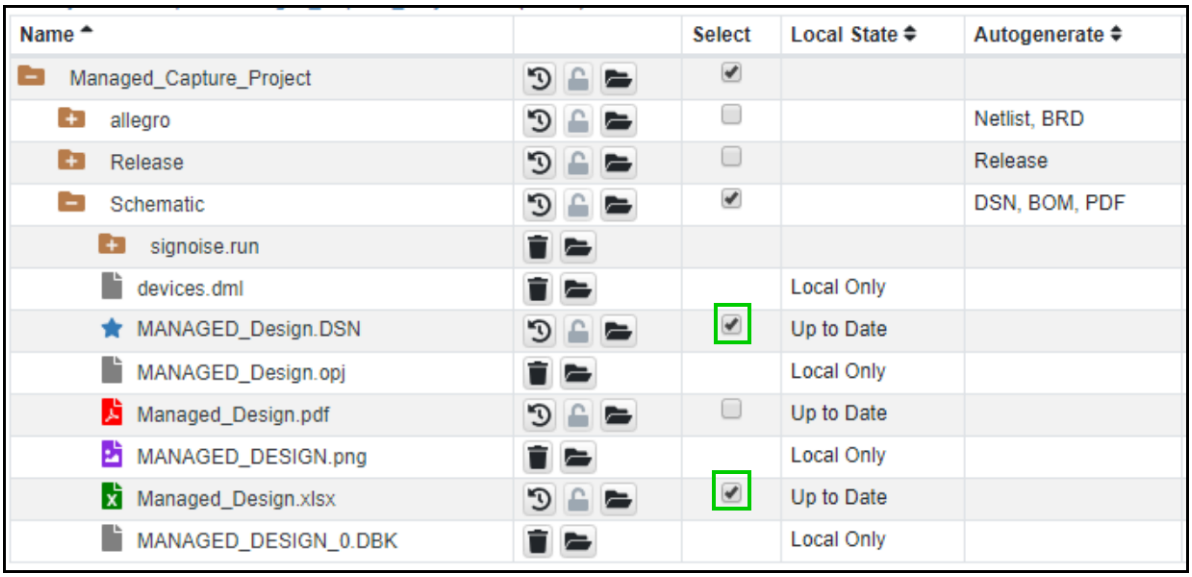


Initial Download of Capture Project

Once the new files have been imported or checked in the first time, they will be available for Check Out as described in Check Out Files, EDM notifies users when an action taken may cause the files related to the managed project (designated for auto-generation) to become out of sync with the managed schematic or board. Files generated from the managed design are DSN, Netlist, PDF, and BOM. Files that are generated to the managed board are DB Doctor, EDM Validation, and PDF.

Note: The managed design and Capture project files only auto-generate when EDM is running from Capture. Files associated to the managed board are auto-generated from the EDM Desktop.

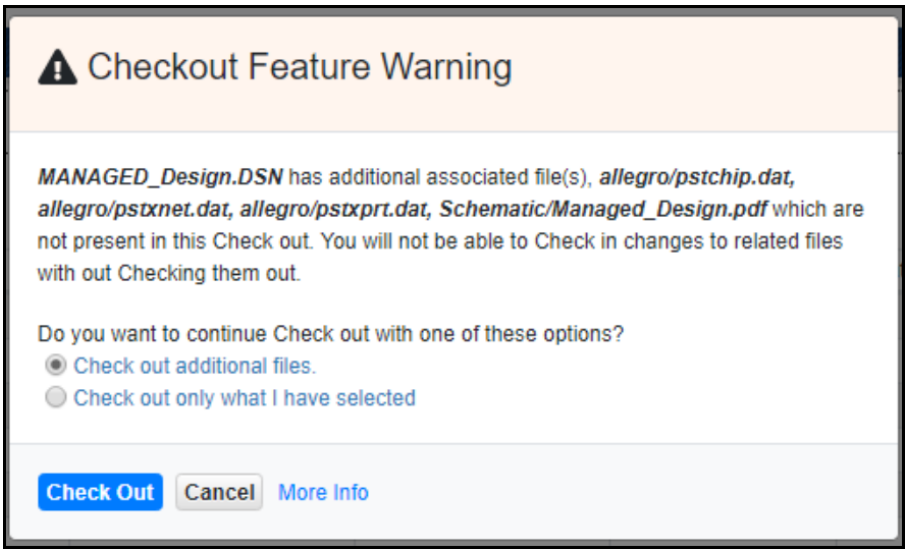
When you check out a managed design file, EDM may alert you regarding the check out of files related to the managed design. The following figure shows an example with the managed design and the associated BOM selected for check out. The managed design project file (.**opj**) is stored locally rather in the vault. If you do not have a local copy of the .opj file, it is created when the managed design is checked out.



Example of Check Out Selection for Managed Design

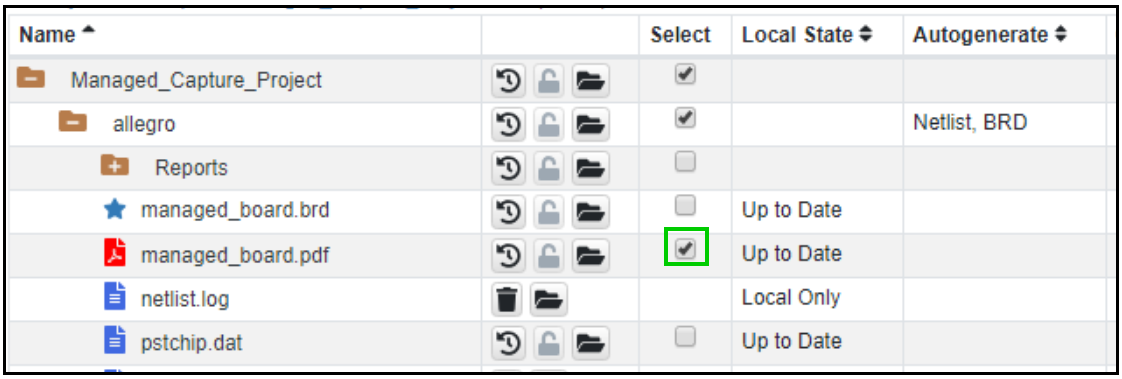
The following is the *Check Out Message* for the selection of files associated to the managed design shown in the previous figure. Only the managed schematic design and BOM were selected for check out,

but the Checkout Feature Warning prompts you to optionally check out the related Netlist and (Schematic) PDF files.



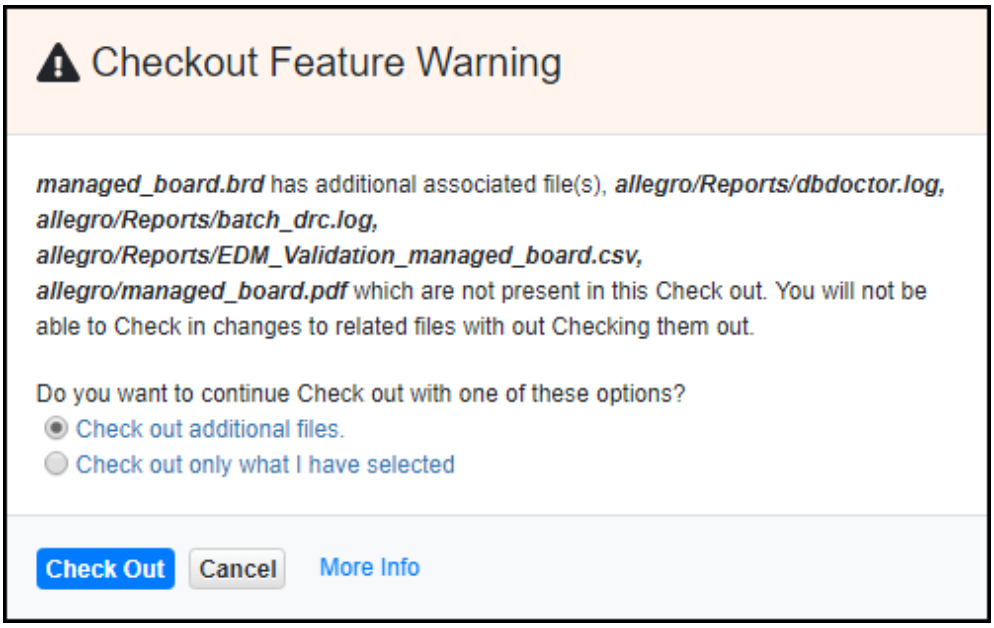
Check Out Pop-up Message for Associated Design Files

When you check out a managed board, EDM may alert you regarding the check out of files related to the managed board. The following figure shows an example with the managed board selected for check out.



Example of Check Out Selection for Managed Board

The following is the *Check Out Message* for the managed board shown in the previous figure. Only the managed board was selected for check out, but the Checkout Feature Warning prompts you to optionally check out the related DB Doctor, EDM Validation Report, and (Board) PDF file.



Check Out Pop-up Message for Board Files

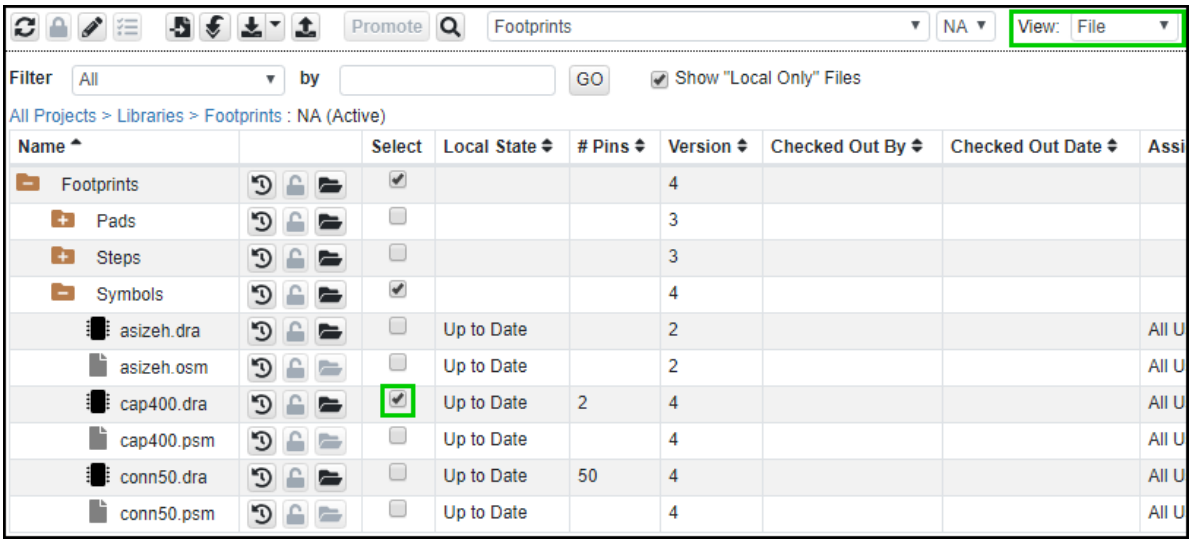
Note: For a full list of messages with descriptions, refer to Warnings and Errors.

7.3 Check Out Footprint Files

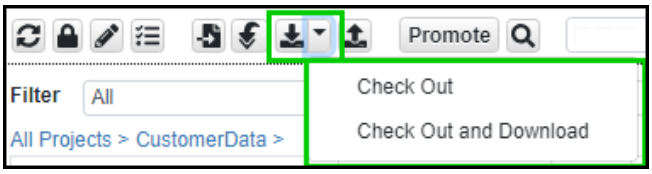
To obtain a copy of all footprints in the footprint project, click the **Full Project Download** button located with the Footprint Project Action buttons in the upper left corner of the footprint project view. This downloads all files without checking any files out. The **Select** column lets you select the item(s) you want to Check Out. You may check out footprint files from any Footprint Project View.

Locate your file by using Views and filter options. Select the checkbox in the **Select** column for the file(s) you want check out, then click the **Check Out** button to check them out. The check out will not download the files, so if you haven't yet downloaded them, the Local State will still show No Local Copy after you checked the files out in this scenario.

If you'd like to download the files during check out, click the **Check Out** drop arrow and select **Check Out and Download.** Alternatively, selecting **Check Out** is the same as clicking the **Check Out** button described above.



Sample Selected Footprint File for Check Out from File View

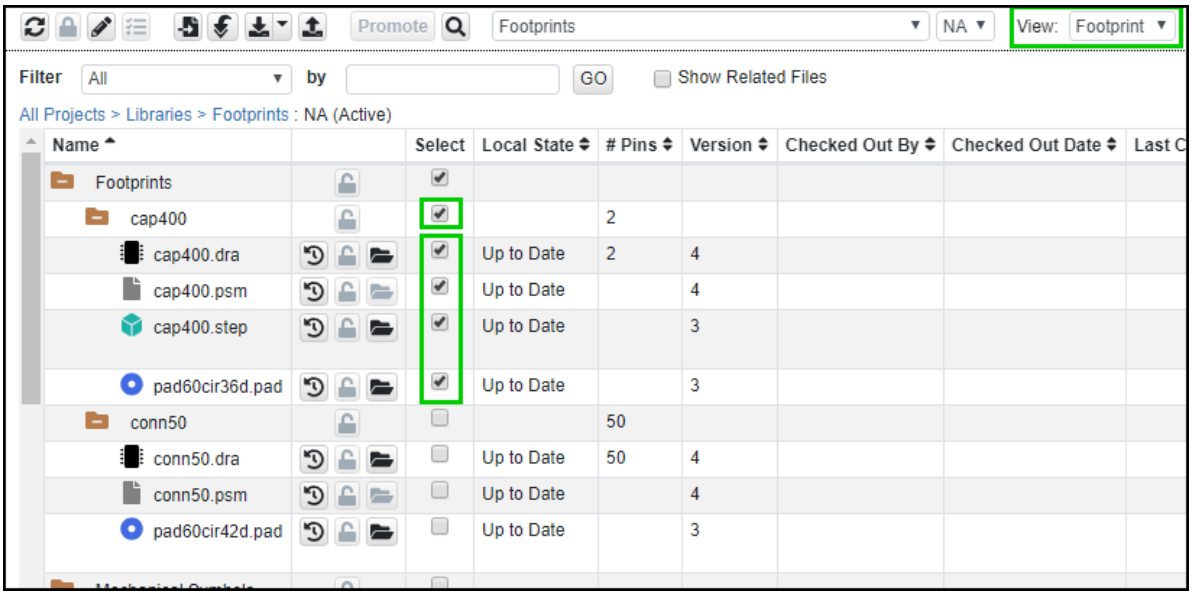


Check Out Options

When you check out of a .dra file or its corresponding compiled (.psm, .bsm, .fsm, .osm, or .ssm) file, EDM automatically checks out the corresponding file and the success message displays with the total files checked out. Similarly, canceling checkout of a .dra file or its corresponding compiled (.*sm) file will cancel checkout for both files in the (.dra/*.sm) pair.



To check out a group of files, select the check box for the group of files and all files within the group will automatically be selected. The group names will vary based on the selected View. In the following example, clicking the footprint group *cap400* automatically selects all 6 files belonging to that footprint group (shown as a folder).



Sample Selected Footprint Group for Check Out from Footprint View

7.4 Check Out Team Design Pages

When a design is enabled as a Team Design, users are able to download all pages of the design and check out individual pages from the Team Design Details View. Check out permissions are based on page assignments, similar to the process for Checking Out Files. Any user who is assigned to a project, can download a read-only version of the full design from the Team Design folder. However, only users specifically assigned to pages are allowed to check out those pages.

Hierarchical designs have pages that belong in schematics. Pages in a flat design can be organized in groups. Groups are a grouping of pages in a flat design. Grouping of pages help with management of the pages. The selection of pages is similar to selection of files, as are the selection of groups/schematics to folders.

When you select or deselect a group or schematic for check out, all pages associated with the group or schematic are automatically selected or deselected. To download a copy of all pages in the Team design folder and merge pages to recreate your design file, click the **Full Download** button. This can also be done using a **Full Project Download** in the Project Details view or **My Projects** list from the Home page. The following options are available for Full Download, all of which require the Capture Client plugin to merge the pages into a design:

- **Full Download** - All items are merged to a single file. When a full download is performed to a team design that has been previously downloaded, pages with the same name will be updated. Any new pages will be added. Pages in your local design that haven't been check in are left untouched.
- **Full Download with Update Cache** - With this option, in addition to the **full download**, a forced update cache is performed on the local workspace design file as well as on each downloaded page before merging into your local design file. If the pages in the Team Design were imported from a different design file, the symbols of the original pages may

differ. The update cache option ensures symbols are updated to the latest library symbol before being merged. The Update Cache option takes longer for the download.

- **Delete Local and Clean** - With this option, you local design is first deleted and then a **Full Download with Update Cache** is performed. If pages within the design have been renamed, a Full download will add pages with the new names to your local design containing the older page names. This option enables you to get a clean copy from the Vault. Any new pages that are in your local design and and haven't been checked in will be lost when using this option

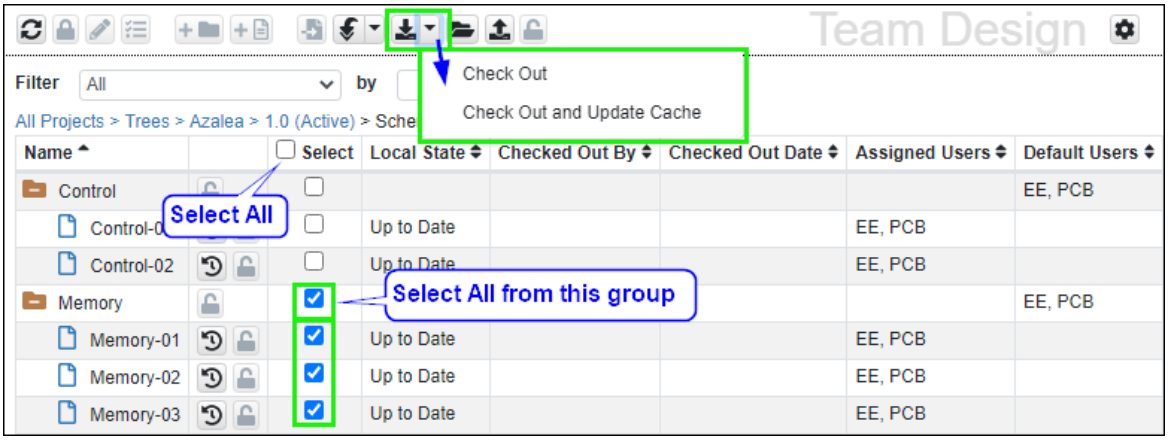
Note: If you change a page name, anyone who downloaded the team design prior to the name change and downloads the team design after will see both pages. EDM will not remove any pages in a user's local design, since there is no way to differentiate new pages added (and awaiting check in) from pages that no longer belong in the design file. When pages have been renamed by anyone, all other users should delete their local copy of the merged design to get a fresh copy of the design file.

After making your selections for check out, click the **Check Out** button to get new pages and merge them into the design (.DSN) file in your local workspace. The design filename of the Team Design is set by the assigned leader and should not be changed. The following options are available for Check Out, all of which require the Capture Client plugin to merge the pages into a design:

- **Check Out** - A **Full Download** is automatically performed when you check pages out with this option. This option is faster since no additional processing is required. If you find that the downloaded pages contain symbols with problems, you may use the **Check Out and Update Cache** option to ensure that the symbols are updated to the latest library symbol.
- **Check Out and Update Cache** - A **Full Download with Update Cache** is automatically performed when you check pages out with this option.

Note: The central library location path must be included in your Capture.ini file.

The following figure shows how the user has permission to check out Memory Group using one of the two available check out options.



Team Design Download Options

Status of Pages

After download, the Local State, Checked Out By, and Checked Out Date fields update to reflect the latest state of the pages as explained in the following descriptive list.

- **Checked Out By** - shows the user who has the file checked out. If you need to work on a page that is checked out, contact that user or an assigned leader to Cancel the Check out.
- **Checked Out Date** - shows the date the page was checked out.
- **Local State** - indicates the state of pages in your local workspace. Groups do not have a Local State. Detailed information can be found in Local State Definitions.

Local Design File

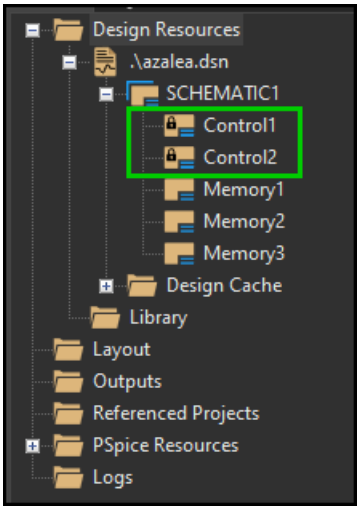
Once you receive the confirmation that the download or check out is complete, you may open your local design file in Capture by selecting the **Open Design** button that is located with Team Design Action buttons in the upper left corner of the Team Design View.



Select Open Design

When you check out pages or conduct a full download of a flat design, all pages are placed into SCHEMATIC1 of the assigned design file name regardless of the group.

Pages that are not checked out become locked and may not be modified, as shown in the following figure:

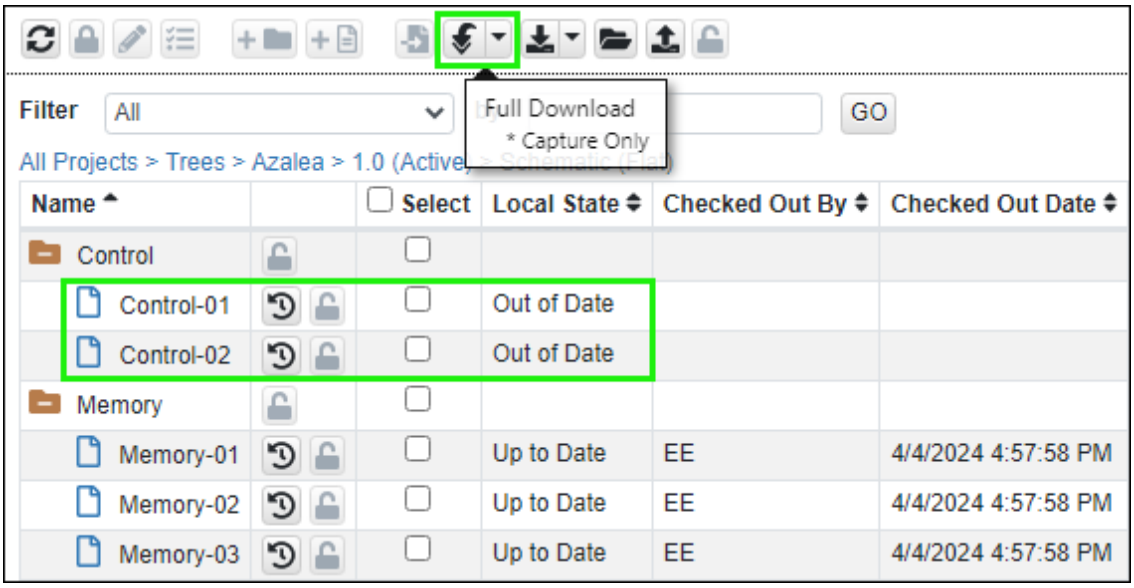


Pages Locked in Capture

For hierarchical designs, the full download or check out of pages are placed into corresponding schematic of the assigned design filename. If selected for download, the root schematic becomes the root schematic of the downloaded design.

Update Out of Date Pages in Local Design

When multiple users work on a single design, you may see out of date pages when other users check in their updated pages. To update the pages that are out of date, click the **Full Download** button. The updated pages will download and merge into your local design. The following figure provides an example of out-of-date pages in a local design:



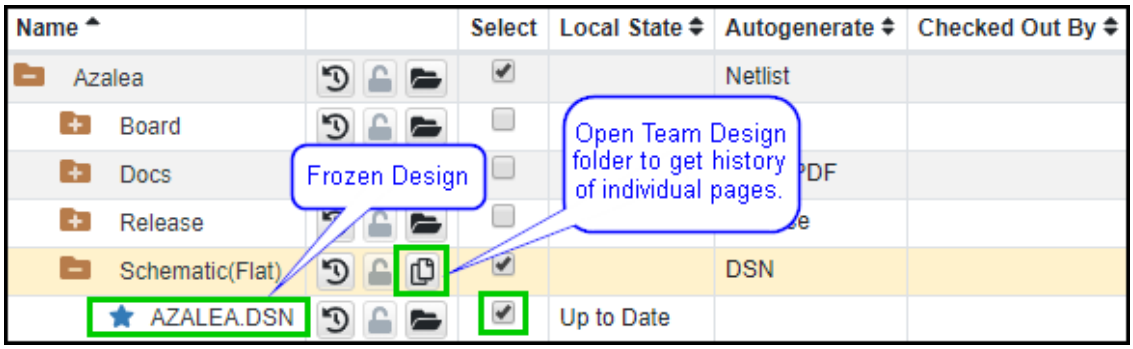
Full Download to Replace Out of Date Pages

7.5 Frozen Design Check Out

The Team Design folder, when frozen, looks like a standard folder with a merged design. You can check out the frozen design file similarly to the way you check out any file.

If a team design folder is designated as the Managed Design, there is a blue star ★ next to the file name. To check out the design file for editing, place a check mark in the **Check Out** box of that row.

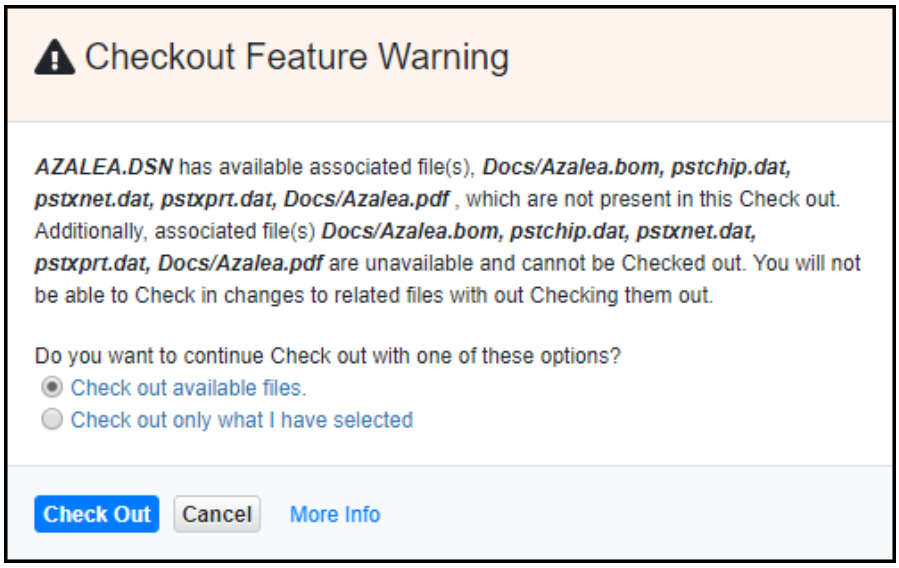
Then click the **Check Out** button.



Check Out Frozen Team Design

You cannot check out individual pages of a Team Design once it has been frozen (refer to Freeze Team Designs). Users are only permitted to check out the full design file at that point. Pages may only be downloaded from the history within the Team Design folder. To view individual page history, click the Team Design Pages button.

The check out of a Frozen Managed design is the same as Check out of Managed Design. When you are attempting to check out a frozen design, the Check Out Feature Warning appears when your related files are checked in but not selected for check out.



7.6 Check Out Team Library Symbols

You may download a full project or check out individual symbols from the Details View. This process is similar to the download/check out process conducted from the Team Design Details View. Check-out permissions for symbols are based on assignments, similar to the permissions established when checking out files. Any user assigned to a project can download symbols in the Schematic/Team Library folder as read only. However, only users specifically assigned to the symbol are allowed to check them out.

There are two ways you can check out symbols using the Check Out button.

Schematic symbols can be checked out from Schematic/Team Libraries using the following methods:

- **Check Out** button - Check out selected symbols and refresh symbols in your library that are not checked out. Select this option when you want your local library to include all available symbols, whether checked out or not.

- **Check Out** drop arrow , then:

  - **Check Out (Refresh symbols not checked out)** - Check out selected symbols and refresh symbols in your library that are not checked out. Select this option when you want your local library to include all available symbols, whether checked out or not. This is the same action taken when you click the **Check Out** button without the drop arrow

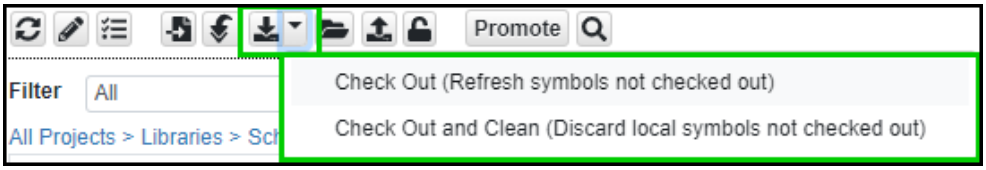
- **Check Out and Clean (Discard local symbols not checked out)** - Check out selected symbols and remove symbols from your local .OLB library that are not checked out . Select this option when you want your local library to include only symbols that are checked out.

Note: Check Out and Check In actions do not alter promoted schematic symbol libraries.

To check out symbols

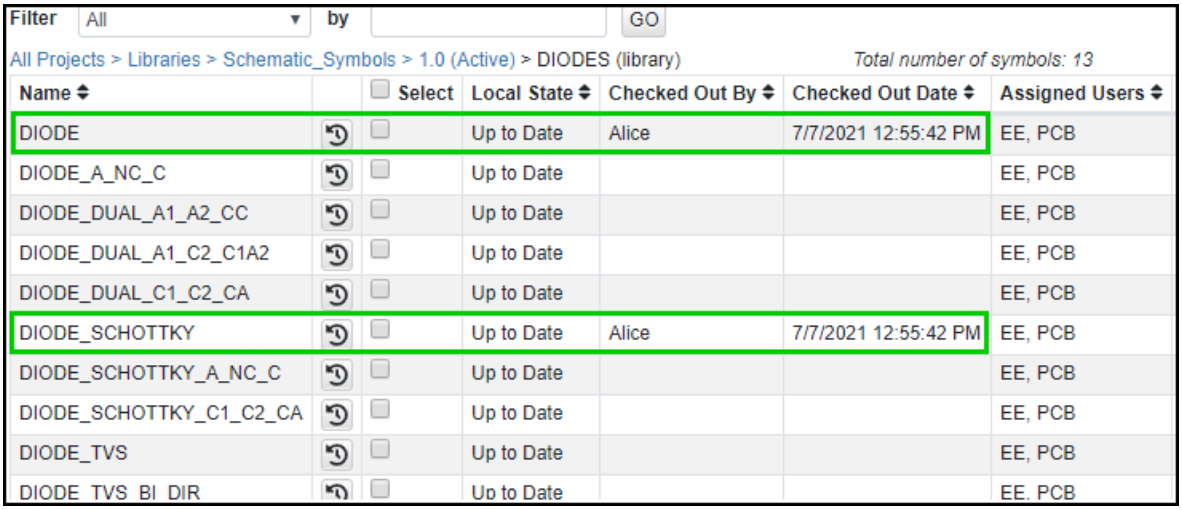
1. Place a check mark in the **Select** column to indicate each symbol you want to check out.
2. Click the **Check Out** button or click the **Check Out** drop arrow .

   If you click the Check Out drop arrow, select either **Check Out (Refresh symbols not checked out)** or **Check Out and Clean (Discard local symbols not checked out).** As described above, the former option will download all the library's symbols into the library (.OLB) file in your local workspace, while the latter will only download the symbols that are checked out.



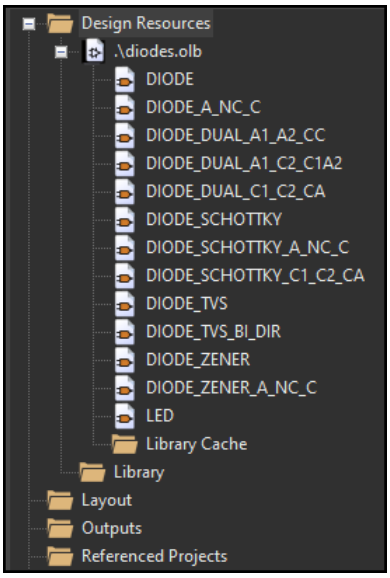
   The schematic symbols you selected that are assigned to you are checked out. The **Checked Out By** and **Checked Out Date** columns are updated. EDM downloads and merges the symbols into the library (.OLB) file in your local workspace and refreshes all symbols in your .OLB.

3. View the **Checked Out By** column to identify the symbols you have successfully checked out.

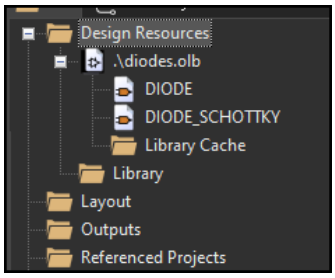


Note: Before library symbols are available for placement, the project leader needs to ensure the symbols are promoted symbols. To view symbols that have been promoted, click **View Promoted Library** .

4. Click the **Open Library** button to begin editing your symbols. The first image below shows a refreshed symbol library that includes all schematic symbols in the local library (.OLB) downloaded from EDM via the **Check Out** or **Check Out (Refresh symbols not checked out)** options. Notice how all symbols are downloaded, checked out or not. The second image shows a symbol library that only includes checked out symbols, which helps avoid confusion when checking out symbols from a large library.



All symbols downloaded in local library

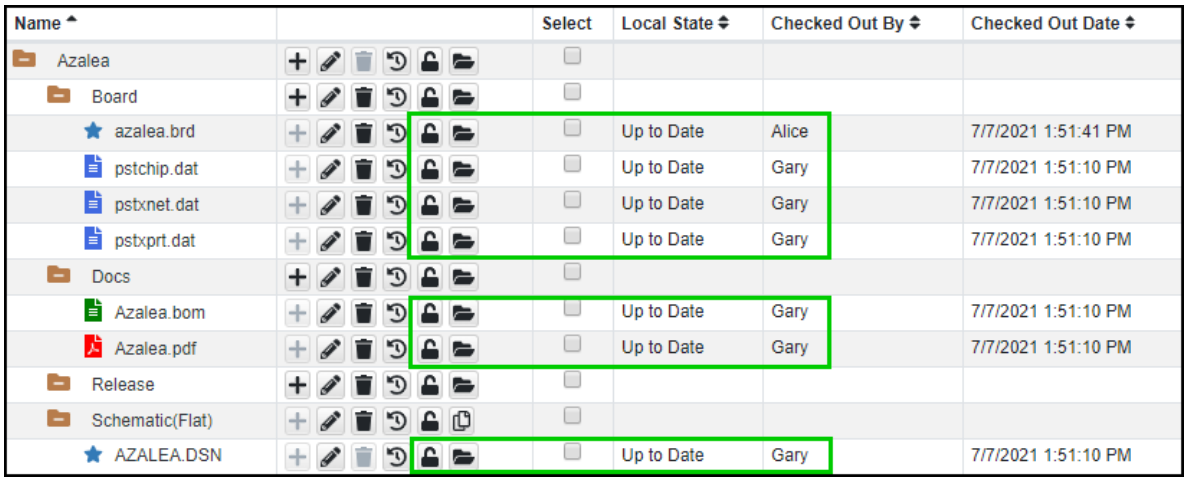


Only checked-out symbols in local library

7.7 Cancel Check Out

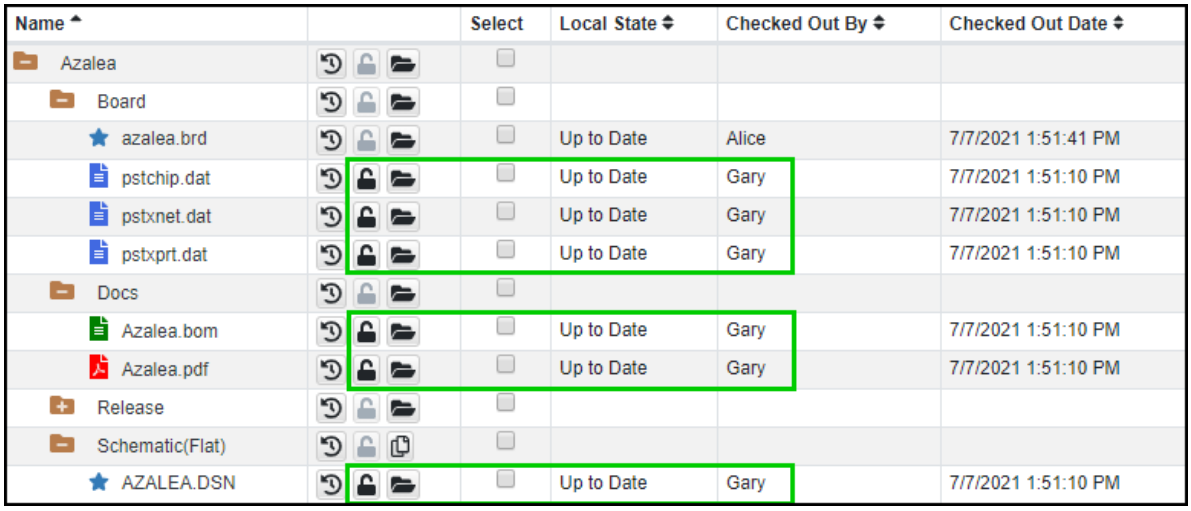
If you are no longer working on a checked out item, you can **check in** the item or **cancel** the check out. Only an assigned leader can cancel check out for project items checked out by other users. To cancel a check out, click the **Cancel Check Out** button next to the project, folder, or file name. The item then unlocks and becomes available for other assigned users or assigned leaders to check out. If the **Cancel Check Out** button is selected at the project or folder level, all files, pages, and symbols contained in any sub-folders will also be unlocked. You will need navigate directly to **Team Design** or **Team Library (Schematic Symbol)** folders to cancel check out for specific pages or symbols.

The figure below shows a detailed project view for Alice, an assigned leader. The **Cancel Check Out** buttons are enabled for all items and folders, regardless of who checked them out.



Cancel File Check Out as Assigned Project Leader

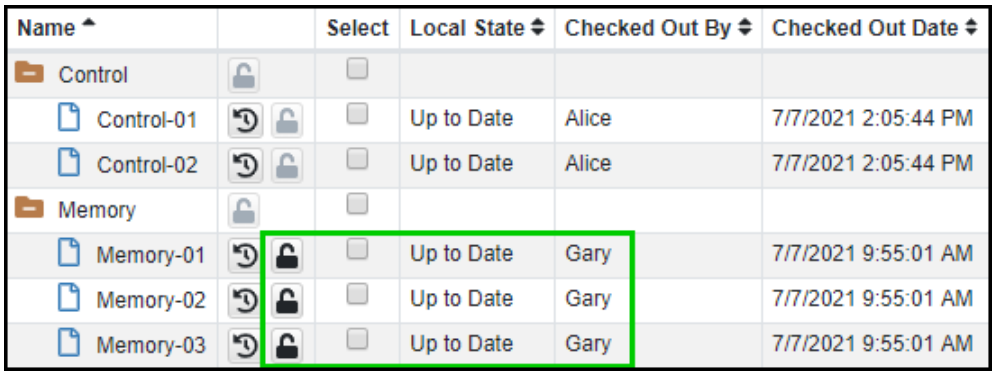
The figure below shows a detailed project view for Gary, an assigned Project User. The **Cancel Check Out** buttons are only enabled for items he has checked out. Cancel buttons are disabled for Folders.



Cancel File Check Out as Assigned User

Team Design Cancel Check out

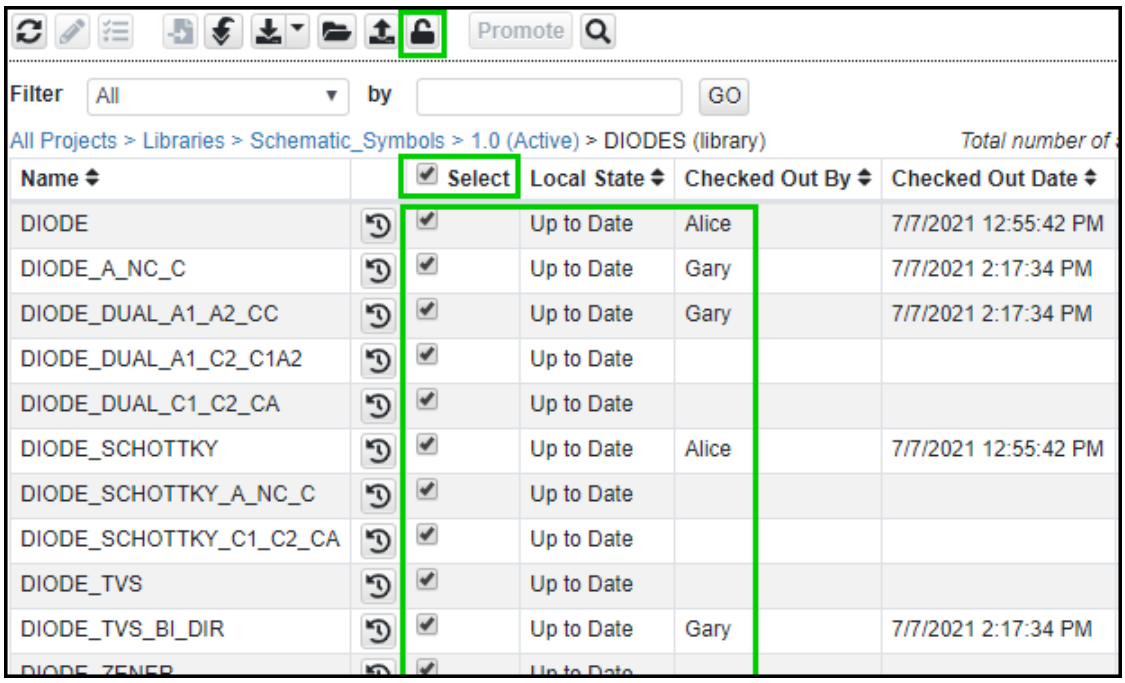
Cancel check out for pages is performed the same way as cancel check out for files. In the figure below, the **Cancel Check Out** buttons are only enabled for pages that Gary, an assigned Project User, has checked out. Cancel buttons are disabled for pages assigned to other users, such as Alice, and for Folders (Groups, Schematics).



Cancel Team Design Page Check Out as Assigned User

Team Library Cancel Check out

Cancel check out for schematic symbols is performed by selecting the desired symbols (individually or using the select all box) and then clicking the **Cancel Check Out** button. As an assigned Project User, only the selected symbols that you have checked out will be canceled, even if you select others. As an assigned leader, all selected symbols will be canceled.

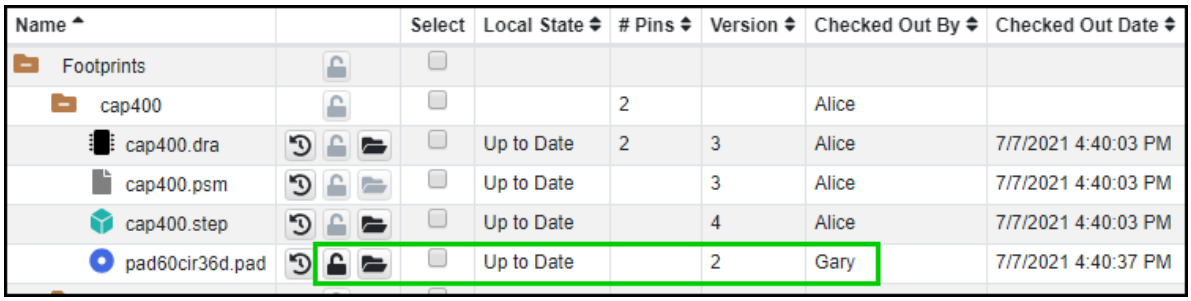


Cancel Schematic Symbol Check Out as Assigned User

Footprint Cancel Check out

Cancel check out for footprint files is performed in a similar way as cancel check out for files. They can be canceled individually, at a folder (group) level, or a combination of both, from any of the Footprint Project Views: File, Footprint, Padstack, or Step. Canceling checkout of a .dra file or its corresponding compiled (.*sm) file will cancel checkout for both files in the (.dra/*.sm) pair. In the figure below, the

Cancel Check Out buttons are only enabled for files that Gary, an assigned Project User, has checked out.



Name	Select	Local State	# Pins	Version	Checked Out By	Checked Out Date
Footprints						
cap400			2		Alice	
cap400.dra		Up to Date	2	3	Alice	7/7/2021 4:40:03 PM
cap400.psm		Up to Date		3	Alice	7/7/2021 4:40:03 PM
cap400.step		Up to Date		4	Alice	7/7/2021 4:40:03 PM
pad60cir36d.pad		Up to Date		2	Gary	7/7/2021 4:40:37 PM

8 Check In

Each time you complete local modifications, you need to check updated items into the vault. After check in, your updated modifications become available for other project users to download. It is good practice to either check in your work daily or at the completion of a given task. This practice ensures your work is backed up in the vault frequently, as you continue to make changes to your project.

The EDM vault archives each file version you check in. This enables you to download a previous file version whenever desired.

The check-in process begins from the Project Details page that provides access to the Check In button. When you check in updates to Team Folder items, such as Team Design Pages and Schematic/Team Library Symbols, the check in process begins from the Team Folder Details page. You may check in new files/pages/symbols as well as those that have been checked out and modified.

Note: The Check-In form excludes files that are ignored based on a folder's File Extension Rules, shown in the **Ext Rules** and **Extension** columns. EDM also excludes the project file (OPJ), back up files (DBK), and lock files (LCK) from check in, and these files are automatically archived in the local workspace. The project file is recreated each time the managed design is checked out.

Refer to the Appendix for details regarding Warnings and Errors a user may encounter during Check In.

8.1 Check In Files and Folders

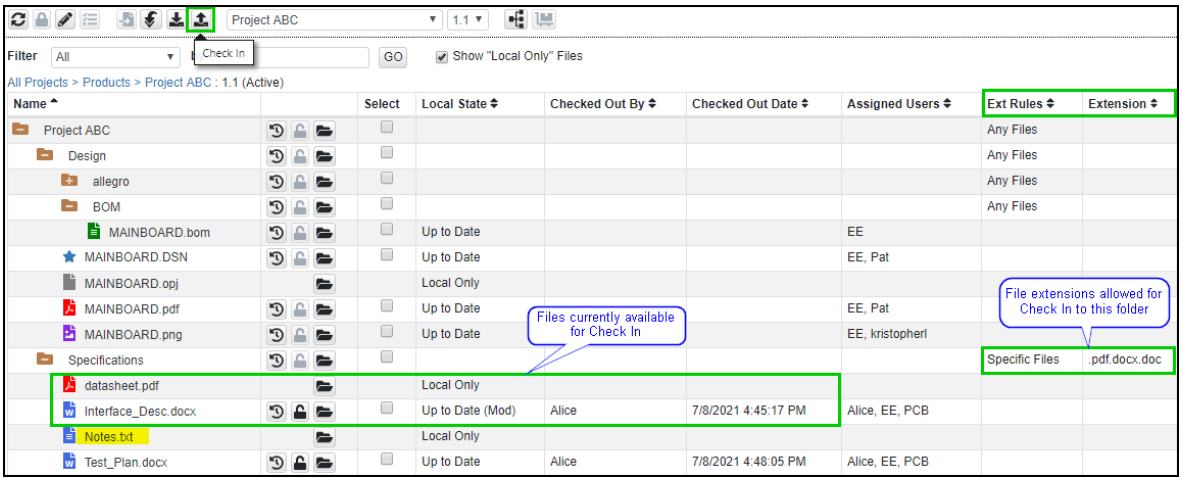
The top row of the Project Details page contains action buttons such as the **Check In** button. This button opens the Check In form, which shows a user all files available for check in and includes their Status and any enabled File User Fields configured in Customizable Fields.



Identify files available for check in

The Project Details page provides information that helps users quickly identify files that are available for check in using the **Local State** column. Files that are new (**Local Only**) or checked out and modified (**Up to Date (Mod)**) will be available for check in, and those with **Up to Date (Mod)** state will be auto-selected when opening the check in form. The Check In form excludes files that are ignored based on a folder's File Extension Rules, shown in the **Ext Rules** and **Extension** columns.

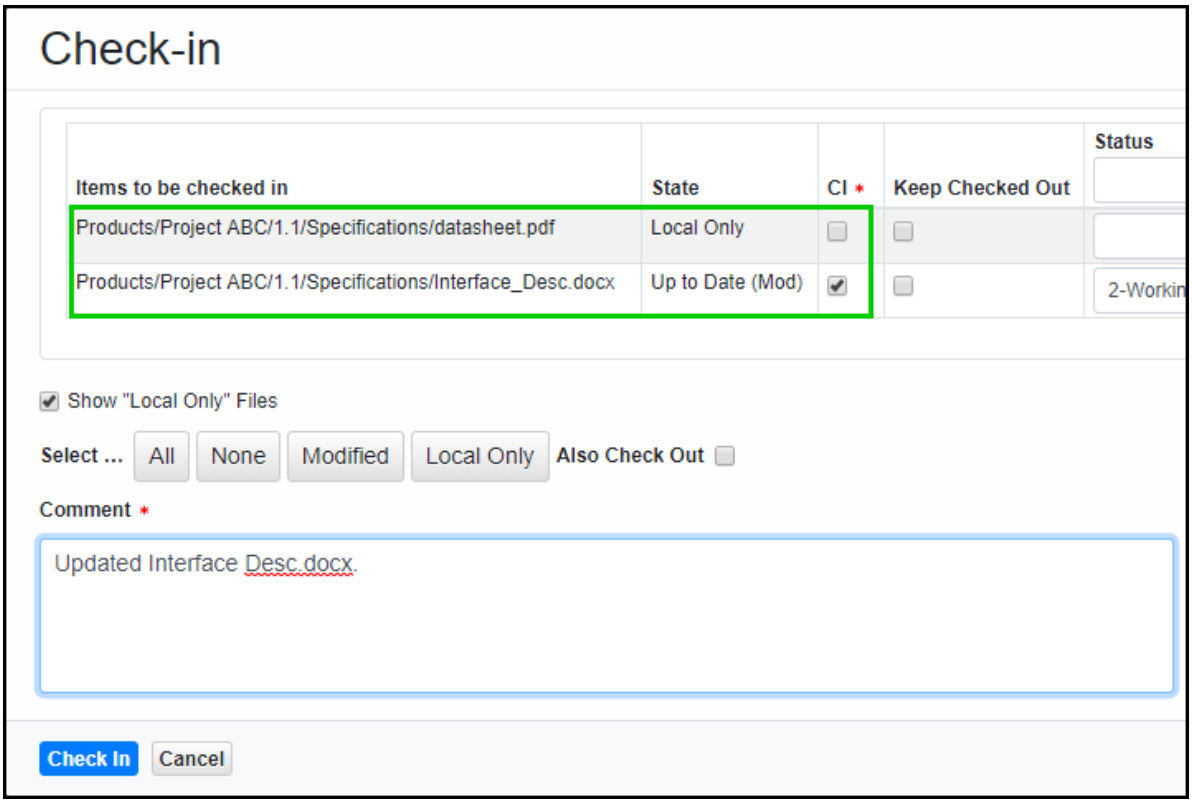
The example below shows the user Alice checked out two files (**Interface_Desc.docx** and **Test_Plan.doc**) and modified one of them (**Interface_Desc.docx**). The **datasheet.pdf** and **Interface_Desc.docx** files may be checked in because their local states are **Local Only** and **Up to Date (Mod)**, respectively, and they meet the file extension criteria (.pdf, .docx, .doc). Alice may begin the check in process by clicking the Check In button from the project details page as shown. This opens the Check In form.



Detail Project View Shows Files Available for Check In and File Ext Rules Applied

As shown in the figure below, the Check In form will display all items available for check in and automatically select the Check In (**CI**) option for files that have been checked out and modified (State is **Up to Date (Mod)**). In this example, the **CI** option was auto-selected for **Interface_Desc.docx** because it is **Up to Date (Mod)** and it is not excluded based on the file extension rules for the "Specifications" folder. The new (**Local Only**) **datasheet.pdf** file was not auto-selected for Check In, but may be selected individually or via the group **Select... Local Only** button under the list of files. To filter out Local Only (New) files from the list, you can uncheck **Show "Local Only" Files**.

Notice that the **Notes.txt** file highlighted in yellow in the figure above is excluded from the Check In list in the figure below. The file extension rules of the parent "Specifications" folder do not include **.txt** files so they are excluded from check in. EDM also excludes the project file (OPJ), back up files (DBK), and lock files (LCK) from check in.



Check In Form for Files

Check In Files and Folders

1. Click the **Check In** button that is part of the action buttons located along the top row of the Project Details page. This opens the Check In form.



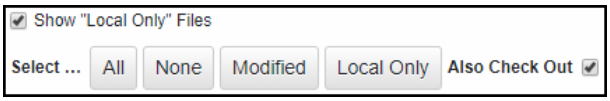
2. Select files to be checked in. To filter out Local Only (New) files from the list, you can uncheck **Show "Local Only" Files**. If you'd like files to remain checked out after check in, this can be done in groups (by State via **Select...** buttons) or individually (via **CI** buttons), and may be done in combination with each other. You may wish to group-select files by State first, then individually select/deselect items after that.

   a. **Group select files (by State)**: The **Select...** buttons allow groups of files to be selected based on the (Local) State of the files shown in the list of **Items to be checked in** (Check In list). The buttons affect the checkboxes in the **CI** and **Keep Checked Out** columns, respectively.

      First, select the **Also Check Out** checkbox if you are choosing a group of files that you wish to remain checked out after the check in completes. This is optional and is usually done when you want to check files into the vault, but keep them checked out to continue working on them. This

option is saved per user and your last-used selection will be applied when opening the Check In form.

Next, click one of the **Select...** buttons (located below the list of **Items to be checked in**) to select a group of files with a given (Local) State.



Select... Buttons with Local Only Files



Select... Buttons without Local Only Files

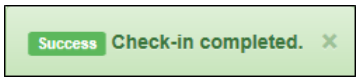
- **All** - selects all files in the list.
- **None** - deselects all files in the list. This can be useful if you accidentally selected files and want to start again with all files unchecked.
- **Modified** - selects the files shown as **Up to Date (Mod)** under the **State** column. It does not deselect any previously-selected files, regardless of their State.
- **Local Only** - selects only the files shown as **Local Only** under the **State** column. It does not deselect any previously-selected files, regardless of their State. This button is hidden when **Show "Local Only" Files** is unchecked.

b. **Individually select files**: The individual **CI** and **Keep Checked Out** boxes for each file allow individual files to be selected on-the-fly, regardless of State.

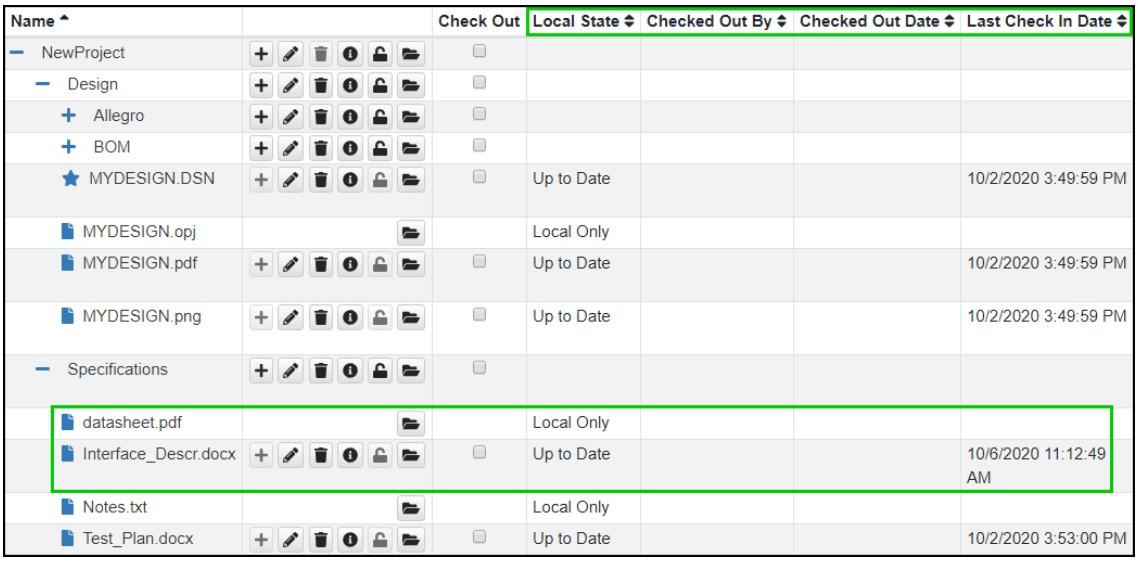
Select or de-select individual checkboxes in the **CI** (Check In) column for each file you want to check in. Select the **Keep Checked Out** checkbox for each file that you wish to remain checked out after the check in completes. This is optional and is usually done when you want to check a file into the vault, but keep it checked out to continue working on it.

3. Enter a value into the **Status** field and any other File fields shown (configured in Customizable Fields configured for Files) if you want to update the file fields for the items you are checking in. Updating these fields is optional and can be done individually using each file's text box or pick list for each field, or as a group using using each file's top-level text box or pick list for each field. This may be done in combination with each other and you may want to update file fields by group, then individually change the ones that should be different. For new files, the configured default values will be auto-populated for Status and each enabled file field.
4. Enter a comment to describe the check in. The comment will be saved to the specific historical record in the project history.
5. Click the Check In button to check in the selected file(s), or click the Cancel button to close the **Check In** form.

After successful check in, the following success message displays. You may dismiss the message manually by clicking the **X** in the message or it automatically closes after a few seconds. Refer to Warnings and Errors for information about other check in messages.



The **Checked Out By**, **Checked Out Date, and Last Check In Date** columns will be updated accordingly. The **Status** field and file fields will be updated as well, based on the values entered during check in. The following figure shows an example of the updated Local State after check in.



Status Updated After Check In

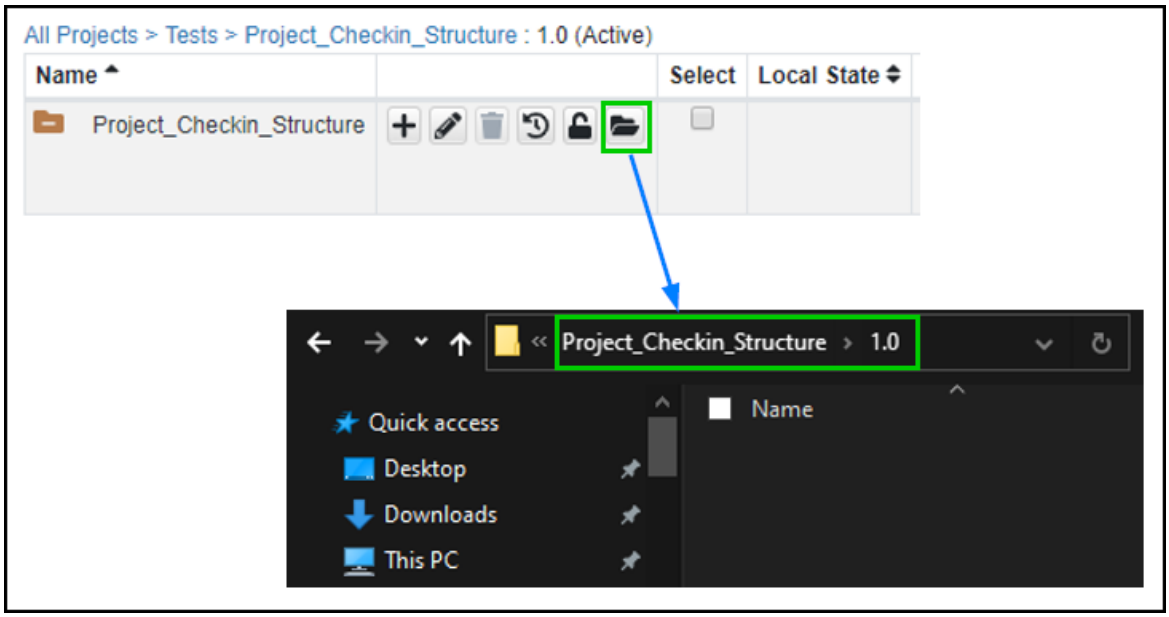
Check In Folders or Project Structure

An entire project structure may be checked in instead of creating a project through adding folders or from a template. This applies to General and Capture Design project types.

If you want EDM to manage the entire structure of a project, you can create an empty project of the desired Project Type and enable the option to **Allow Users to Check in New Folders**. Only assign yourself to as a leader and do not assign any users for now.

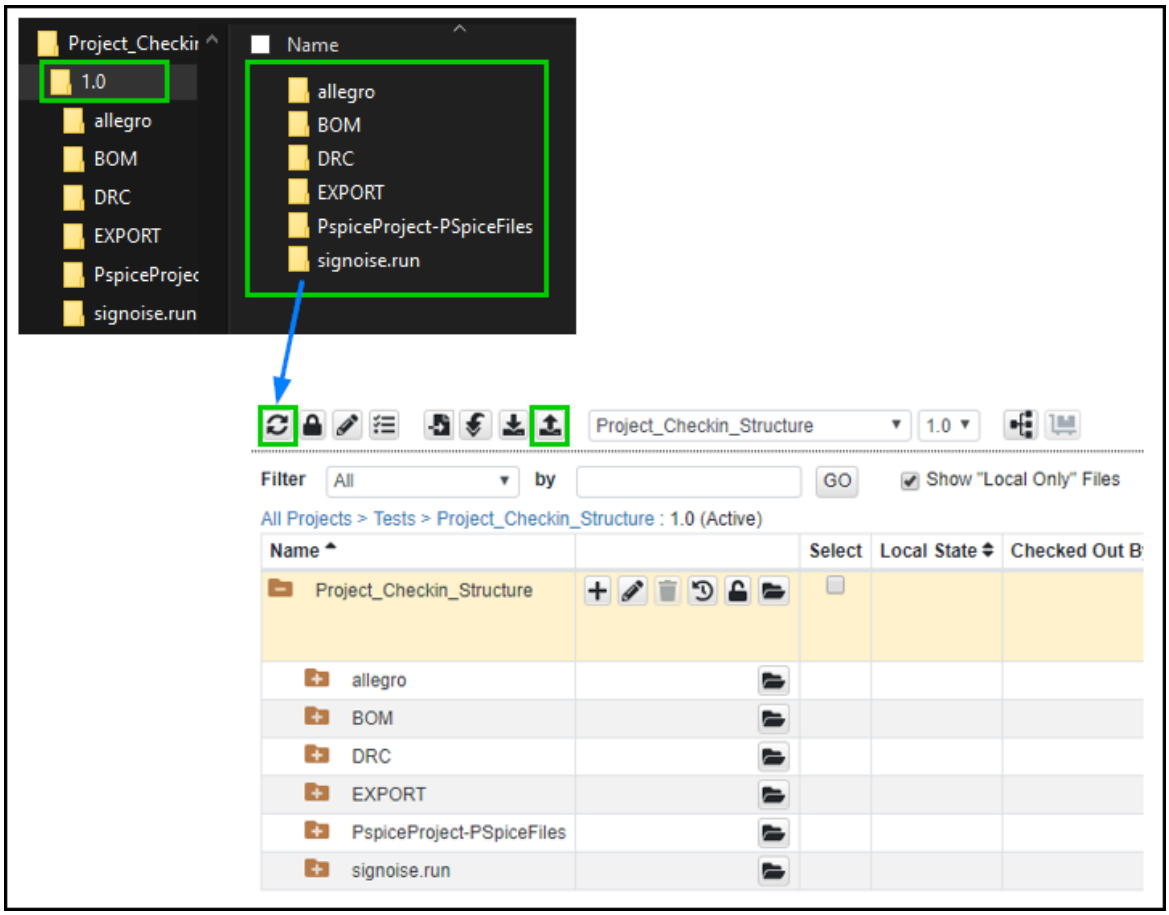
To check in a folder or entire project (after project creation):

1. Open the project revision.
2. Click the **Open Folder** button next to the top-level project folder to open it as shown below. For Capture Design projects, this will also create an empty Capture design (.dsn) in the project folder using the project's designated DSN Filename. You can delete the new file if you plan on copying, optionally renaming, and checking in an existing design, or you could use the **Import > Design (.dsn)** option as described in Import Design and Board Files. Similar steps can be followed for the managed Board (.brd).



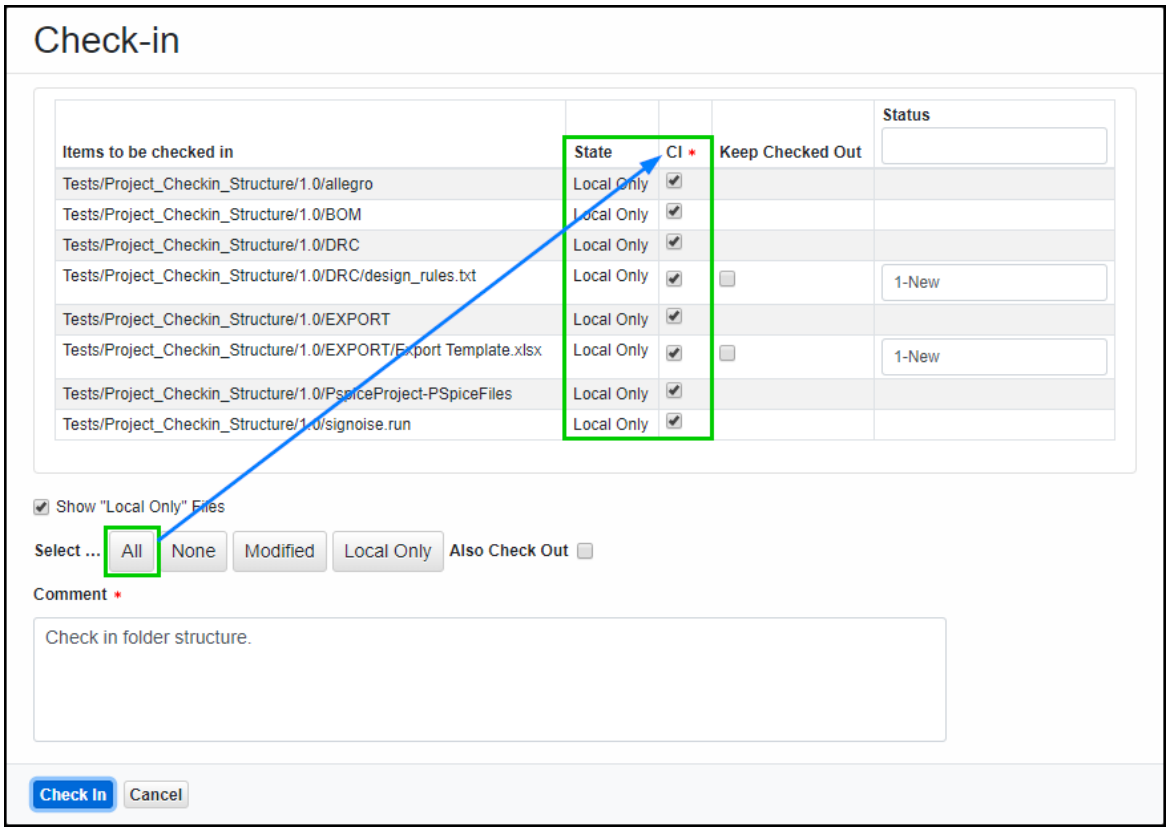
Open Folder of Empty Project

3. To check in a folder, file, or folder structure, copy the folders and files from your existing project folder (or any other folder) to the opened (new project) folder. Refresh the project details in EDM and all files appear as Local Only. A refresh is not necessary before Check In unless you would like to see the updated structure in the project details before opening the check in form.



Copied Structure View in Project Details

4. Click the **Check In** button to access the check in form and check all files into the vault. The project file (OPJ), back up files (DBK), and lock files (LCK) are excluded from the check in form.



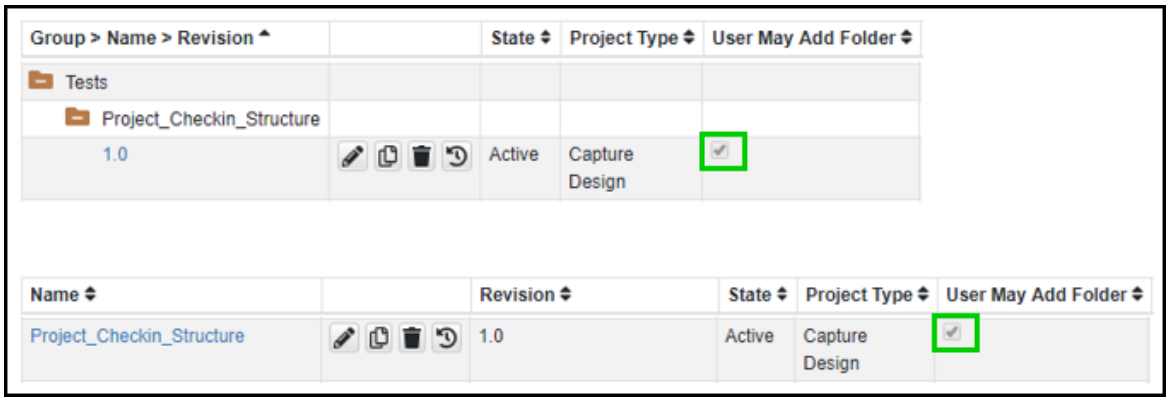
Select All Files in Copied Folder Structure

5. Click the **All** button to select all files and folders available for check in. Since these are all new files (Local Only), clicking the **Local Only** button would have the same result. If **Show "Local Only" Files** is unchecked, you'll need to check it to display the Local Only files in the list and show the **Local Only** select button.

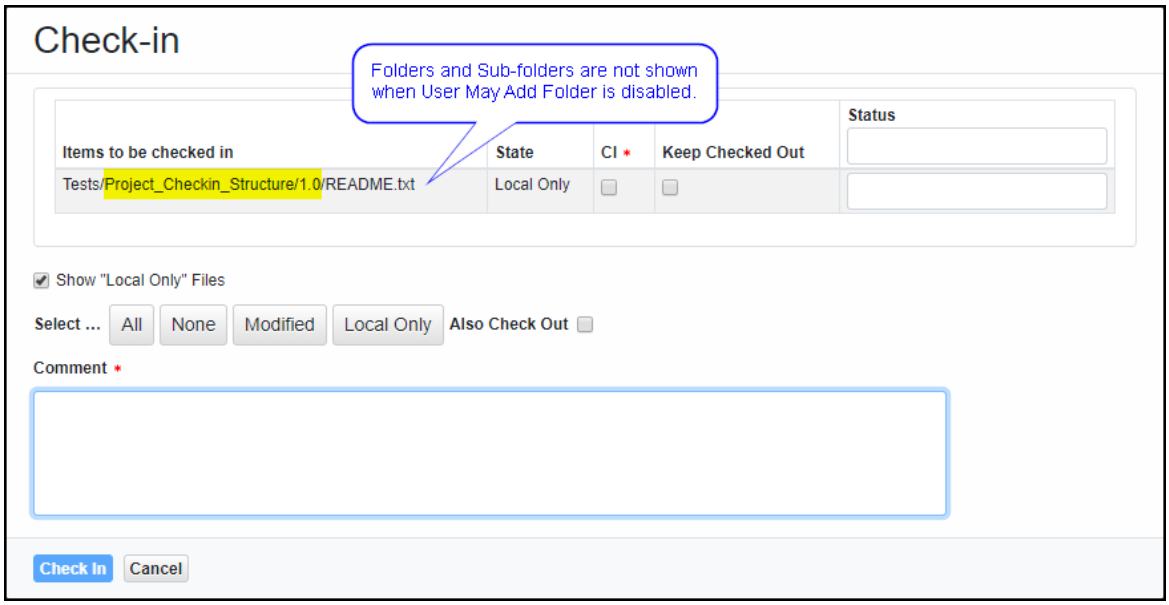
6. Enter the new **Status**, update any user fields if enabled, and enter a **Comment**. Only the comment is required.

7. Click the **Check In** button to check in the selected file(s), or click the **Cancel** button to close the **Check In** form.

Note: If the project is not enabled to allow users to check in new folders, then only files at the project level are included in the check in list. You can check the **User May Add Folder** option in the Project Dashboard. Admins and Assigned Project Users can also see the option in the Edit Project form. Folders, sub-folders, and files within a folder are excluded from the check in.



User May Add Folder option shown in Project Dashboard for Hierarchical View (Top) and Flat View (Bottom)



Only Project Level Files (No Folders) are included in Check In form when User May Add Folder option is disabled

Files Excluded from Check In

Since all prior versions of the files checked into the vault may be retrieved, Capture back up files (DBK) as well as Capture and PCB Editor lock files (LCK) are excluded from the list of files available for check in. Project files (OPJ) are archived locally in the users local workspace and are also excluded from check in.

Note: When the version of a file in the vault is identical to the checked out version in the local workspace, the local version is excluded from check in. Only new (**Local Only**) and checked out and modified (**Up to Date (Mod)**) files are available for check in. Additionally, the check in form is not available when EDM is opened from a web browser.

8.2 Check In Managed Design Files

EDM treats files associated with a managed Capture design differently from those not part of a managed Capture Design project. Any type of file may be checked into the vault if permitted by the file extension rules, excluding project files (OPJ), back up files (DBK), and lock files (LCK). Files that are associated to a managed Capture design, (BOM, Netlist, and PDF) may be automatically generated when you check in a design file.

When checking in a managed design file, the Check In form includes options that are specifically applicable to the managed design. The user instructions presented in this section describe in detail the options dedicated to managed designs, while the user instructions provided in the section Check in Files explain in detail the fields that are common to all project types (Capture Designs, General projects, and Footprint projects).

Note: When you check in a managed design that has hierarchical blocks external to the design file, EDM notifies you about schematic files not included in the project.

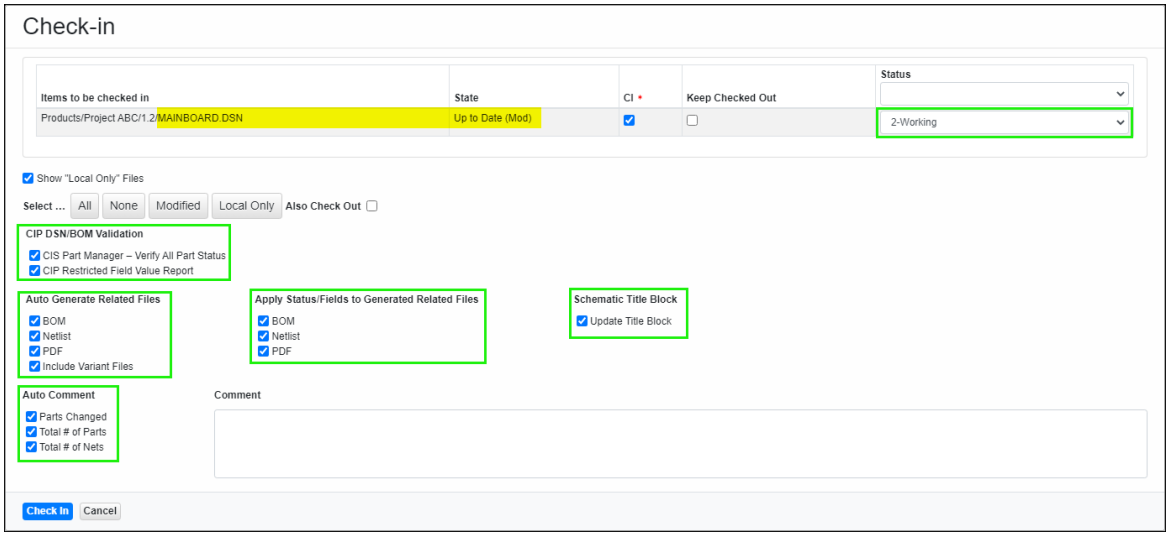
Check In Managed Design Files

1. Open the project details of your Capture project as described in the section, Check In Files and Folders. Click the **Check In** button to open the Check In form. Files that are new (**Local Only**) or checked out and modified (**Up to Date (Mod)**) will be available for check in, and those with **Up to Date (Mod)** state will be auto-selected when opening the check in form. Select the design file(s) that you want to check in (CI column) if they are not already selected by default.
2. Select checkboxes in the form to indicate the managed design options you want to apply. These option are only shown when the managed design is selected for **CI** (check in).
   - **CIP DSN/BOM Validation:**
     - **CIS Part Manager – Verify All Part Status** – automatically run CIS Part Manager to verify that the statuses of all parts in your managed design are up to date (**Approved: Current**) with respect to your CIS database. Settings configured in the user's CIS .DBC file, Capture.ini, and Capture CIS are used when verifying part status, just as they are when manually running Part Manager. When selected, the CIS part status of all parts on the design must be **Approved: Current** to proceed with the check in, otherwise an Unable to Complete Check In error will appear. Deselect this checkbox to check in your managed design without requiring all parts to have a status of **Approved: Current**. Refer to the OrCAD Capture CIS documentation on Part Manager usage for more info.
     - **CIP Restricted Field Value Report** – warn or prevent check in of a managed DSN and its auto-generated files (BOM, PDF, etc.) when any of the configured CIP restricted field values are found. When this happens, the CIP Restricted Field Value Report window is displayed. This will present the user with buttons to **Proceed** or **Cancel** (for Warning) or **OK** (for Error), depending on the configured **Response Type** you've configured for the CIP BOM Restricted Field Value Report under BOM Validation.
   - **Auto Generate Related Files** – Select checkboxes to identify the files you want to auto generate. The files you select will be auto-generated with information derived from the managed design file. EDM will place the files into the configured project folders. The related files you may choose are:
     - **BOM** (Bill of Materials) – create and check in the BOM(s) using the BOM template(s) associated with the project. The format of the BOM(s) is defined by the Admin or Project leader as described in the section topics, Manage BOM Template and the BOM folder associated to the project.

- **Netlist** – create standard Capture Netlist files for PCB Editor.
- **PDF** – create the intelligent PDF. Before you can successfully select this checkbox you need to configured the PS interpreter. Follow instructions in EDM Settings for PDF configuration.
- **Include Variant Files** – open the managed design and variants with a Capture CIS license. The PDF and BOM generated will include files for the variants where the variant file names are based on the names of the variants.

- **Apply Status/Fields to Generated Related Files** – enter values into the **Status** and any other File user fields shown (configured in Customizable Fields configured for Files) if you want to update the fields for the items you are checking in. The values you enter will be applied to the auto-generated files for each type of file (BOM, Netlist, PDF) selected. If left empty, the Status or user fields, the pre-existing status will be applied to the selected auto-generated files.
- **Schematic Title Block** – select the **Update Title Block** checkbox if you want to automatically update the title block properties during check in based on the title block mappings in Admin > Configuration > Capture.
- **Auto Comment** – select checkboxes to **Auto Comment** in Comment input field during check in.
  - **Parts changed** – compare parts in your local design file to the version in the vault. Reference Designators of parts are compared. Part reference designators that have been added and removed are added to the comments field. Parts with properties that changed are also added to the comments.
  - **Total # of Parts** – identify the total number of parts in the managed design file and record the number in the comment field.
  - **Total # of Nets** – identify the total number of nets in the managed design file and record the number in the comment field.

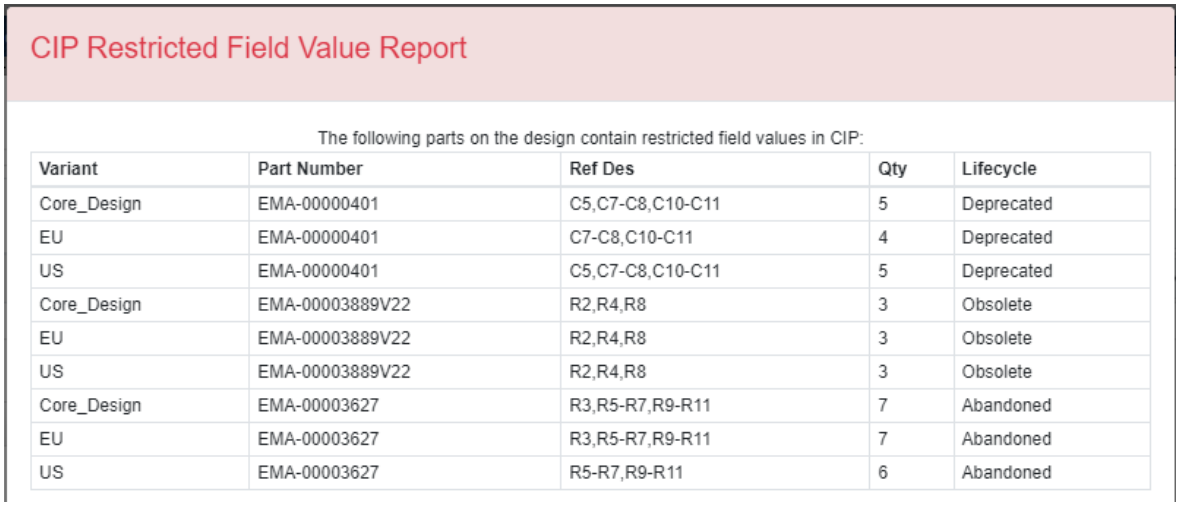
The figure below provides an example of the Check In form when a managed Capture design (MAINBOARD.DSN) is selected. The checkbox items outlined in green indicate the options available for a managed design (.DSN file).



Check In Form with Managed Design Selected

3. Click the Check In button after you complete the form. This executes the selected auto-generation of files, selected CIP DSN/BOM Validation options, and completes the check in process. Or click the Cancel button to cancel the **Check In** form.

When enabled, the **CIP Restricted Field Value Report** will run prior to checking in the managed design, its selected auto-generated files, and any other files selected for check in. The pop-up window below will appear if the report finds configured restricted values in the CIP field in question for any of the CIP parts on your schematic design.

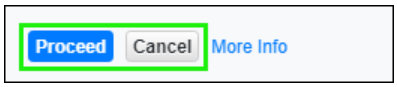


CIP Restricted Field Value Report

The following parts on the design contain restricted field values in CIP:

Variant	Part Number	Ref Des	Qty	Lifecycle
Core_Design	EMA-00000401	C5,C7-C8,C10-C11	5	Deprecated
EU	EMA-00000401	C7-C8,C10-C11	4	Deprecated
US	EMA-00000401	C5,C7-C8,C10-C11	5	Deprecated
Core_Design	EMA-00003889V22	R2,R4,R8	3	Obsolete
EU	EMA-00003889V22	R2,R4,R8	3	Obsolete
US	EMA-00003889V22	R2,R4,R8	3	Obsolete
Core_Design	EMA-00003627	R3,R5-R7,R9-R11	7	Abandoned
EU	EMA-00003627	R3,R5-R7,R9-R11	7	Abandoned
US	EMA-00003627	R5-R7,R9-R11	6	Abandoned

CIP Restricted Field Value Report



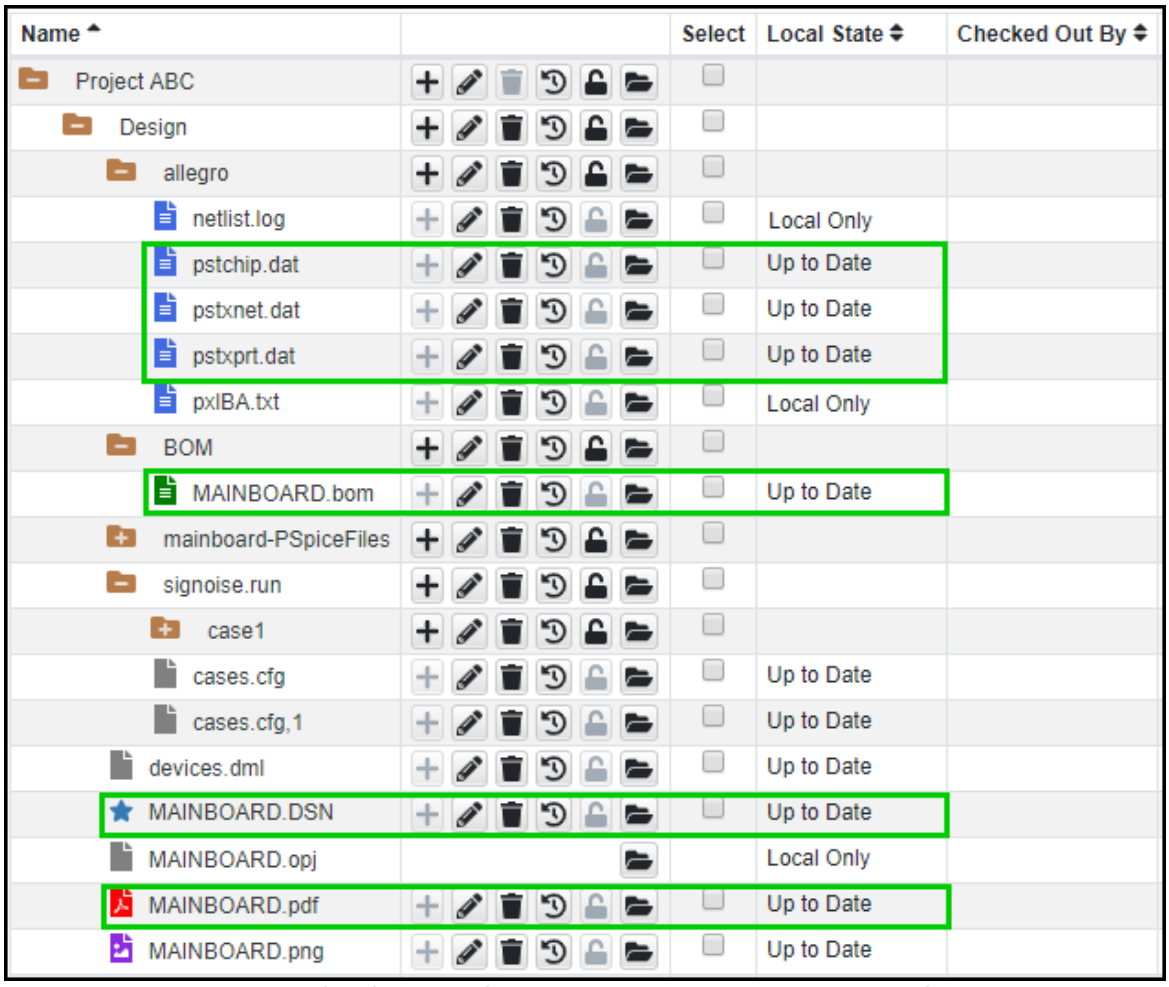
Response Type set to Warning – Prompt to proceed with Check In



Response Type set to Error – Prevent Check In

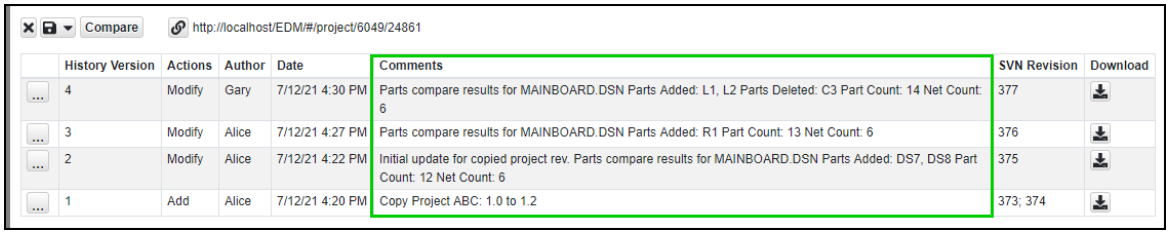
The window will either have Proceed Cancel buttons when the Response Type is configured for **Warning – Prompt to proceed with Check In** (default) or an OK button when configured for **Error – Prevent Check In**. Click the **Proceed** button to continue with the check in, or click **Cancel** or **OK** to close the report. In the latter case, you'll need to resolve the reported issues on the design before checking it in.

After check in is complete, the project details are displayed. The following figure shows the updated status of the managed design in the Local State and auto-generated files that were checked in.



Project Detail View After Check In of Managed Design and Autogeneration of Related Files

User comments and selected auto-generated comments will become part of the check-in history. The image below shows the history of a managed design file that includes user input and auto-generated comments.



Comments in History

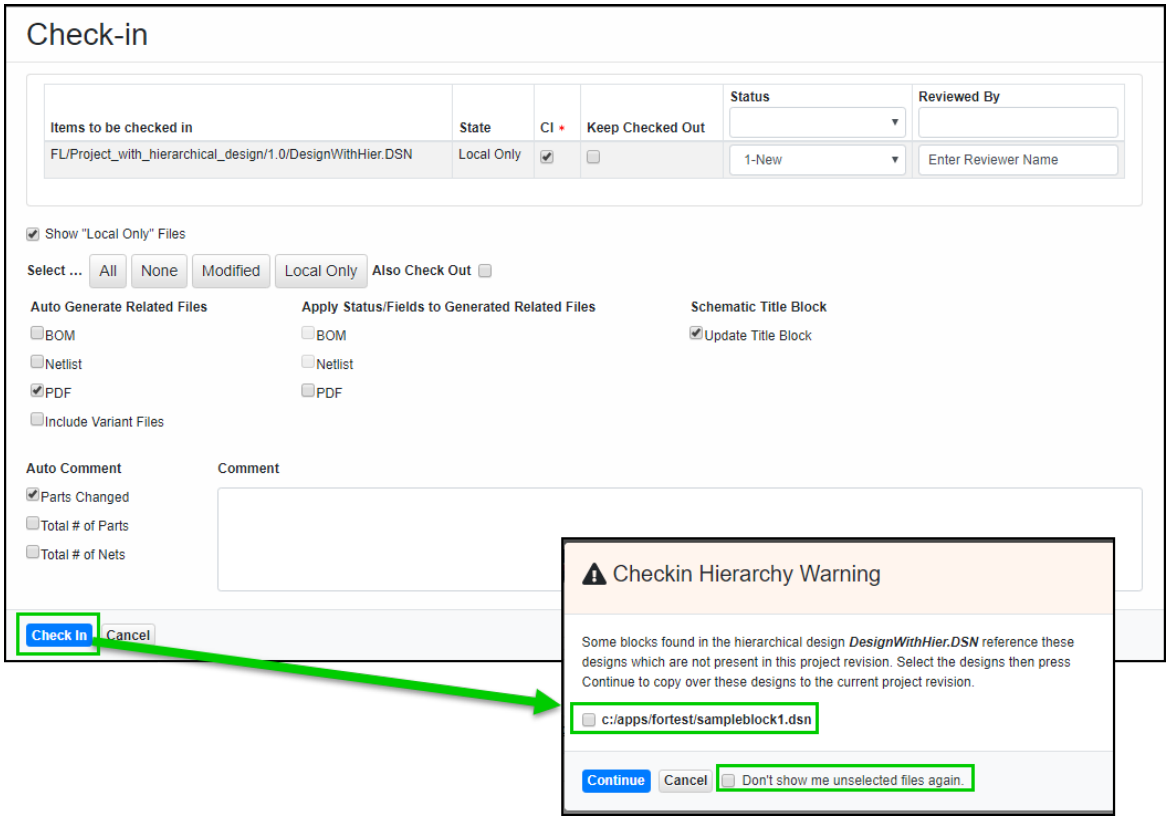
The files listed below are often generated automatically when saving and checking in a managed design. You may manually select these files for check in if desired.

- <manageddesignname>.png
- netlist.log
- signoise.run folder and its content
- case1 and its contents
- devices.dml

You can use the **Open** button to view any file from the EDM user interface. This also includes the automatically generated files.

To check in a managed hierarchical design

When you check in a managed design that has a hierarchical block, EDM notifies you if schematic files are referenced that are not included in the project. You have the option to have EDM copy the files into the project and check them in. You also have the option to stop the message from displaying again if you do not want to copy the files.



Hierarchical Design Options For Managed Design Check In

8.3 Check In Managed Board Files

Check in of a managed board and associated auto-generation functionality is available when you open the EDM Desktop application.

Check In Managed Board Files

1. Open the project details of your Capture project as described in the section, Check In Files and Folders. Click the **Check In** button to open the Check In form. Files that are new (**Local Only**) or checked out and modified (**Up to Date (Mod)**) will be available for check in, and those with **Up to Date (Mod)** state will be auto-selected when opening the check in form. Select the files you want to check in (CI column) if they are not already selected by default.

2. Select checkboxes in the form to indicate the managed design options you want to apply.

(2-a) Auto Generate Related Files – Select checkboxes to identify the files you want to auto generate (call out 2-a). When you select the **DB Doctor** or **EDM Validation** option, a **Reports** folder is auto-created and the output files are placed in this folder.

- **DB Doctor** – generates two log reports (dbdoctor.log and batch_drc.log) that logs DRC results of the board. (This report may also be manually generated from the Allegro PCB Designer menu: Tools > Database Check or OrCAD PCB Designer: Check > Database Check...)
- **EDM Validation** – generates a report that compares the internal version ID of the footprint .dra that was generated during promotion to the footprint .psm version placed on the managed board.
- **PDF** – generates a PDF file using the board's PDF domain films if they exist in the board's Film Domain Settings. If not found, all films will be included.

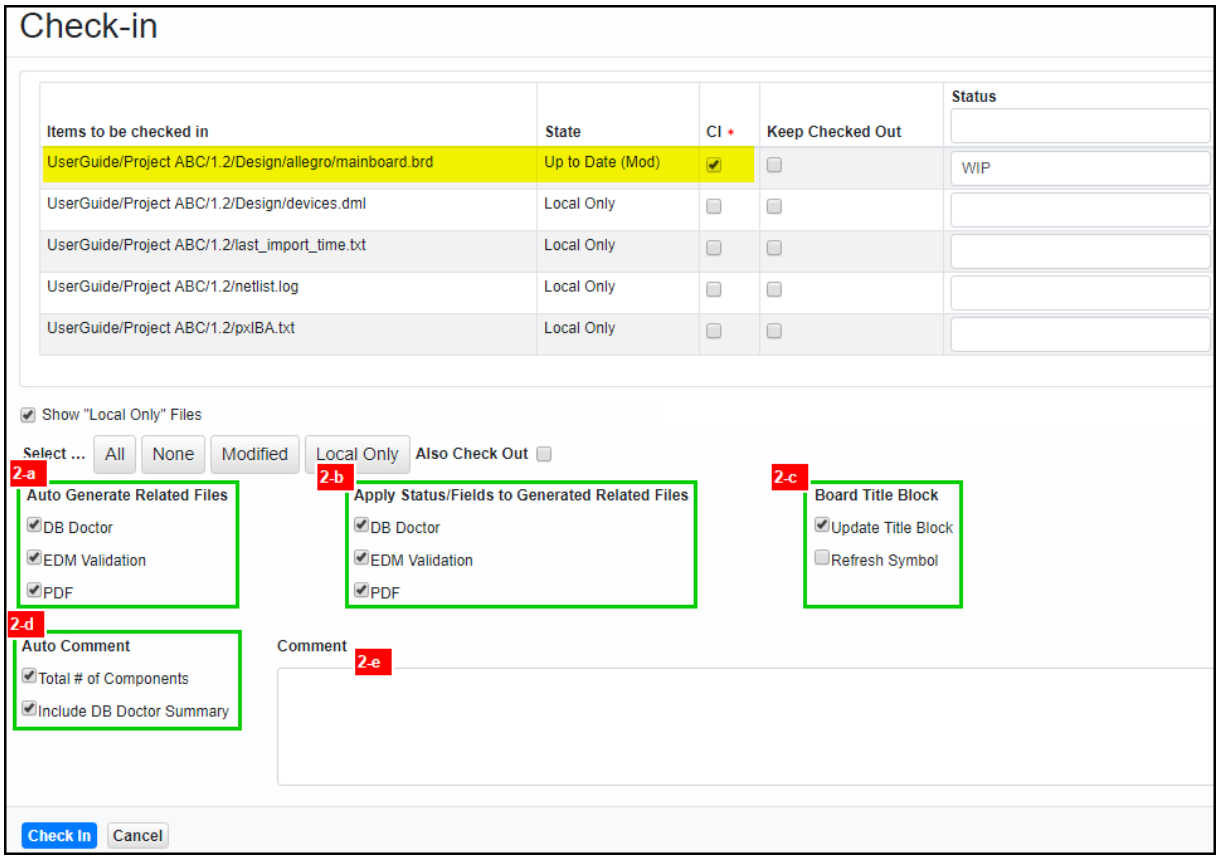
(2-b) Apply Status/Fields to Generated Related Files – if you enter values into the Status and user fields (in the upper-right corner of the form), the values you enter will be applied to the auto-generated files you select (call out 2-b). Files you may select are DB Doctor, EDM Validation, or PDF. checkbox(es) to associate the entry of the Status field in the upper part of the form to each automatically generated file you select. If you leave the Status or user fields blank, the pre-existing status will be applied to the files you select (call out 2-b).

(2-c) Board Title Block

- **Update Title Block** – select the Update Title Block checkbox to update the title block properties (call out 2-c). Updates are based on the title block mappings set up through the menu selection: Admin > Configuration > PCB Editor.
- **Refresh Symbol** - select the Refresh Symbol checkbox to update the title block symbol in your board file.

(2-d) Auto Comment – select a checkbox to automatically add the **Total # of Components** on your board and/or **Include DB Doctor summary** into the **Comment** field.

(2-e) Comment – enter a comment that will be archived in the project and file history. If you select an Auto Comment checkbox, the manual entry of an additional comment is optional. However, if the Auto Comment checkboxes are not selected, manual entry of a comment is required.



3. Click the **Check In** button to save and apply the items you selected in the Check In form to your managed board file, or click the **Cancel** button to close the **Check In** form.
   After check in and auto-generation is complete, the updated project details are displayed.

EDM Validation Report

When you select the **EDM Validation** checkbox on the Check In form (call-out 2-a in above image), EDM automatically creates a **Reports** sub-folder (if it does not exist) in your managed board directory with the auto-generated reports. The report lists all the footprint files that existed on the managed board at the time of check in. Below is a sample EDM Validation report.

Footprint Name	Pin Count	Validation Status	Footprint Location	Project Name: Rev	Last Promote Date	Last Promoted Version	Last Promoted By	Version	Footprint Status
BGA48C75P6X8_600X800X100	48	Out of Date (BRD FP Older)	//engineering01/promoted_footprints/symbols/bga48c75p6x8_600x800x100.psm	Footprints: Ongoing	6/20/2019 10:51	2	Pat	2	3-QA
CAPC1005X56N	2	Up to Date	//engineering01/promoted_footprints/symbols/capc1005x56n.psm	Footprints: Ongoing	10/7/2022 10:26	1	Pat	1	4-Production
CAPC1608X87N	2	Up to Date	//engineering01/promoted_footprints/symbols/capc1608x87n.psm	Footprints: Ongoing	8/15/2022 8:31	1	Kevin	1	4-Production
CAPC2012X100N	2	Out of Date (BRD FP Newer)	C:/SPB_Data/local_workspace/EDM/Libs/Footprints/Ongoing/cap2012x100n.psm	Footprints: Ongoing	6/20/2022 10:30	2	Joanna	3	1-New
CAPC2012X145N		Not Found in Promoted Location(s)	C:/Cadence/CIP/Allegro_Library/symbols/capc2012x145n.psm						

EDM_Validation_projectname Test

EDM Validation Report

The EDM Validation report generates a CSV file that contains the following data columns:

- **Footprint Name** – This footprint file name found on the board during check in.

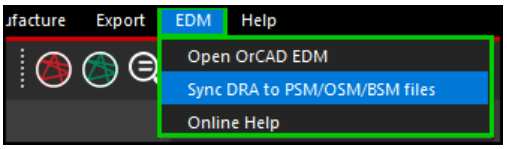
- **Pin Count** – The number of pins on the footprint when last checked into EDM. It will match the **Pin Count** column in the footprint project.
- **Validation Status** – This column lists the status of footprints placed on the board file as compared to the last time the footprints were promoted in EDM. There are four possible statuses.
  - Up to Date – The version of the footprint on the board is the same as the version recorded when the footprint was promoted.
  - Out of Date (BRD FP Older) – The version of the footprint on the board is older than the version recorded when the footprint was promoted. This would occur if a footprint was promoted after the time it was placed on the board. A board symbol refresh is required in this case.
  - Out of Date (BRD FP Newer) – The version of the footprint on the board is new than the version recorded when the footprint was promoted. This would occur if a placed footprint was modified locally and not yet promoted.
  - Not found in Promoted Location(s) – The footprint on the board is not found in the configured Footprint Promote Path(s). It would occur if the footprint file was (1) not managed in EDM, or (2) managed in EDM but not yet promoted.
- **Footprint Location** – The path and footprint file name that was placed on the board.
- **Project Name: Rev** – Project Name and Project Revision that contains the footprint.
- **Last Promote Date** – The date/time recorded in EDM the last time the footprint file was promoted, as shown in the **Last Promote Date** column in the footprint project.
- **Last Promoted Version** – The footprint file version in EDM the last time it was promoted, as shown in the **Last Promoted Version** column in the footprint project.
- **Last Promoted By** – The user who last promoted the footprint file, as shown in the **Last Promoted By** column in the footprint project.
- **Version** – The current version of the footprint file, as shown in the **Version** column in the footprint project. It may be newer than the Last Promoted Version if an update was checked in and not yet promoted.
- **Footprint Status** - The footprint file status, as shown in the **Status** column in the footprint project.

Note: In determining the **Validation Status** for each footprint file, **only** the built-in **Version_ID** for .dra and .psm files are compared. The Version_ID of the files match when .psm files are updated at the same time .dra files are saved. If your Footprints were uprev'ed using DB Doctor, the Version_IDs of your .dra and corresponding .psm files will not match and you'll need to sync the files in groups or individually as shown below.

You can synchronize the Version_IDs of a .dra/.psm pair individually (per footprint file) or in groups (per folder). To sync individual footprint files using PCB Editor, open and save each .dra file to synchronize the Version_IDs. If you have many files, you may use the **Sync .DRA to .PSM files** feature of EDM. **This functionality overwrites existing files**. If you want to maintain a copy of your original files, make a backup copy of the folder before you begin syncing.

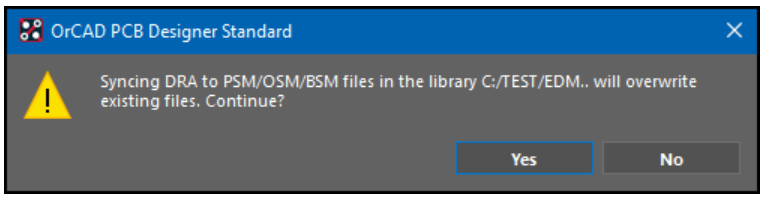
Sync multiple footprint (.dra to .psm) files in a directory using EDM > Sync DRA to PSM in PCB Editor

1. Select the **EDM > Sync DRA to PSM/OSM/BSM files** menu item.



Uprev and Sync DRA files to PSM/OSM/BSM files

2. The **Pick Library directory** window appears. **Select** the folder location of your .dra and .psm files to be synced, then click **Choose**.
3. A warning appears, stating the Sync will update all your footprint files in the selected folder. Click **Yes** to confirm that you want to overwrite existing files in the selected folder with the updated versions that will be created during sync. Or click **No** if you wish to cancel and close.



Once confirmed, PCB Editor will begin opening each footprint in the selected folder saving it to create new .dra and .psm/.osm/.bsm file pairs. When complete, PCB Editor will stop opening and closing the footprint files automatically.

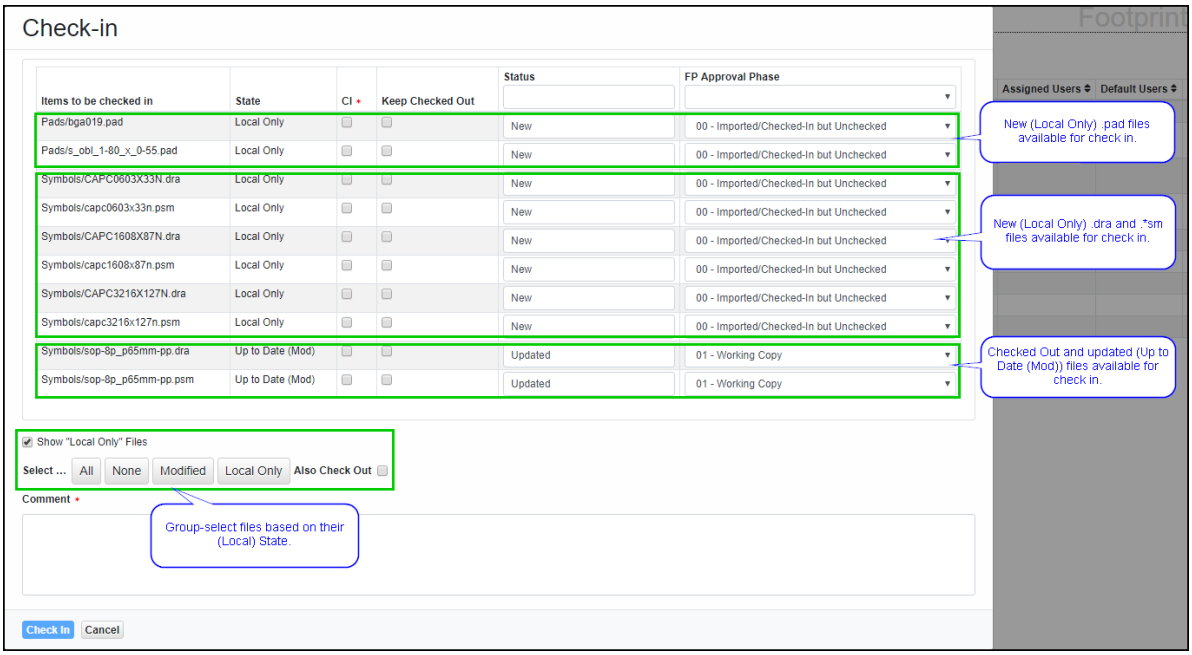
4. Now that the footprint .dra/.*sm pairs are in sync (i.e. have matching Version_IDs), they can be imported into EDM, checked out/in as needed, promoted when ready, and placed on your boards. When checking in your managed board files, the EDM Validation Report will use the updated Version_IDs when comparing the placed footprints on the board and the promoted footprints in the footprint promote path(s).

8.4 Check in Footprint Files

The project details of a Footprint project may be viewed from the **File**, **Footprint**, or **Padstack** view. If Step Models are enabled, project details can also be viewed from the **Step** view. The **Check In** button is available in each footprint view.

When you are ready to check in your footprint files, click the **Check In** button along the top row the Project Details. EDM will auto-launch PCB Editor (if not already open) and read the files and relationships for the new (**Local Only**) and checked out and modified (**Up to Date (Mod)**) files. PCB Editor will minimize when finished and the **Check In** form will appear. Files with **Up to Date (Mod)** state will be auto-selected when opening the check in form.

Files that are new (**Local Only**) or checked out and modified (**Up to Date (Mod)**) will be available for check in. When you create a Footprint Project, you have the option to either create and name the designated **Symbols** and **Pads** folders as sub-folders within the project, or to store all files in a **single folder**. If Step Models are used, you can name the designated **Steps** folder as well. Before new footprint files can be checked in, you need to copy the files into their respective project folders (Symbols, Pads, and Steps). Only footprint and step files (**.dra, *.sm, .pad, and .stp/.step**) will be shown in the Check In form and available for check in. For example, .log files are excluded from the checked in even if you copy them into a footprint project folder.



Footprint Check in form

Note: The Version_ID of the footprint files needs to match in order for EDM Validation Report to work properly with footprint file comparison between managed boards and promoted footprint files. They match when .psm files are updated at the same time .dra files are saved. If your Footprints **were uprev'ed using DB Doctor**, the Version_IDs of your .dra and corresponding .psm files **will not match**. You can synchronize the Version_IDs of a .dra/.psm pair individually (per footprint file) or in groups (per folder). To sync individual footprint files using PCB Editor, open and save each .dra file to synchronize the Version_IDs. If you have many files, you may use the Sync .DRA to .PSM/.OSM/.BSM files feature of EDM. **This sync functionality overwrites existing files**. If you want to maintain a copy of your original files, make a backup copy of the folder before you begin syncing. Refer to Check In Managed Board Files for more info.

Check in Footprint Files

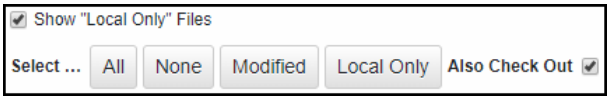
1. Click the **Check In** button that is located in the upper left corner of the Project Details page. This opens the Check In form.



2. Select files to be checked in. To filter out Local Only (New) files from the list, you can uncheck **Show "Local Only" Files**. If you'd like files to remain checked out after check in, this can be done in groups (by State via **Select...** buttons) or individually (via **CI** buttons), and may be done in combination with each other. You may wish to group-select files by State first, then individually select/deselect items after that.
   a. **Group select files (by State)**: The **Select...** buttons allow groups of files to be selected based off of the (Local) State of the files shown in the list of **Items to be checked in** (Check In list). The buttons affect the checkboxes in the **CI** and **Keep Checked Out** columns, respectively.

First, select the **Also Check Out** checkbox if you are choosing a group of files that you wish to remain checked out after the check in completes. This is optional and is usually done when you want to check files into the vault, but keep them checked out to continue working on them. This option is saved per user and your last-used selection will be applied when opening the Check In form.

Next, click one of the **Select...** buttons (located below the list of **Items to be checked in**) to select a group of files with a given (Local) State.



Select... Buttons with Local Only Files



Select... Buttons without Local Only Files

- **All** - selects all files in the list.
- **None** - deselects all files in the list. This can be useful if you accidentally selected files and want to start again with all files unchecked.
- **Modified** - selects the files shown as **Up to Date (Mod)** under the **State** column. It does not deselect any previously-selected files, regardless of their State.
- **Local Only** - selects only the files shown as **Local Only** under the **State** column. It does not deselect any previously-selected files, regardless of their State. This button is hidden when **Show "Local Only" Files** is unchecked.

b. **Individually select files**: The individual **CI** and **Keep Checked Out** boxes for each file allow individual files to be selected on-the-fly, regardless of State.

Select or de-select individual checkboxes in the **CI** (Check In) column for each file you want to check in. Select the **Keep Checked Out** checkbox for each file that you wish to remain checked out after the check in completes. This is optional and is usually done when you want to check a file into the vault, but keep it checked out to continue working on it.

3. Enter a value into the Status field and any other file fields shown (configured in Customizable Fields configured for Footprints) if you want to update the file fields for the items you are checking in. Updating these fields is optional and can be done individually using each file's text box or pick list for each field, or as a group using using each file's top-level text box or pick list for each field. This may be done in combination with each other and you may want to update file fields by group, then individually change the ones that should be different. For new files, the configured default values will be auto-populated for Status and each enabled file field.
4. Enter a comment to describe the check in. The comment will be saved to the specific historical record in the project history. The red asterisk indicates required field entry.
5. Click the Check In button to check in the selected file(s), or click the Cancel button to close the **Check In** form.

After successful check in, the following success message displays. You may dismiss the message manually by clicking the **X** in the message or it automatically closes after a few seconds. Refer to Warnings and Errors for information about other check in messages.

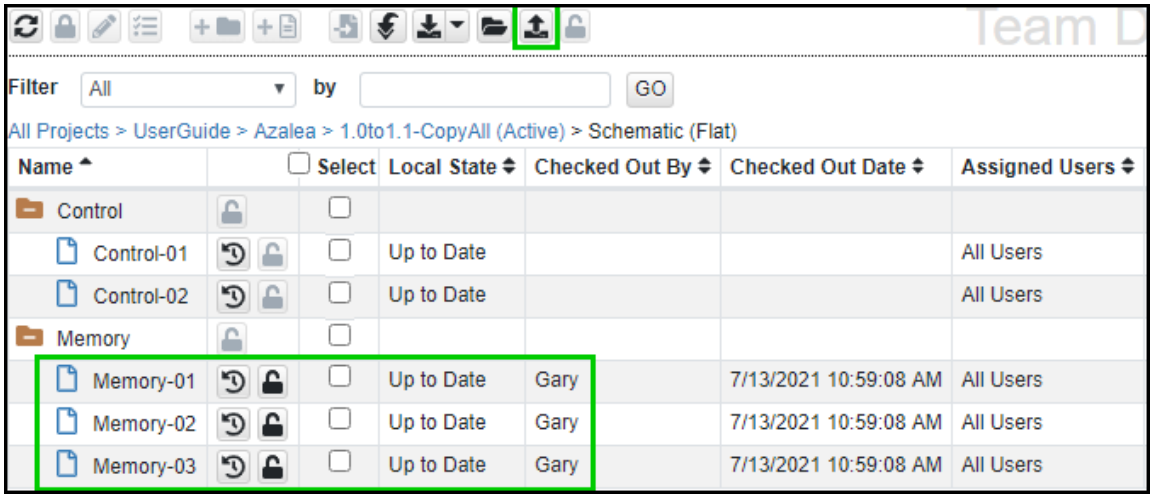


The **Checked Out By**, **Checked Out Date, and Last Check In Date** columns will be updated accordingly. The **Status** field and Footprint user fields will be updated as well, based on the values entered during check in.

8.5 Check In Team Design Pages

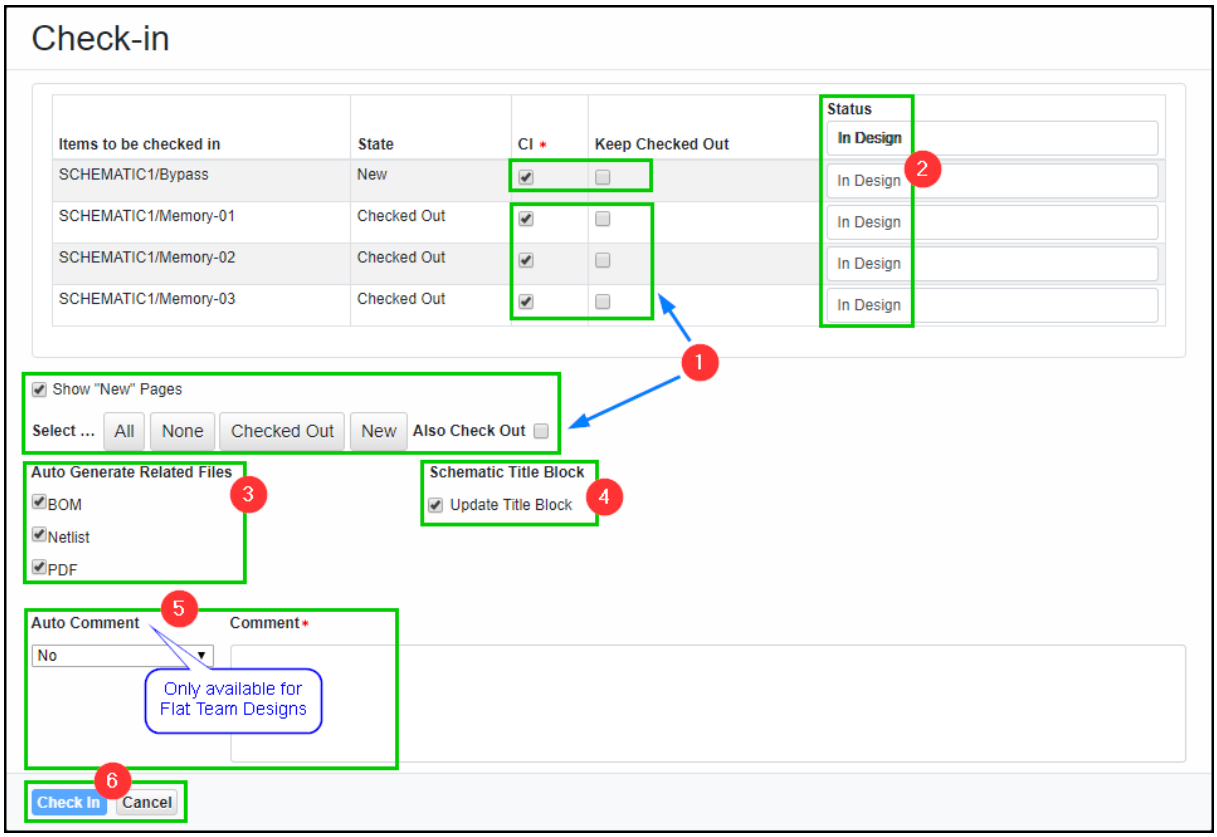
Individual pages of a team design must be checked in from the Team Folder Details page. The top row of the details page contains action buttons such as the **Check In** button. This button opens the **Check In** form, which shows a user all team design pages available for check in and includes their Status and any enabled Team Design User Fields configured in Customizable Fields.

The example below shows the user Gary has 3 pages checked out and Up to Date. The state of modification to pages in the merged design is not reflected in the Local State.



View the Team Folder Details to Identify Checked Out Pages

The **Check In** button opens the Check In form, which automatically lists pages that are available for check in. The list includes pages that are **New** or have been **Checked Out**, regardless of whether they have been modified. Files with **Checked Out** state will be auto-selected in the check in form.



Check In Pages Form

Check In Form

The Check In form only lists pages available for check in. This list includes new pages as well as checked out pages. If there are new or modified pages that you do not want to check into the vault, do not select those items. The Check In form needs to be completed before you are able to proceed.

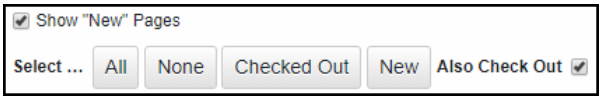
Check in Team Design Pages

1. Select pages to be checked in. To filter out New pages from the list, you can uncheck **Show "New" Pages**. If you'd like pages to remain checked out after check in, this can be done in groups (by State via **Select...** buttons) or individually (via **CI** buttons), and may be done in combination with each other. You may wish to group-select pages by State first, then individually select/deselect items after that.

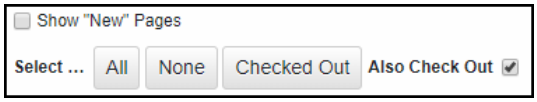
   a. **Group select files (by State)**: The **Select...** buttons allow groups of files to be selected based off of the (Local) State of the files shown in the list of **Items to be checked in** (Check In list). The buttons affect the checkboxes in the **CI** and **Keep Checked Out** columns, respectively.

   First, select the **Also Check Out** checkbox if you are choosing a group of files that you wish to remain checked out after the check in completes. This is optional and is usually done when you want to check files into the vault, but keep them checked out to continue working on them. This option is saved per user and your last-used selection will be applied when opening the Check In form.

Next, click one of the **Select...** buttons (located below the list of **Items to be checked in**) to select a group of files with a given (Local) State.



Select... Buttons with New Pages



Select... Buttons without New Pages

- **All** - selects all pages in the list.
- **None** - deselects all pages in the list. This can be useful if you accidentally selected pages and want to start again with all pages unchecked.
- **Checked Out** - selects the pages shown as **Checked Out** under the **State** column. It does not deselect any previously-selected pages, regardless of their State.
- **New** - selects the pages shown as **New** under the **State** column. It does not deselect any previously-selected pages, regardless of their State. This button is hidden when **Show "New" Pages** is unchecked.

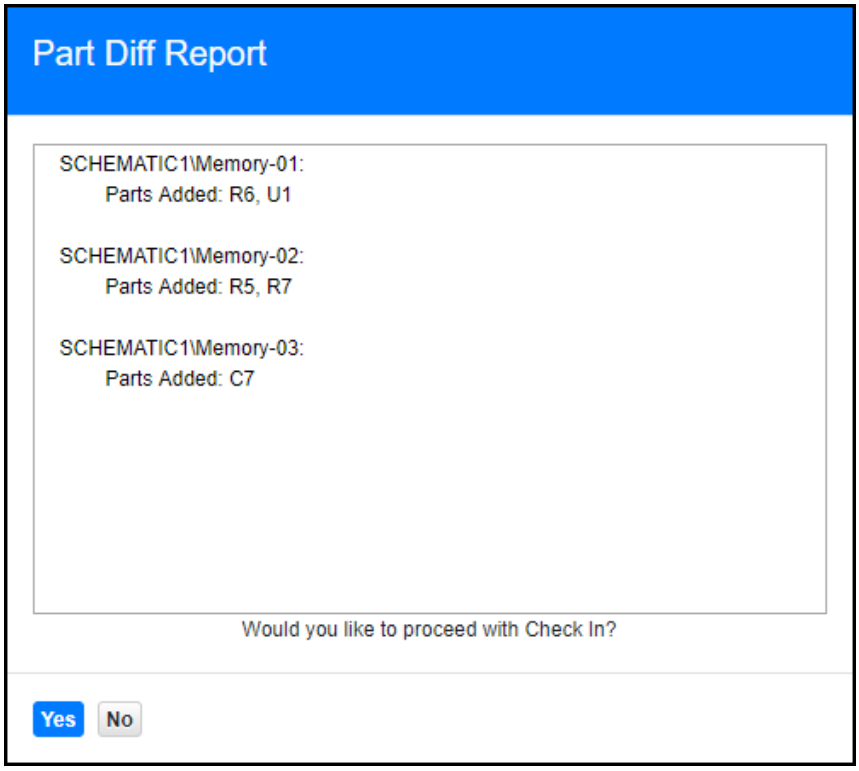
b. **Individually select files**: The individual **CI** and **Keep Checked Out** boxes for each file allow individual files to be selected on-the-fly, regardless of State.

Select or de-select individual checkboxes in the **CI** (Check In) column for each file you want to check in. Select the **Keep Checked Out** checkbox for each file that you wish to remain checked out after the check in completes. This is optional and is usually done when you want to check a file into the vault, but keep it checked out to continue working on it.

2. Enter a value into the **Status** field and any other Team Design fields shown (configured in Customizable Fields configured for Team Designs) if you want to update the design page fields for the items you are checking in. Updating these fields is optional and can be done individually using each file's text box or pick list for each field, or as a group using using each file's top-level text box or pick list for each field. This may be done in combination with each other and you may want to update file fields by group, then individually change the ones that should be different. For new files, the configured default values will be auto-populated for Status and each enabled file field.
3. **Auto Generate Related Files** – Select checkboxes to identify the files you want to auto generate. The files you select will be auto-generated with information derived from the Team Design Pages. EDM will place the files into the configured project folders. **Note**: These auto generate options are only available when the Team Design folder is designated as the Managed DSN folder.

   The related files you may choose are:
   - **BOM** (Bill of Materials) – create and check in the BOM(s) using the BOM template(s) associated with the project. The format of the BOM(s) is defined by the Admin or Project leader as described in the section topics, Manage BOM Template and the BOM folder associated to the project.
   - **Netlist** – create standard Capture Netlist files for PCB Editor.
   - **PDF** – create the intelligent PDF. Before you can successfully select this checkbox you need to configured the PS interpreter. Follow instructions in EDM Settings for PDF configuration.

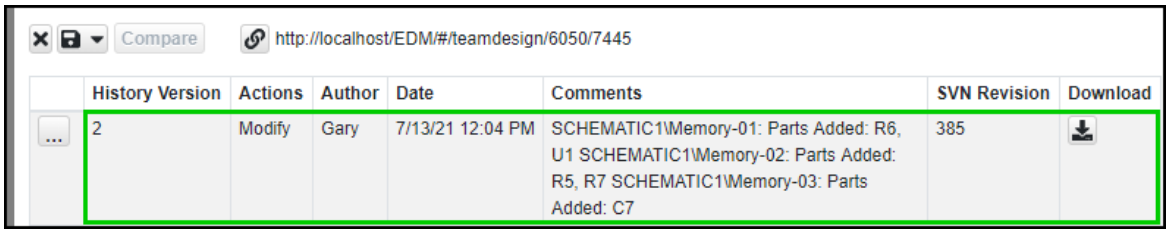
4. **Schematic Title Block: Update Title Block** - EDM updates the Title Block for each selected page in the design file as defined in the Title Block Mapping of the EDM Configuration.
5. Enable an Auto Comment (**Compare** or **Compare with Preview**) or enter a comment to describe the check in. Auto comments and user-entered comments will be saved to the specific historical record in the project history.
    a. **Auto Comment** - only available for Flat Team designs. When auto comment is enabled, EDM compares each page in the local design that is selected for check in to the corresponding page in the vault. New pages don't yet exist in the vault so they will not appear in the report. The following options for auto comment are provided:
    - **No** - no comparison will be made.
    - **Compare** - pages will be compared and differences will be added to the comments during the check in. Part Reference Designators (RefDes) will be compared, then additions and deletions will be added to the comments for the check in. Parts with properties that have changed will also be added to the comments.
    - **Compare with Preview** - this is the same as the **Compare** option, but the compared differences will be shown in a Part Diff Report window before the actual check in. Click **Yes** to proceed with the check in continues and the comments are added as shown in the **Team Design - Page History** figure below, or click the **No** to close the **Part Diff Report** and return to the **Check In** form.



Part Diff Compare Report

    b. **Enter a Comment** to describe the check in. This is optional if used in combination with the Auto Comment.

    The following figure shows an example of history with auto comment enabled.



Team Design - Page History

6. Click the Check In button to check in the selected page(s), or click the Cancel button to close the **Check In** form.

   After successful check in, the following success message displays. You may dismiss the message manually by clicking the **X** in the message or it automatically closes after a few seconds. Refer to Warnings and Errors for information about other check in messages.

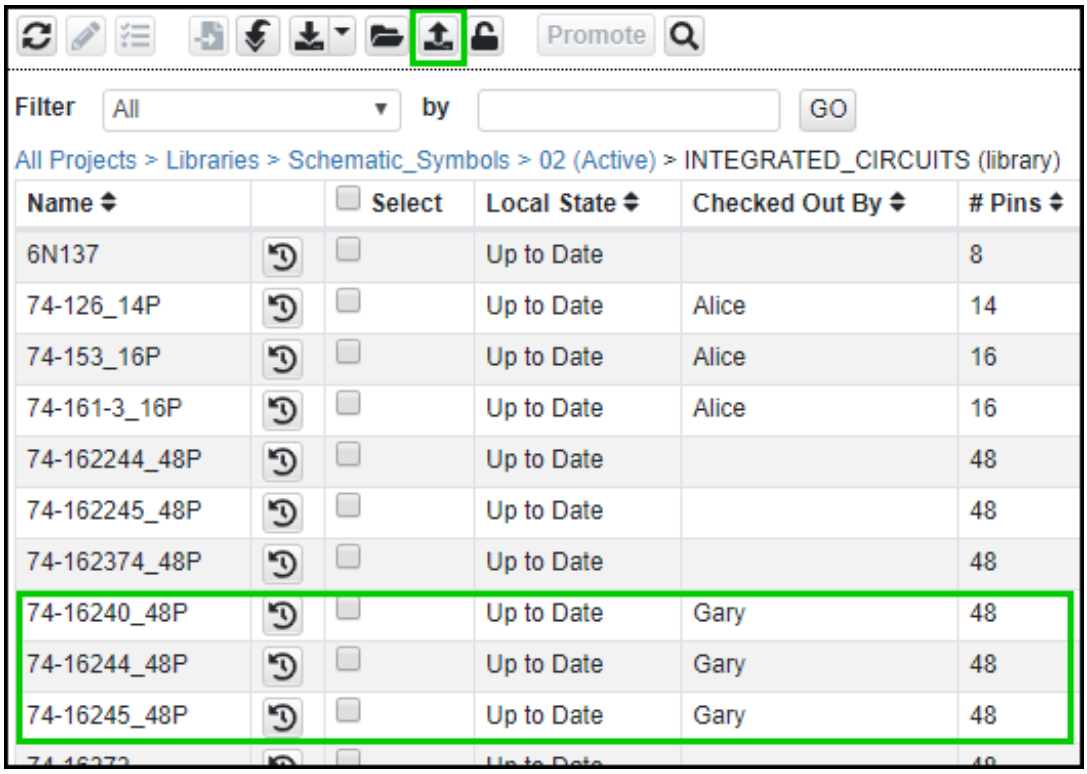
   

   The **Checked Out By**, **Checked Out Date, and Last Check In Date** columns will be updated accordingly. The **Status** field and Team Design fields will be updated as well, based on the values entered during check in.

Note: Once a Team Design has been frozen, the frozen design can be checked in from the Project Details page. If the Team Design folder is designated as the managed design, follow instructions to Check In Managed Capture Files. Otherwise, check in the frozen design file the same way you would check in any non-managed file (e.g. .txt, .docx, .xlsx, etc.).

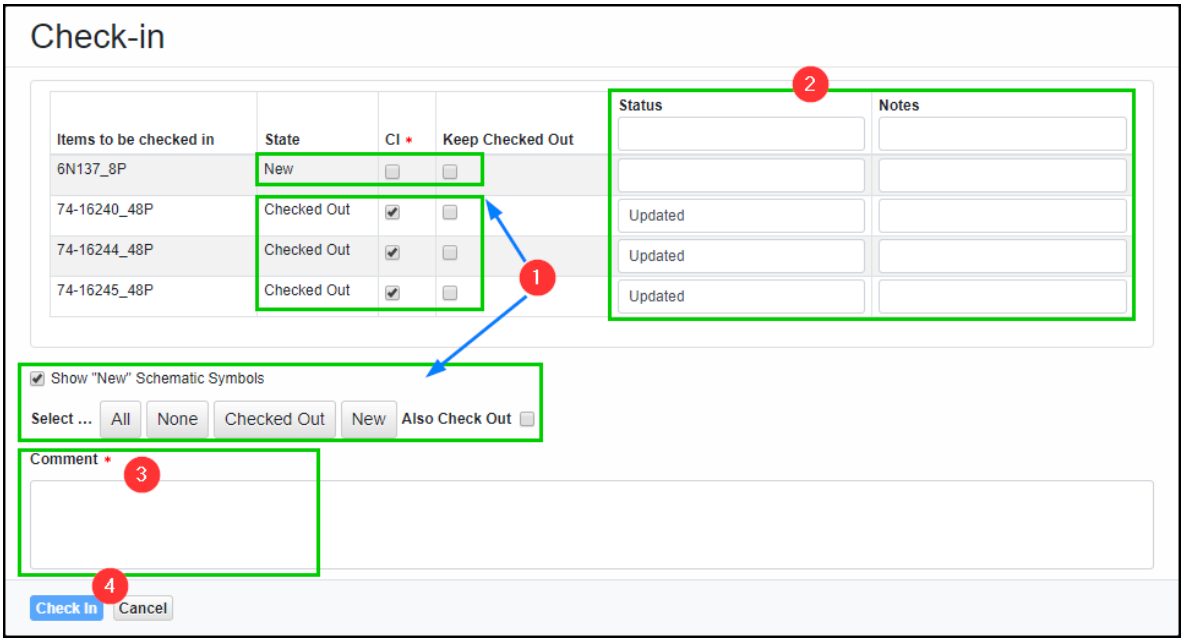
8.6 Check In Team Library Symbols

Users can check in individual schematic symbols from a (Team) Library Folder. Before checking in one or more symbols, verify the **Checked Out By** property to ensure you have symbols checked out. EDM system-defined properties such as the Local State, Checked Out By, and Pin Count (shown in the image below) are updated during the check in process. User-configured Schematic Symbol fields can be manually updated in the Check In form or the project details page.



(Team) Library View for Schematic Symbols

Click the **Check In** button in the upper left corner of EDM to open the Check In form, which automatically lists symbols that are available for check in. The list only includes symbols that are **New** or have been **Checked Out**, regardless of whether they have been modified. Files with **Checked Out** state will be auto-selected in the check in form.



Library Check In Form

The process for checking in symbols is nearly the same process followed for Check In of Team Design Pages. The only difference is the auto comment issue

Check In Form

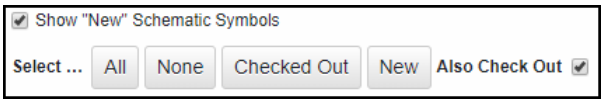
The Check In form only lists symbols available for check in. This list includes new symbols as well as checked out symbols. If there are new or modified symbols that you do not want to check into the vault, do not select those symbols. The Check In form needs to be completed before you are able to proceed.

Check In Team Library Schematic Symbols

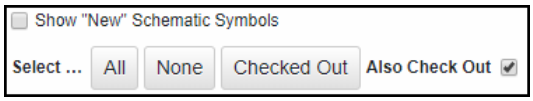
1. Select schematic symbols to be checked in. To filter out New symbols from the list, you can uncheck **Show "New" Schematic Symbols**. If you'd like symbols to remain checked out after check in, this can be done in groups (by State via **Select...** buttons) or individually (via **CI** buttons), and may be done in combination with each other. You may wish to group-select symbols by State first, then individually select/deselect symbols after that.
   a. **Group select symbols (by State)**: The **Select...** buttons allow groups of symbols to be selected based off of the (Local) State of the symbols shown in the list of **Items to be checked in** (Check In list). The buttons affect the checkboxes in the **CI** and **Keep Checked Out** columns, respectively.

      First, select the **Also Check Out** checkbox if you are choosing a group of symbols that you wish to remain checked out after the check in completes. This is optional and is usually done when you want to check symbols into the vault, but keep them checked out to continue working on them. This option is saved per user and your last-used selection will be applied when opening the Check In form.

      Next, click one of the **Select...** buttons (located below the list of **Items to be checked in**) to select a group of symbols with a given (Local) State.



Select... Buttons with New Schematic Symbols



Select... Buttons without New Schematic Symbols

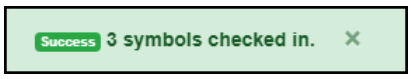
- **All** - selects all symbols in the list.
- **None** - deselects all symbols in the list. This can be useful if you accidentally selected symbols and want to start again with all symbols unchecked.
- **Checked Out** - selects the symbols shown as **Checked Out** under the **State** column. It does not deselect any previously-selected symbols, regardless of their State.
- **New** - selects the symbols shown as **New** under the **State** column. It does not deselect any previously-selected symbols, regardless of their State. This button is hidden when **Show "New" Schematic Symbols** is unchecked.

   b. **Individually select symbols**: The individual **CI** and **Keep Checked Out** boxes for each item allow individual symbols to be selected on-the-fly, regardless of State.

Select or de-select individual checkboxes in the **CI** (Check In) column for each file you want to check in. Select the **Keep Checked Out** checkbox for each file that you wish to remain checked out after the check in completes. This is optional and is usually done when you want to check a file into the vault, but keep it checked out to continue working on it.

2. Enter a value into the **Status** field and any other Team Library fields shown (configured in Customizable Fields configured for Team Libraries/Schematic Symbols) if you want to update the symbol fields for the ones you are checking in. Updating these fields is optional and can be done individually using each file's text box or pick list for each field, or as a group using using each file's top-level text box or pick list for each field. This may be done in combination with each other and you may want to update file fields by group, then individually change the ones that should be different. For new symbols, the configured default values will be auto-populated for Status and each enabled file field.
3. Enable a comment to describe the check in. Comments will be saved to the specific historical record in the project history.
4. Click the Check In button to check in the selected page(s), or click the Cancel button to close the **Check In** form.

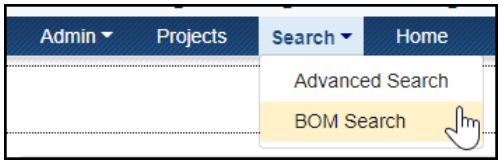
After successful check in, the following success message displays. You may dismiss the message manually by clicking the **X** in the message or it automatically closes after a few seconds. Refer to Warnings and Errors for information about other check in messages.



The **Checked Out By**, **Checked Out Date, and Last Check In Date** columns will be updated accordingly. The **Status** field and Team Library fields will be updated as well, based on the values entered during check in.

9 Search

From the **Search** menu, you may select either BOM or Advanced Search.



BOM Search enables you to search for BOMs and associated projects that contain specific criteria. For example, if a part has become obsolete and you want to identify projects that use that part number, you can enter the obsolete part number into the search string.

Advanced Search enables you to search for files, pages, symbols, and footprints that match specific search criteria. For example, if you need to find footprints promoted between certain dates or files checked out to a specific user, you can define a search that retrieves files and projects that match that criteria.

9.1 BOM Search

The BOM Search screen contains the following sections.

1. Define Search Criteria - You need to define or specify search criteria that controls how the query is conducted.
2. Add BOM File Fields - You may add fields of information to appear in your search results.
3. Search results - You can organized search result by changing the order of search results.
4. Saving search criteria - You may save the search criteria as a global or personal search that can be used at a future time.

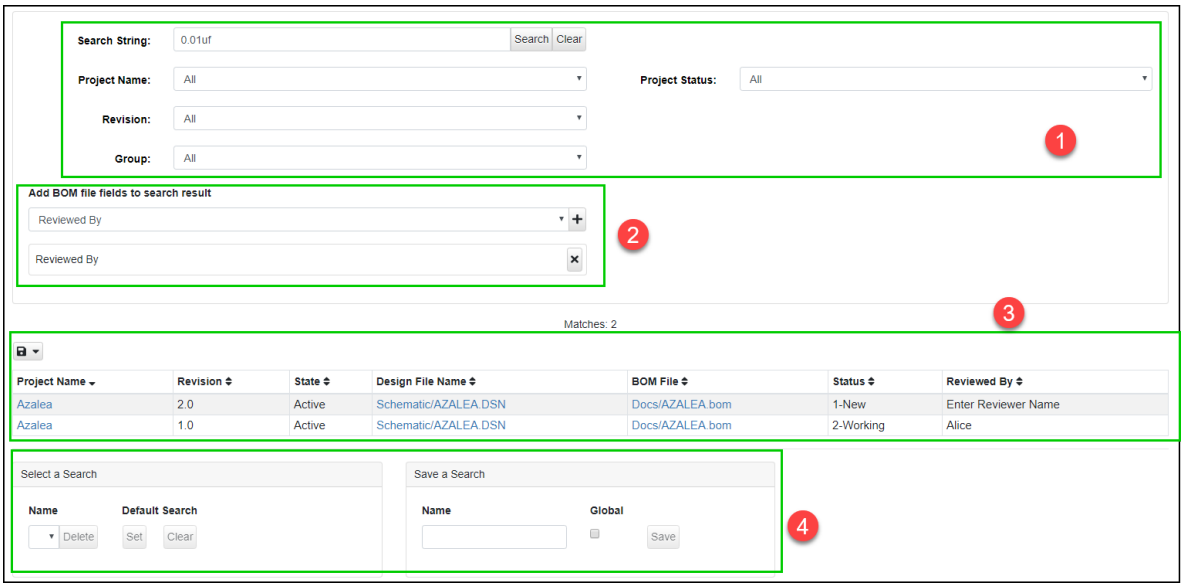
You can export the results of your BOM search to either a CSV or Excel file that can easily be referenced at a later time.

Define Search Criteria

You can search for a BOM using any of the following criteria:

- **Search String** - the search string may be a part number or any parameter value in a BOM that has been auto-generated and checked in with a managed design.
- **Project Name** - select a specific project to narrow your search for a specific BOM. Select the project name from the drop-down list.
- **Revision** - select a revision to narrow your search criteria.
- **Group** - select the project group that you wish to search through.
- **Project Status** - enter a status to narrow your search criteria. Left it blank to include all project statuses.

Click on **Search** button to perform the search. Click on the **Clear** button to remove all criteria and BOM file fields.



BOM Search

Add Fields to Search Results

If you have defined any customizable file fields in **EDM Configuration**, you may add them to the displayed fields in the search results. You may select any additional fields from the drop-down list and include them by using the + button. Multiple additional fields can be added and the fields will be presented in the Search Results regardless of whether data populates the field. To remove any additional fields, click the × button. Fields may also be reordered by clicking and dragging the fields up or down in the list.

Search Results and Ordering

The results from your search will show the number of files matching your criteria as well as a list of the file. The listing will include the following columns:

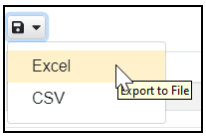
- **Project Name** - name of the project as a link. Selecting the project name will open the project revision that the BOM file was found.
- **Revision** - Revision of the project associated with the BOM file
- **State** - State of the project revision
- **Design File Name** - File name of the Capture Design file as a link. Selecting the design filename will open the design file in Capture.
- **BOM File** - File name for the BOM file as a link. Selecting the BOM filename will open the file with the associated program.
- **Status** - Status of the file
- **User fields** - If configured, value of the user field for the BOM file.

The list order may be changed by selecting the column heading. You may reverse the order by selecting the column heading again.

Export BOM Search Results

You can export the results of your BOM search to either a CSV or Excel file.

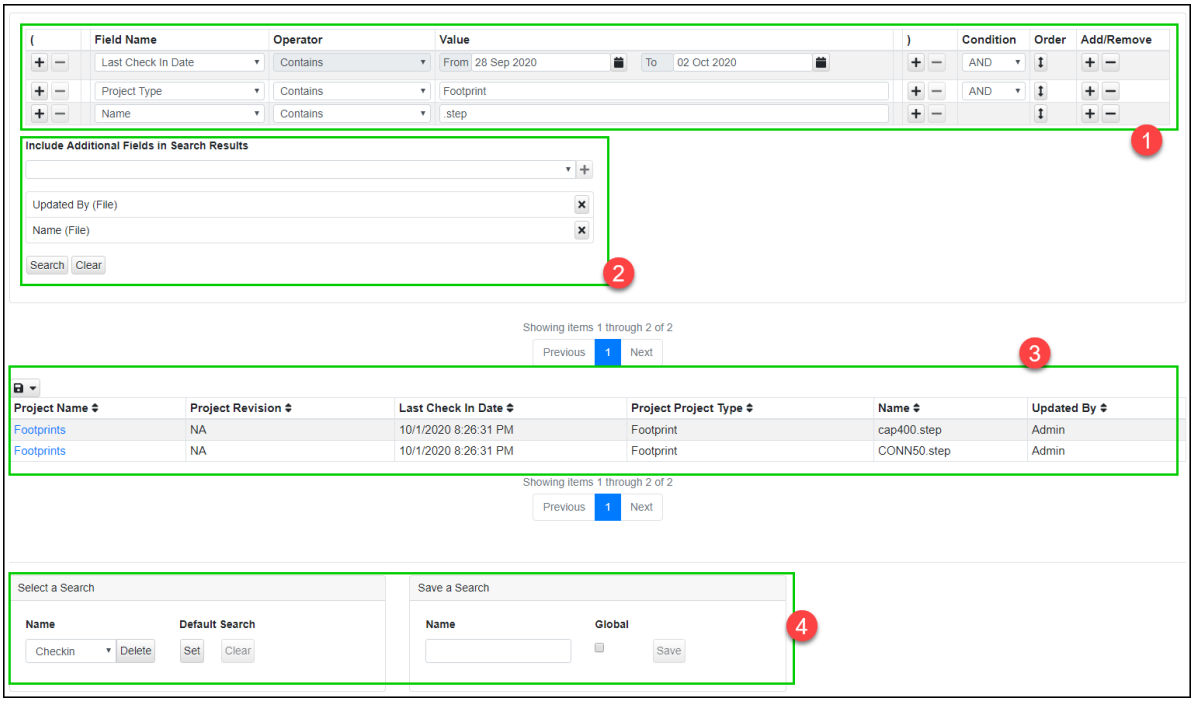
To export your BOM Search results, click the **Export to File** drop arrow that is located before the search results and select either Excel or CSV from the menu. Save and name the BOM search results when prompted.



9.2 Advanced Search

The Advanced search screen contains the following sections.

1. Define Search Criteria - You can specify a basic search or more complex searches by adding conditions and place appropriate parentheses to refine your search criteria.
2. Include Additional Fields - You may add fields of information to appear in your search results.
3. Search results - The search results are retrieved and displayed when you click the **Search** button. You can organize your search result by changing the order of the results.
4. Save search criteria - You may save the search criteria as a global or personal search that can be used at a future time.



Advanced Search

Define Search Criteria

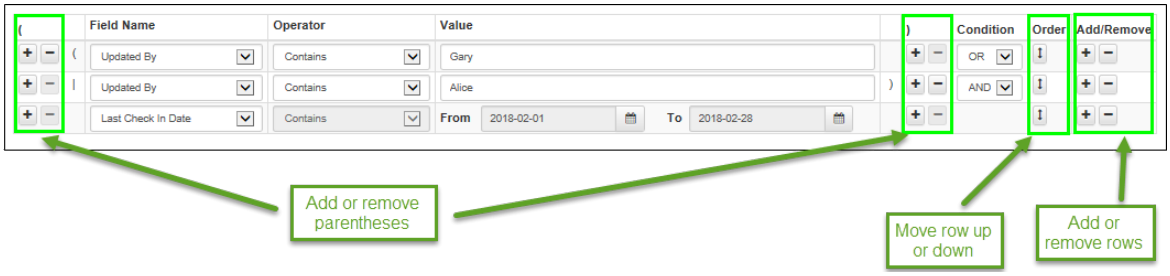
To specify search criteria:

1. Configure the following fields as desired: (see call-out #1 in the previous graphic)

- **Field Name** - select an item from the drop down list that specifies the subject of your search.
- **Operator** - select an operator from the drop down list that specifies how your search will perform the search for the "Field Name" you selected. Available operators are Contains, Does Not Contain, Contains Data, Does Not Contain Data, Starts With, Does Not Start With, Ends With, Does Not End With, and Is, and Is Not.
  When a date is selected for the Field name, you need to specify from and to dates.
  When Pin Count is selected as the Field name, you may select one of the additionally listed operators: Greater Than (>), Less Than (<), Greater Than or Equal To (>=), or Less Than or Equal To < =).
- **Value** - enter any alpha-numeric value that you want the operator to apply to the selected field.
-  - There are three sets of buttons you can use to build complex queries. Selecting the Add  button adds a left or right parentheses or a row for additional criteria. Selecting the Remove  button removes the item added. This is used for complex searches.
- **Condition** - used to specify the logic between each row's criteria. You may specific AND or OR condition. This is used for complex searches.
-  - select the Move Row Up or Down  button and move the row a new position.

2. After defining your search criteria, click the **Search** button to view search results. Or you may select the **Clear** button to clear your search criteria.

The figure below shows an example where you want find files that were check in by either Gary or Alice between Feb 1, 2018 and Feb 28, 2018. This complex query is built with three simple queries joined by OR and AND conditions.



Query for Advanced Search

Include Additional Fields in Search Results

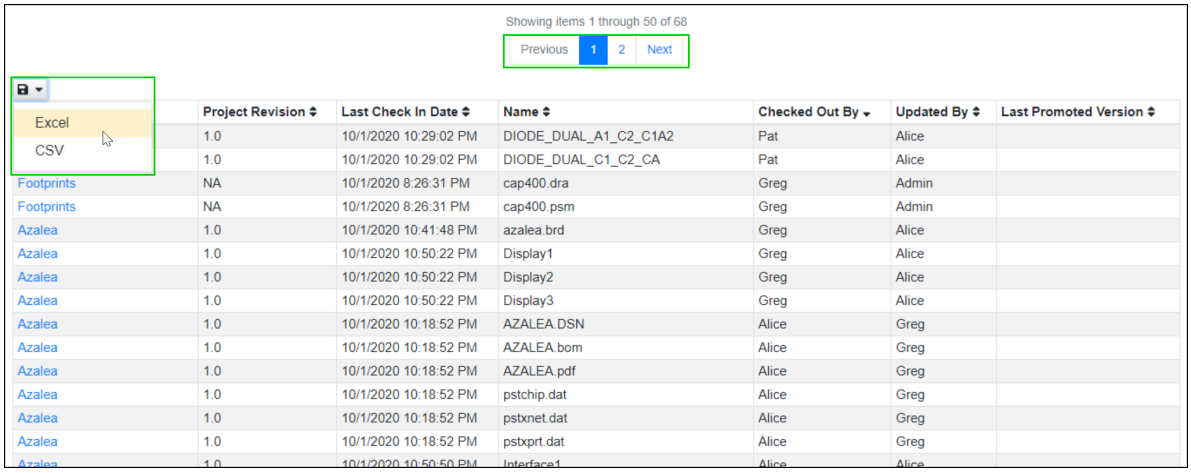
Additional fields may be displayed with the results of a search. You may select any additional fields from the drop-down list in the **Include Additional Fields in Search Results** section and include them by using the **Add**  button. Multiple additional fields can be added and the field will be presented in the Search Results regardless of whether data populates the field. Fields may also be reordered by clicking and dragging the fields up or down in the list.



After adding fields, select the **Search** button to see results. You may select the **Clear** button to clear your search criteria.

Search Results and Ordering

Search results lists the number of files that match the search criteria you specified. The files are presented in a list that displays 50 files at a time. You can cycle through pages of results using the buttons at the top and bottom of the search results area. You may export the list of results to Excel or CSV file format by clicking the **Export** button and selecting your preferred format.



Results from Advanced Search

The listing includes the following columns of information:

- **Project Name** - name of the project as a link. Selecting the project name will open the project revision that the BOM file was found.
- **Provision Revision** - revision of the project associated with the BOM file
- **Field names** of field in the search criteria **-** field in your search criteria will be in the columns following project name and revision.
- **Added field -** fields in the order added and adjust.

The list result order may be changed by selecting the column heading. You may reverse the order by selecting the column heading again.

9.3 Saving Search Criteria

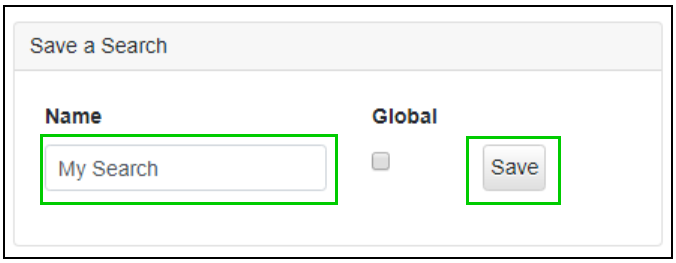
Search criteria can be saved for later use. The criteria and fields that define how a search is performed can be saved as personal or global search criteria. Personal search criteria are only available to the user that saved the search. Global searches can be used by all users. Only users whose role has been granted the Save Global Searches permission can save global searches.



BOM and Advanced Search criteria is saved for later use after the search is defined and conducted.

Save A Search

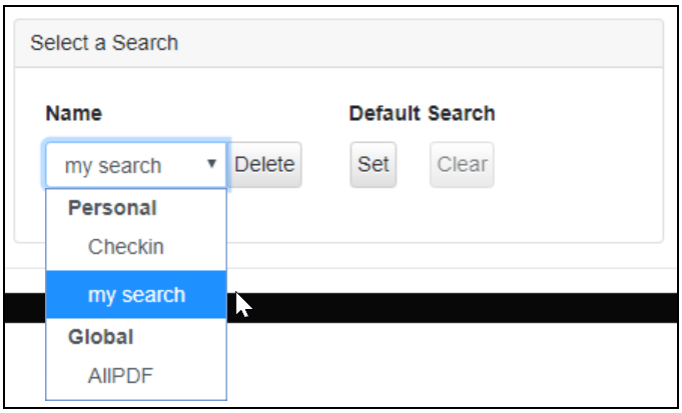
To save a search, in the **Save a Search** section, enter a name into the Name field and click the **Save** button. Users will appropriate permission may select the **Global** checkbox to save it as a global search.



Save A Search

Select A Search

To select a previously defined search, select the name of the saved search from the drop-down list. The list groups searches by Global and Personal searches. After you select a search name, the criteria of the saved search populates input fields. To conduct the search, you can click the **Search** button that is located at top of the screen for the BOM Search and under the Include Additional Fields section for an Advanced Search.



If you want to setup an Advanced Search or BOM Search to open and show a specific saved search, select the saved search name and click the **Set** button under **Default Search** area.

If you no longer want a search to be set as your default, click the **Clear** button. To delete a saved search, select it from the drop-down list and click the **Delete** button.

Edit A Saved Search

To edit a saved search, select the saved search; make changes to criteria and added fields as desired. Then name and save the search.

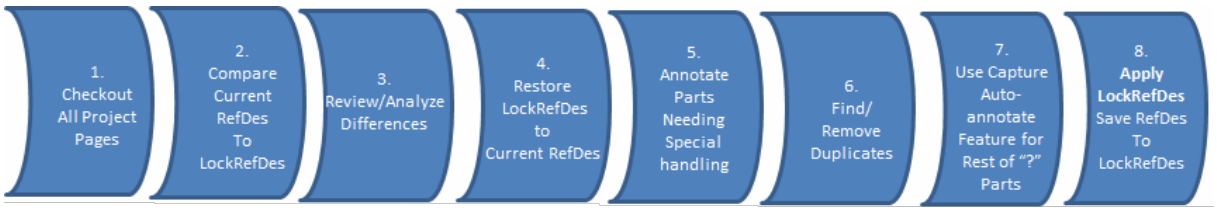
10 Reference Designator Management For Hierarchical Team Designs

In any schematic, a basic requirement is that each reference designator be unique. This is an essential part of the overall flow between schematic design and PCB layout. Likewise, information coming back after PCB layout is complete will need to be applied to the schematic. The information is in the form of a back-annotation file. Capture uses the back-annotation file to identify changed component reference designators and then update them accordingly in the schematic.

Whether blocks of reference designators are manually or automatically assigned, managing reference designators early and frequently reduces surprises and rework later in the design cycle. As long as EDM or RefDes is enabled, EDM will help you to save and restore a copy of the reference designator for later use.

The EDM Reference Designator feature, when enabled, will add a "**LockRefDes**" property to each symbol as it is placed on an EDM schematic. The **LockRefDes** property is used by EDM to help the assigned leader manage references across all the pages of a design. As long as EDM or RefDes is enabled, the **LockRefDes** property will be added as read only. Users will not be able to edit the property. Pasting of a copied part will clear the **LockRefDes** property.

With multiple users working on various schematic pages, duplication of reference designators may occur. EDM has the ability to aid in the management of reference designators ensuring uniqueness as well as allowing the project manager to restore saved references. Typical steps for effective reference designator management are shown in the following figure:



Reference Designator Management Process

A description of these steps is provided here.

1. Before using any of the RefDes commands, you should make sure that you have Checked out all pages for the Team Design. This is a very important first step. If you do not check out all the pages, another user may add parts with references that may conflict with what you are setting. If the Team Design is a hierarchical Team Design, a restore will automatically occur when you check out the pages.

2. After Checking out all pages, you should perform Compare LockRefDes. The first time you do this on a Team Design, you should see all yellow parts. Yellow is expected when an

Apply LockRefDes hasn't been performed yet. Parts will also be yellow if the reference is "?". Comparing before making changes gives you a chance to see whether users have specifically changed the reference after a LockRefDes has been applied. If a user has done so, you can investigate the need for certain reference before making any changes.

3. Reviewing and understanding differences is important so that work for users who need certain references isn't destroyed. Red parts show that a user has edited parts after a LockRefDes has been applied. Green parts show that RefDes has not been changed since the LockRefDes has been applied. Users who do not have EDM opened or RefDes Enable may get green parts for parts copied.

4. If you find that there is no need to have any RefDes changed by users, you can run a Restore Lock. This command will restore all references that have been saved (applied) to LockRefDes previously. All parts that do not have a LockRefDes value will be restored to "?".

5. After the Restore, you may manually edit parts that need specific references. If you do not have any, you can skip this step.

6. Next, you will need to remove all duplicates. You can use Find Duplicates to identify all parts that have a corresponding duplicate. After finding duplicates, you may resolve this by manually editing them or use the Remove Duplicate command. The Remove Duplicate command will set all references with duplicate values to "?".

7. When you are sure that all duplicates are set to a reference of "?", you may use the built in Annotate feature in Capture. If you have a hierarchical block, you will need to make sure that the references for the blocks are not duplicated as well.

8. After all duplicates have been removed, you can **Apply LockRefDes** to save the references.

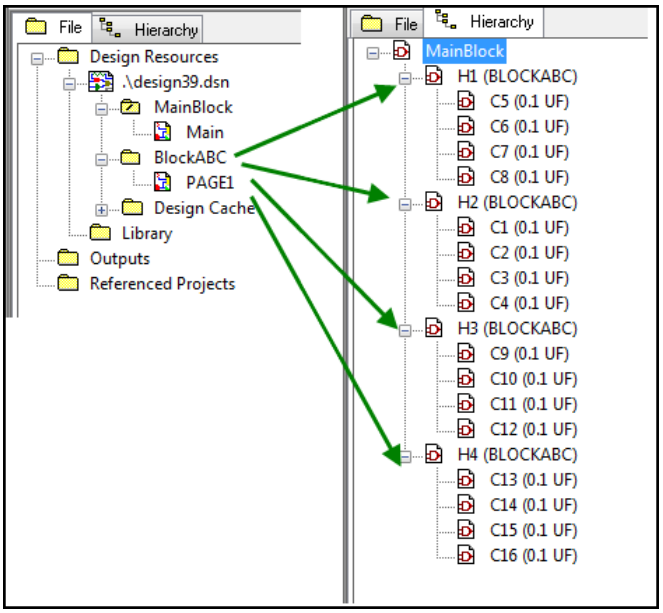
10.1 Reference Designators in Hierarchical Designs

Reference designators may change depending on the schematics checked out. This may occur even after you apply **LockRefDes**.

When you check out a schematic or page within a hierarchical design and open the schematic, Capture has no knowledge that the checked out schematic is part of a larger design or how many times it may be used. As such, it will default to using the original instance reference assigned when parts were placed.

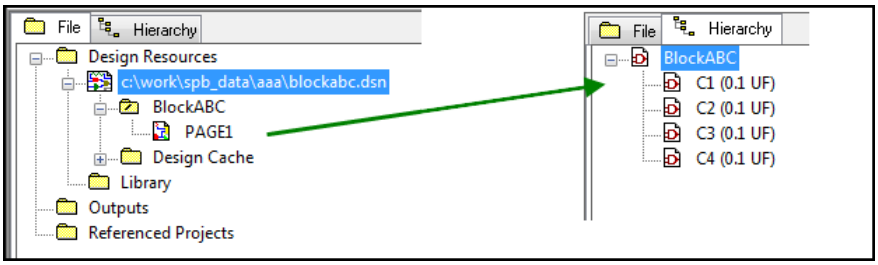
EDM will try to restore any applied **LockRefDes** when schematics and pages are checked out. Even though some of the pages may be checked out as read only, EDM will be able to restore **LockRefDes** if enough of the hierarchy has been checked out.

For example, if you have a design that uses BLOCKABC four times in the Main block (as shown on the left half of the following figure), the reference designators will look like the right half of the following figure:



Main Block Example

If you checked out just BLOCKABC, you will only see one of four sets of reference designators.



BLOCKABC Example

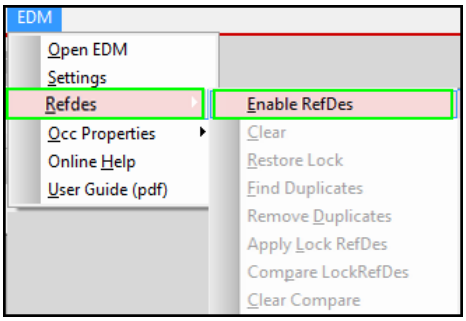
10.2 EDM RefDes Commands

Menu items available to manage reference designators include the following commands:

- Enable RefDes - enables Reference Designator functionality without opening EDM tab.

- Clear - clears RefDes values for only the parts that do not have a saved LockRefDes value.
- Restore Lock - restores parts that have LockRefDes values to RefDes property.
- Find Duplicates - identifies all parts that have more than one occurrence of the same RefDes value.
- Remove Duplicates - sets RefDes values to "?" for all the parts identified from the Find Duplicate command.
- Apply Lock RefDes - saves the RefDes values to the LockRefDes property (available to users with applicable permissions).
- Compare LockRefDes - compares and color codes parts during comparison of the RefDes values to the LockRefDes values.
- Clear Compare - resets the color codes to user preferences.

EDM Ref Des features are enabled by either opening EDM or selecting **Enable RefDes** from the **EDM > Refdes** Menu item. The **Apply LockRefDes** menu item will only be enabled for users whose role has been granted Apply Lock to Team Designs permissions.



Enable RefDes

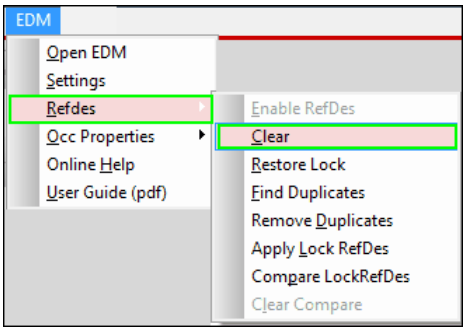
10.2.1 Clear

The **Clear** menu command only resets the currently visible RefDes on all parts that do not have a saved LockRefDes value. Resetting the RefDes changes currently visible RefDes to "?".

Parts that are placed on a design without opening EDM or without enabling EDM RefDes will not have any LockRefDes property. The **Clear** command will clear the RefDes for these types of parts. Performing a **Compare LockRefDes** will turn these types of parts yellow.

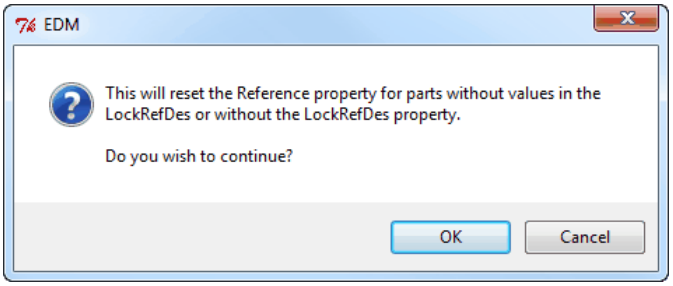
This command is similar to the **Restore Lock**. The major difference is that **Clear** will not clear the RefDes if a **LockRefDes** has been applied or saved. If a user changes a RefDes for some reason, you will be able to see the change when you run a **Compare LockRefDes**.

To clear your Reference Designators, under the **EDM** menu option select **Refdes > Clear**.



Clear Reference Designators on Parts

The EDM pop-up will display with the following message: *This will reset the Reference property for parts without values in the LockRefDes. Do you wish to continue?* Click **OK** to continue. Click **Cancel** to cancel the **Clear** command.



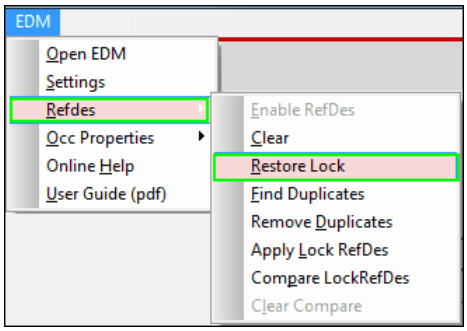
Note: Pages may need to be refreshed. To refresh the page, you will need to navigate to another page and then return to the current page.

10.2.2 Restore Lock

The **Restore Lock** menu command, will copy previously saved LockRefDes values back to the RefDes property. This will overwrite all references. For parts that do not have a saved LockRefDes values, they will be reset to "?".

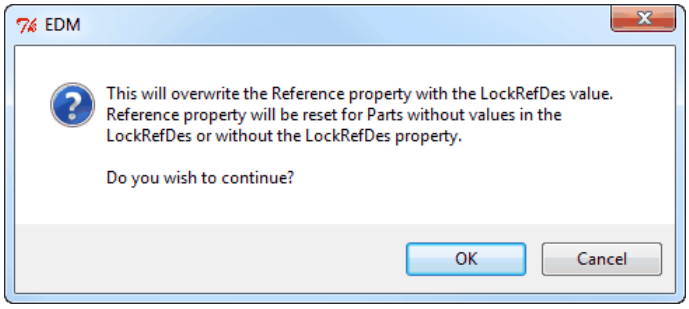
Although this is the same as **Clear**, you may not want to use this once layout has started in order to avoid rework.

To Restore the Lock on Reference Designators under the **EDM** menu option, click **Refdes > Restore Lock.**



Restore Lock

The EDM pop-up will display with the following message: *This will overwrite the Reference property with the Lock RefDes value*. Click **OK** to continue. Click **Cancel** to cancel the Restore Lock command.

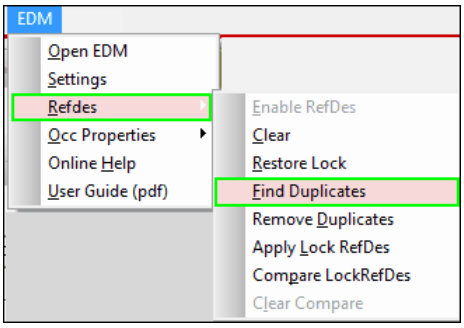


Note: Pages may need to be refreshed. To refresh the page, you will need to navigate to another page and then return to the current page.

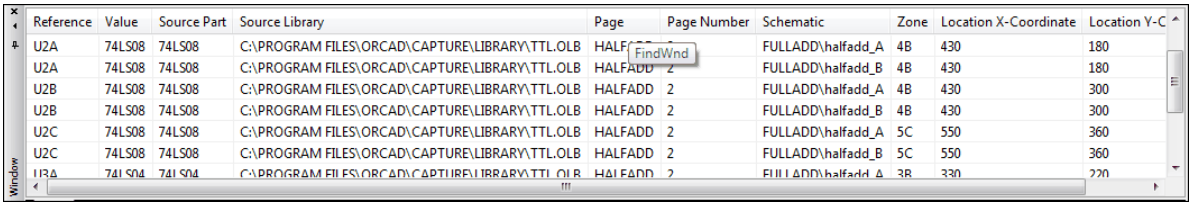
10.2.3 Find Duplicates

The **Find Duplicates** menu command identifies all parts with duplicate references. You can use this command if you need to selectively retain reference for certain parts. The list of duplicate parts will be shown in the Find Window. Double clicking on an item in the list will open the page that the part is on and the selected part will be shown in a different color. Parts with a LockRefDes when copied and pasted without opening EDM or without enabling **EDM RefDes** will have duplicate LockRefDes properties as well. A **Restore Lock** will restore duplicate Ref Des. When this happens, you will need to manually change the duplicates before you **Apply LockRefDes**. If resetting the saved LockRefDes is not critical, you may use the built in annotate feature in OrCAD Capture (**Tools** > **Annotate**) to re-annotate all RefDes.

To Find Duplicate Reference Designators, under the **EDM** menu option select **Refdes > Find Duplicates.**

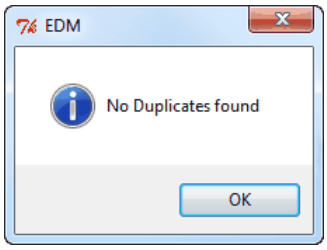


Find Duplicates



Duplicate RefDes Report

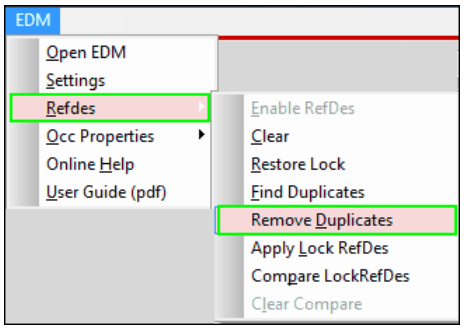
If no duplicates are found, the following pop-up will display. Click **OK** to continue.



10.2.4 Remove Duplicates

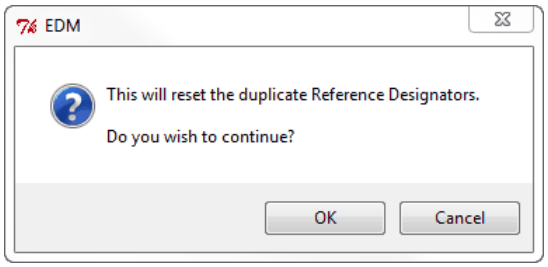
The **Remove Duplicates** menu command, will reset all RefDes parts that have duplicate values. This includes parts that have LockRefDes already applied.

To Remove Duplicates, under the **EDM** menu option select **Refdes > Remove Duplicates.**



Remove Duplicates

The **EDM** pop-up displays with the following message: *This will reset the duplicate Reference Designators. Do you wish to Continue?* Click **OK** to continue. Click **Cancel** to cancel the Remove Duplicates command.



Pop-up- Reset Duplicate RefDes

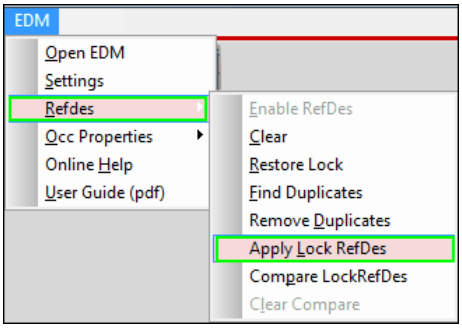
Note: Pages may need to be refreshed. To refresh the page, you will need to navigate to another page and then return to the current page.

10.2.5 Apply Lock RefDes

The **Apply Lock RefDes** menu command is only available for users whose role has been granted the Apply Lock to Team Designs permission. The **Apply LockRefDes** command will save the RefDes value to the LockRefDes property. If a prior saved value is already in the LockRefDes property, applying LockRefDes will overwrite the saved value with the new value.

You should make sure that the entire design has been checked out before you apply LockRefDes. If you do not, you may potentially apply the same Reference to the LockRefDes and the built in **EDM RefDes** commands. This may not be as useful in helping you resolve duplicates. When this occurs, you will need to find the duplicates and manually change these. You should save the design after using this command.

To Apply Lock RefDes, under the **EDM** menu option select **Refdes > Apply Lock RefDes.**

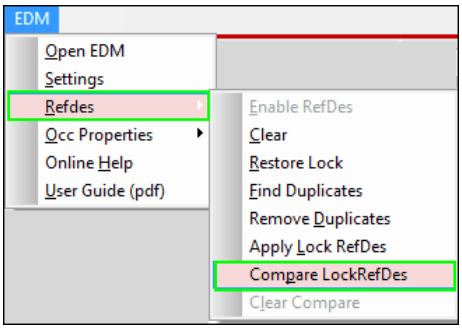


Apply Lock

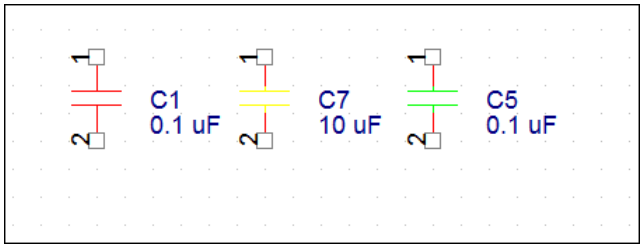
10.2.6 Compare LockRefDes

The **Compare LockRefDes** menu command option compares the currently visible RefDes to the saved value of LockRefDes. The parts are color coded based on the result from the comparison. After checking out all pages, you will want to do a compare first to see if any one has manually changed the RefDes since the LockRefDes was applied. This will give you a chance to investigate the need for a particular designator. If you check out all pages for a hierarchical design, the LockRefDes will automatically be restored. If the LockRefDes has not been applied, the instance value of the reference designator will be shown. If any of the occurrences of hierarchical blocks are yellow or red, all corresponding parts will be color coded the same.

To Compare LockRefDes, under the **EDM** menu option select **Refdes > Compare LockRefDes.**



Compare Lock



Color Coded View

Color Coding of Parts

- **Red** – RefDes does not equal saved LockRefDes on the Part. This occurs when a user changes the RefDes after an apply LockRefDes was done.
- **Yellow** – Part does not have LockRefDes property. This occurs when a user starts working on a schematic that has been checked out from EDM and Capture had been closed. The user might have started Capture back up without enabling RefDes or starting EDM.

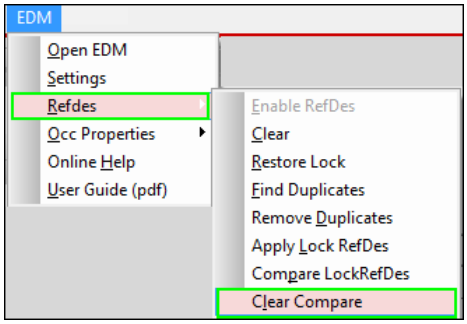
- **Green** – RefDes is the same as saved LockRefDes.

Note: Pages may need to be refreshed. To refresh the page, you will need to navigate to another page and then return to the current page.

10.2.7 Clear Compare

The **Clear Compare** menu command resets all parts to the color specified in the user's color preferences in Cadence OrCAD Capture.

To Clear Compare, under the **EDM** menu option select **Refdes > Clear Compare**.



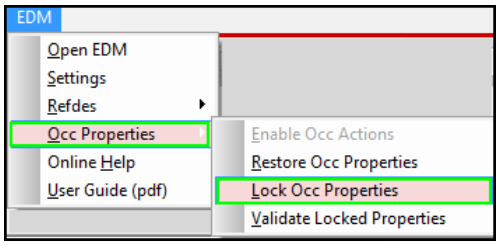
Clear Compare

Note: Pages may need to be refreshed. To refresh the page, you will need to navigate to another page and then return to the current page.

11 Lock and Restore Occurrence Properties

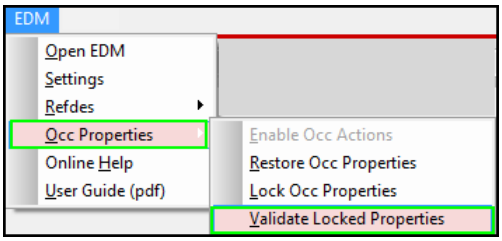
The Lock and Restore Occurrence Properties will save and restore configured occurrence properties. The saved property values relate to the locked reference designators.

Before you can save your occurrence property values, you will need to make sure that your reference designators have been locked. Refer to reference designator management for information on how to apply the lock on your reference designators. When you are ready to save your occurrence property values, select **Occ Properties > Lock Occ Properties** from the **EDM** menu.



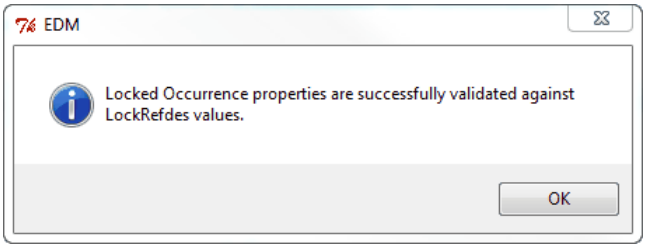
Lock Occ Properties

The occurrence property value for parts where the reference designator has not been annotated will be ignored when you lock the occurrence. If reference designators are changed after either references or occurrence properties have been locked, you will need to apply the lock to both again. Since the occurrence property values are stored using reference designators information, you will need to ensure that there are no duplicates or unannotated references. You can run **Validate Locked Properties** under the **EDM > Occ Properties** menu to check for consistency of locked properties.



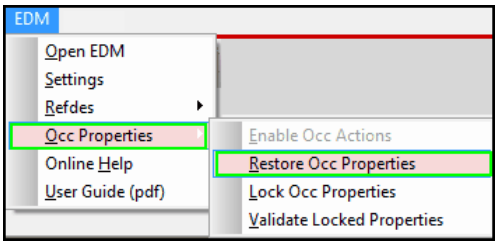
Validate Locked Properties

The following message displays: *Locked Occurrence properties are successfully validated against LocRefdes values*.



When you check out an entire design, the occurrence properties will be restored automatically. When only a portion of the design is checked out, then the restore will restore the occurrence information to the occurrences available in the check out.

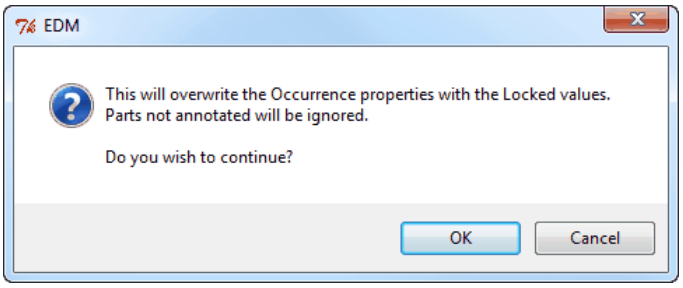
The occurrence property values can be manually restored by selecting the menu option **Occ Properties > Restore Occ Properties** from the **EDM** menu**.**



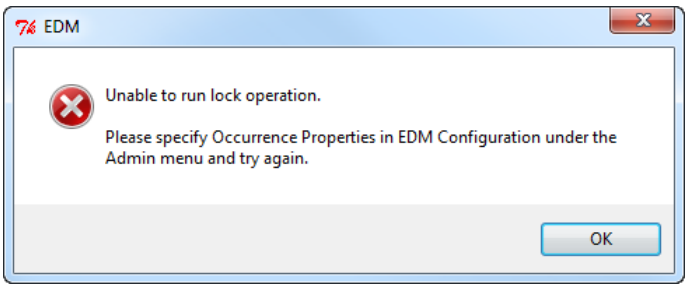
Restore Occ Properties

The following message displays: *This will overwrite the Occurrence properties with the Locked values. Parts not annotated will be ignored.*

Do you wish to continue? Click **OK** to continue. Click **Cancel** to cancel from the Restore Occ Properties command.



Note: If the property is not setup in **EDM Configuration** in the **Occurrence Properties**, you will receive the following message: *Unable to run lock operation. Please specify Occurrence Properties in EDM Configuration under the Admin menu and try again.* Click **OK** to continue.



Note: If the design is not opened, you will receive the following message: *No active design found.* Click **OK** to continue.

12 EDM Client Installation Guide

12.1 Client Installation and Setup

This Client Installation Guide provides the following instructions:

1. Installation of EDM client files.
2. Set up EDM URL within EDM Client installer or manually from EDM Settings.
3. Set Cadence Version within EDM Desktop Settings.
4. Verify EDM client installation.

Make sure your client version is the same as the EDM Web version to ensure your log in is successful. The matching version may be downloaded from EDM Web page.

12.1.1 Required Information

To proceed with your client installation, you need the EDM Client installer as well as the following information:

- EDM URL for your EDM Web application
- Location of your Windows **HOME** environment variable. Capture loads TCL files from **HOME.** Although **CDS_SITE** may be used, it may greatly increase Capture start up time.

Prerequisites

To fully obtain the full functionality of EDM, the following applications are required:

- Cadence/OrCAD Capture or Capture CIS versions 25.1, 24.1, or 23.1
- Cadence/OrCAD PCB Editor versions 25.1, 24.1, or 23.1
- Windows 11 (64-bit Professional or Enterprise Edition) (versions supported by Microsoft)
- Ghostscript PS interpreter or Acrobat Distiller *Required for PDF output

EDM uses the FlexLM license manager that is included with Cadence License Manager version **22.01 or higher**. EDM Client requires a **25.1, 24.1, or 23.1** license.

12.1.2 Installing EDM Client

This section contains instructions for installing EDM Client only. The functionality of the EDM client will only work with the corresponding version of EDM Server. Consult with your IT or project leader to determine whether the database and web application are set up. This client plugin will load into Capture version 25.1, 24.1, and 23.1.

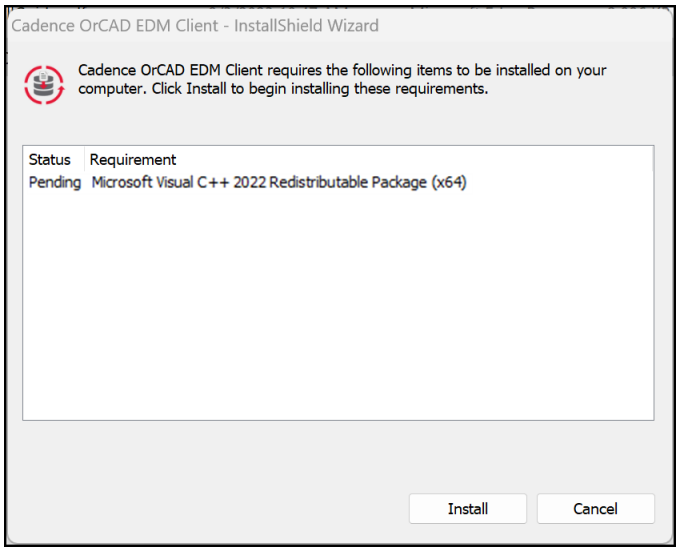
OrCAD Capture must be installed before installing EDM Client. Prior to installing EDM Client, ensure Capture is closed. This EDM Client plugin loads in Capture versions 25.1, 24.1, and 23.1.

To install a new EDM Client

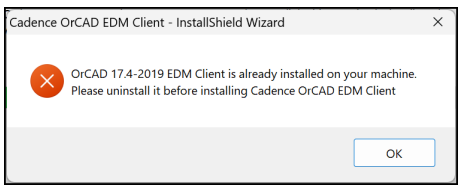
1. Launch the EDM Client installation wizard by double clicking the EDM Client installation program:

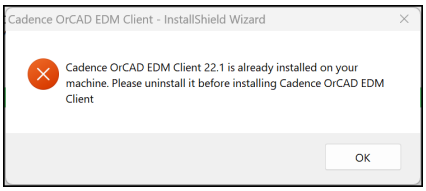
   CadenceOrCADEDMClientSetup.exe

   The EDM Client requires the Microsoft Visual C++ distributable package. You will be prompted for its installation to continue. Click **Install** to continue.

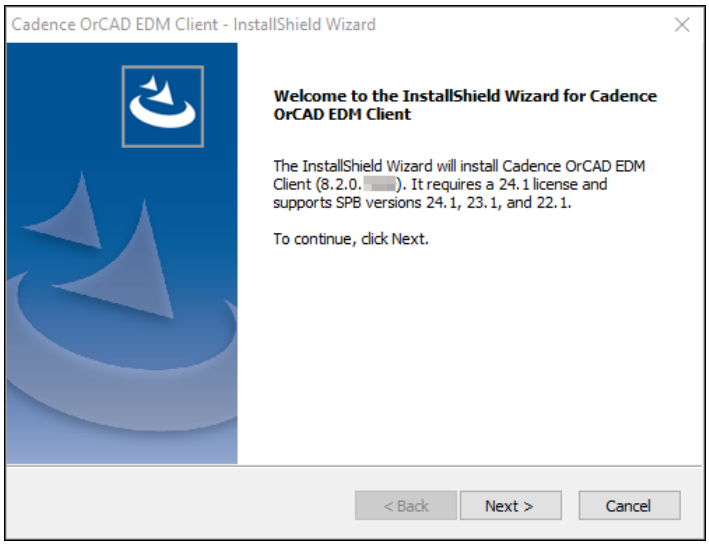


   If you have a prior version of the EDM Client installed, you may be prompted to remove it if it is not able to be upgraded (examples shown below), before continuing with the installation.





   The installation of EDM Client must be completed on each client machine set up. The product Install Wizard opens to the **Welcome** dialog box. Click **Next**.

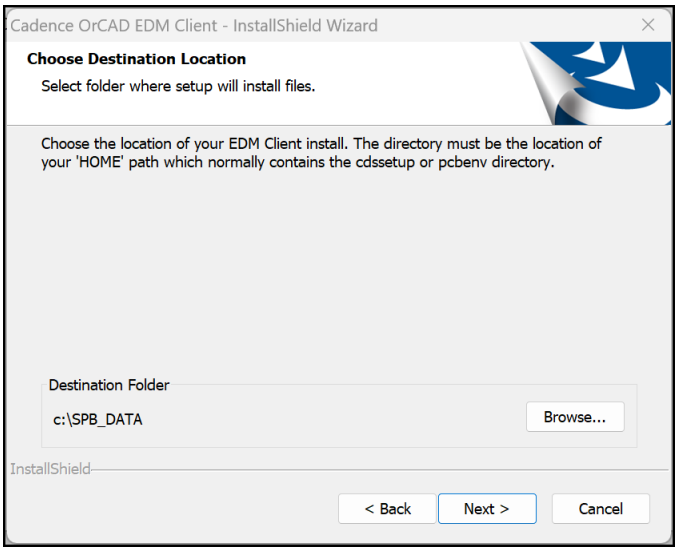


Installation Welcome Screen

2. Click on **Next** to bring up the **License Agreement** dialog. You will need to read and agree to the End User License Agreement (EULA) before proceeding. After reading the EULA, click on the **"I accept**…" button and click **Next** to continue. If you do not have OrCAD Capture installed, the installation will terminate.

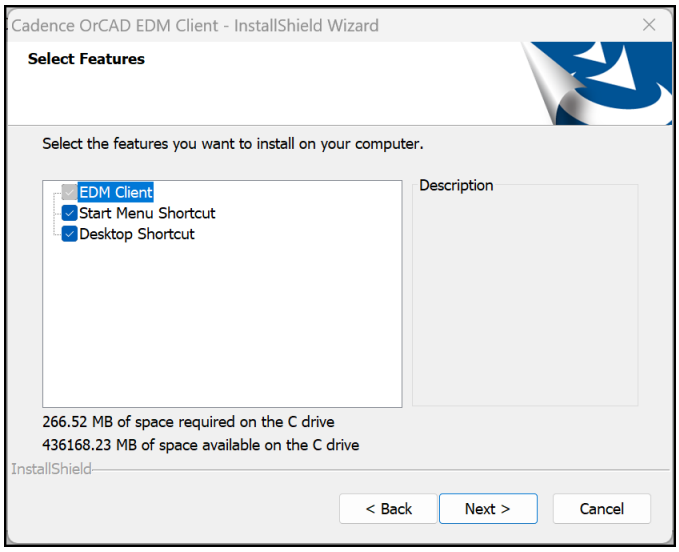
3. On the **Choose Destination Location** dialog box, you may change the location to the EDM installation. The default installation directory will be your **HOME** environment variable location. If you wish to change the installation location, click **Browse**… and navigate to the folder in which you wish to install EDM. If an **ALLEGRO_PCBENV** environment variable is defined, your PCB Editor related files (**allegro.ilinit**, **edm.il**) will be added or updated in that location instead of your **HOME** location.

   Note: The location must be a valid location where Capture loads TCL files, or PCB Editor looks for the **Allegro.ilinit** file. Select the directory and click **Next** to continue. If you do not have a **HOME** environment variable defined, you will need to add it in order for EDM to load correctly into Capture.



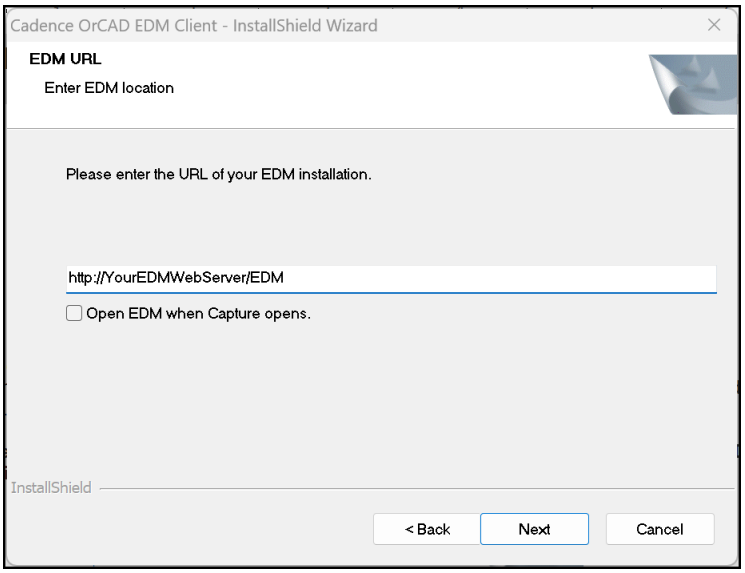
EDM Client Install Location Screen

4. The **Select Features** window will display. Options for **Start Menu Shortcut** and **Desktop Shortcut** are available. They are checked by default. Uncheck either if you do not want shortcuts added during installation. Then click **Next** to continue.



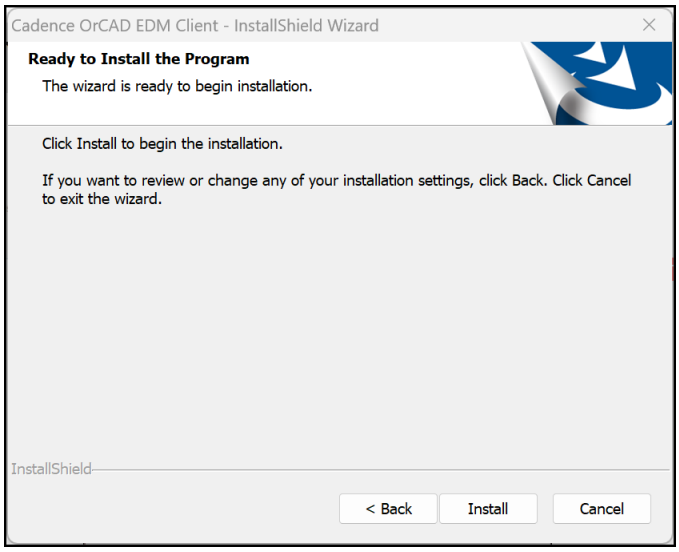
5. On the **EDM URL** dialog, you may enter the URL of your EDM web server. If you do not know the URL, you may **Skip** this step and enter the URL manually later. If you have the URL, enter the URL and click **Next**.

   Note: The URL is required to open EDM. To enter or change the URL after installation, select **EDM > Settings** from the Capture menu item.



Enter EDM Web Application URL

6. In the **Ready to Install the Program** dialog box, click **Install** to start the installation.



7. When the EDM Client installation is complete, click **Finish**.

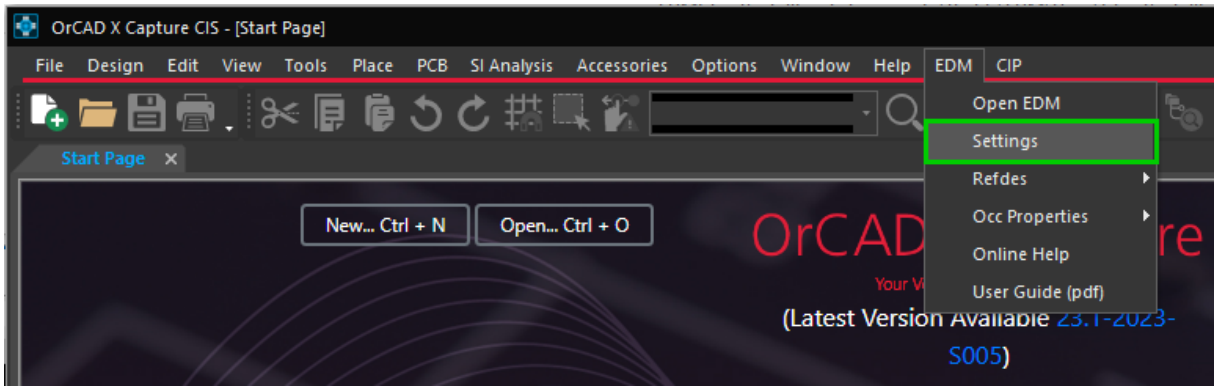
Follow the Getting Started section in the EDM user guide to start using EDM.

12.1.3 Updating EDM URL

To change the EDM URL when opened in Capture

1. Open OrCAD Capture.
2. Select **EDM** from the top menu bar and then click **Settings...**

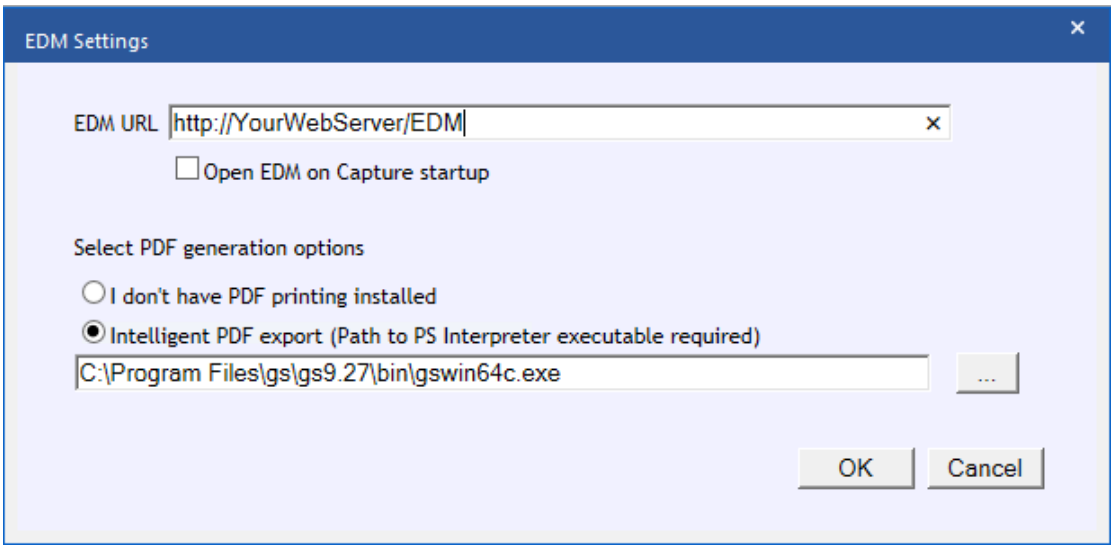
Note: If **EDM** is unavailable from the top menu bar, see the Troubleshooting Installation section.



EDM menu > Settings

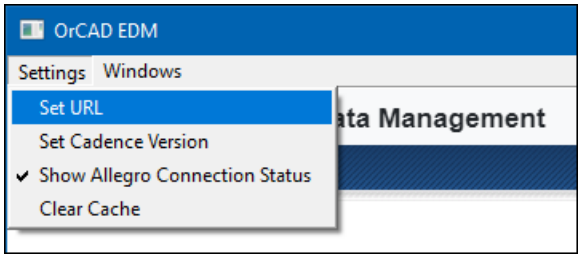
The EDM Settings pop-up opens as shown in the following figure.

3. Enter your company's EDM URL. If you want EDM to automatically open when you open Capture, select the check box to **Open EDM on Capture startup**.

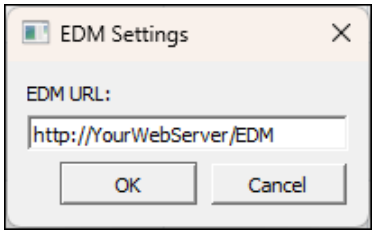


To change the EDM URL when opened in EDM Desktop

1. Open EDM Desktop application.
2. Select **Settings** from the top menu bar and then click **Set URL**.



3. Enter your company's EDM URL and click **OK** to save and open EDM.

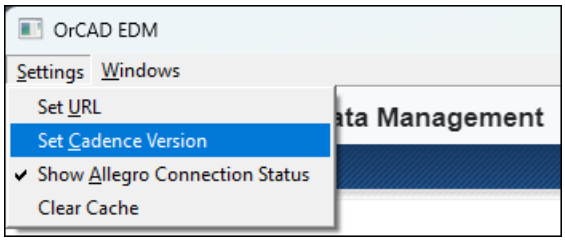


12.1.4 Updating Cadence Version

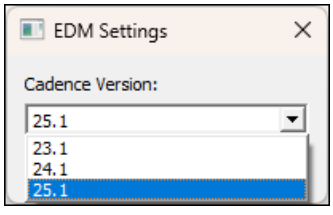
The **Set Cadence Version** setting in EDM Desktop tells the tool which version of Cadence/OrCAD PCB Editor to use for operations (Open, Import, Check In, etc.) that are performed on footprint files and managed board files. The default for new installs is the lowest supported version of SPB, unless the user has Capture open, in which case it will use that version as the default. Upgrades will retain the previous setting unless that version is no longer supported.

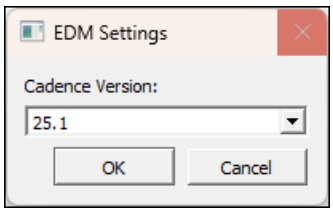
To change the Cadence Version for PCB Editor operations

1. Open EDM Desktop application.
2. Select **Settings** from the top menu bar and then click **Set Cadence Version**.



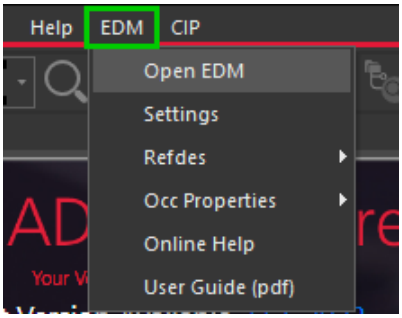
3. Select your Cadence PCB Editor version (e.g. 25.1) from the drop-down list and click **OK** to save.



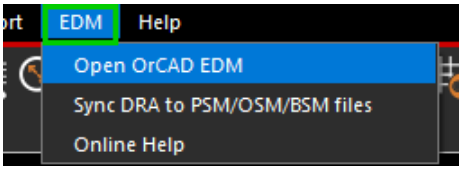


12.1.5 Verify EDM Installation

Open Capture and look for the **EDM** menu item. Review steps in troubleshooting EDM menu problems in Capture if you do not see this menu.



Open PCB Editor and look for the **EDM** menu item. Review steps in troubleshooting EDM menu problems in PCB Editor if you do not see this menu.



12.2 Troubleshooting Client Installation Problems

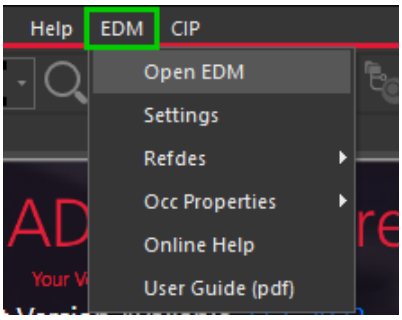
The troubleshooting instructions that follow help you debug typical installation problems that are associated with the EDM Client installer. Please contact your Cadence channel partner if you have any questions or continue to have an installation issue.

The troubleshoot section provides information on resolving the following issues:

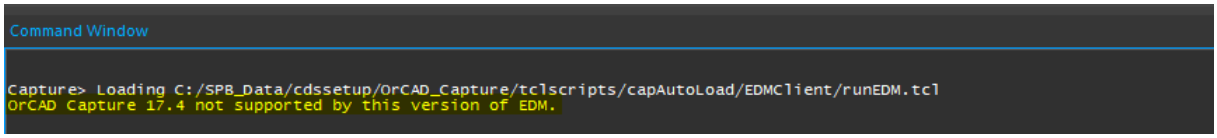
- EDM menu item is not available or correct in Capture
- EDM menu item is not available or correct in PCB Editor
- EDM fails to connect to PCB Editor
- Incompatible version message after log in
- License error after log in

12.2.1 Trouble with EDM Menu Item in Capture

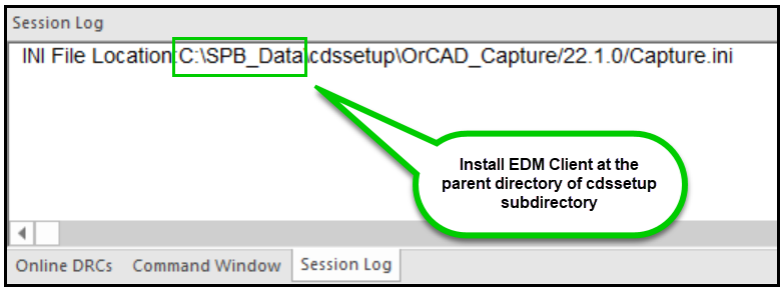
The **EDM** menu will only show for a supported version of Capture (part of the Cadence/OrCAD SPB suite). After you open Capture, the menu option to **Open EDM** will be missing from the menu bar if launched in an unsupported version.



Check to make sure that you are running one of the supported releases of Capture and that you installed to the corresponding "HOME" directory. Supported releases are specified in the prerequisites. The Command Windows inside Capture will show an error when attempting to load EDM in an unsupported version. The **EDM** menu will not be shown in such cases.

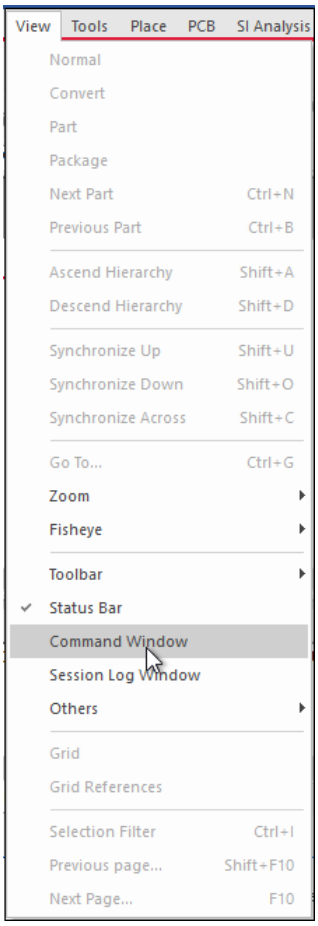


To determine the proper installation directory, open Session log after Capture opens. Review the path of the Capture.ini loaded by Capture. See example in the following figure.

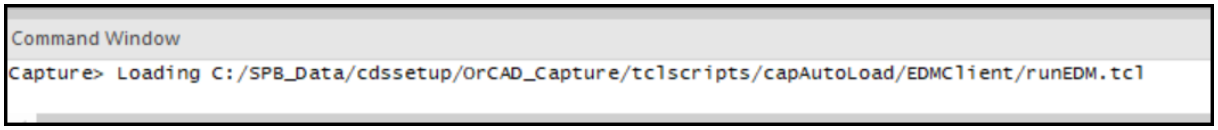


If you see EDM but the rest of the menus options are incorrect, make sure that you did not move any of the files after installation. If you did, remove all files and re-install. (Moving files can cause Capture to do partial loading of EDM Client files. Making copies of the files and putting them in various Cadence directories may also cause problems as you upgrade, since Capture will load TCL files from various locations.)

If you are unsure whether the EDM files loaded without errors, open your command window. While in Capture, select **View -> Command Window** (see figure below).



Locate where runEDM.tcl is loading. The command window should look similar to what is shown in the following figure. If you don't see the following lines in your command window, then EDM client most likely did not install in a location where Capture loads TCL files from. If there are errors shown for loading this file in the command window, you will need to follow the error message to see the source of the problem. You can try removing the installation and all files and install the files again, or contact your Cadence channel partner.



Possible causes:

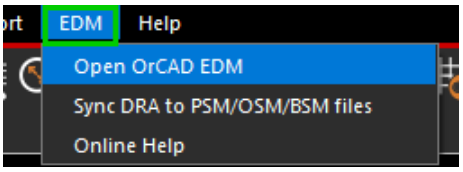
1. You may have either a **TCL_Library** or **TK_LIBRARY** environment variable defined.
2. You may have another TCL application installed that redefines the TCL library environment.

Solution:

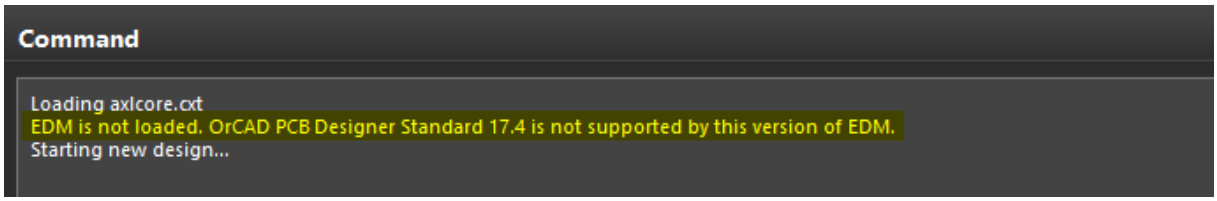
1. You will need to remove these environment variables. Further explanation can be found from the following link: https://wiki.tcl-lang.org/page/TCL_LIBRARY.
2. If you do have either environment variable defined, try removing each of the other TCL applications to see which of the applications is causing the problem.

12.2.2 Trouble with EDM Menu Item in PCB Editor

The **EDM** menu will only show for a supported version of PCB Editor (part of the Cadence/OrCAD SPB suite). After you open PCB Editor, the menu option to **Open EDM** will be missing from the menu bar if launched in an unsupported version.



Check to make sure that you are running one of the supported releases of PCB Editor and that you installed to the corresponding "HOME" directory. Supported releases are specified in the prerequisites. The Command Windows inside PCB Editor will show an error when attempting to load EDM in an unsupported version. The **EDM** menu will not be shown in such cases.



During the installation of the EDM Client Plug in, the installer updates your Allegro.ilinit file by inserting the following lines the file. Un-installation and re-installation of the EDM Client plugin multiple times may lead to addition of these lines in unexpected ways. You may manually fix the **allegro.ilinit** file or remove all lines added by the EDM installation and run the installer after removing unwanted lines.

To edit the **allegro.ilinit** file:

1. Open (Windows) File Explorer.
2. Navigate to the location of your **HOME** environment variable and open the **pcbenv** folder. You can copy the string **%HOME%\pcbenv** into your address bar, then hit Enter.



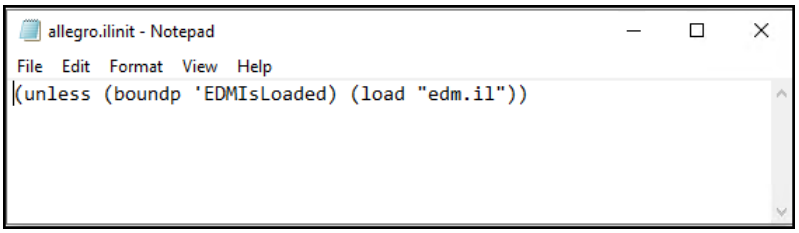
3. Open **allegro.ilinit** in a text editor and check for any axlSetVariable lines that refer to older versions of EDM. This will be the case if an older version of the EDM client has been installed at some point. Installation of multiple versions of EDM Client may also lead to incorrectly formed Allegro.ilinit. The following is a sample of an **outdated** allegro.ilinit file.

```
(axlSetVariable "EMA_EDMInstall" "C:/SPB_Data/cdssetup/OrCAD_Capture/17.2.0/tclscripts/capAutoLoad/EDMClient/EDM")
;(axlSetVariable "EMA_EDMInstall" "C:/SPB_Data/cdssetup/OrCAD_Capture/tclscripts/capAutoLoad/EDMClient/EDM")
;(axlSetVariable "EMA_EDMInstall" "C:/SPB_Data/cdssetup/OrCAD_Capture/16.6.0/tclscripts/capAutoLoad/EDMClient/EDM")
(axlSetVariable "EMA_EDMRoot" (strcat (axlGetVariable "EMA_EDMInstall") "/Desktop"))
```

Example of an Outdated allegro.ilinit File

4. Delete any axlSetVariable lines that are related to EDMInstall or EDMRoot and reinstall the EDM client. The following is a sample of a **correct** allegro.ilinit file.

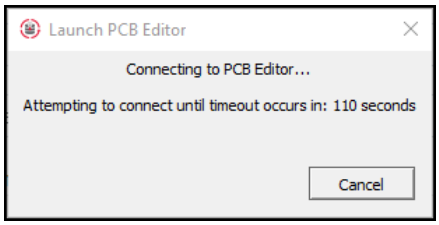
Note: You may have non-EDM related lines in your allegro.ilinit file that are supported by PCB Editor. That is okay so long as you have the EDM entry as shown.



Example of a Correct allegro.ilinit File

12.2.3 EDM Fails to Connect to PCB Editor

When EDM Desktop application is open and you select an option that requires PCB Editor, a connecting to PCB Editor pop-up window appears while EDM launches and attempts to connect to PCB Editor.



EDM will try for 2 minutes before declaring a failure.

Possible causes for connection failure:

1. The configured PCB version selected in EDM > Settings is not a version that is installed on your PC.

2. A Cadence Allegro Product Choice pop-up window is open, but hidden behind other windows and awaiting your selection of the product to open. To resolve this issue, locate the Product Choice pop-up window and select a product.

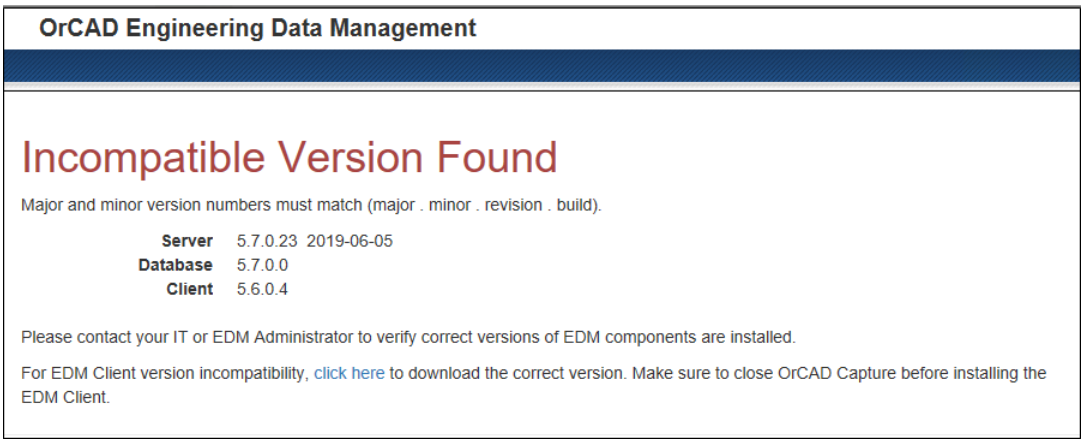
3. The Allegro.ilinit file was corrupted in some way during installation or an update of this file. For this case, you will not see the EDM menu Item in PCB Editor when opened. To resolve this issue, review information in section for Trouble with EDM menu Item in PCB Editor.

12.2.4 Incompatible Version Error After Log In

EDM Server and Client must at compatible level to function properly. There are 2 ways that incompatible versions can occur:

1. Your Administrator updated the EDM server and you have not
2. You may have downloaded and installed a later version before the EDM Server has been upgraded.

When you see an error message as shown in the following figure, click on the link to download and install the compatible version of the EDM Client.

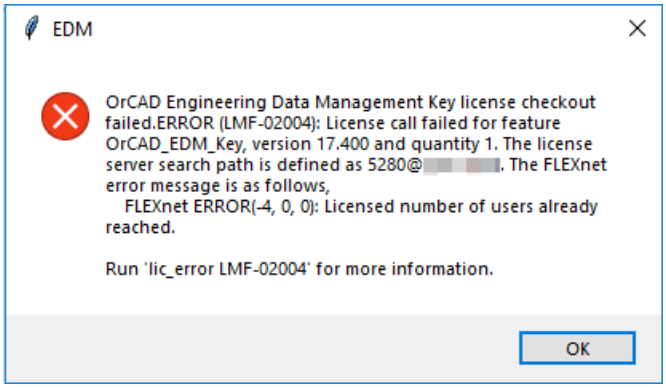


Note: You may need to uninstall first.

12.2.5 License Error After Log In

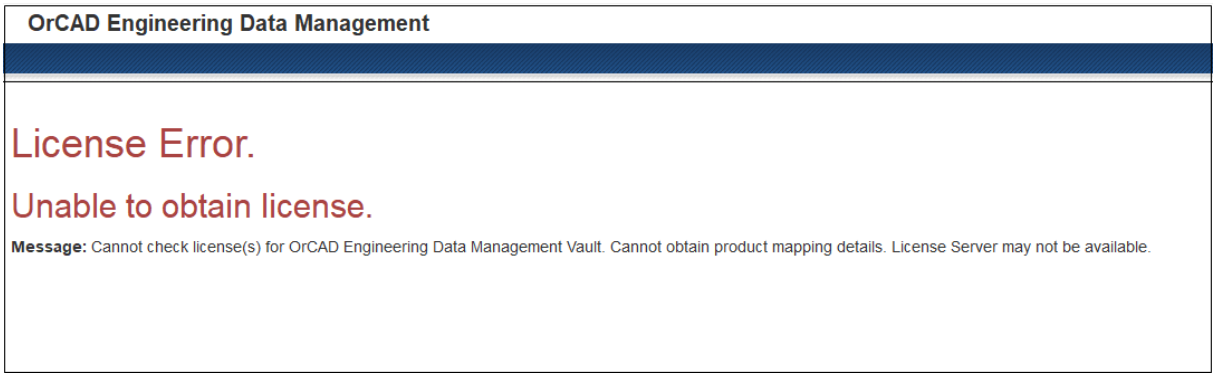
Both EDM Server and Client components require separate licenses. Errors from failing to check out a license by EDM Client is different than by EDM Server.

When the EDM Client (the Capture plug-in) fails to get a license, a pop up identifying the license issue appears when attempting to open EDM from within Capture or opening the EDM desktop.



You may need to contact your license administrator to help you resolve your problem.

When EDM server fails to get a license, the error message appears from the web page after logging in. To confirm that the EDM server has a problem checking out a license, bring up a browser, type the EDM URL and log in.



You will need to contact your IT or EDM administrator to resolve this problem.

12.2.6 EDM Shows Blank Screen

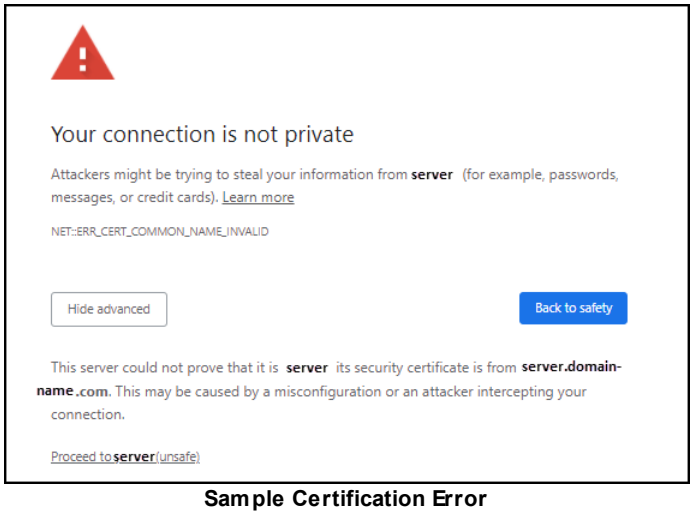
Blank EDM screen inside Capture may be caused by one of two scenarios.

1. Certificate Warning not shown in Capture - When you open EDM while in Capture, a blank screen shows whereas when you open EDM in a standard browser, you see a certificate warning where you can accept risks to continue.

2. Domain Login issue - When you log in from inside Capture using Windows Domain login, you see a blank screen. However, when you log in using a standard browser with Windows Domain login, you are able to see the EDM landing page after you enter your credentials.

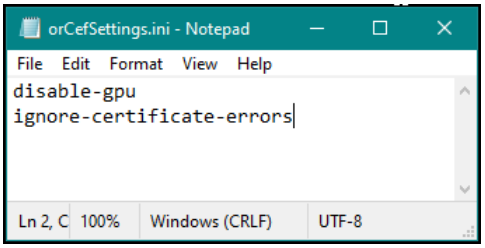
Certificate Warning Issue

If your EDM website is set up with a certificate that is not recognized as a valid certificate by a browser, the browser shows a certificate warning. A sample figure of the warning is shown below.



Sample Certification Error

Each browser shows this warning until you accept the risk and proceed to the site. To accept the risk while using the browser inside Capture, insert "ignore-certificate-errors" into the **orCefSettings.ini** located in **<Cadence_SPB_install_directory>\tools\bin** (e.g. **C:\Cadence\SPB_24.1\tools\bin**) corresponding to the version you are using, as shown in the following figure. Update this file for all client users.



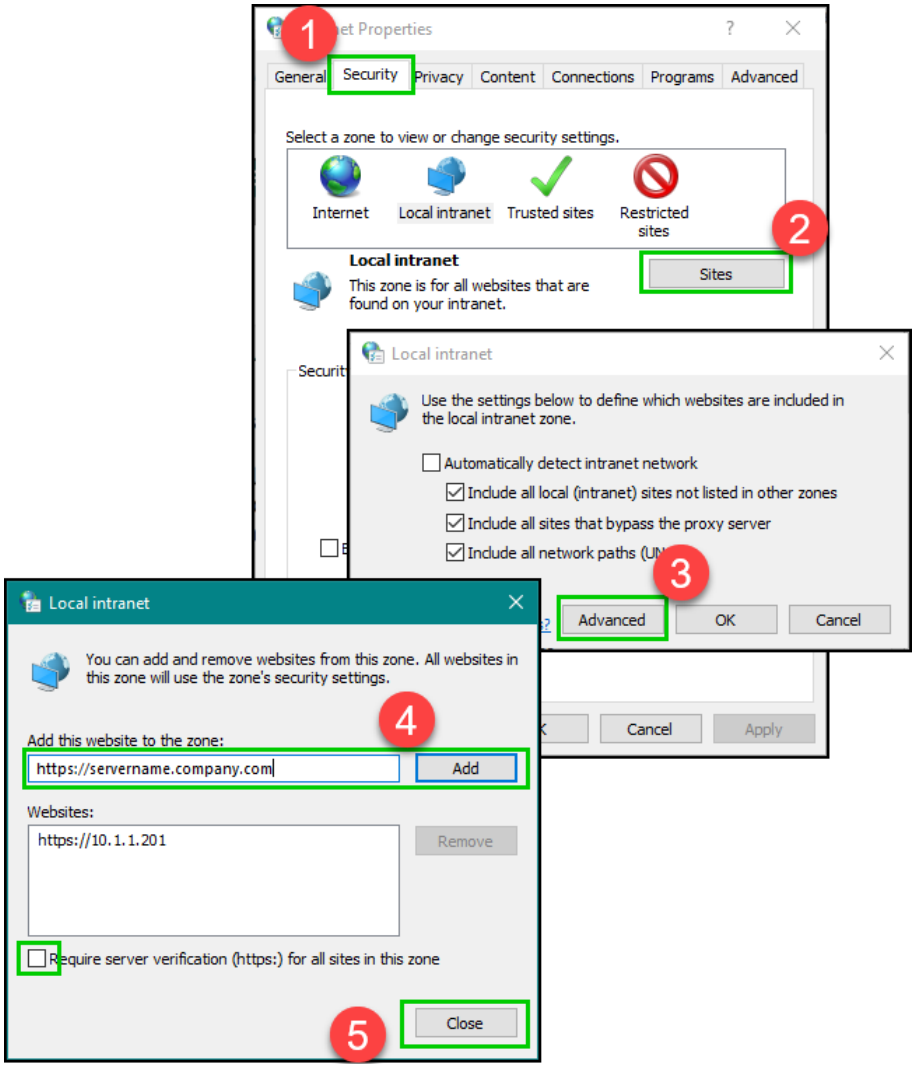
EDM Blank Screen with Windows Domain Log In

When you log in using Windows domain login from a standard browser and you are required to re-authenticate, the browser will automatically open a pop up for you to enter your login information. The need to re-authenticate is related to trust not having been established between the client browser and the domain where EDM is installed. Possible causes include:

- Your PC or VM hasn't been joined to the domain.
- You logged into your PC or VM using a local login.
- Your EDM website URL is entered using an IP address.
- Your EDM website URL is entered using a fully qualified domain name (FQDN).

Potential solutions include:

- Ensure that your PC or VM is joined to the domain and is trusted by the domain.
- Ensure that you are logged into your PC using your windows domain (not a local) login.
- If you must use an IP address or FQDN, add the domain to the list of trusted sites. To add a domain to trusted sites, select Start > Control Panel > Internet Options. Follow the steps shown in the following figure.


1
Properties
General
Security
Privacy
Content
Connections
Programs
Advanced
Select a zone to view or change security settings.
Internet
Local intranet
Trusted sites
Restricted sites
2
Local intranet
This zone is for all websites that are found on your intranet.
Sites
Local intranet
Use the settings below to define which websites are included in the local intranet zone.
Automatically detect intranet network
Include all local (intranet) sites not listed in other zones
Include all sites that bypass the proxy server
Include all network paths (UN
3
Advanced
OK
Cancel
Cancel
Apply
Local intranet
You can add and remove websites from this zone. All websites in this zone will use the zone's security settings.
4
Add this website to the zone:
https://servername.company.com
Add
Websites:
https://10.1.1.201
Remove
Require server verification (https:) for all sites in this zone
5
Close

13 Appendix

Supplemental information is provided in this appendix for terminology, licensing, file type associations, Technical Support, and hierarchical designs.

13.1 Definition of EDM Terms

This glossary of terms briefly defines the items and concepts associated to the EDM menus, dialog boxes, and tables.

Action Buttons

Action buttons are included in the Project Dashboard and Project Details. Action buttons may be displayed or hidden. Settings that control the visibility of the buttons are available from the Gear icon  , "Set Field Order and Visibility" dialog. Action buttons include: Add  , Copy  , Edit  , Delete  , History  , Cancel Checkout  and Open  .

Additional buttons are located along the upper right corner of the EDM screen. The display of these buttons vary with the assigned EDM user role and include: Refresh  , Baseline  Import  , Full Project Download  , Check Out  , and Check In  .

Activate

Activate is an action applied to a project that enables check out and check in of files, symbols and pages. This task can be performed by a user that is granted permission to *activate and baseline* a project.

Active project

Leaders who are assigned to a project *activate* the project before users are able to check out files, begin working on their assignment, and check the updated files back into the vault. Active is one of the possible project states.

Admin

The *Admin* is one of four default EDM roles (Admin, Project Leader, Project Leader will All Copy, General User). Examples of tasks the *Admin* has permission to perform include: template management, user management, EDM configuration, and vault setup.

Alias

EDM accepts alias names for setup of the EDM vault server and automated email notifications.

Artwork

Artwork is the film, usually mylar, that contains an accurately scaled representation of each layer of a printed circuit/substrate design. In the layout editor, creating artwork is the process of generating files that are used by the manufacturer to physically put the printed circuit design onto film.

When the Artwork option is selected during the development of the EDM Package for Release, artwork output files are created and placed into the zipped output folder.

Assigned Leaders

Assigned Leaders (Assigned Project Leaders) are able to perform actions determined by the permissions assigned to their user role. Users having any of the permissions in the role permission groups identified as No Assignment Required or Requires Assignment as Leader can be assigned to a project as a leader. Assigned Leaders are able to perform the permissions granted within that project.

Assigned Users

Assigned Users are granted access to project files by their Admin or Assigned Leader. The users who are assigned to a project may view the project in the Project Dashboard and check out/check in files from the Project Details page.

The Project Dashboard and Project Details includes a field column named **Assigned Users**.

Autogenerate

When an updated managed Capture Design is checked into the vault, the user who is conducting the check in, may optionally select checkboxes to *auto-generate* files such as a BOM, PDF, Netlist or BRD. Once the files are *auto-generated* and the check in is complete, the files are placed into the folders designed during folder creation or editing.

Baseline

Baseline is an action applied to a project that prevents users from checking in or checking out files, symbols, and pages. This task can be performed by a user that is granted permission to *baseline* a project. Baselined is one of the possible project states.

BOM template

Users whose role has been granted Manage Template permission may configure a *BOM template* from the Admin > Manage Templates > BOM Template menu selection. *BOM templates* help to maintain consistency from one BOM to another. The template defines column headings, property mappings to the Capture, CIS, and title block properties. The *BOM template* may be formatted by uploading a BOM template file that is formatted as an Excel or CSV file.

BOM template file

When a BOM template is configured, a *BOM template file* may be uploaded to more easily configure the setup of the BOM template. The BOM template file may include a company logo, formatted headings and text, as well as fields that are automatically linked to a title block or Capture and CIS properties.

Build BOM Search

The *Build BOM Search* is an EDM feature that permits users to index BOMs that were generated prior to EDM version 4.2.0. Once the earlier BOMs are indexed, users can conduct a BOM search that includes results from both recent BOMs as well as the earlier BOMs.

Cancel check out

Cancel check out is an action where the user removes the check out status and thereby *cancels the check out* without checking in the file, folder, project, etc.

Capture Design Project

There are four types of EDM projects: *Capture Design*, General, Schematic Symbol, and Footprint. A *Capture Design project* supports managed designs and boards where EDM can automatically generate files such as BOM(s), PDF(s), Netlist, and various reports when project files are checked in. Selection of the auto-generated files is optional during check in and most can can be created on-the-fly via the auto-generate button.

Check in

Check in is a task performed by a user to bring files from a local workspace into the vault. Users can *check in* a folder or file only when they are assigned to the folder or file and the revised content of the item has not yet been uploaded to the vault.

Check out

Check out is a task performed by a user to bring files, symbols and pages from the vault to a local workspace for the purpose of editing. Users can only *check out* files, symbols or pages that are not currently checked out. A user can only check out and edit an item that is specifically assigned to them.

The *check out* feature is a technique that manages control of local files and vault versions.

Checked out

A file, symbol or page in the *checked out* state shows that a user is working on a file, symbol or page. The lock symbol next to the file, symbol or page shows that it is locked. This feature prevents multiple users from making changes to the same file, symbol or page and potentially undoing changes checked in by other users.

CI

Abbreviation for Check in.

Customizable field

A set of ten *customizable fields* (user-defined fields) are available for footprints, projects, team designs, and team libraries. Users whose role has been granted Configuration permission may configure a user-defined field through the Admin > Configuration > General menu. Each customizable field may be configured to display a menu pick list or text value the user types into the field.

Users can customize the display of a *customized field* as it appears in the Project Dashboard or Project Details. For example, the user defined column may be hidden or unhidden, column order may be re-arranged, and the order of items within the column may be reordered.

DB Doctor

An executable that automatically updates the database version of files to ensure design and board compatibility.

Default search

When a user saves and names search criteria, EDM enables that user to save the named search as the default search criteria. Next time the user wants to conduct a search, the default search criteria is automatically selected.

Default users

A *user* who is assigned to a project and given access to project folders is a *default user*. This contrasts to *assigned users* who are granted access to specific project files. Default users and assigned users are both field columns in the Project Dashboard and Project Details. New files added to a folder in which a user is already assigned as a default user will automatically have permission to check out and check in the newly checked in file.

Default users are assigned to a project when a folder is either created or edited. When the folder created or edited is a Team Folder, default users for the team folder will be the same as the project default users.

EDM Capture plugin

EDM Client plugin

The *EDM Capture/Client/Key plugin* allows EDM to be used inside Capture, which allows users to check out and check in files, pages, and symbols, auto-generate related Capture files, promote schematic symbols, use Team Design functionality, etc. Refer to the Introduction for more details.

EDM Key

This is one of three ways to open EDM.

EDM Desktop application

The *EDM Desktop application* opens without the need to open Capture. The EDM Desktop application enables a user to check out and check in files and footprints, auto-generate related PCB Editor files, promote footprint files, create release packages, etc. Refer to the Introduction for more details.

This is one of three ways to open EDM.

EDM from a web browser

Launching *EDM from a web browser* allows users to perform actions that do not require Capture or PCB Editor, including, but not limited to: View projects and statuses, Update user assignments, status, and properties of projects, Configure templates, (excluding functionality requiring Capture or PCB Editor), and Configure EDM settings. Refer to the Introduction for more details.

This is one of three ways to open EDM.

EDM Validation Report

A CSV-formatted report that compares footprint versions in a promoted library to those placed on a managed board (.brd).

Email notification

Users whose role has been granted Configuration permission can configure notifications that automatically send users an email message that notifies them when the status, revision, title block mapping, or use of the customized user field of a file, folder, or project changes.

Database Mail must be configured on the server before the setup of this feature becomes available.

Export report

A command users may execute when viewing a project or folder history. The report can be formatted as either an Excel or CSV file. The export report feature is accessed from the Save  button that is included in the project or folder

History  . All information included in the project, folder, or file history is included in the exported report.

Extension A group of letters occurring after the period in a file name, indicating the format of the file. When a leader creates a folder, rules may be associated to the file extension to specify the file types that may be placed in the folder.

Ext Rules When a project leader creates a folder or sub-folder, file extension rules may be applied to the folder to specify which file extensions or file types are permitted to be placed in the folder.

Fabmaster When the Fabmaster option is selected during the development of the EDM Package for Release, <filename>.cad output files are created and placed into the zipped output folder.

File Server The File Server is one of two types of servers (File Server and SVN Server) that may set up to host the EDM vault.

Flat team design Team folders may be created as a flat or hierarchical folder. When a team folder is used to manage a Capture schematic design that does not contain hierarchical blocks, the team folder must be defined as a flat team design.

Flat view When projects are viewed in the Project Dashboard the project list may be toggled between the Flat View and the Hierarchical View. The Flat view lists all projects by name without grouping the projects by work group or revision.

Folder Defaults Users assigned to a folder are default users for a folder.

Footprint project One of the three EDM project types: Capture Design, General, and Footprint.

Freeze design Assigned leaders typically *freeze a team design* when the design is complete and ready for PCB layout. Freezing the design folder keeps all design pages in a single file. This feature helps preserve the back annotation function.

Full Project Download Downloads the entire project structure, which includes all files in the current version from the vault. Files are downloaded to the local workspace.

Gear icon or Gear button The Gear button  is located in the upper right corner of the Project Dashboard and Project Details page. Users may access settings using this button to control and re-arranged the display of project columns.

General account A dedicated account that provides access to the EDM SVN. The login to the General Account provides access to project revision history and opening of files when EDM is accessed from the Web browser. Login settings for the General Account are defined through the menu selection: Admin > Vault Setup.

General user One of the four EDM default roles, (Admin, Project Leader, Project Leader with All Copy and General user). The *general user* has permission to view projects and project details for which the user is assigned. The general user can download, check out, check in, cancel check out, and perform all RefDes functionality with the exception of Apply Lock for projects for which they are assigned.

Global search Search criteria a user names and saves that is accessible to all EDM users. Users whose role has been granted Save Global Searches permission are able to save or alter search criteria as a global search.

Group *Group* identifies a container for projects to be stored.

Hierarchical design A Capture schematic design that contains hierarchical blocks.

Hierarchical team design Team folders may be created as a flat or hierarchical folder. When team folders are used to manage a Capture schematic design that contains hierarchical blocks, the team folder must be defined as a hierarchical team design.

Hierarchical view When projects are viewed in the Project Dashboard the user may toggle the list of projects between the Flat View and Hierarchical View. The Hierarchical View lists projects by work group, name, and revision in an expandable/collapsible format.

History A listing of all action performed in regard to a project, folder, or file. The history is accessible through the History  button. The History button also provides download access to previous versions of project files.

Home page The EDM *Home page* shows the user's most recently accessed projects and files that are checked out. Users may click a project name listed on the Home page to view project details. Users may configure their Home page as their landing page.

Import The *import* process retrieves files, pages, symbols or footprints from a user's local or network drive and saves them to a project or team folder. This task can be performed by a user that has permission to import a project.

Inactive project An *inactive project* excludes files from the checked in and check out process. Assigned leaders may *activate* the project and end the inactive status which then allows users to check out files assigned to them so they may begin working on their assignment. Inactive is one of the possible project states.

Intelligent PDF Export *Intelligent PDF export* automatically generates and saves a PDF copy of the design when the design or project is checked in. This feature is only available

for managed Capture Designs. The feature is controlled with the selection or deselection with the PDF checkbox.

IPC-2581

IPC2581 is a standard data exchange format for using ECAD data in the fabrication and assembly of printed circuit boards. IPC2581 is an XML based format that contains intelligent data used to describe printed boards and printed board assembly products. This XML file is included in the Package for Release zipped output folder when the IPC-2581 option is selected.

IPC 356

The IPC D 356 data format supports standard electrical TEST record structure as well as the IPC-D-356A format that additionally supports buried and blink via extended records.

When the IPC 356 option is selected during the development of the EDM Package for Release, an .ipc output file is created and placed into the zipped output folder.

Local State

The *Local State* indicates the status of a file that resides in a user's local workspace. The value assigned to the Local State automatically updates when the user modifies or checks in a file, page or symbol. Team libraries and team designs provides a subset of the possible local states. Detailed information can be found in Local state Definitions.

Local workspace

A location on a user's computer where the user temporarily saves their locally modified files before checking them into the vault. The *local workspace* must be defined before a user can check out a file or object. Any change to the workspace does not affect projects in the vault until the user checks in the changes. The structure of the local workspace is the same as the structure of the vault and consists of the following: Local Workspace > Project Group > Project Name > Project Revision > Project Folders/Files.

LockRefDes

The read-only symbol property (when EDM RefDes is enabled) that facilitates the management of all references across multiple pages of a design. Only users whose role has been granted Apply Lock to Team Designs are able to perform this function.

Managed board

A managed board is a BRD file that supports the auto-generation of the following files and reports: DB Doctor, EDM Validation, and PDF when the managed board is checked into the vault. EDM also can reference a managed board to generate a "Package for Release" that can be archived or forwarded to manufacturing. See Check in Managed Board Files and Package for Release.

Managed design

A managed design is a Capture design (.DSN) that includes options for auto-generating .BOM, .PDF, and Netlist. There can only be one managed design per project revision. EDM identifies a managed design by placing a blue star symbol next to the file name. Managed

team designs are only identified with the blue star after the design is frozen.

NC Drill When you are ready to manufacture your design, the layout editor can generate output files for numerically controlled (NC) drill holes. This output file that may optionally be included in the EDM Package for Release zipped output folder.

NC Route When you are ready to manufacture your design, the layout editor can generate output files for numerically controlled routing. This output file may be included in the EDM Package for Release.

ODB++ The *ODB++* is an option that may be included in the EDM Package for Release dialog if the *ODB++* Inside package is installed.

Package for Release The Package for Release functionality generates a zipped file that can be archived or forwarded to manufacturing. The zip file may include: NC Drill and Route output, artwork files, TestPrep, and other files related to the physical board design.

Permission Ability to perform tasks in the system. *Permissions* are assigned to roles. Users belonging to certain roles have permissions that are granted to that role.

Pick and Place When the Pick and Place option is selected during the development of the EDM Package for Release, a report file named, pcp_rep.rpt is included in the zipped output folder. Information included in this report identifies the position and orientation of placed components.

Profile The EDM link to *profile* settings enables users to update their local workspace, select a default login page, change their password, and identify their assigned role. Access to the user profile is provided through the profile link located in the upper right corner of the EDM screen.

Project A *project* provides a structure in which folders and files are managed and checked into the vault. Any number of projects can be added to a project group. EDM supports three different types of projects: Capture Design, General, and Footprint. Only users who are granted access (either explicitly or implicitly) to a project are able to view the project or project group.

Project Dashboard A high level view where a high level summary of all projects are listed. Users whose role has been granted View All projects are able to view all projects from the Project Dashboard. All other users will only see projects that they have been assigned.

The *dashboard* is an EDM screen that shows all projects assigned to a user. The dashboard provides a high level status of the project(s).

Project details An EDM page that lists all project folders, files, user assignments, and attributes. The project details page is opened from the Project Dashboard. Users may customize the details that display. For example, columns can be hidden or unhidden, the column order may be re-arranged, and the order of items within a column may be reordered.

Project Group When a new project is created, the project may be added to a new or existing project group. When a new project group is created, a directory folder is created in the local workspace and vault where all files and sub-folders associated to that group are archived.

Project Leader One of the four default EDM user roles (Admin, Project Leader, Project Leader with All Copy, General User). The Project Leader role is granted all leader permissions to create or edit project folders, and assign users and leaders access to folders and files. The default Project leader role may also manage templates and EDM configuration.

Project status The status of a project. Examples of user-defined *project status* may include: New, Work In Progress, In Review, Inactive, and Baseline. The variables for a project status are created by the users whose role has been granted Configuration permission.

Project structure The project structure is the directory structure of a project.

Project Types There are four EDM project types: **Capture Design** (managed designs that include options for auto generation of files such as BOM, PDF, and Netlist files), **General** (all file types are processed the same way; no auto-generation functionality is provided), **Schematic Symbol** (specifically dedicated for schematic symbols), and **Footprint** (specifically dedicated for footprints).

Promote library A promoted library or promoted library location is a shared directory folder location where assigned leaders can promote symbols, and footprints to a shared network location where others are able to access symbols, or footprints.

RefDes commands Menu commands that manage reference designators include: Enable RefDes, Clear, Restore Lock, Find Duplicates, Remove Duplicates, Apply Lock RefDes, Compare LockRefDes, and Clear Compare. Each of these menu commands are described in EDM RefDes Commands

Refresh The Refresh  button reloads the view of the Project Dashboard, Project Details, or Team Design/Library folder.

Repository Server Type The **Repository Server Type** selection is used for configuring the type of Server that will store your Vault(s), which will be used by EDM as you create and manage projects and their contents, including items that are checked in and out. Only one type of repository can be selected: **SVN** or **FileSystem**. Once a vault is created and in use, you are not able to change the repository server type.

Role See User Role.

Schematic Symbol Project There are four types of EDM projects: Capture Design, General, *Schematic Symbol*, and Footprint. A *Schematic Symbol project* enables management of schematic symbols. It only consists of Team Library folders, each of which can contain any number of schematic symbols. No additional configurations (Folders, Files, Auto-generation, etc.) are provided with this project type.

State There are three states: Inactive, Active, and Baselined. The "Active" state of a project enables assigned users to check in and check out project files. The "Inactive" and "Baseline" states prevent users from checking in or checking out project files.

Status Options that identify a project status are configured by the Admin through the Admin > Configuration > General menu selection. Depending upon the way the Admin configures the Status property, users may edit or update the status of their project by selecting an option from a pick list or by entering descriptive text when they check in their project files.

SVN Server The SVN (subversion) server is a software versioning and revision control system that tracks changes made to files. When the *SVN server* is used as your Vault, the SVN server is installed on a network server.

Team Design Management of designs that requires multiple engineers to work in a team environment. Team design functionality enables teams of engineers to work on a single design file. This permits concurrent development on a single design.

Team Folder Details Similar to project details, the Team Folder Details is an EDM page that lists schematics, pages, symbols, user assignments, and other attributes associated to a team folder. The Team Folder may be flat or hierarchical. Users may customize the details that display in the Team Folder Details. For example, columns can be hidden or unhidden, the column order may be re-arranged, and the order of items within a column may be reordered.

Team Library A Team Library may be created when a Team Library (aka Schematic Symbol) Folder is added to a project to support librarian management of symbol creation.

Testprep The testprep process scans the board database to create test points for each net, according to the established criteria and design rules. Due to the high

density of design elements on PCBs, this test is recommended to optimize testing. Testprep is a report that can be included in the EDM Package for Release.

UNC (Universal Naming Convention) A standard for identifying servers, printers and other resources in a network. A UNC path uses double slashes or backslashes to precede the name of the computer. This method is recommended in EDM when specifying file location instead of map drives since all users must have same drive letter mapping.

Unlock When a user checks out a folder or file from the vault, other users are able to access files, but are unable to edit the files that became locked. However, users whose role has been granted Cancel Project Checkouts permission, or person who checked out the file may click the Cancel Checkout  button to cancel or unlock the check out and restore other users' ability to edit.

Unversioned The version of a new file created in the local workspace before it is checked into the vault.

User (Account) Anyone logged into EDM, regardless of role assignment.

User Group A group of EDM users an Admin may create (Admin > Manage Users > User Groups) to easily assign multiple users to projects and their underlying contents. Users whose role has been granted the Manage Users permission can create user groups.

User Interface (UI) The EDM application may be opened from three different user interfaces: 1) the EDM Desktop application or launched from PCB Editor; 2) from the Capture plugin; and 3) from a standard web browser.

Although the general look and feel of EDM is similar regardless of the user interface used to open EDM, the full functionality of EDM varies with the user interfaced used to open EDM.

For example, auto-generated files that require Capture for creation (e.g. Schematic PDF, BOM, or Netlist) need to be checked in when EDM is opened from Capture. Auto-generated files that require PCB Editor for creation (e.g. DB doctor, EDM Validation, or Board PDF) or Package for Release require the EDM Desktop.

User assignments, templates, and project properties may be managed from any user interface.

User Role A User Role is a name given to a group of users who are assigned a common set of permissions that enable them to conduct work tasks. Roles can be created and managed by users whose role has been

granted the Manage Users permission. There are four pre-configured EDM roles: Admin, Project Leader, Project Leader with All Copy, and General User.

Vault — A top level location defined by an Administrator in which projects can be created for storage. Locations for vaults must be installed/configured before adding them to EDM. Users who are granted permission to a vault and the ability to add projects are able to add projects to assigned vaults. Vaults will be stored under their configured repository location, which can be createder an **SVN** or **FileSystem** repository server type.

Web Application Identity — The process identity is the name in which the EDM web application is running as. One way to determine the process identity is to launch Windows Task Manager (on the Web Server) and locate the w3wp.exe process in the Details tab. You may need to open the EDM web page from a browser to start the process.

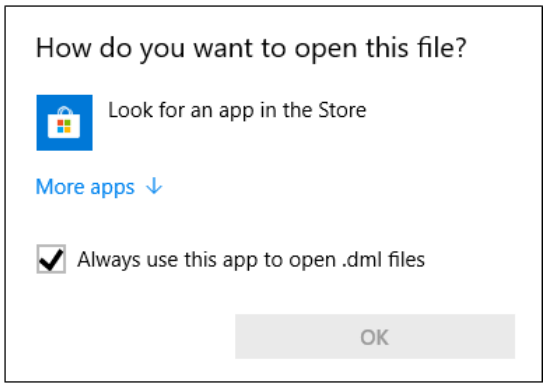
Windows Folder Naming Convention — Folder names for a Windows OS do not allow the following characters:

\ / : * ? < > | " .

Windows user — A user who accesses EDM through Windows authentication. A Windows user opens the application without use of a login name or password.

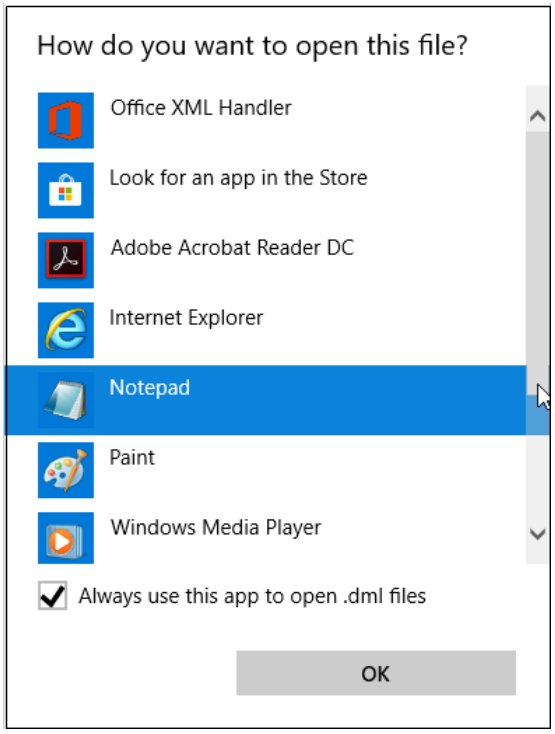
13.2 File Type Associations

When you use the **Open** button to view a file in your local workspace folder that does not have a Windows association, you will receive a Windows prompt asking you how you want to open the file.



File Type Association

To associate a specific file extension to a program click the More apps link and select the application you want to use to open the file. You may need to click the right border of the message window and use the scroll bar to find the application you want to select.



Application Selected for File Association

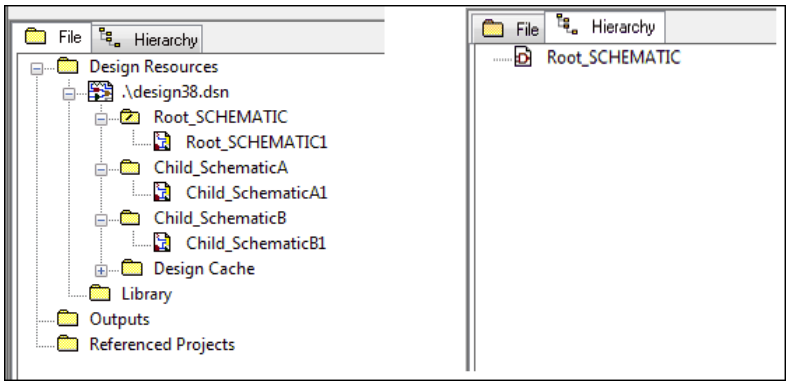
Check the checkbox, **Always use this app to open files** to continue using this application to open files with the same extension. Click the **OK** button to save your selection.

13.3 Hierarchical Designs

Hierarchical designs, unlike flat designs have information regarding relationship of schematics embedded in the design details. The relationship is added by placing hierarchical blocks onto a page of a schematic using OrCAD Capture. EDM does not add or alter hierarchical blocks. This section will guide you through different approaches for adding hierarchy to your design depending on whether you are building the hierarchy from existing design or all newly created schematics.

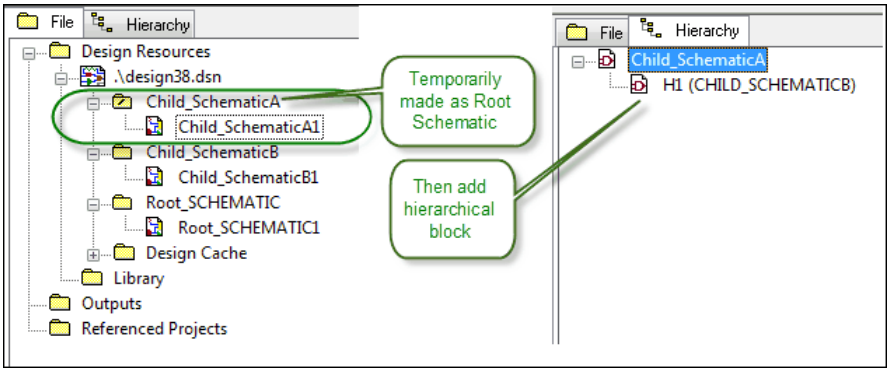
New Schematics created using EDM

When a new project is created with new root schematic and all child schematics using EDM, no hierarchical blocks are placed on the pages. Since all the pages are blank, when you check out all schematics to a design, Capture will open the design and will only show that the root schematic is root with any hierarchy. If you have a hierarchical project that has all schematics created and you need to have complex hierarchy where one child schematic will be a hierarchical block with another child schematic, you will need to add the hierarchy information in one of two ways. The following figure shows a project with the root and all child schematics checked out. Since no blocks were added, no hierarchy is shown.



No Hierarchy

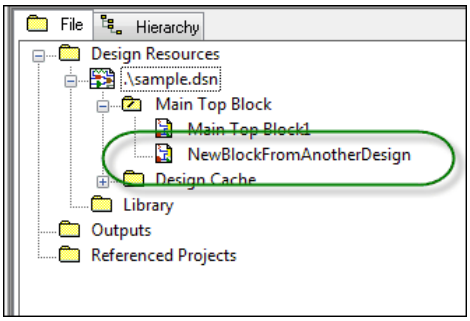
If you want to make Child_SchematicB a block within Child_SchematicA and you checked out the Root as read-only, you may have problems adding the hierarchy as is. See warnings and errors with downloaded designs for possible errors. You can either check out just two child schematics without the root or make Child_SchematicA the root temporarily. While Child_SchematicA is the root, you can add Child_SchematicB as a hierarchical block in A. After adding the hierarchical block and making any needed changes, you can check in the child schematics. The following figure shows Child_SchematicA made as root temporarily:



Child_SchematicA Made As Root

Import an existing schematic to an existing hierarchical design project

If you have a schematic from an existing design that you want to reuse, you can import just the schematic that you want to add to an existing project. You can then add a new page to the root schematic with a hierarchical block that uses the imported schematic. See the figure below showing the new page. You can import the new page and it will be added to the root schematic with the same name.



Import Existing Schematic

Note: If you are trying to import a new schematic to a hierarchical project where the new schematic is the root in the selected upload design, the root schematic for the project will change. An Admin will need to change the root schematic to the correct one.

13.4 Warnings and Errors

There are several types of warnings and errors that EDM may present to a user after taking an action. Warnings generally alert the user of a potential issue, but allow the operation to continue if desired. Errors indicate there was an issue that prevented the operation from completing successfully. Most of these warnings and errors fall under the topics below.

- Open File Warnings
- Checkout Feature Warnings
- Checkin Feature Warnings
- Unable to Create Project
- Unable to Download
- Unable to Import
- Unable to Check Out
- Unable to Complete Check In
- Unable to Promote or View Promoted Location
- Unable to Rename
- Unable to Backup and Clear Local Workspace
- Unable to Add Vault
- Browser Caching or Database not reachable issue
- Other Warnings and Errors

13.4.1 Open File Warnings

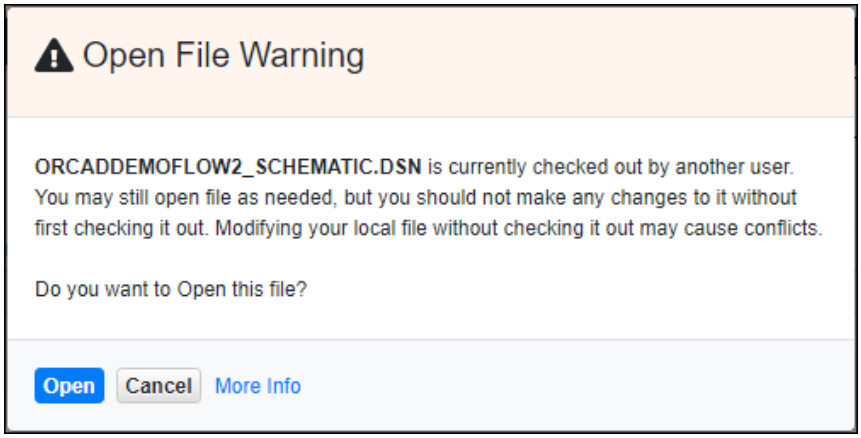
Open File Warning: File Currently Checked Out by Another User

The following warning appears when you click the Open button on a file that is already checked out by another user. Click **Open** or **Cancel**.

In the example below, the user can see the managed DSN is **Checked Out By** by another user (Admin). User clicked the Open File button for the managed DSN and was warned, then prompted to proceed with **Open** or **Cancel**. While the user could proceed with opening the file, no modifications should be made to it without first checking it out.



DSN is Checked Out By Another User (Admin)

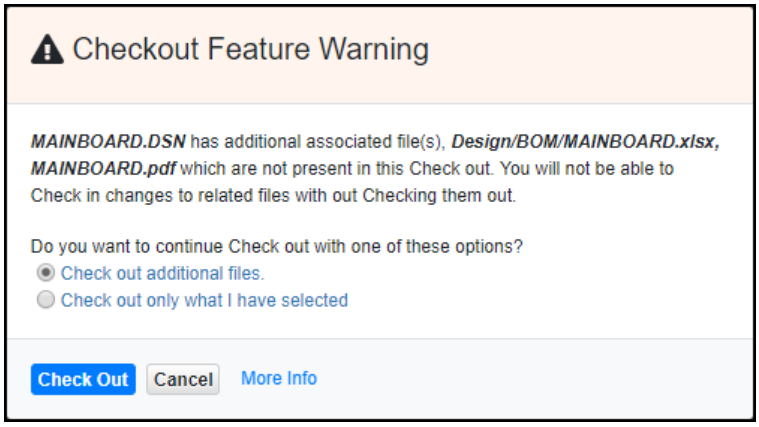


Open File Warning

13.4.2 Checkout Feature Warnings

Checkout Feature Warning: Associated DSN Files Not Present in Check Out

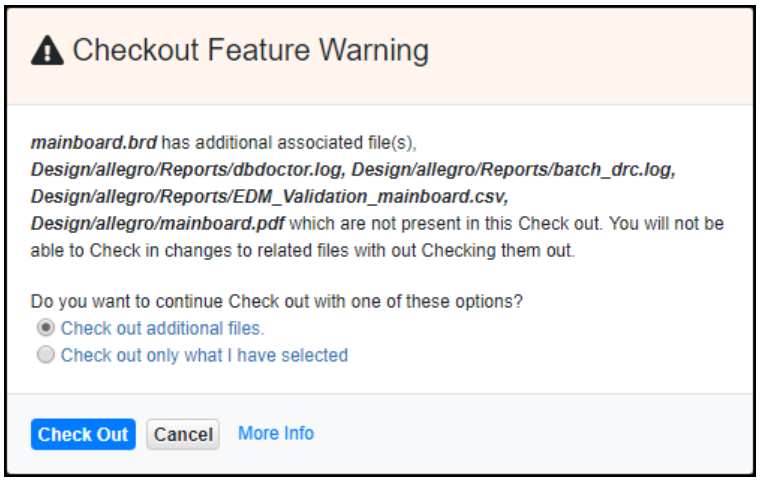
If you check out a managed design without checking out all of its associated files, the following pop-up warning will list the additional files and prompt you to choose whether to **Check out additional files** or **Check out only what I have selected**. After picking an option, click **Check Out** or **Cancel**. In the example below, the managed DSN and Netlist files were selected for check out so the prompt provides the option to include the BOM and PDF.



Associated DSN Files Not Present in Check Out

Checkout Feature Warning: Associated BRD Files Not Present in Check Out

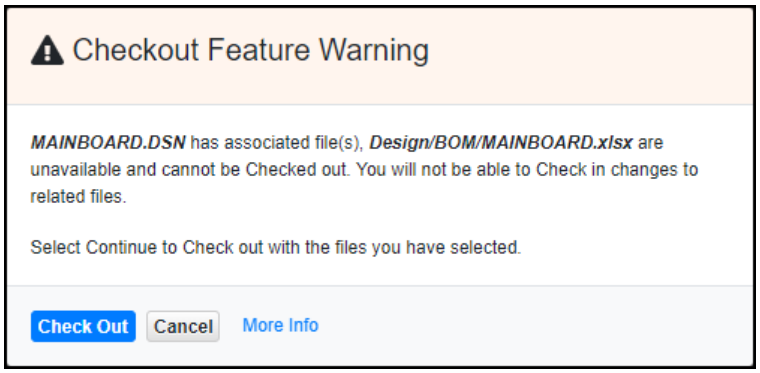
If you check out a managed board without checking out all of its associated files, the following pop-up warning will list the additional files and prompt you to choose whether to **Check out additional files** or **Check out only what I have selected**. After picking an option, click **Check Out** or **Cancel**. In the example below, the managed BRD was selected for check out so the prompt provides the option to include the DB Doctor, EDM Validation Report, and (Board) PDF.



Associated BRD Files Not Present in Check Out

Checkout Feature Warning: Associated Files are Not Available for Check Out

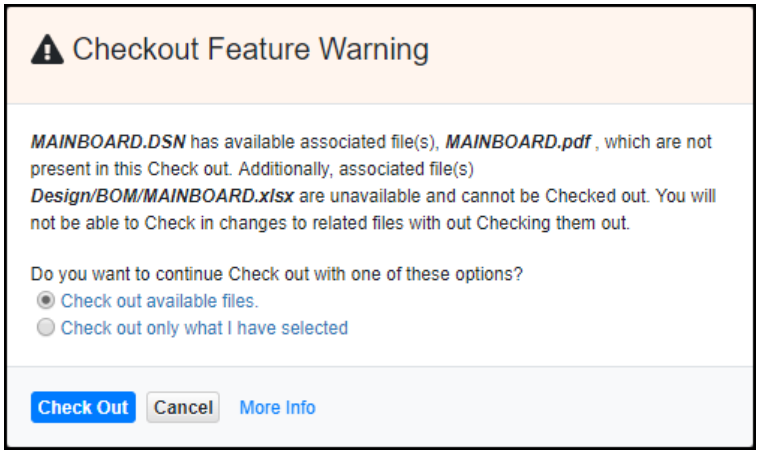
If you check out a managed design or board and one or more of the associated files are not available for Check Out, the following pop-up warning will list the unavailable files and prompt you to choose whether to continue with **Check Out** or **Cancel**. In the example below, the managed DSN, BOM, Netlist, and PDF files were selected for check out, but the prompt states the BOM is not available for check out (because another user has the file checked out or it is no longer assigned to you).



Associated Files Not Available for Check Out

Checkout Feature Warning: (1) Associated Files Not Present in Check Out, and (2) Files are Not Available for Check Out

If you check out a managed design or board (1) without checking out all of the associated files, and (2) some associated files are not available for Check Out, the following pop-up warning will list both the additional files and unavailable files. It will prompt you to choose whether to **Check out available files** or **Check out only what I have selected**. After picking an option, click **Check Out** or **Cancel**. In the example below, the managed DSN and Netlist files were selected for check out so the prompt (1) provides the option to include the PDF file, and (2) the BOM is not available for check out because another user has the file checked out or it is no longer assigned to you.

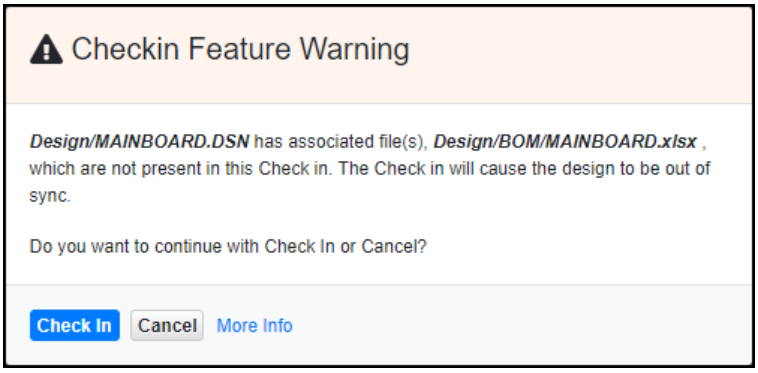


Associated DSN Files Not Present in Check Out, and Files are Not Available for Check Out

13.4.3 Checkin Feature Warnings

Checkin Feature Warning: Associated Files Not Present in Check In

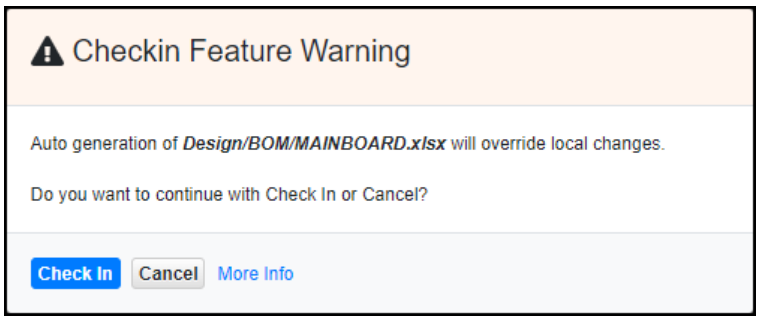
When you check out managed design files and choose not to auto generate them when checking in your existing managed design, a warning pop-up will appear, saying that continuing will cause the associated files to be out of sync with the managed design. The following pop-up warning will prompt you to continue with **Check In** or **Cancel**.



Associated Files Not Present in Check In

Checkin Feature Warning: Auto-generated File will Override Local Changes

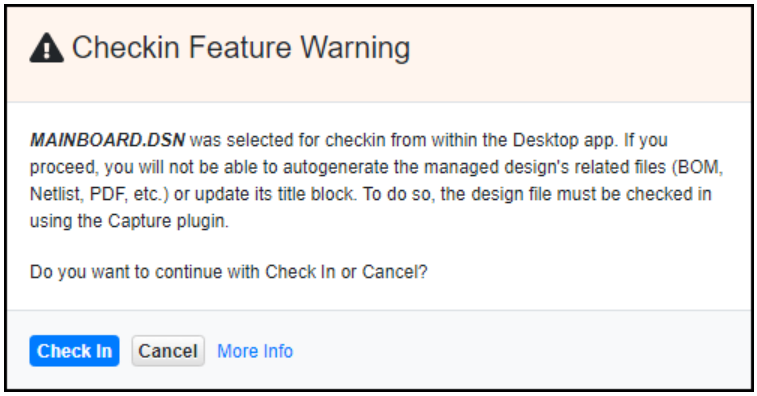
If you manually update your local copy of a managed file that you've checked out, then choose to auto-generate (e.g. BOM file) it, the following pop-up warning will prompt you to continue with **Check In** or **Cancel**.



Auto Generation During Check In will Override Local Changes

Checkin Feature Warning: Check In from EDM Desktop App Will Not Auto-generate Managed Files

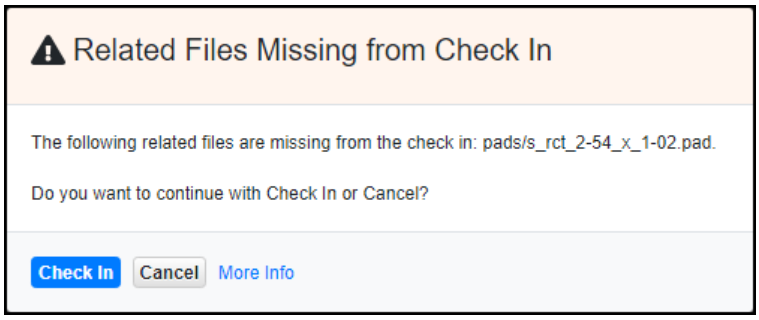
When you attempt to check in a managed design using the EDM Desktop app, a warning will appear to let you know that the design's associated managed files (e.g. BOM file) cannot be autogenerated. The following pop-up warning will prompt you to continue with **Check In** or **Cancel**.



Checking In Managed DSN from Desktop App

Checkin Feature Warning: Related Files Missing from Check-In

If you attempt to check in a footprint draw (.dra) file that references a file missing from the checkin (.pad, .step, etc.), the following pop-up warning will prompt you to continue with **Check In** or **Cancel**.

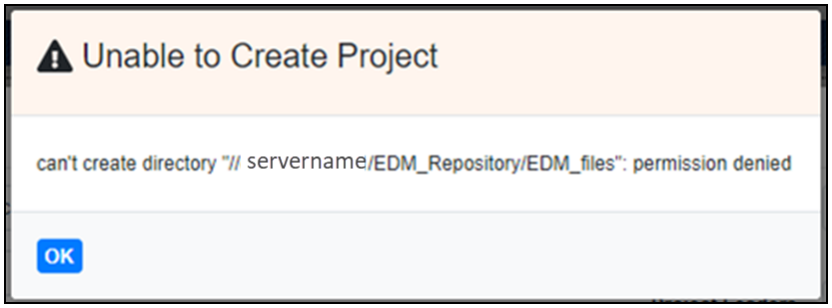


Related Footprint Files Missing from Checkin

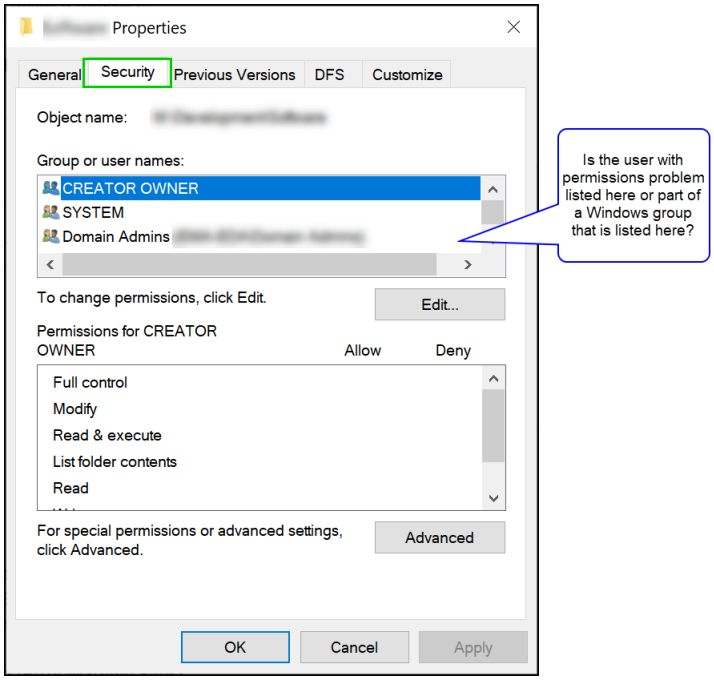
13.4.4 Unable to Create Project

Unable to Create Project: Can't create directory ... permission denied

For EDM systems set up with File System (FS) as the Vault, the vault directory location must be configured to enable **Full control** permission for all EDM users and the EDM web application's AppPool identity. If you see an error message similar to the one in the following figure, contact your EDM or IT administrator to grant you access to the directory location identified in your error message.

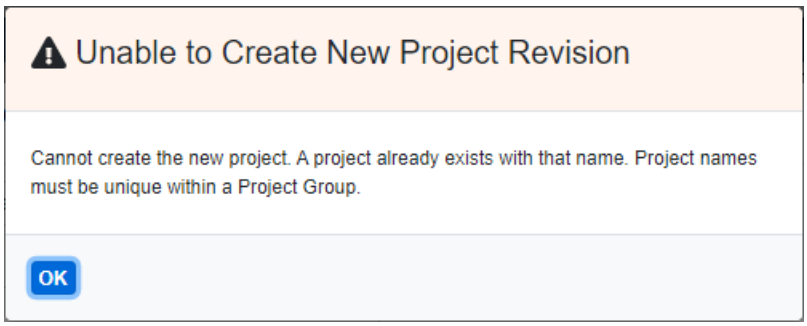


To confirm that you have the proper permissions, navigate to the directory identified in your error message and examine the folder properties. Review the list in **Group or user names** to ensure your username, or the Windows group to which you belong, are granted **Full control** permissions on your designated EDM Vault folder (FS only).



Unable to Create New Project Revision: A project already exists with that name...

This error describes how project names must be unique within a Project Group. You will see this if you (1) use the Add Project form to create a new project revision when the project name already exists in the project group, or (2) use the Copy Project to New Revision form to copy a project revision whose name already exists in the selected destination project group. If the project name already exists in a given project group, you can only add new revisions to the project by copying a project revision that already exists in the project group.



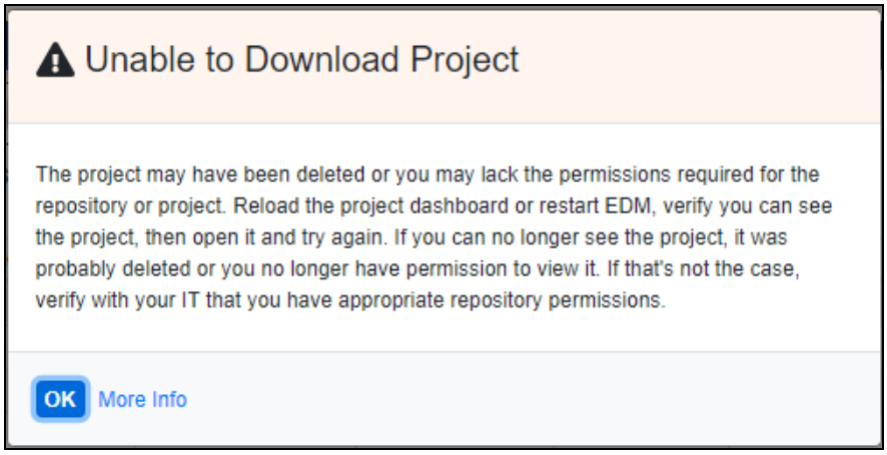
13.4.5 Unable to Download

Unable to Download Project: Project was Deleted, Project/Group was Renamed, or You Lack Required Project or Repository Permissions

The following error occurs when the you try to download a project and any of the following is true:

1. You have the project open and another user takes any of the following actions before you attempt to download the project.
   a. Deletes the project.
   b. Renames the project or its project group.
   c. Removes your role permissions to view/open the project.

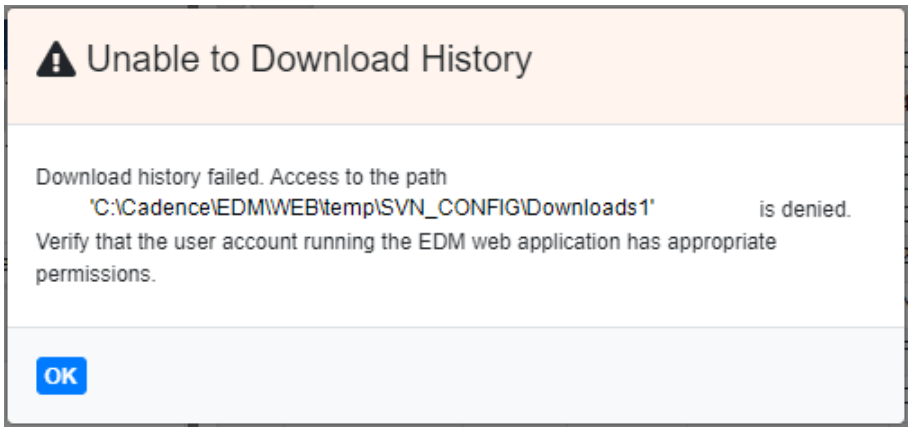
2. You lack the required permission on your vault repository. You will need to contact your IT dept. if that is the case and the process to apply your permissions will vary based on your repository server type (FS or SVN).



Unable to Download Project

Unable to Download History

The following error occurs when EDM is open in a web browser and a failure occurs while downloading project history. Ensure the user account running the EDM web application has the required **Modify** permissions on its local **temp** folder (C:\Cadence\EDM\WEB\temp) and its underlying contents. Refer to the EDM Install Guide section titled **Verify Permissions on EDM Web Folders** for more info.

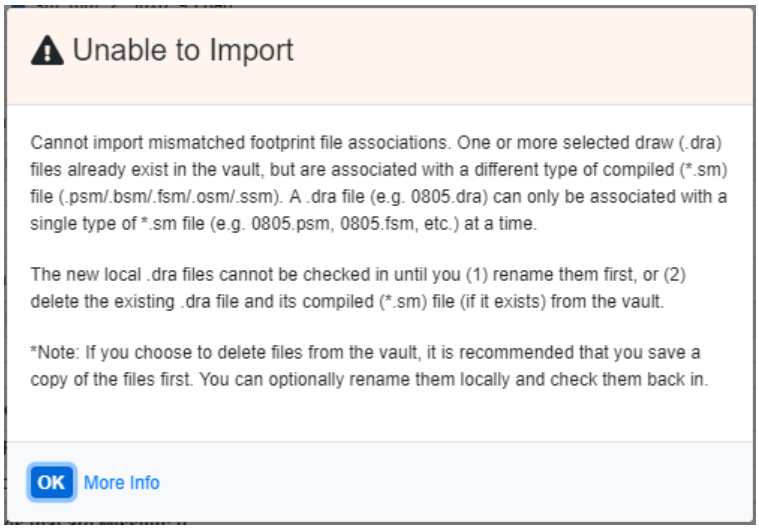


Unable to Download History from Web Browser

13.4.6 Unable to Import

Unable to Import: Attempting to Import Mismatched Footprint File Association(s)

The following error occurs when you attempt to import mismatched footprint file associations. A draw (.dra) file can only be associated with a single compiled (.*sm) file at a time. Review the options in the message to resolve the issue. Click **OK** to close the import error.



Attempting to Import Mismatched Footprint File Association(s)

13.4.7 Unable to Check Out

No Files Checked Out: Checkouts are disabled for baselined projects.

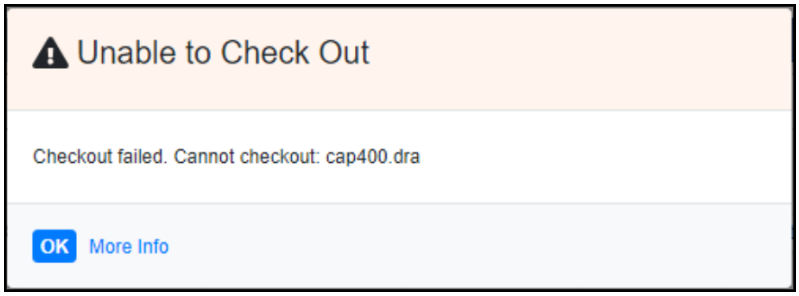
The following error occurs when a project is baselined after you started viewing a project and then you try to check out files. After a project is baselined, check in and check out are disabled. Click **OK** to dismiss this message and refresh the project detail page.



Check Outs Are Disallowed for Baselined Projects

Unable to Check Out: Checkout failed.

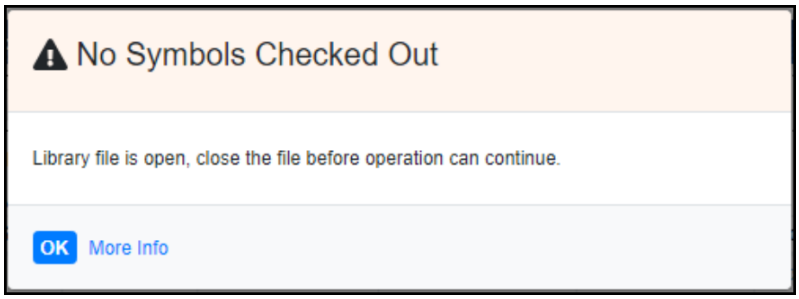
The following error occurs when a user tries to check out a file that hasn't been assigned to the user. The file that cannot be checked out is identified in the error message. Only files that have been assigned to a user or group that the user belongs can be checked out by the user. Check with your Project Leader to confirm whether you should be assigned to a footprint. Click **OK** to dismiss this message.



Checkout Failed. Cannot Checkout

No Symbols Checked Out: Library file is open, close the file before operation can continue.

The following error occurs when a user tries to check out a symbol but has kept the library file open in which the symbol belongs. Check out of a symbol downloads and merges the latest version of the symbol from the EDM Vault to the local library file. The library file must be closed first before the symbol can be successful checked out. Click **OK** to dismiss this message.



No Symbols Checked Out: Library File Is Open.

No Pages Checked Out: Design file is open, close the design before update.

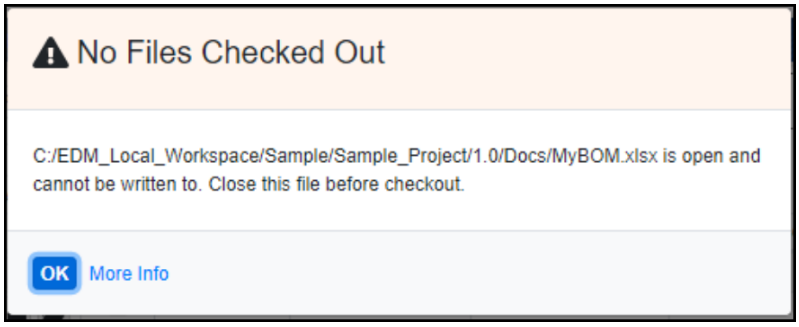
The following error occurs when a user tries to check out a page in a Team Design but has kept the design file open in which the page belongs. Check out of a page downloads and merges the latest version of the page from the EDM Vault to the local design file. The design file must be closed first before the design can be successful checked out. Click **OK** to dismiss this message.



No Pages Checked Out: Design File Is Open

No Files Checked Out: ... file is open... close this file before checkout.

The following error occurs when a user tries to check out a file from a project that has a file opened and locked by an application such as Excel or Capture. Check out of any files first downloads the entire project bringing the local project up to date with the EDM Vault. The file identified in the message must be closed before files can be successful checked out. Click **OK** to dismiss this message.

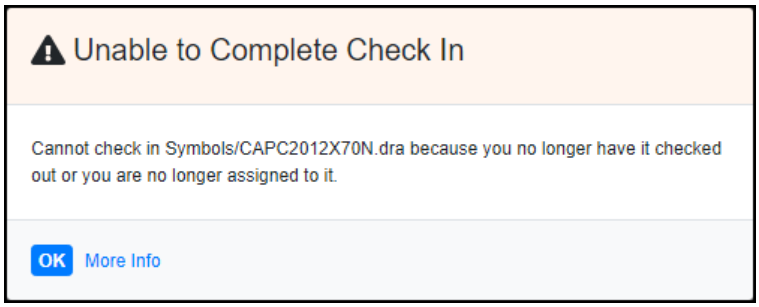


No Files Checked Out: ... File is Open

13.4.8 Unable to Complete Check In

Unable to Complete Check In: File No Longer Checked Out or Assigned to You

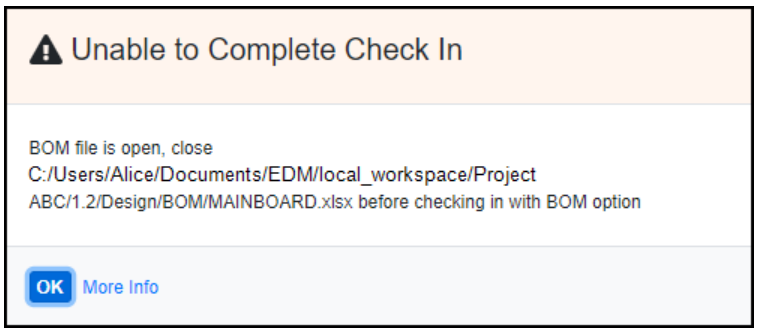
The following error occurs when the you no longer have the file checked out or it is no longer assigned to you. Click **OK** to cancel check in.



File No Longer Checked Out or Assigned to You

Unable to Complete Check In: File is Open or Locked

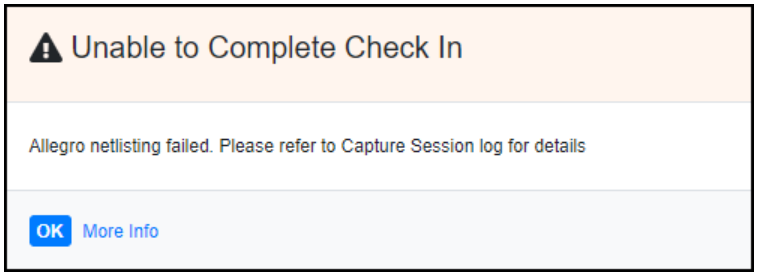
When auto-generating related files from a managed design, errors may occur if the auto-generated files are in use or the managed design file encounters errors. If a file is open during auto generation, the following *Unable to Complete Check In* error displays (e.g. BOMt is opened with Excel). To continue, close the file identified in the error message before proceeding.



File is Open or Locked

Unable to Complete Check In: Netlisting Failed

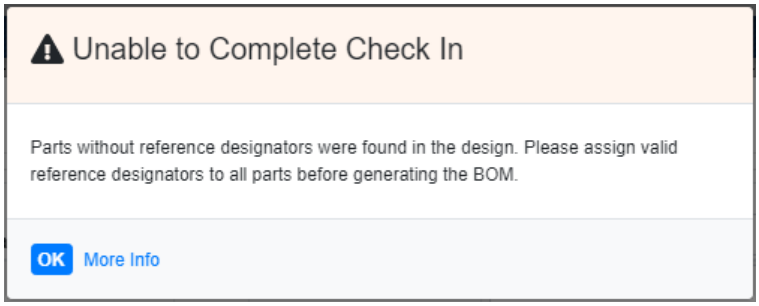
The following error occurs if there is an issue with generating the Netlist files during check in. If you receive this error message, review the Capture Session log for details and correct the error causing the Netlist problems before proceeding with the check in. You may deselect the Netlist option for check in and correct the problem later. This is not recommended however, because other users may be unaware the files are out of sync.



Netlisting Failed

Unable to Complete Check In: Parts without reference designators were found in the design.

The following error occurs when the BOM auto generation is selected and the design contains a part with a missing reference designator (RefDes). Annotate the design or manually assign a valid reference designator to all parts before generating the BOM.



Parts Without RefDes are Present on Design

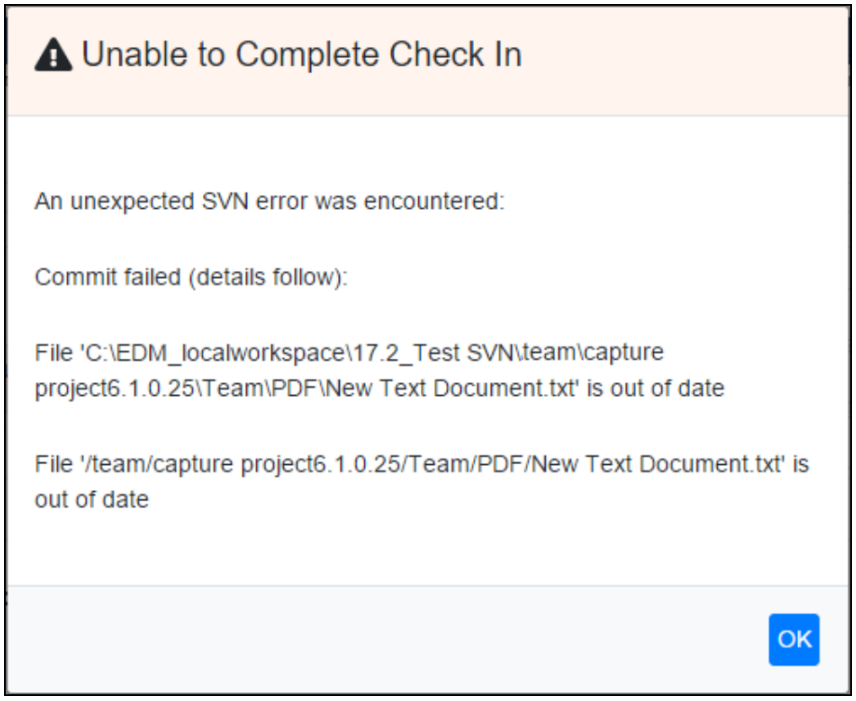
Unable to Complete Check In: File Already Exists in EDM

The following error occurs when your vault uses SVN and you are checking in a file that has different capitalization then one that is already check in. If you receive this error message, follow the steps below to investigate your problem:

1. Open the folder in your local workspace and change the name of the file with the issue.
2. Perform a full project download to review the capitalization of the file in your vault.
3. Check out the review the Capture Session log for details and correct the error causing the Netlist problems before proceeding with the check in. You may deselect the Netlist option for check in and correct the problem later. This is not recommended however, because other users may be unaware the files are out of sync.

Unable to Complete Check In: Unexpected SVN error

Issue: While performing a check in, you receive an error "Unable to Complete Check In - An unexpected SVN error was encountered: Commit failed (details follow):" as shown in the example below



Unexpected SVN Error

Cause: An unexpected SVN error may be due to various reason, some of which are included below:

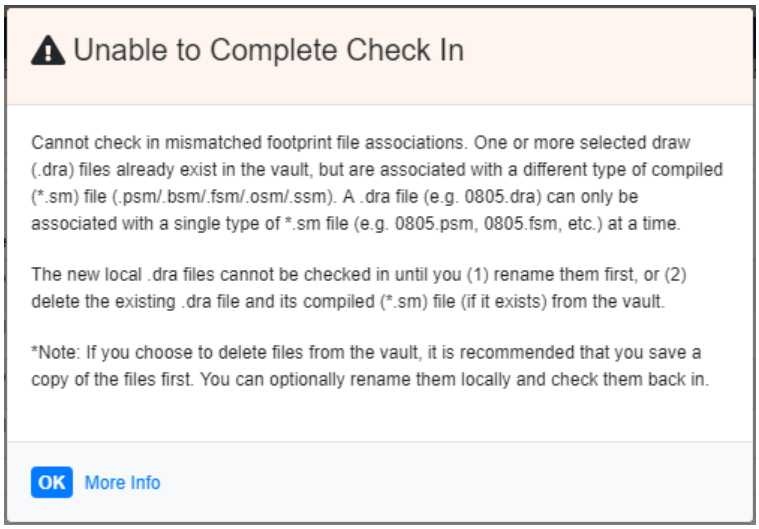
- An assigned leader changes something in EDM while a user is in the midst of an action.
- A separate SVN client is used instead of EDM.

Resolution: Perform the following steps to troubleshoot your issue.

1. Take a backup of your last change, saving the file in a temporary location that is outside of your local workspace.
2. Perform a full project download.
3. Check out the file in question if it is not already checked out.
4. Compare the file you backed up locally to the one from the project download to ensure changes by another user are not lost. Update the file as needed.
5. Copy your updated file replacing the one from the full project download.
6. Check in the file and verify the status after it finishes.

Unable to Complete Check In: Attempting to Check In Mismatched Footprint File Association(s)

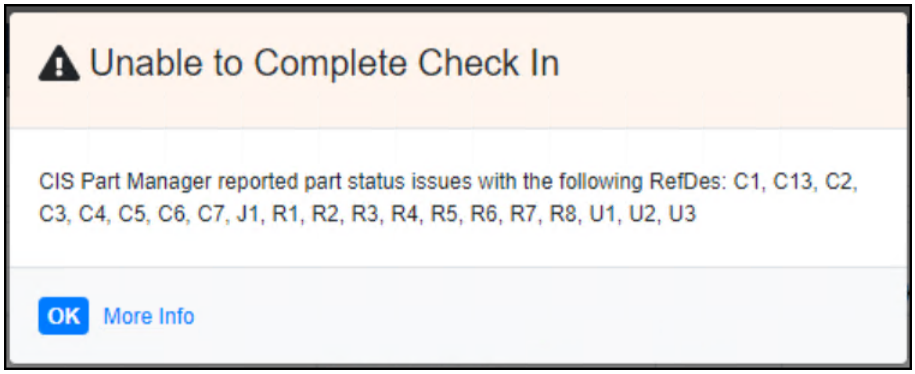
The following error occurs when you attempt to check in mismatched footprint file associations. A draw (.dra) file can only be associated with a single compiled (.*sm) file at a time. Review the options in the message to resolve the issue. Click **OK** to close the check-in error.



Attempting to Check In Mismatched Footprint File Association(s)

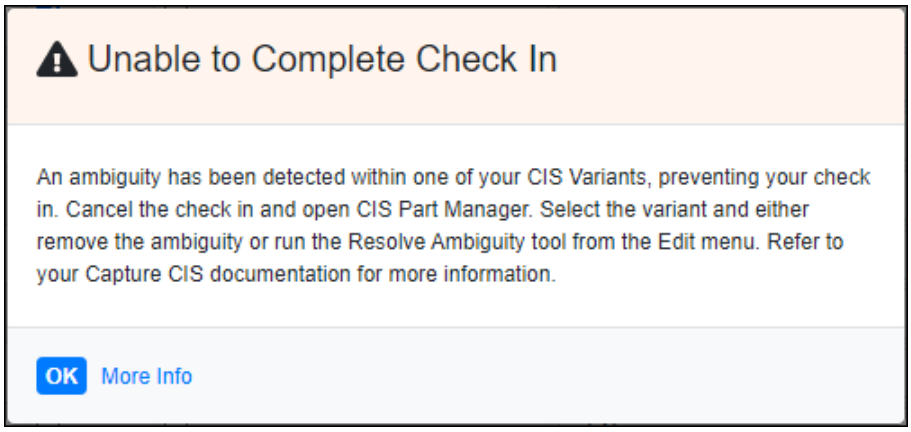
Unable to Complete Check In: CIS Part Manager reported part status issues

The following error occurs when you attempt to check in a managed design with the CIS Part Manager - **Verify All Part Status** option enabled and one or more parts in your design do not have the status of **Approved: Current.** To review part statuses in your design, select Tools > Part Manager > Update in the Capture CIS menu. To proceed with checking in the managed design, make sure that all parts in the Core and Variants have an Approved: Current status, or deselect the **Verify All Part Status** option.

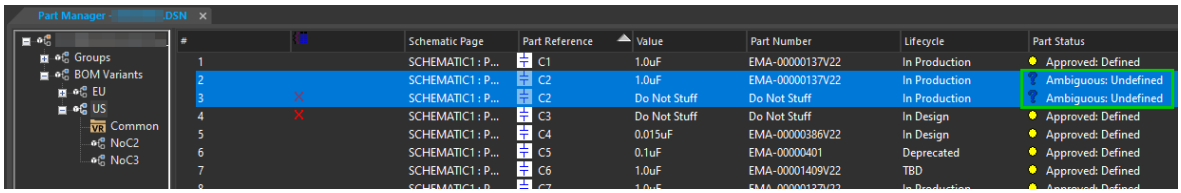


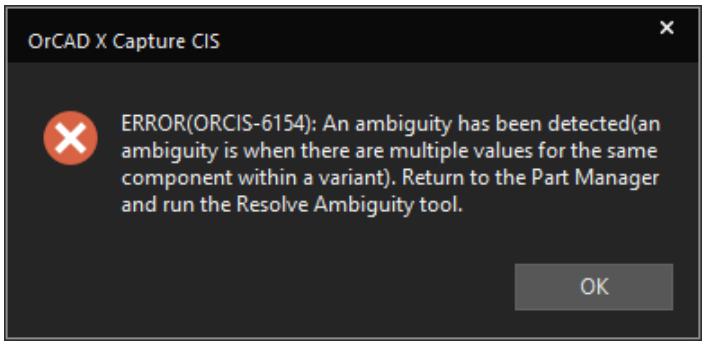
Unable to Complete Check In: Ambiguity detected in CIS Variant(s)

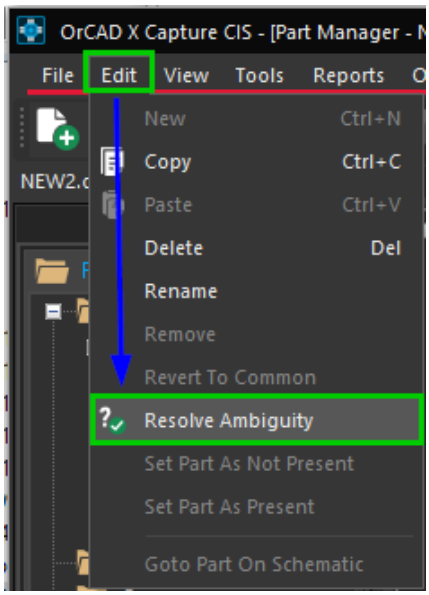
The following error occurs when you attempt to check in a managed design with CIS Variants and one or more of the variants contains an ambiguity. For instance, if the same Part Reference (e.g. C2) is both present and not present (Do Not Stuff) on the same variant, there is an ambiguity. While EDM did not cause the error, it is reporting that the ambiguity exists and you'll need to resolving using one of the two methods described below.



When one or more ambiguities exist in your CIS Variant(s), the Part Status for such Part References (RefDes) will show **Ambiguous: Undefined**. Attempting to run CIS Part Manager's update part status will display the ambiguity error shown below. You may resolve the ambiguity by (1) removing the conflicting RefDes in the CIS Variant from CIS Part Manager, or (2) manually marking it as resolved by using the Edit > Resolve Ambiguity tool. Refer to your Capture CIS documentation for more information regarding how to use the Resolve Ambiguity tool.



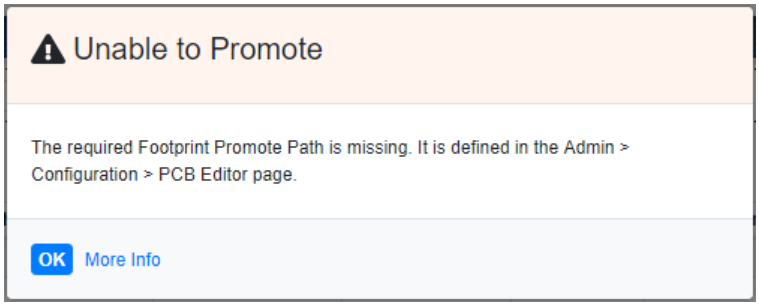




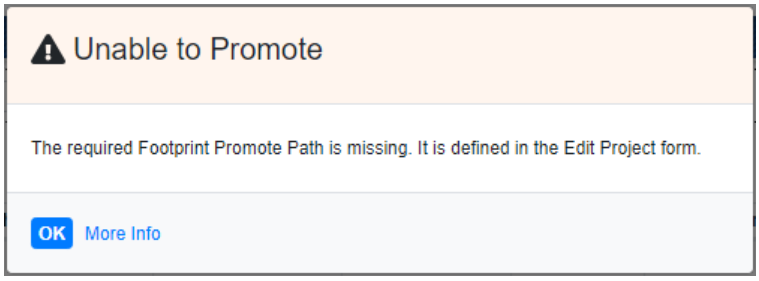
13.4.9 Unable to Promote or View Promoted Location

Unable to Promote: Footprint Promote Path is missing

The following error occurs when you attempt to promote footprints before configuring the Promoted Location. If the promote location is set to Central Footprint Location, update the promoted location from **Admin** > **Configuration** > **PCB Editor**. If the promote location is set to Each Footprint Project, update it from the Edit Project (Revision) form. Contact your company's EDM administrator to make this change if you do not have permissions to make the change.



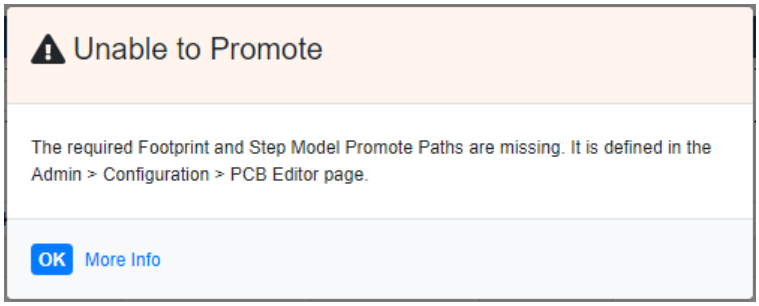
Central Footprint Location



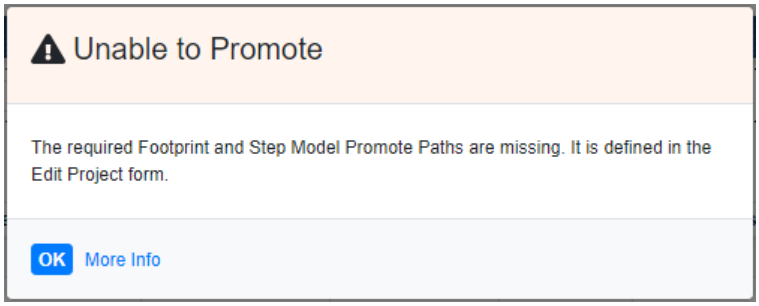
Each Footprint Project

Unable to Promote: Footprint and/or Step Model Promote Paths are missing

The following error occurs when you attempt to promote footprints or step models before configuring the Promoted Location. If the promote location is set to Central Footprint Location, update the promoted location from **Admin** > **Configuration** > **PCB Editor**. If the promote location is set to Each Footprint Project, update it from the Edit Project (Revision) form. Contact your company's EDM administrator to make this change if you do not have permissions to make the change.



Central Footprint Location



Each Footprint Project

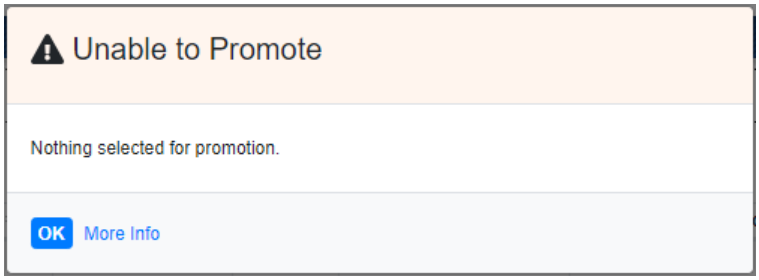
Unable to Promote: Schematic Library Promote Path is missing

The following error occurs when you attempt to promote symbols from a schematic library before configuring the Capture Symbol Promoted Location. To resolve this error, select **Admin** > **Configuration** > **Capture** and follow directions in this section to configure the promoted symbol location. Contact your company's EDM administrator to make this change if you do not have permissions to see the Admin option.



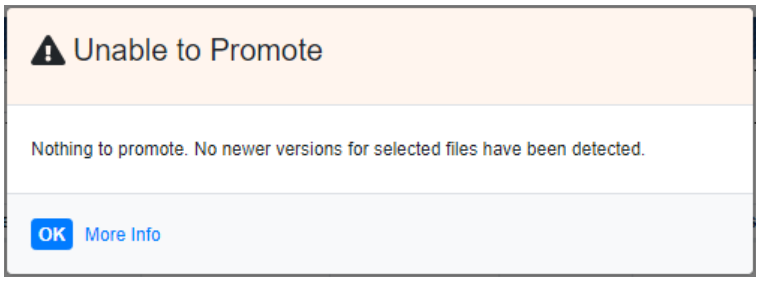
Unable to Promote: Nothing selected for promotion

The following error occurs when you click the promote button before selecting checkboxes for symbols or footprint files to promote. To resolve this, click checkboxes for symbols or footprints you wish to provide before selecting the promote button.



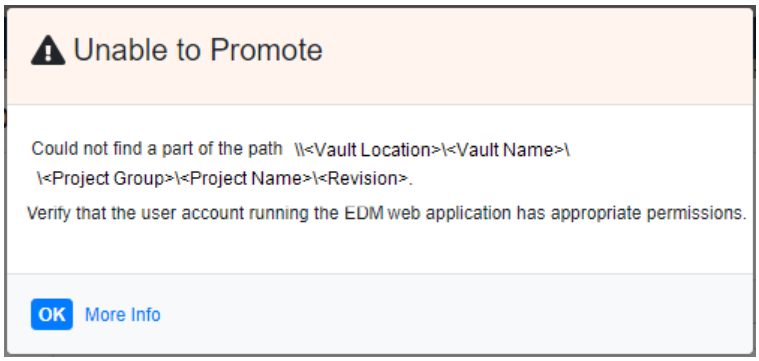
Unable to Promote: No newer version selected

The following error occurs when your EDM is configured to only promote footprints that are newer than files already promoted. EDM will notify you when you attempt to promote footprints that have not changed. To change your configuration to promote files regardless of changes made, select **Admin** > **Configuration** > **PCB Editor** to change rules for footprint Promote and Delete. Contact your company's EDM administrator to make this change if you do not have permissions to see the Admin option.



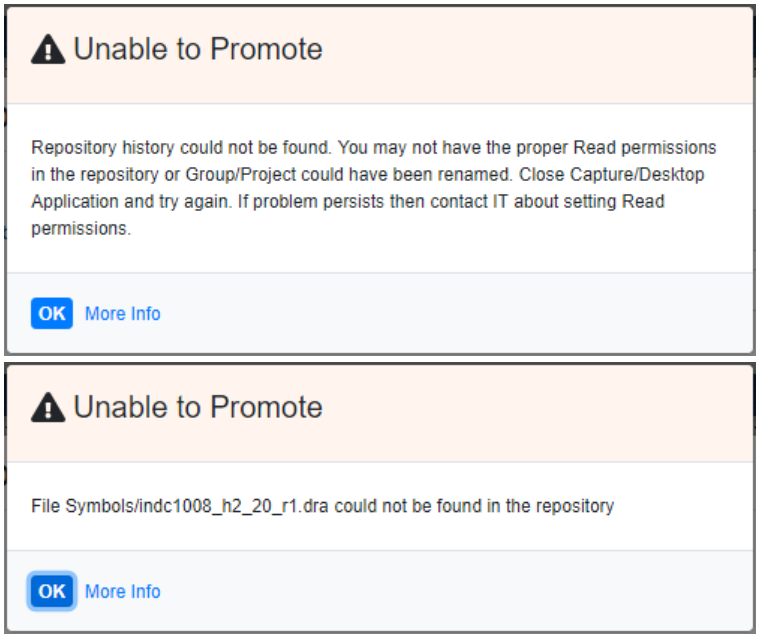
Unable to Promote: Could not find a part of the path...

The following error occurs when you attempt to promote schematic symbols or footprint files and the user account running the EDM web application can't reach the vault path or does not have appropriate permissions. Contact your company's IT or someone else that has access to update the permissions for the vault (<Vault Location>\<Vault Name> under Admin > Vault Setup).



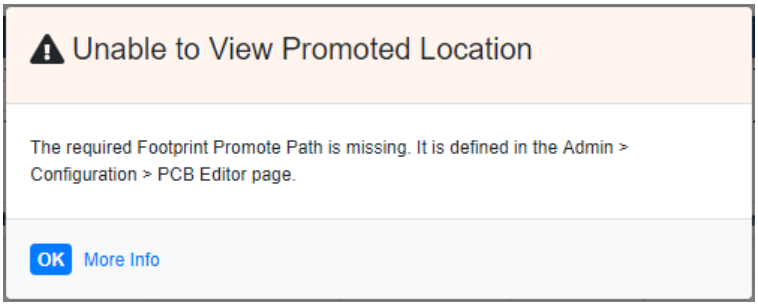
Unable to Promote: Repository History Not Found or File Not Found

This type of error occurs when you attempt to promote schematic symbols or footprints files and they could not be found in the vault repository. This is unlikely to happen, but is possible if folders or files in the vault were directory modified or removed without using EDM. No one should be directly touching the EDM Vault or any of its contents without using the EDM UI.

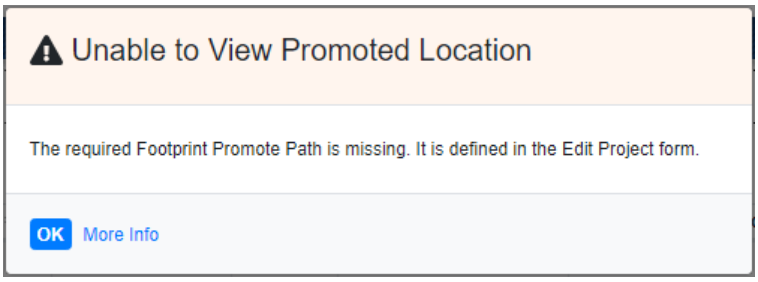


Unable to View Promoted Location: Footprint Promote Path is missing

The following error occurs when you attempt to view promoted footprint library before configuring the Promoted Location. If the promote location is set to Central Footprint Location, update the promoted location from **Admin** > **Configuration** > **PCB Editor**. If the promote location is set to Each Footprint Project, update it from the Edit Project (Revision) form. Contact your company's EDM administrator to make this change if you do not have permissions to make the change.



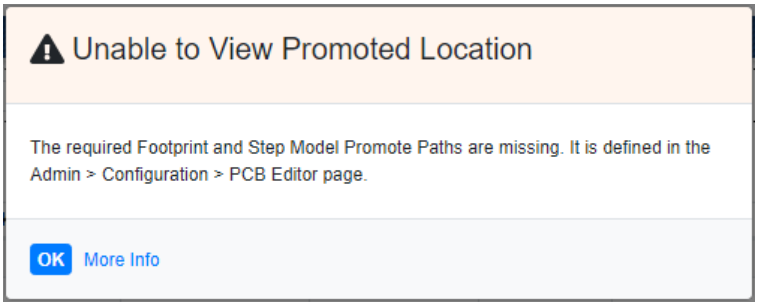
Central Footprint Location



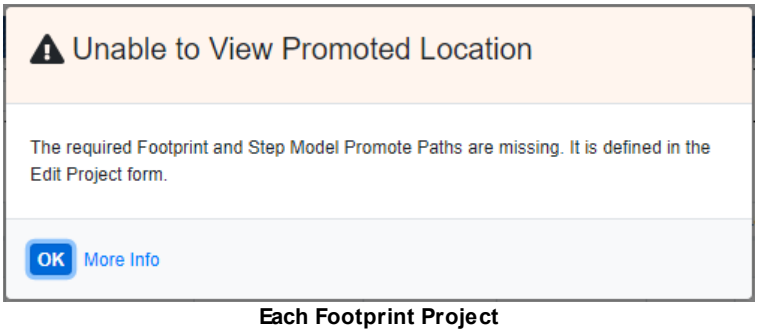
Each Footprint Project

Unable to View Promoted Location: Footprint and/or Step Model Promote Paths are missing

The following error occurs when you attempt to view promoted footprint library before configuring the Promoted Location. If the promote location is set to Central Footprint Location, update the promoted location from **Admin** > **Configuration** > **PCB Editor**. If set to Each Footprint Project, update the promoted location from the Edit Project (Revision) form. Contact your company's EDM administrator to make this change if you do not have permissions to make the change.



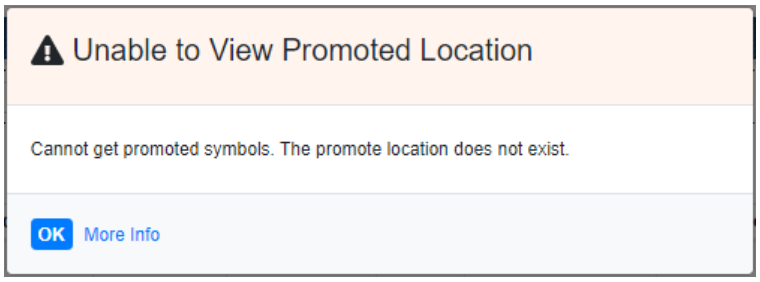
Central Footprint Location



Each Footprint Project

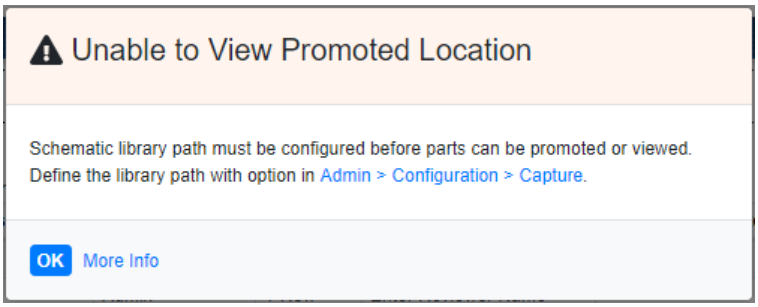
Unable to View Promoted Location:

The following error occurs when the specified promote location does not exist.



Unable to View Promoted Location: Schematic Library Promote Path is missing

The following error occurs when you attempt to view promoted symbols before configuring the Capture Symbol Promoted Location. To resolve this error, select **Admin** > **Configuration** > **Capture** and follow directions in this section to configure the promoted symbol location. Contact your company's EDM administrator to make this change if you do not have permissions to see the Admin option.



13.4.10 Unable to Rename

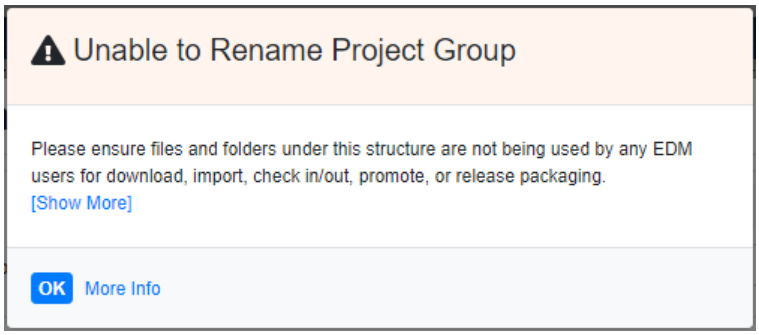
Rename errors generally occur when the structure being renamed is not found in the vault, a file is locked due to an existing action (download, import, check in/out, promote, or release), or the EDM web application does not have proper permissions on the vault.

Renaming affects everyone who has downloaded any of the project structure in question (project group, project, project revision, folder/file/page/symbol). This will cause issues for users of said projects unless they take the following steps before renaming:

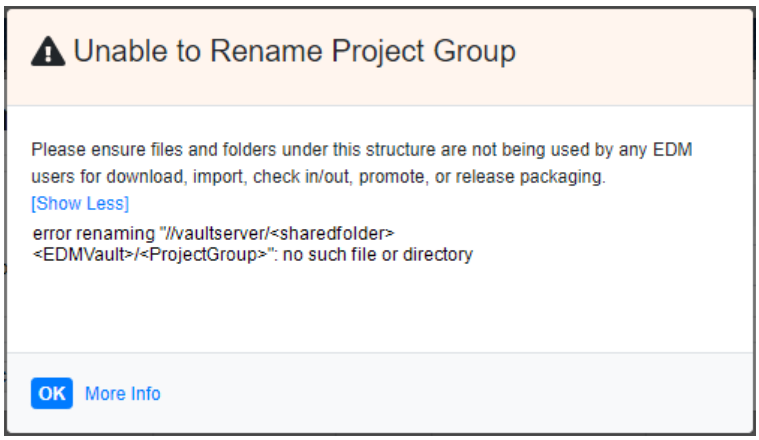
1. All users of affected projects should check in their changes or cancel their check outs.
2. Ensure files and folders under this structure are not actively in use for download, import, check in/out, promote, or release by any user.

Unable to Rename Project Group

The following error occurs when there is a problem renaming a project group. Click the [Show More] link to view more details when needed.



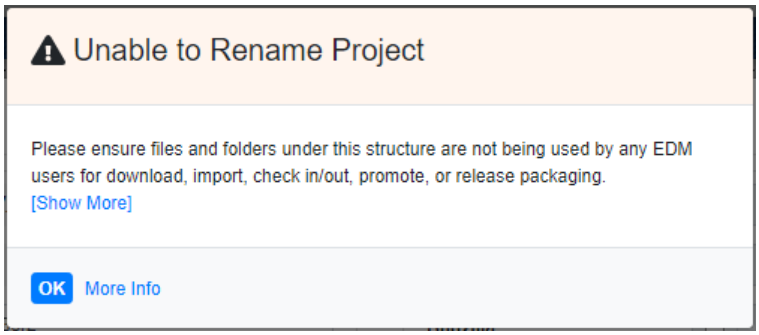
Unable to Rename Project Group



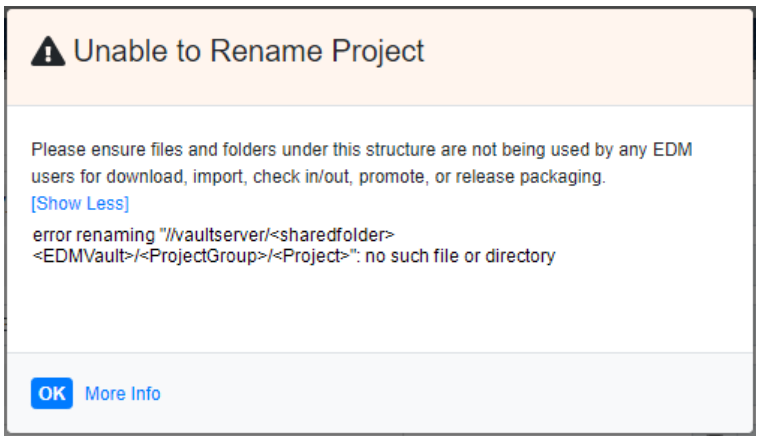
Unable to Rename Project Group - Show More

Unable to Rename Project

The following error occurs when there is a problem renaming a project. Click the [Show More] link to view more details when needed.



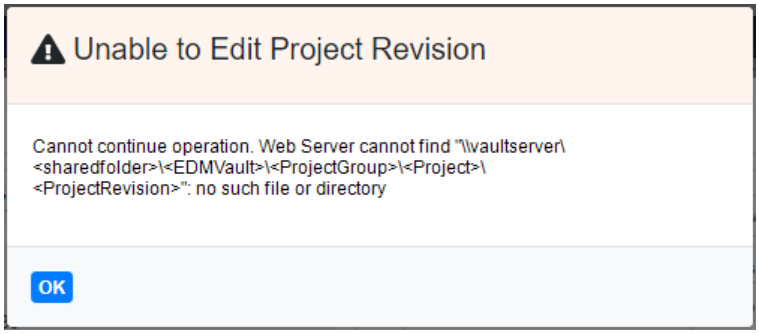
Unable to Rename Project



Unable to Rename Project - Show More

Unable to Rename Project Revision (Edit Project Revision)

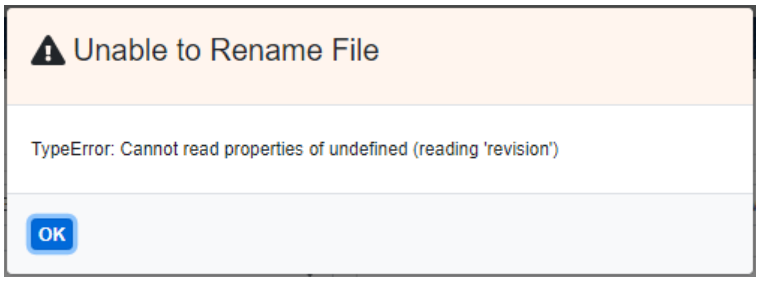
The following error occurs when there is a problem renaming a project revision.



Unable to Rename/Edit Project Revision

Unable to Rename File

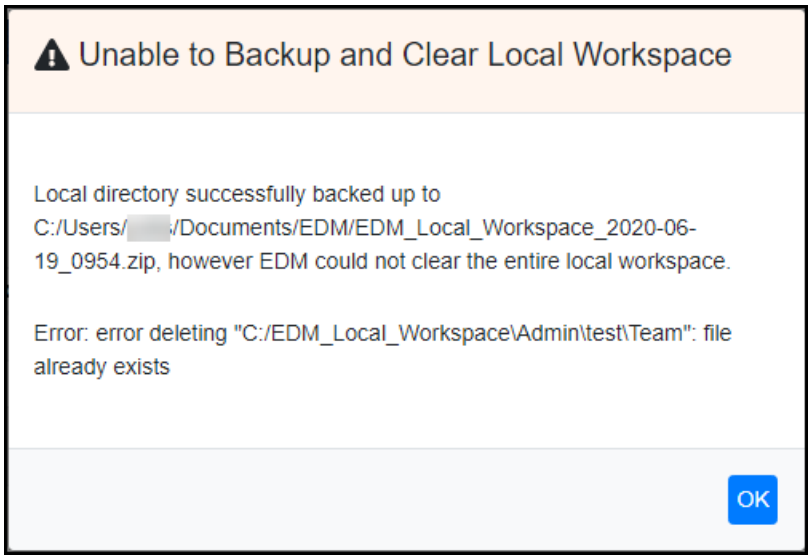
The following error occurs when there is a problem renaming a project file.



Unable to Rename File

13.4.11 Unable to Backup and Clear Local Workspace

Issue: While performing a Backup and Clear Local Workspace, you received an error "Unable to Backup and Clear Local Workspace" as shown in the example below. EDM may have been successful in clearing some of the local workspace files. You must complete the removal of your local workspace files before continuing to use EDM.

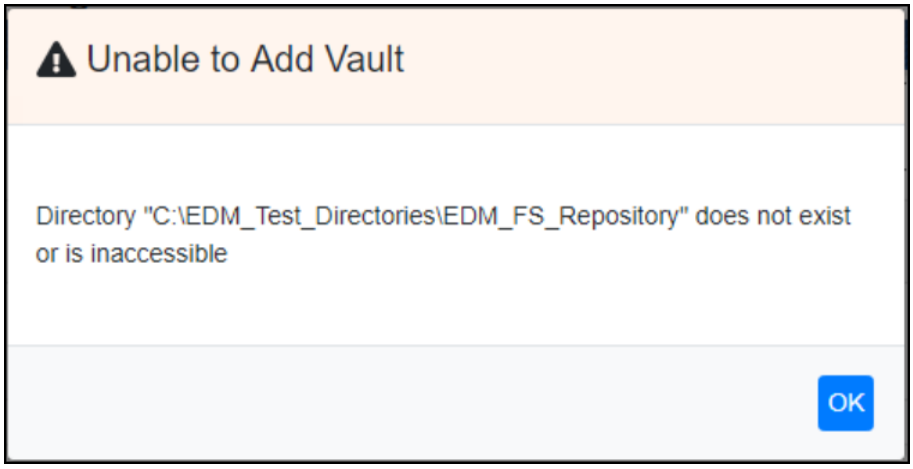


Cause: A file may be locked by another application that is preventing the removal of all files.

Resolution: Open you local workspace using a file explorer window, close any applications that may be locking file(s) and remove all files and folders manually.

13.4.12 Unable to Add Vault

Issue: While configuring the vault for File System to add a Vault, you received an error "Unable to Add Vault" similar to the example shown below.

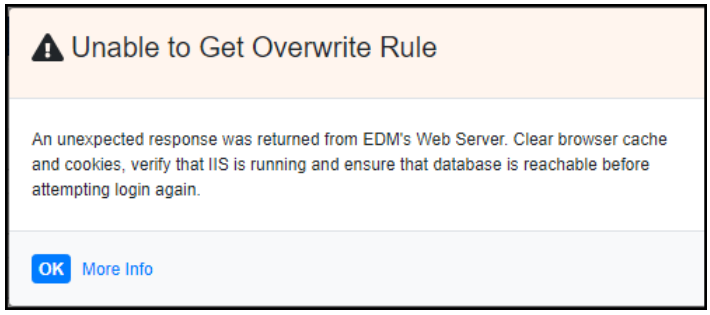


Cause: When a new Vault is saved by EDM, the Web Application verifies that it can access the directory location and that it has sufficient permission. When EDM can't locate the directory or it doesn't have **Full control** permission, an error is presented. The EDM Web Application identity, (the user that EDM runs as) will need to be able to see the location as well as have appropriate access.

Resolution: Make sure that the Web Application identity has **Full control** permission to the directory. If using a mapped drive, may sure that the same mapped drive has been configured for the Web Application identity. UNC path is recommended for configuration of the File System. You may need to contact your system administrator if the file system has not been fully configured as the vault during installation.

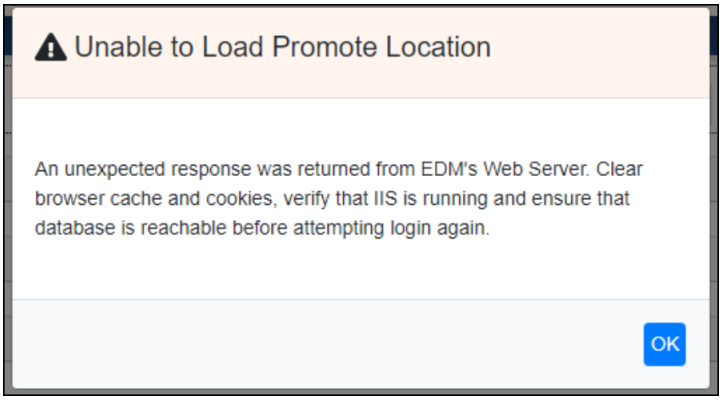
13.4.13 Browser Caching or Database not reachable issue

Issue: While navigating to a page in EDM, you received an error similar to "Unable to Get Overwrite Rule" or "Unable to Load Promote Location" as shown in the example below.



Cause: There are two possible cause for this problem.

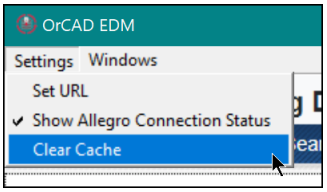
1. There may be times, after an EDM software upgrade, where some of the EDM web pages from are cached in your browser and will need you to take action to clear the browser history before updating. When this occurs, the page may not be able to show correct options or may not open correctly. The following figure is a sample warning when EDM detects that there may be a caching issue.



2. The database is either down or connection string from the EDM web application. This may be result after an upgrade or some other unknown reason. To determine if the problem is related to a connection between EDM Web application and the database, log out, open another browser window to EDM and try to log in. An error "Database is unreachable" shows when this is the problem.

Resolution: The following two actions correspond to the type of problem cause this issue.

1. If the problem is due to a caching issue, you will be able to clear cache on the local PC. To clear cache for the EDM Desktop, select **Settings** > **Clear Cache**.



After doing this, close the EDM desktop application and restart.

2. For the database issue, contact your IT to check the following:
   a. Verify that the connection string in the appsettings.json of the web installation has the correct database name, database username and password.
   b. Verify that the database is up and running
   c. Verify that the user in the appsettings.json has access to the database. A restore of the user will be required if the database has been migrated from one server to another. Information to restore user may be found in the EDM installation guide.
   d. Perform an IISRESET on the IIS server.

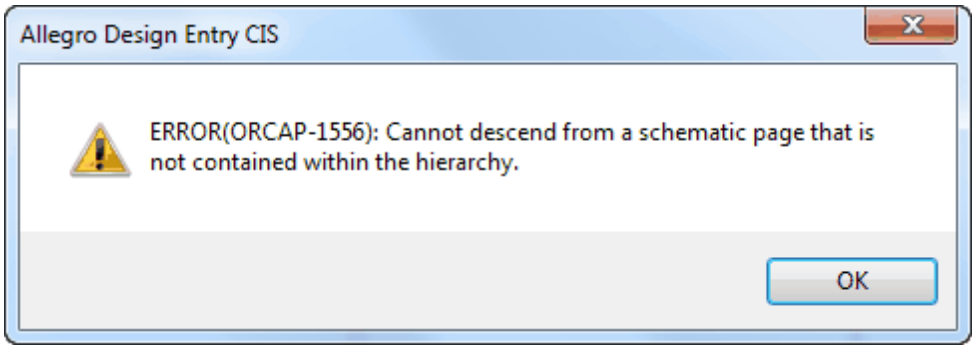
13.4.14 Other Warnings and Errors

When you add parts, pages, and other schematics to a design file using the standard OrCAD Capture graphical interface, Capture's built-in checking of cache or linkage to hierarchy takes place with each item that is added to the design. With EDM, each page or schematic can be added separately to the vault and is checked when the entire design is checked out. Capture routinely prevents you from pasting a schematic that has parts out of sync with other schematics. Capture identifies parts that are out of sync when you attempt to open the downloaded schematic.

EDM has a feature to help you clear such inconsistencies. These inconsistencies may occur when checking out a new file or adding pages or schematics to an existing design. The following list contains information about how to resolve some known problems:

1. After check out of a design, you try to insert a hierarchical block with the name of another schematic as a child, but you get the following error:

ERROR(ORCAP-1556): Cannot descend from schematic page that is not contained within the hierarchy.

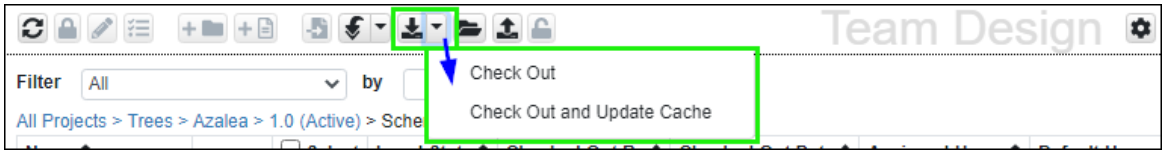


Not being able to descend to a hierarchy occurs when the checked out design has disconnected relationships. To fix this type of problem, you can temporarily change the root schematic in order for you to make your modifications. Refer to hierarchical designs more information on how handle hierarchy.

2. Parts disappear from some of the pages of the schematic after check out. This may occur when parts are out of sync. The errors in the session log may show as:

Page Read Error: Wire read failure
INFO(ORDBDLL-1191): Page <pagename> has database errors; autocorrection in progress. You might lose information.

To clear this type of problem, use the menu option **Check Out and Update Cache** when you check out design pages. This automatically performs a **Full Download with Update Cache** during check out, which forces each page to update with current library symbols. If this does not work, check out each schematic individually and copy and paste each schematic.

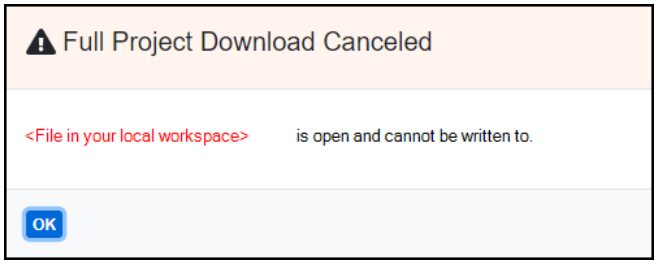


3. Warning appears in the session log regarding not finding library files. Since each user may place parts from various libraries, when you check out a design, local libraries used by one user may not be available in your library or the common library. When this occurs, each symbol not found by Capture will be reported in the session log. Before you finalize your design, it is a good idea that you review this list to ensure that your designs use a common and approved library. The error may show as:

   Could not find <library>

   Example: *Could not find E:\APPS\ORCAD\ORCAD_25.1\TOOLS\CAPTURE\LIBRARY\CAPSYM.OLB*

4. If the design is opened in Capture you will receive the following *Unable to Copy* message. Once you close the design, click the **Check Out** button again to continue the check out process.



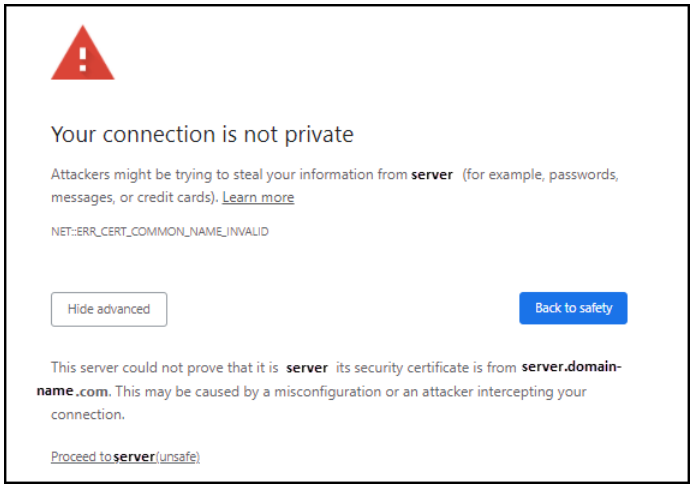
13.5 EDM Shows Blank Screen in Capture

Blank EDM screen inside Capture may be caused by one of two scenarios.

1. Certificate Warning not shown in Capture - When you open EDM while in Capture, a blank screen shows whereas when you open EDM in a standard browser, you see a certificate warning where you can accept risks to continue.
2. Domain Login issue - When you log in from inside Capture using Windows Domain login, you see a blank screen. However, when you log in using a standard browser with Windows Domain login, you are able to see the EDM landing page after you enter your credentials.

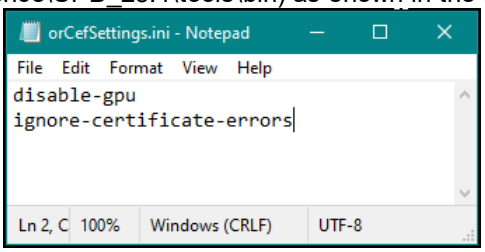
Certificate Warning Issue

If your EDM website is set up with a certificate that is not recognized as a valid certificate by a browser, the browser shows a certificate warning. A sample figure of the warning is shown below.



Sample Certification Error

Each browser shows this warning until you accept the risk and proceed to the site. To accept the risk while using the browser inside Capture, insert "ignore-certificate-errors" into the orCefSettings.ini located in <Cadence_SPB_install_directory>\SPB_**<25.1/24.1/23.1>**\tools\bin\ (e.g. C:\Cadence\SPB_25.1\tools\bin) as shown in the following figure. Update this file for all users.



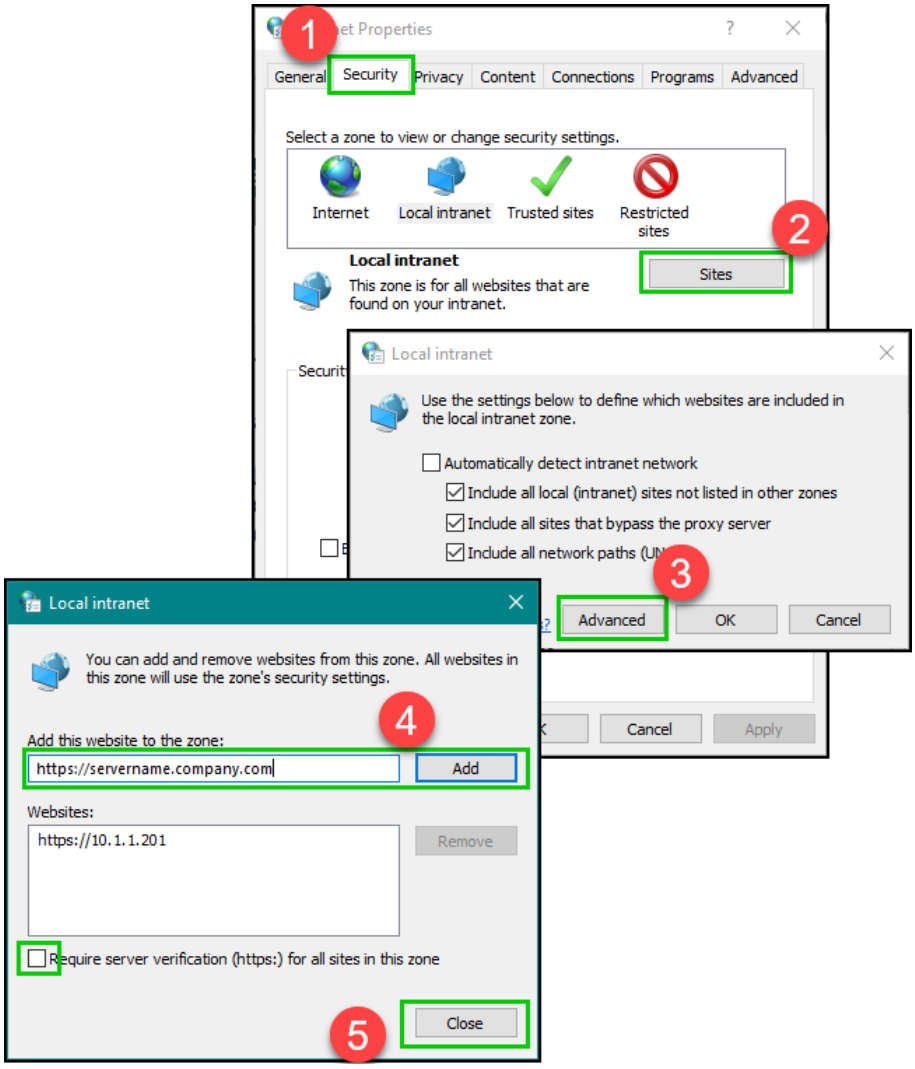
EDM Blank Screen with Windows Domain Log In

When you log in using Windows domain login from a standard browser and you are required to re-authenticate, the browser will automatically open a pop up for you to enter your login information. The need to re-authenticate is related to trust not having been established between the client browser and the domain where EDM is installed. Possible causes include:

- Your PC or VM hasn't been joined to the domain.
- You logged into your PC or VM using a local login.
- Your EDM website URL is entered using an IP address.
- Your EDM website URL is entered using a fully qualified domain name (FQDN).

Potential solutions include:

- Ensure that your PC or VM is joined to the domain and is trusted by the domain.
- Ensure that you are logged into your PC using your windows domain (not a local) login.
- If you must use an IP address or FQDN, add the domain to the list of trust sites. To add a domain to trust sites, select Start > Control Panel > Internet Options. Follow the steps shown in the following figure.

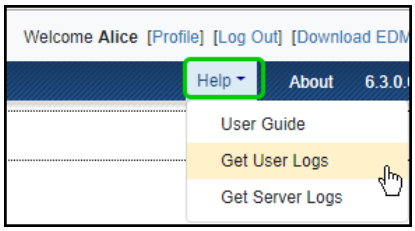


13.6 Technical Support

The EDM **Help** menu enables you to open the **User Guide** in a web browser.

If you have questions or issues that are not addressed in the **User Guide**, contact your local channel partner with questions that require technical support.

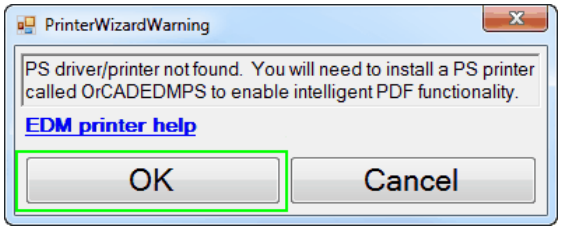
Technical support staff may ask you to forward your log files. To obtain a copy of your local user or server logs, hover over the **Help** menu and select the corresponding log option.



13.7 Modifying PDF Printer Setup

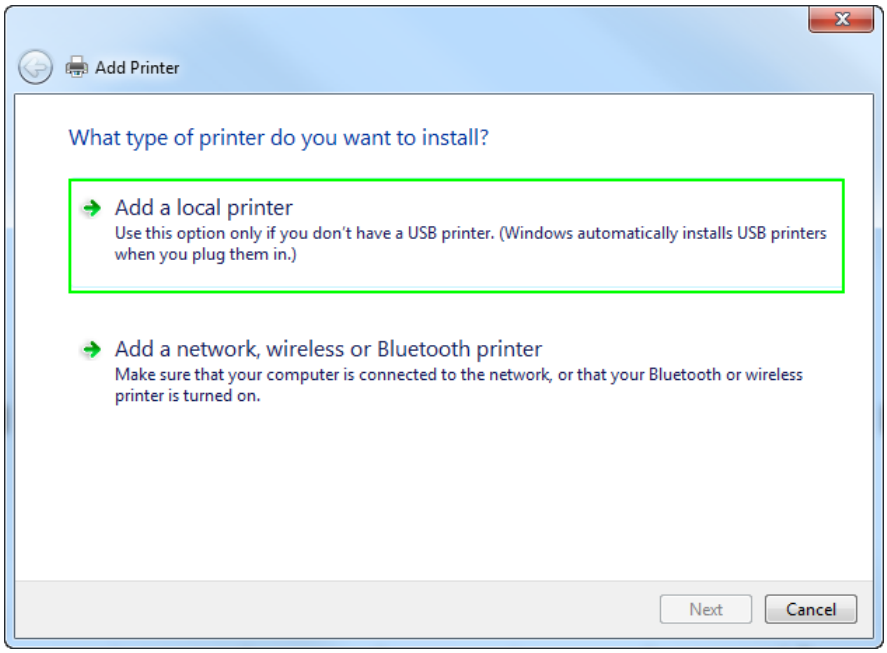
A prerequisite to enable PDF Printing is to have a Post script Interpreter executable such as **Ghost script** (gswin32x.exe or gswin64c.exe) or **Acrobat** (acrodist.exe) available. If you receive a **PrinterWizardWarning** pop-up with the *PS driver/printer not found* message when you attempt to set up Intelligent PDF printing through the EDM Settings dialog, you can follow the steps presenting in this section to install a printer named OrCADEDMPS. After you successfully install this printer you can return to the EDM Settings dialog and enable the Intelligent PDF printing.

1. If you receive the following warning message when setting up Intelligent PDF Printing, click **OK** to open the Add Printer dialog.

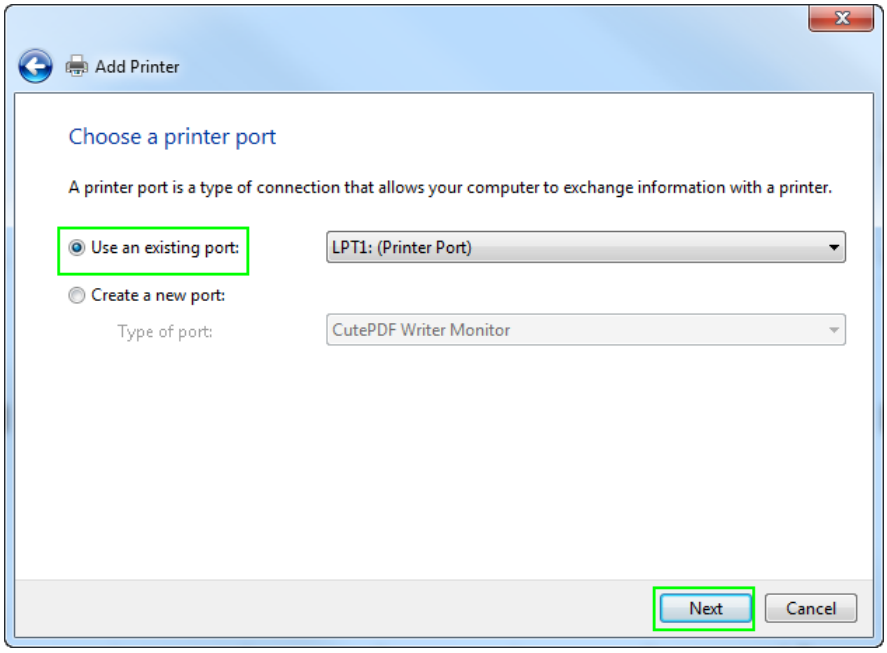


Note: you can alternatively open the Add Printer window, from Settings > Printers & scanners.

2. Select the option to **Add a local printer.**



3. The **Choose a printer port** window will display. Click the **Next** button to **Use an existing port**.

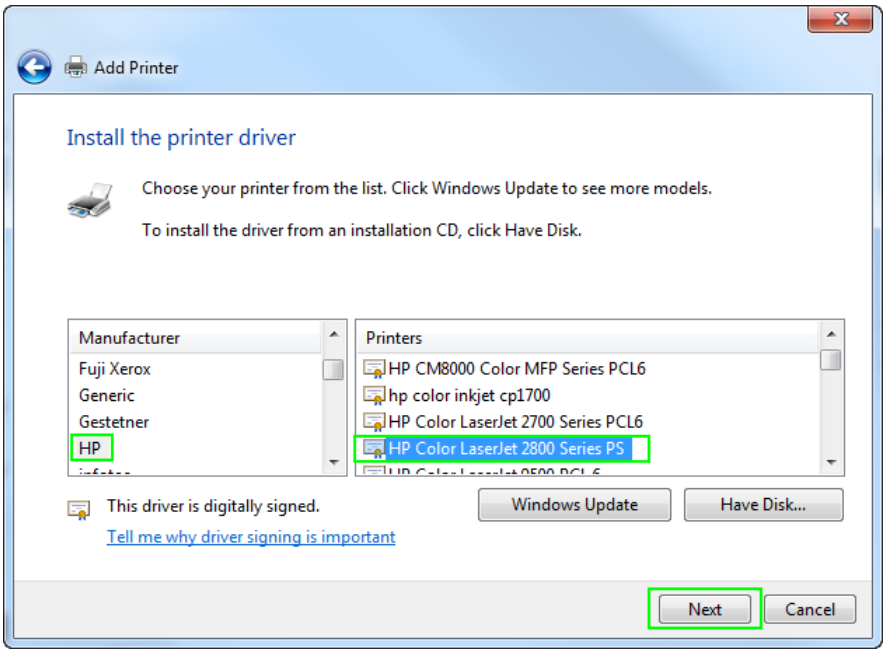


4. The **Install the printer driver** window will display. You will need to select a manufacturer that has **Post Script** capability.

   In the example below, select **HP** under Manufacturer, and then select **HP Color LaserJet 2800 Series PS** under Printers. You can select any printer in your installation that contains the PS for

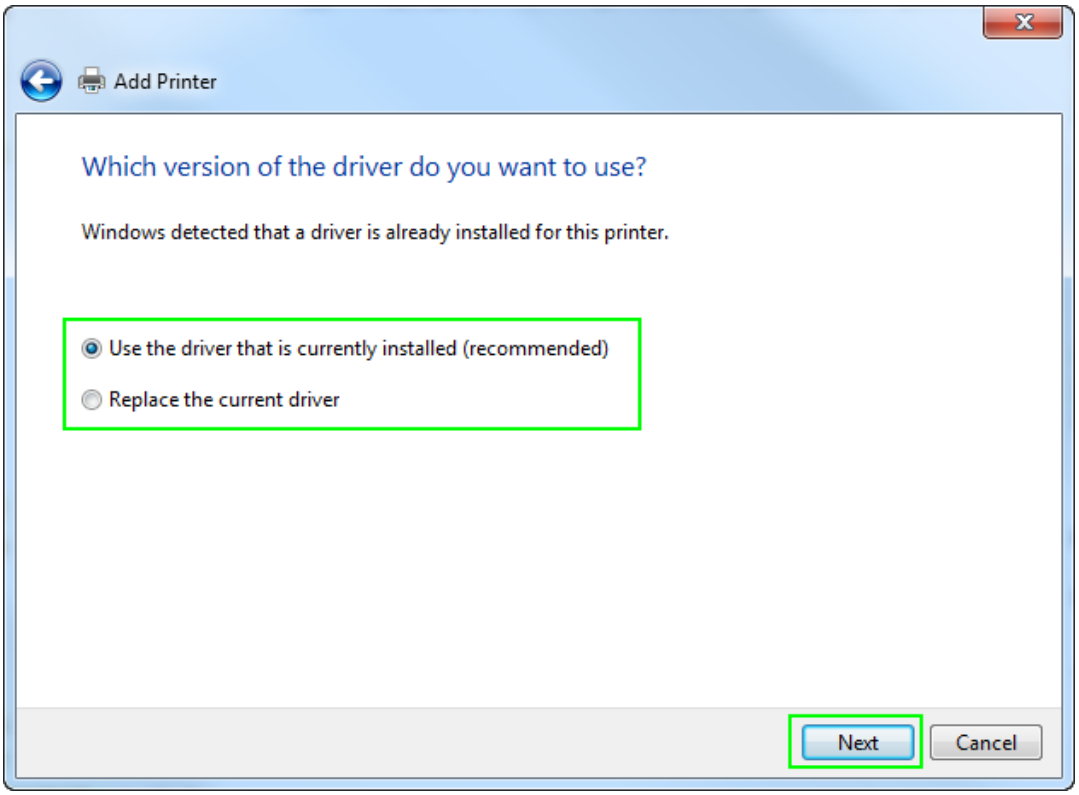
Postscript option.

Then click the **Next** button.

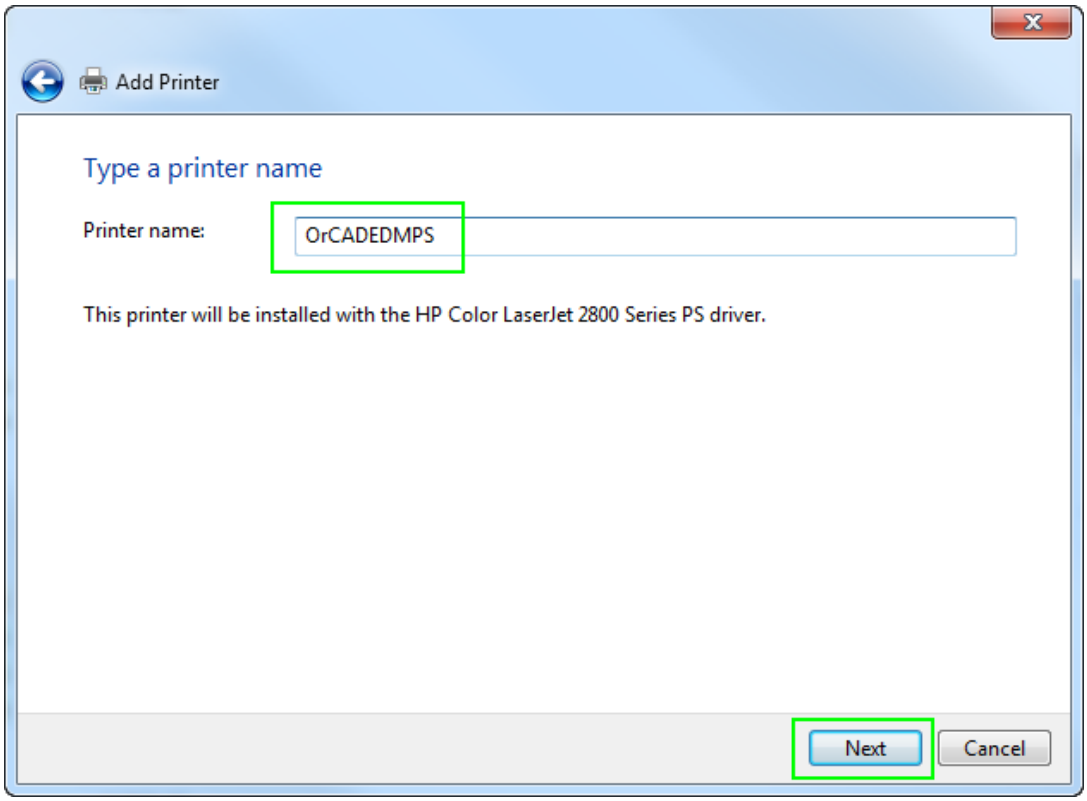


5. If you already have the driver the **Which version of the driver do you want to use?** window will display.

6. Select the option to ether Replace the current driver or Use the driver that is currently installed. When finished, click the **Next** button.

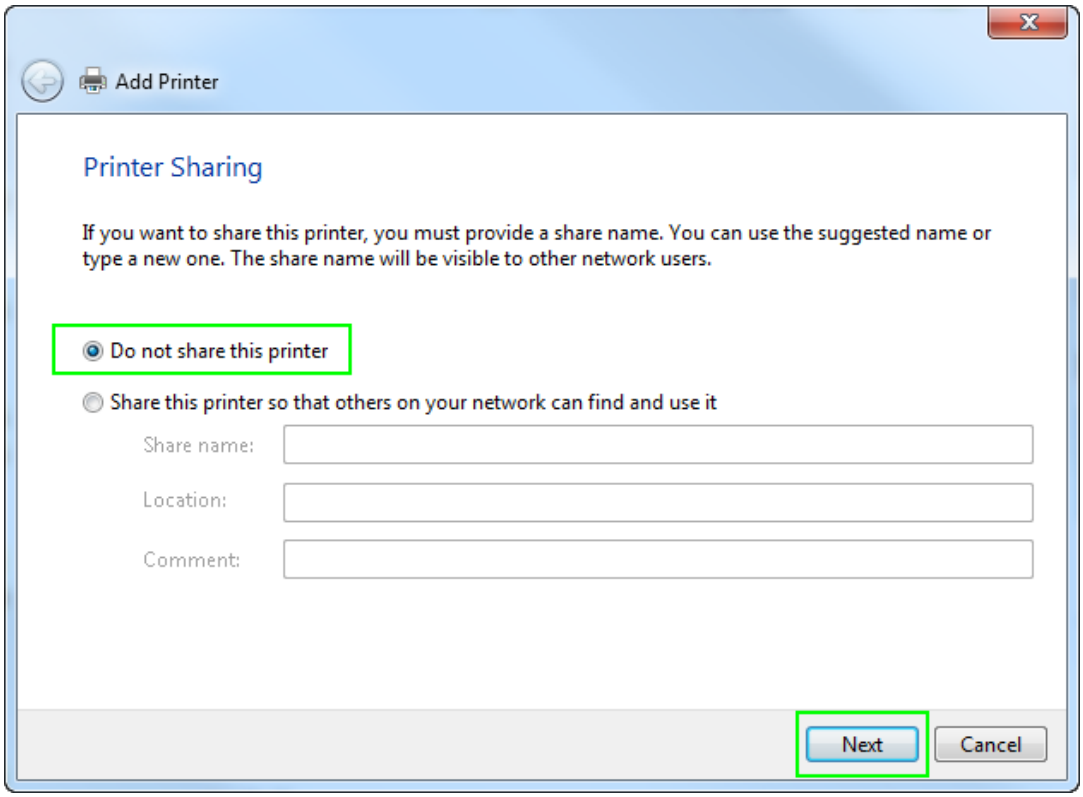


7. In the **Type a printer name** window, enter the printer name: **OrCADEDMPS**, then click the **Next** button.



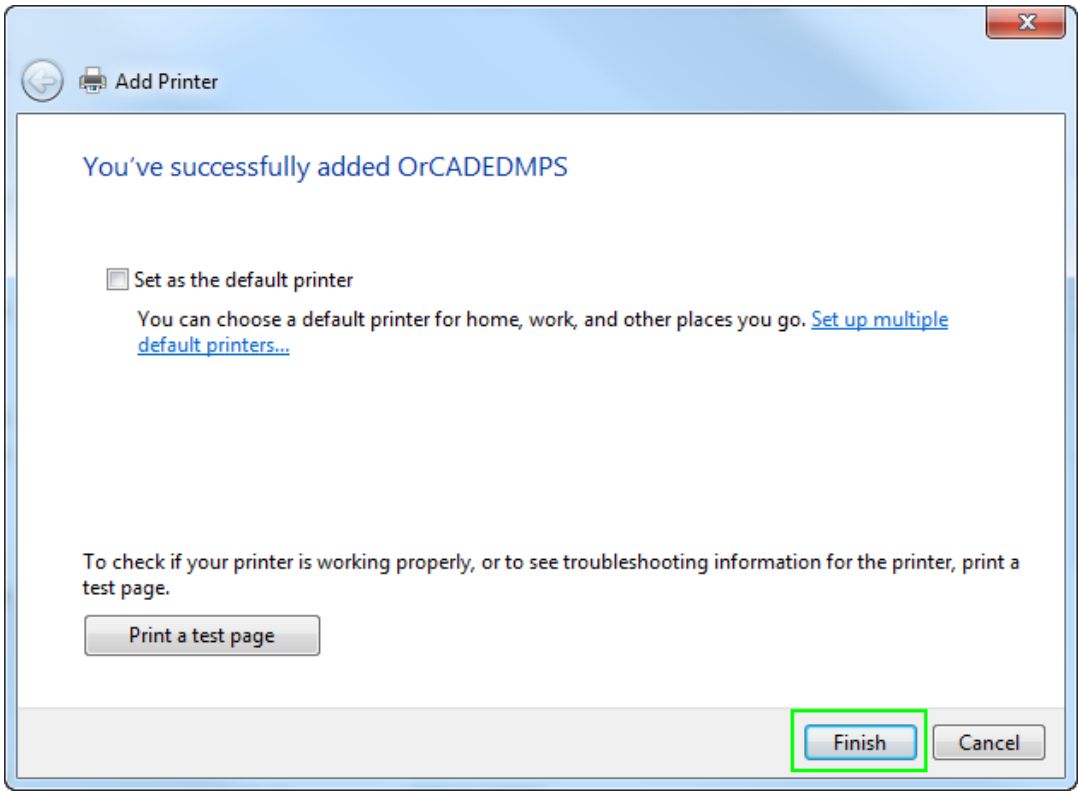
Note: If you chose the **Replace the current driver** option or if you do not have the driver, the **Installing Driver** window will display.

8. The **Printer Sharing** window will display. Leave the default **Do not share this printer** selected. Click the **Next** button.



The **You've successfully added OrCADEDMPS** window will display.

9. Click the **Finish** button.



You are now ready to use the PDF options during **EDM** check in of a Capture Design.