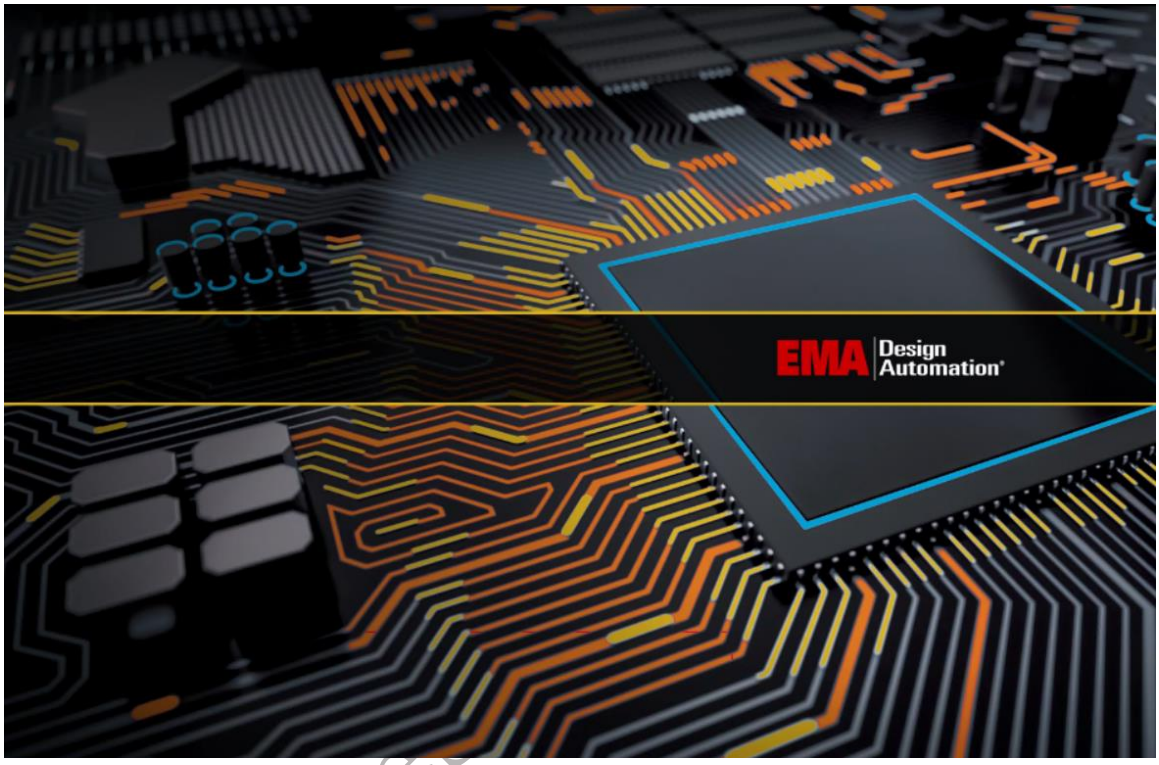


OrCAD EDM

(Engineering Data Management)

Usage Training for Version 6.0.2

EMA Design

©2020 EMA Design Automation, Inc. All rights reserved in the U.S. and other countries. EMA Design Automation, the EMA logo, and TimingDesigner are registered trademarks of EMA Design Automation. Cadence, Allegro, OrCAD, PSpice, and SPECCTRA are registered trademarks of Cadence Design Systems, Inc. All other marks are the property of their respective owners.

Table of Contents

Overview / How to Use This Manual 5
Course Agenda 5
Other Related EMA Courses 6
Formatting Conventions 6
Introduction 8
Roles & Tasks 8
EDM Schematic Flow 10
EDM PCB Flow 10
Lesson 1: Setup and Administration 13
Lesson Objectives 13
Launching OrCAD EDM 13
OrCAD EDM Administration 14
Assigning Roles 14
Important Terms 15
Managing Templates 15
EDM Configuration 19
Lab 1-1: Setting Up a Template 23
Launching EDM Within OrCAD Capture 23
Setting Up a Project Template 23
Adding Folders 25
Add Gerber Sub-Folder 26
Add Schematic Sub-Folder 27
Add PDF Sub-Folder 28
Add BOM Sub-Folder 29
Lab 1-2: Creating a Team Template 31
Creating a Template used for Team Design 31
Add Sub-folders 32
Add PDF Sub-Folder 35
Lesson 2: Working with Schematic Projects 37
Lesson Objectives 37
EDM Terminology 37
Lab 2-1: Creating a Capture Schematic Project 40
Creating a New Project 40
Setting Up the Profile 40
Beginning the New Project 41
Review Settings 43
Lab 2-2: Adding a Schematic Design to the Project 44
Ways to Add a Design to a Project 44
Using Full Project Download 44
Lab 2-3: Performing Work on the Schematic Project 46
Opening the New Design 46

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

Placing a Part on the Schematic Page 47
Save and Check In 48
Checking the Design In 48
Checking Project History 51
Checking Files Out 52
Open Design in Capture & Place Part 52
Checking In the Design After Editing 54
Viewing the Auto-generated PDF 55
Lab 2-4: Creating a New Project 57
Create the New Project 57
Importing the Design 60
Lab 2-5: Applying Lock Ref Des 62
Preparing the Project for Multiple Users 62
Check In After Applying Lock Ref Des 64
More About Locking Reference Designators 65
Assigning Users to Schematic Pages 67
Viewing Team Design Pages in Capture 69
Lesson 3: Working with Schematic Symbol Libraries 73
Lesson Objectives 73
Library Management Flow 74
Lab 3-1: Creating a Library Project 76
Adding a Folder to the Project 77
Lab 3-2: Importing a Library of Symbols into the Project 78
Lab 3-3: Performing Design Work on a Symbol 80
Checking In the Edited Library Part 83
Lab 3-4: Promoting the Symbol 85
Promote Verified Part 85
Lesson 4: Working with PCB Projects 87
Lesson Objectives 87
Generating the Board File 87
Lab 4-1: Opening the EDM Desktop Application 90
Lab 4-2: Importing a Board to a Project 91
Lesson 5: Working with PCB Footprint Symbols 95
Lesson Objectives 95
Working with PCB Footprints 95
Lab 5-1: Generating a New Footprint Project 97
Lab 5-2: Importing a Footprint Library 100
Import Footprint Library 100
Navigating the Footprint Library Views 103
Lab 5-3: Editing the Footprint 105
Checking Out and Editing the File 105
Checking the File Back In 107
Promoting the Completed Footprint 108
Lesson 6: Final Package Release 111

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

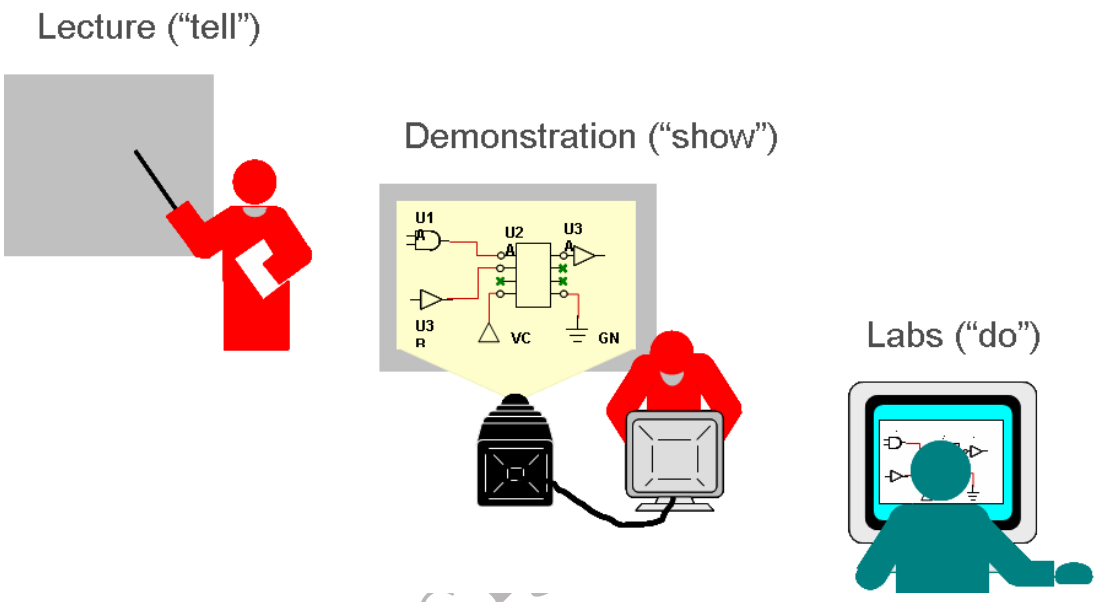
Lesson Objectives 111
Releasing the Package 111
The Package for Release Form 112
The Release Package Zip File 112
Lesson 6-1: Generating a Package Release File for a Project 114

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

Overview / How to Use This Manual

This EMA Design Automation® OrCAD® Engineering Data Management (EDM) Training Manual closely complements the OrCAD Team Design training lectures, slides, and software demonstrations. Concepts and skills taught during this course are based on a "explain," "show," and "hands-on" method.



Each lesson begins with an explanation of application features and associated dialog boxes. Lab exercises follow that provide guided hands-on experience.

Course Agenda

Introduction: EDM Overview
Lesson 1: Setup & Administration
Lesson 2: Working with Schematic Projects
Lesson 3: Working with Schematic Symbol Libraries
Lesson 4: Working with PCB Projects
Lesson 5: Working with PCB Footprint Symbols
Lesson 6: Package for Release

EMA Design Automation, Inc ®© 2020 All rights reserved

Note: It is recommended that students have a pre-requisite knowledge and proficiency in OrCAD Capture Schematic editing and may also wish to have some proficiency in OrCAD PCB Editor.

Other Related EMA Courses

OrCAD PCB Editor (based on Allegro technology): This course will cover all the necessary steps for designing a printed circuit board, from loading logic/netlist data through producing manufacturing/NC output. (3 days) – Also available as Virtual Training (4 days).

OrCAD Capture CIS (Component Information System) enables full database control over schematic symbols, pcb footprints, and all associated part parametrics and manufacturing information. Engineers can confidently choose parts for their designs from the validated, approved database, ensuring that their parts are accurate and available.

OrCAD CIP (Component Information Portal) enables users to completely manage the parts database. It provides access through assigned privileges, enabling teams to acquire new database parts directly from distributors like Digi-Key, Newark, Mouser, Premier-Farnell, Arrow, and RS Components. It also provides capabilities for viewing the schematic symbol for placement, as well as the pcb footprint, and the pcb step model. Additionally, there is a direct link to launch OrCAD Library Builder within CIP.

PSpice® Essentials for Simulation training builds on the design entry essentials learned in the Capture class and adds new skills to enable simulation directly from the Capture environment. This class is ideal for both new PSpice® users and experienced engineers who need to maximize the performance of their circuits.

Please visit the EMA website at ***www.ema-eda.com*** for course descriptions and schedules.

Formatting Conventions

The following formatting conventions are used throughout this training manual:

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

When lab procedures instruct you to click a dialog box button, tab, option, or toolbar icon, the item is formatted in ***bold, italic*** text.

When lab procedures instruct you to select a file name, folder, or schematic page in either the OrCAD Capture Project Manager window or a directory tree, the name of the file, folder, or page is formatted in `courier` text.

When lab procedures instruct you to access a directory path, the path is formatted in courier text.

When you are instructed to select a menu option, the option is formatted in ***bold, italic*** text.

When you are instructed to select a series of menu options, the primary and secondary menu options are separated by the ***(>)*** symbol.

When lab procedures instruct you to press a key on your PC keyboard, the name of the key is enclosed in brackets. For example: ***<key>***.

When lab procedures instruct you to use ***<LMB>*** you are expected to press and hold the ***<Ctrl>*** key on your keyboard while you click design objects using your left mouse button. This technique if often used to select multiple design objects.

Throughout this manual ***(LMB)*** refers to your "**L**eft-**M**ouse-**B**utton". The instructor may also refer to this as the "***Control, Select"*** method of choosing objects.

When lab procedures instruct you to select a design name or object, the name or object is formatted in `courier` text.

If you need assistance contact your Sales Account Representative at EMA Design Automation or you can contact EMA Technical Support at techsupport@ema-eda.com – OR by calling (585)334-6001 Option 5.

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

Introduction

OrCAD EDM (Engineering Data Management) is a collaborative environment that enables teams of engineers to work on a single Cadence OrCAD Capture design file. Administrators can define and set preferences for the team to collaborate with, and Project Managers can then create projects and assign users to work on specified pages within a schematic. Multiple users can check out and check in individual or groups of pages they have been assigned to from a design file thus allowing for concurrent development on a single design. Management of reference designators aid in the removal of duplicate reference designators as well as with the restoration of saved reference designators. Statuses of various portions of a design can be viewed and used for tracking project progress. At the end of the schematic capture design phase, the design can be frozen, or ***baselined***, before progressing onto PCB layout.

OrCAD EDM also handles PCB files in the same manner, enabling teams to work concurrently. Once a design has reached a level of completion EDM provides a Package Release capability that generates a complete zip file including optional auto-generated files and reports.

Roles & Tasks

People who have OrCAD EDM will need to be assigned a role based on the types of things they need to accomplish.

Administrative

There are 3 commonly recognized roles within OrCAD EDM. The administrative role is generally someone who has been designated to set up preferences and determine how EDM will behave with certain commands. They would also be the one who handles SVN installation and setting up the vault. Privileges include:

- Managing Templates
- Generating new projects
- Adding or modifying users
- EDM Configuration settings which include
  - Customizable Fields
  - Title Block Field Mappings
  - Email Notification

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

- Promoted Library Locations
- Occurrence Properties

Project Leader

If a person has been designated as a Project Leader, they will have access to the Projects tab under the Admin menu. This will enable them to generate new projects for team collaboration. Typical tasks would include:

- Create/Modify/Delete projects
- Copy or Freeze Projects
- Create/Modify/Delete schematics and pages
- Import schematics and pages
- Add other Project Leaders to projects
- Assign users to projects and pages
- Promote symbols to libraries
- Remove locks from any pages that may have been locked for editing
- Initiate Package Release

General User

Someone who has been added to a project as a user would perform the following typical tasks:

- View projects in which the user is assigned
- View details for projects in which the user is assigned
- Download project elements as read-only for projects in which the user is assigned
- Lock schematic pages for editing on download of schematic pages that have been assigned
- Upload schematic pages that have been downloaded with lock for editing
- Remove lock for schematic pages that have been locked by the user
- Perform all RefDes functionality except for apply lock
- Relevant privileges apply to the General User who may also work on the board

Once administrative preferences have been defined, users can be assigned to projects. Users will most likely be assigned one or more pages of a project if working in the schematic. Once the user opens OrCAD EDM in their preferred environment, they can see the projects they have been assigned to work on. They can then download those project elements to their local machine and can lock them for editing.

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

This cycle may repeat until all necessary changes and edits have been implemented. When all pages have been completed and restored to the repository the Project Leader can freeze the project and then complete the necessary tasks such as final DRC, Annotation, and Netlisting. Once the design has been completed in PCB Layout, the Project Leader would also handle any back-annotation that might be needed. The Project Leader may also choose to perform a Release Package which includes all project data into a zipped file.

EDM Schematic Flow

The typical flow of schematic data into and out of EDM happens as follows within the OrCAD Capture environment:

- Project Leader creates a schematic project within the Capture EDM environment. The project may either be for a flat or hierarchical schematic design and may include multiple users.
- To generate a schematic project, the Project Leader must access EDM *via the Capture environment*, which enables access to all the relevant tasks within Capture.
- The Project Leader may also generate Schematic Symbol Library projects to manage schematic symbol generation and validation.
- A General User who has been assigned to the Capture project will be able to access their assigned schematic (or schematic pages) by checking them out through EDM. They can then perform the necessary work, save the data, and check the data back in to EDM for revision management.

EDM PCB Flow

The typical flow of pcb data into and out of EDM happens as follows within the OrCAD PCB Editor environment:

- Project Leader will have generated a schematic project in the EDM Capture environment. Now, to generate the associated board file, the Project Leader will open OrCAD PCB Editor and open EDM via the menu ***EDM >> Open***.
- To generate the board, the Project Leader must access EDM *via the PCB Editor environment*, which enables access to all the relevant tasks within OrCAD PCB Editor. Optionally the Project Leader may open the Desktop EDM version to import the board file (or board template) to the project, but to complete any work on the board, the PCB Editor must be opened.

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

- The Project Leader may also generate Footprint Symbol projects to manage pcb footprint symbol generation and validation. This must be done either through the EDM Desktop application or via EDM from within the PCB Editor.
- A General User who has been assigned to the project will be able to access their assigned board by checking them out through EDM via the OrCAD PCB Editor. They can then perform the necessary work, save the data, and check the data back in to EDM for revision management.

NOTE: For this training, Lessons 1 through Lesson 3 will focus on working with EDM via the OrCAD Capture environment in the EDM Schematic flow. Lessons 4 through Lesson 6 will focus on working with EDM via the OrCAD PCB Editor in the EDM PCB flow.

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

EMA Design Automation

Lesson 1: Setup and Administration

Lesson Objectives

After you complete this lesson you will be able to:

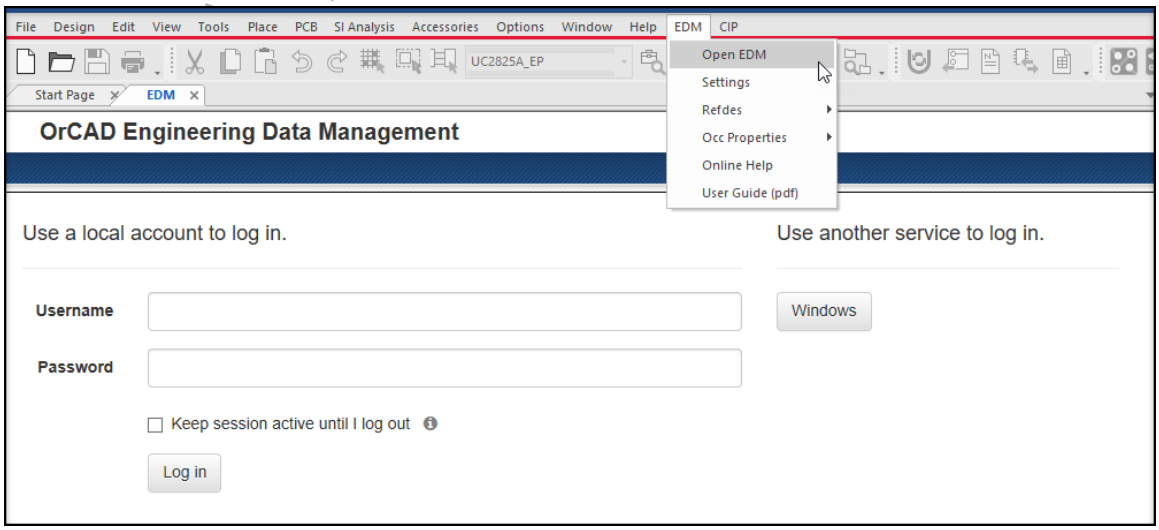
- Launch OrCAD EDM
- Understand about assigning roles
- Understand important terms that apply to EDM file management
- Create Templates

Launching OrCAD EDM

There are 3 ways to access OrCAD EDM:

- From within the OrCAD Capture environment from the ***EDM >> Open*** menu
- From the OrCAD PCB Editor environment from the ***EDM >> Open*** menu
- From the EDM Desktop Application

Once EDM is installed, you will see the EDM menu on the OrCAD Capture main toolbar. Selecting it will provide options, including Open EDM. Once it opens you will need to input your credentials to log in.

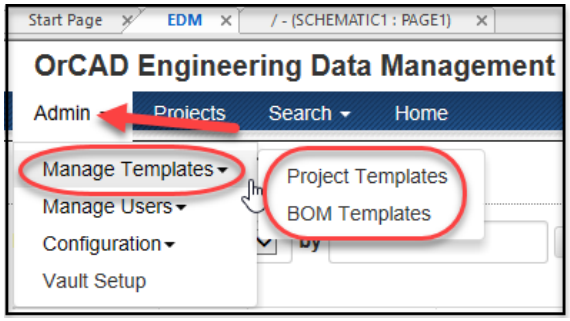


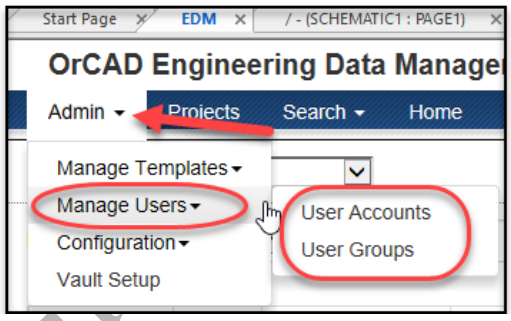
Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

OrCAD EDM Administration

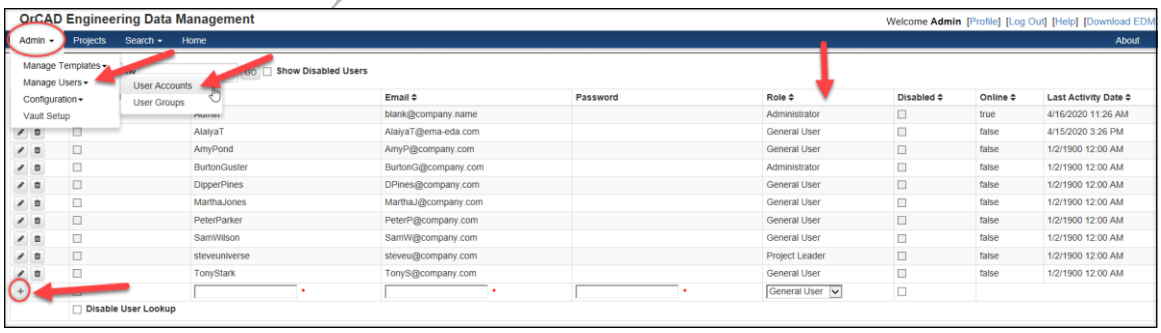
Administration features are available on the Admin Tab. The Admin tab includes options to Manage Templates (Project Templates & BOM Templates), Users (User Accounts, User Groups), Configuration and Vault Setup.





Assigning Roles

Administrative privileges include managing templates, assigning users, setting up EDM configuration, and setting up the vault. Before anyone else can access EDM, the administrator will need to grant access and assign roles to users. Users can be designated as Project Leaders or General Users, or as an Administrator. Username, email, and password are required to add a new user. The default role is General User.



Design Automation

EMA Design Automation, Inc ®© 2020 All rights reserved

Important Terms

Beyond understanding the roles and tasks of individuals using EDM it is also very important to understand what "Team Design" means within EDM. We will be exploring this term as we get into template generation and project generation.

Team Design

If a project has folders that have been designated as **team-design** folders that means that multiple users can be assigned to a project and that assigned users can check out their assigned pages and work on them at the same time as other assigned users. This is conducive to managing larger schematics, either flat or hierarchical.

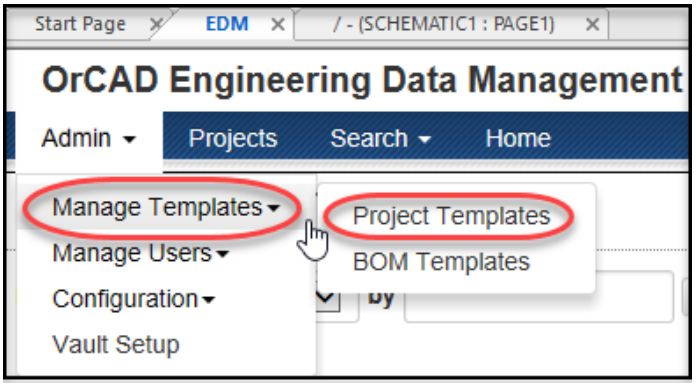
If a project has been set up with **non-team-design** folders, it means that multiple members can be assigned to a project but ***only one assigned member can have it checked out for work at a time***.

We will see samples of these later in the training.

Managing Templates

Project Templates

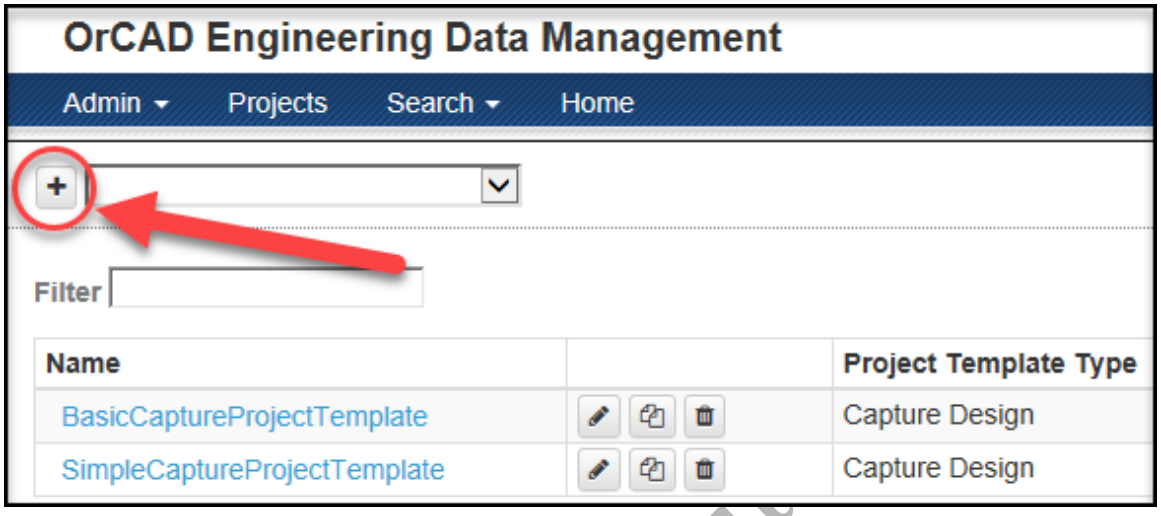
Templates can be generated and reused making project setup more convenient. Both Administrators and Project Leaders can generate templates.



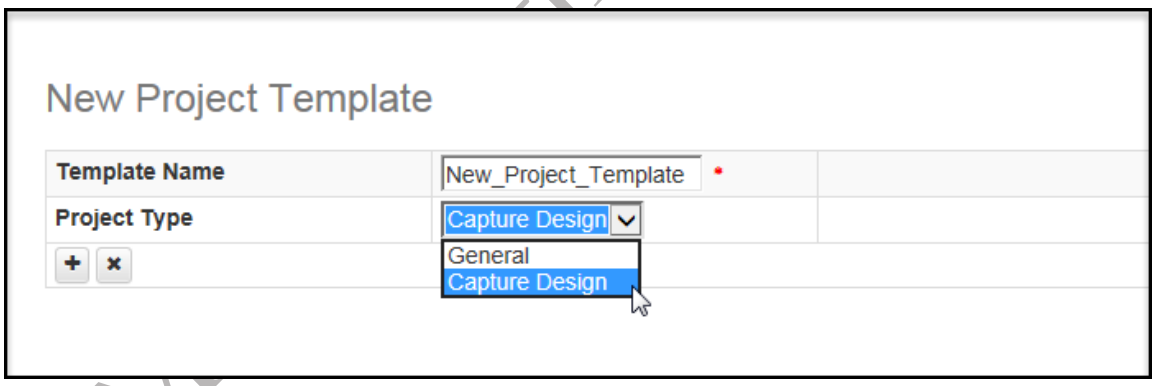
EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

The **Manage Templates > Project Templates** page opens, here you can see some exsisting pre-made templates. A new template can be generated by selecting the '+' plus sign as illustrated in the following graphic.



A name must be assigned to the template. Templates can be applied to either of two types of projects; Capture_Design, or General.



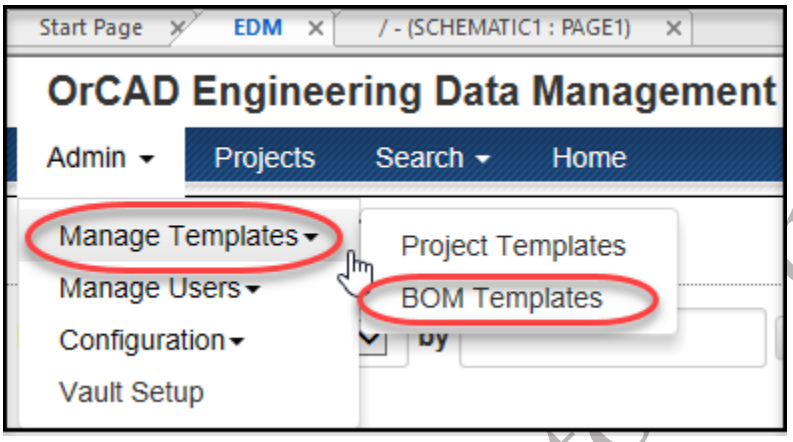
If the project type is General then no Capture files will be generated upon check-in. If the project type chosen is a Capture Design type then Capture files can be auto-generated during check-in.

BOM Templates

Auto-generated BOMs within a project are created using the format of the associated BOM Template. BOM Templates may be created using Capture or CIS BOM formats.

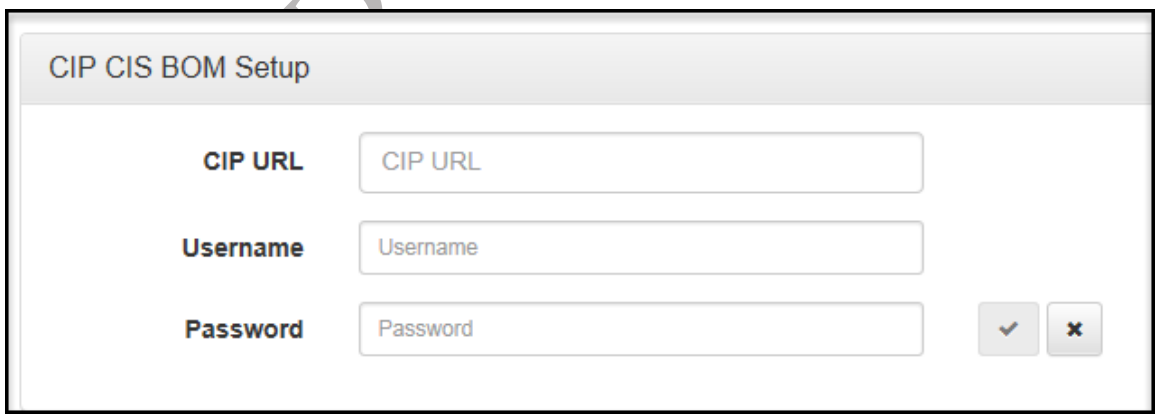
EMA Design Automation, Inc ®© 2020 All rights reserved

While Capture BOM Templates may be used for any design *without variants*, if your design includes variants then the CIS BOM Templates *must* be used to auto-generate variant BOMs. When using CIS BOM templates, the schematic design must be opened using OrCAD Capture CIS.



CIP CIS BOM Setup

The functionality of the CIS BOM template works with OrCAD CIP (Component Information Portal). If you have an invalid connection to CIP, the CIS BOM template uses local properties that were transferred to the design file. If you are using CIS Variants, variant information is pulled from the design and each variant BOM can be auto generated with the check-in of the managed design.

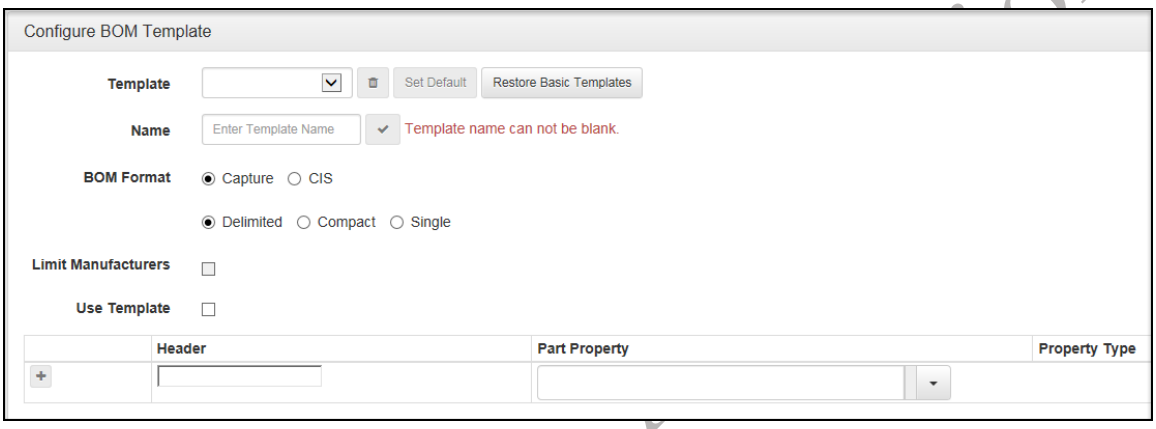


Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

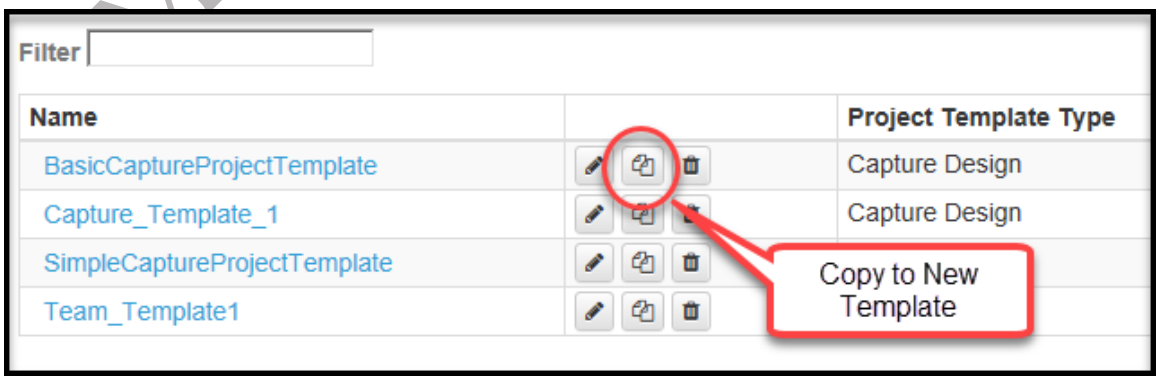
Configure BOM Template

Two BOM Templates come pre-installed and they can be used "as is" or you can modify them as desired. Although you may define as many templates as you need, only one BOM Template can be assigned to a given project's assigned BOM folder. You may select or change the BOM Template assigned to a project when you are adding or editing the BOM. If you have deleted or changed the pre-installed templates and wish to restore them to their original default settings, click the **Restore Basic Templates** button.



Copy Templates

Copy to a New Template is useful when you have a similar structure that requires minor modifications. This allows you to copy the template and then make the modification to the new template.

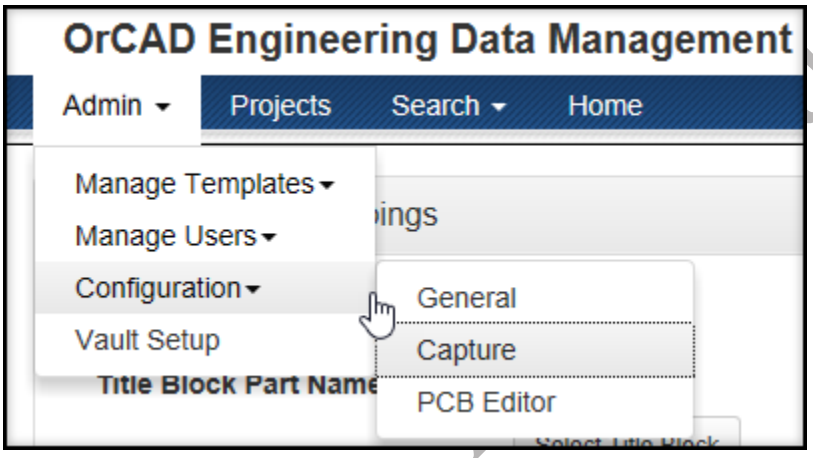


 EMA Design Automation, Inc ®© 2020 All rights reserved

EDM Configuration

The EDM Configuration settings enable you to specify the following:

- General
- Capture
- PCB Editor

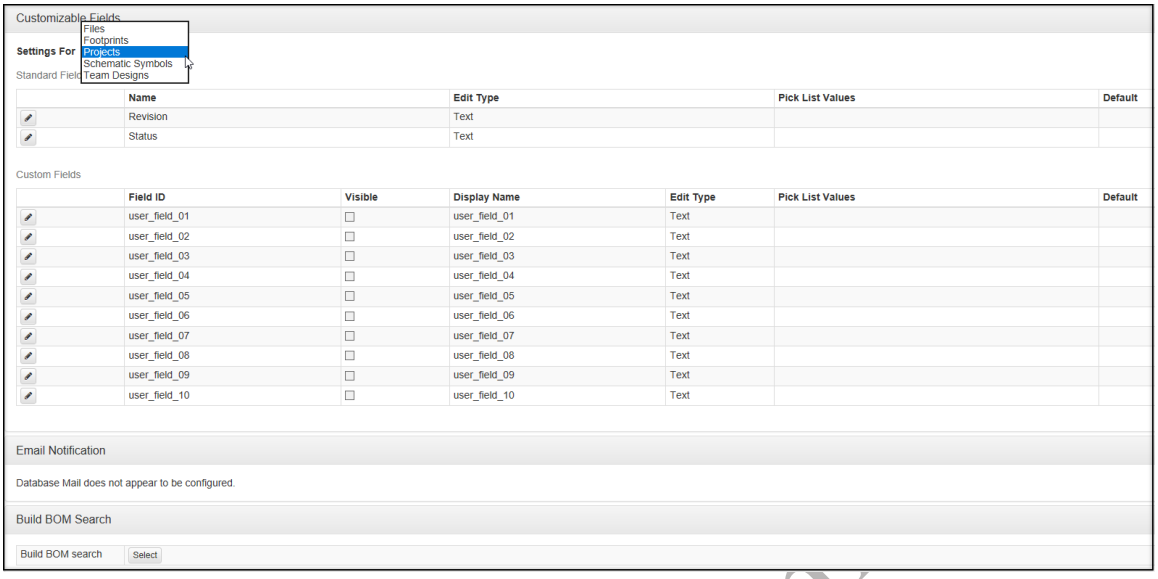


General

- Standard Fields
- Custom Fields
- Email Notification
- Build BOM Search

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

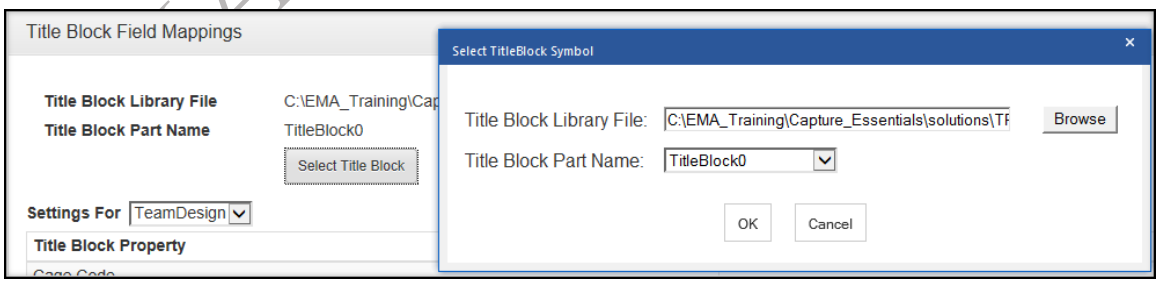


Capture

- Title Block Field Mappings
- Promoted Schematic Symbol Library Locations
- Netlist Generation
- Occurrence Properties
- OrCAD Capture Preferences

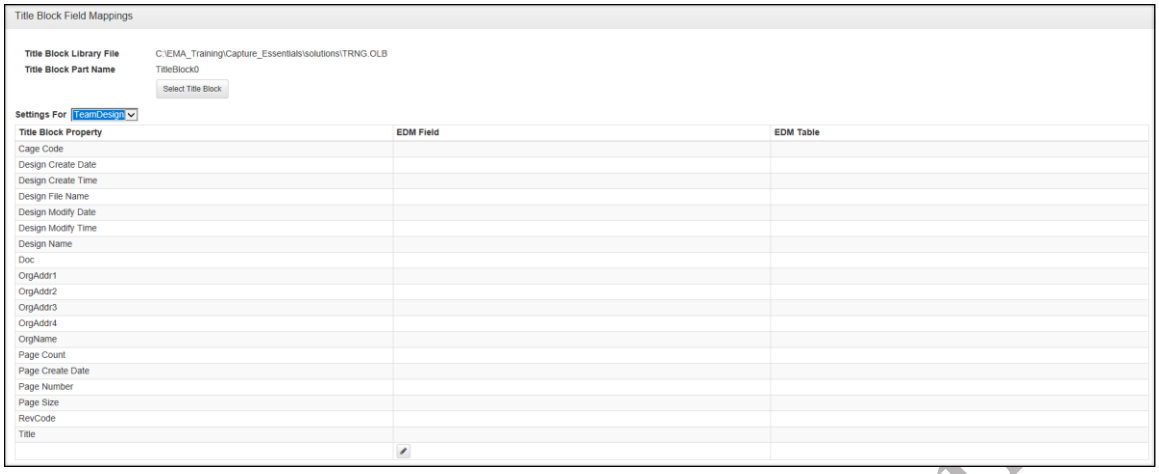
Title Block

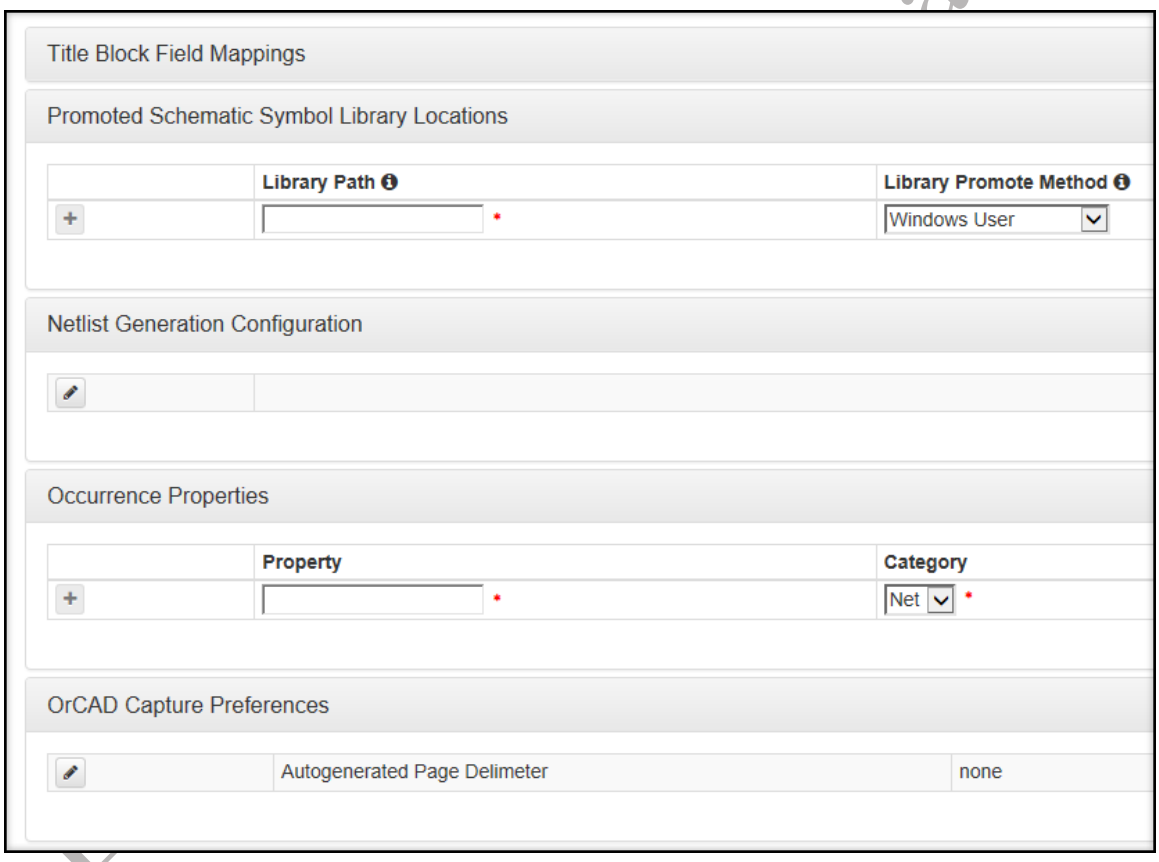
Choose the title block you would like to use and configure field mappings using EDM



EMA Design Automation

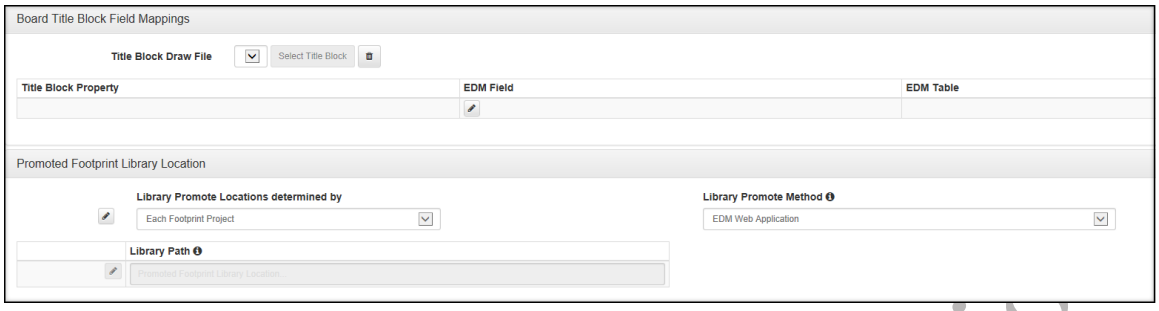
 EMA Design Automation, Inc ®© 2020 All rights reserved





 EMA Design Automation, Inc ®© 2020 All rights reserved

PCB Editor



Administrators and Project Leaders have access to the EDM Configuration settings. For this training class, the configuration settings have already been completed.

EMA Design Automation

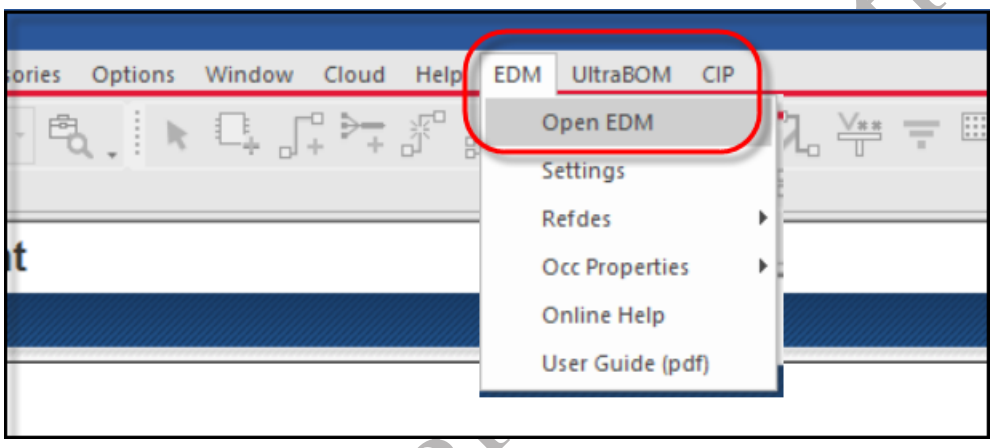
 EMA Design Automation, Inc ®© 2020 All rights reserved

Lab 1-1: Setting Up a Template

In this lab you will set up a new template within EDM as a Project Leader.

Launching EDM Within OrCAD Capture

1. From the Start menu choose ***Cadence PCB 17.4-2019 >> Capture CIS 17.4***
2. When Capture launches choose ***EDM >> Open EDM***



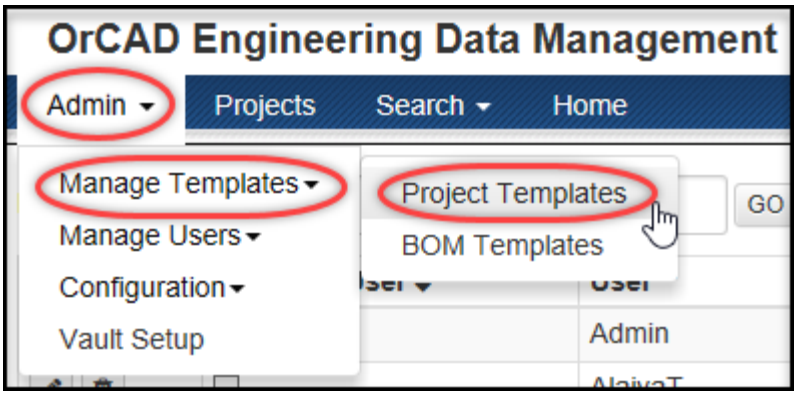
Setting Up a Project Template

Templates enable reuse for new projects. Administrators and Project Leaders can manage templates. Templates can be generated to accommodate the types of projects your team may be working on. In the next lab you will generate a "non-team design" template.

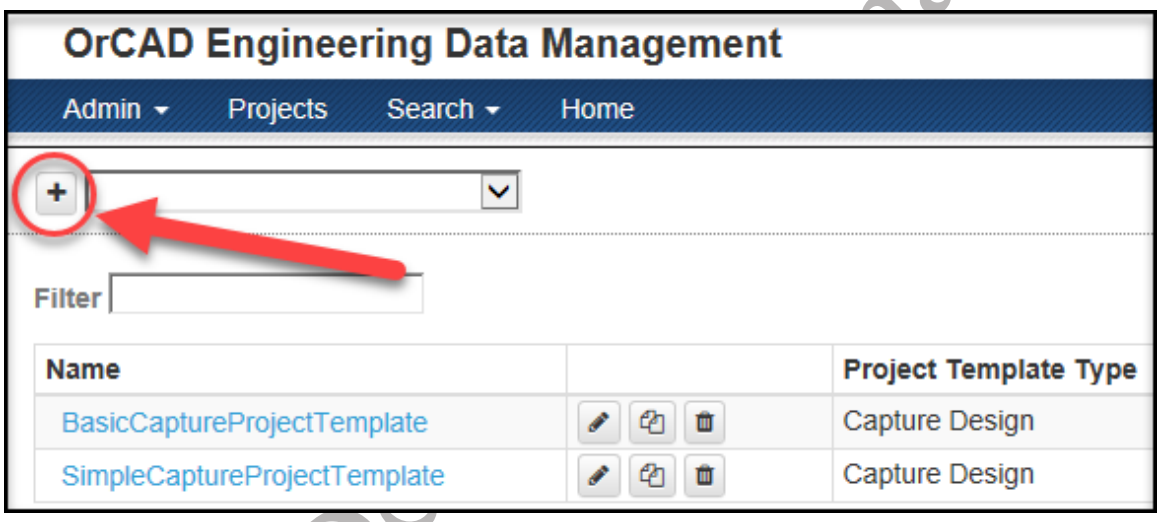
1. Login to EDM as: Username: **Project_Leader** Password: **Pleader_01**
2. Choose **Admin > Manage Templates > Project Templates.**

EMA Design Automation

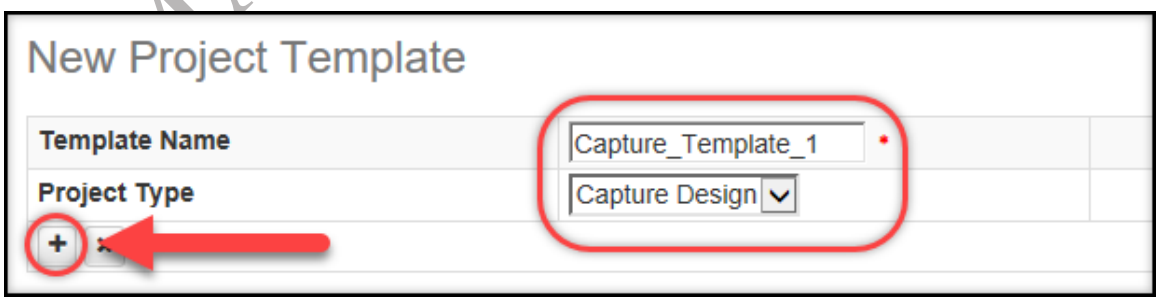
 EMA Design Automation, Inc ®© 2020 All rights reserved



3. When the templates page opens click on the + plus sign to add a new template.



4. Enter the field information as shown in the following illustration. If a template already exists with the same name, append the Template name with a number. For example: If Capture_Template_1 already exists, use Capture_Template_2



- Template Name: **Capture_Template_1**
- Project_Type: **Capture Design**

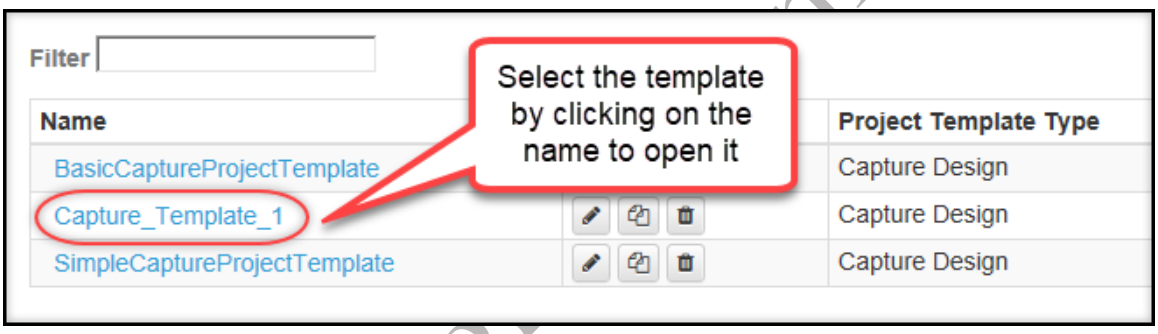
5. Once the data is entered **click the + plus sign** to add the new template.

EMA Design Automation, Inc ®© 2020 All rights reserved

Adding Folders

The template is now ready to be refined. In this next lab you will add folders to house the supporting documents like the bill of materials, PDF, netlist files, and Allegro and Gerber files.

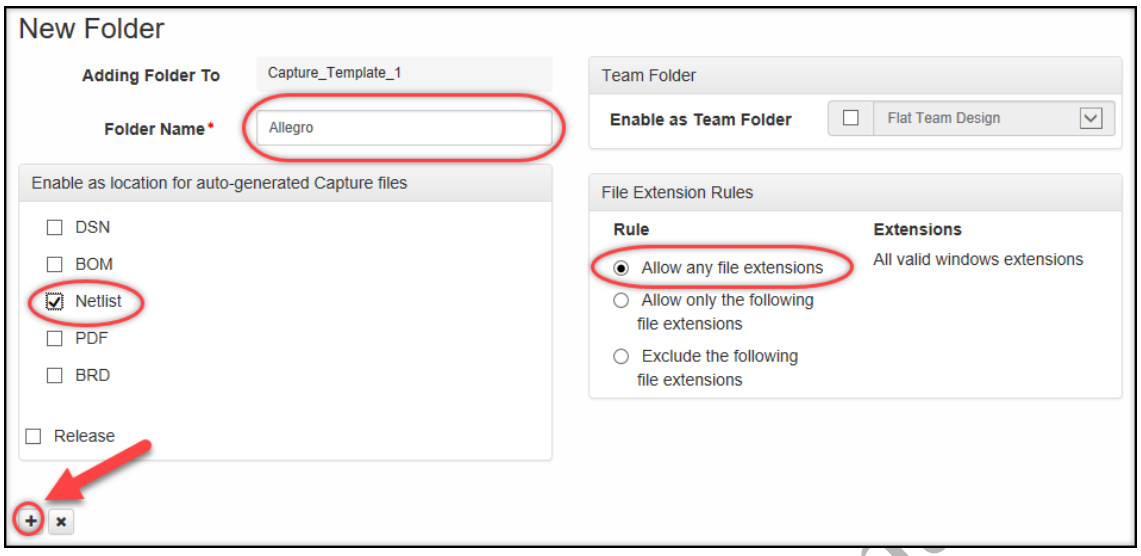
1. Select the name of the template you just created: **Capture_Template_1** (or the one with the name you just created) in the list of templates (**Admin > Manage Templates > Project Templates**) See the following illustration.



2. Next, select the + plus sign to begin adding new folders as is shown in the following illustration.

3. Once the New Folder page opens add the folder name and select the options as in the illustration below. Click the + plus sign to add the new folder.

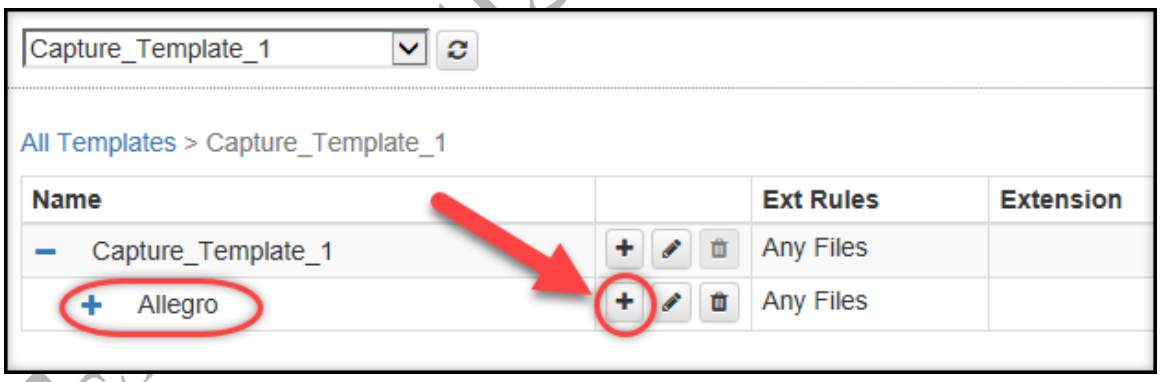
EMA Design Automation, Inc ®© 2020 All rights reserved



Next you will create a sub-folder under the Allegro folder.

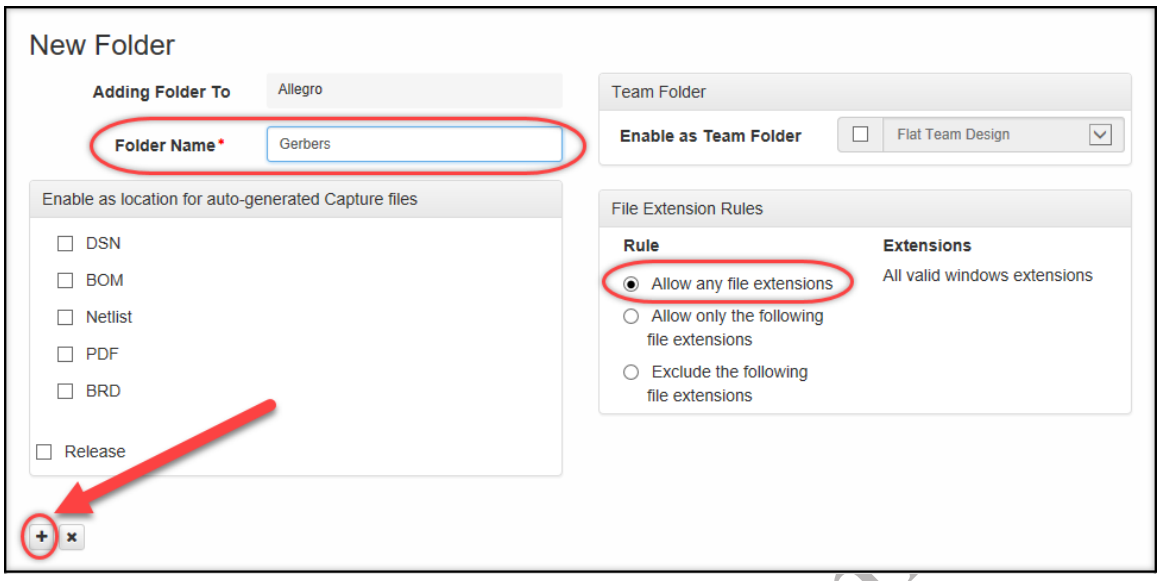
Add Gerber Sub-Folder

1. Click the **+** plus sign in the Allegro folder to add a new sub-folder.



2. Once the New Folder page opens add the name and select the options as shown in the following illustration. The folder name will be **Gerbers**.

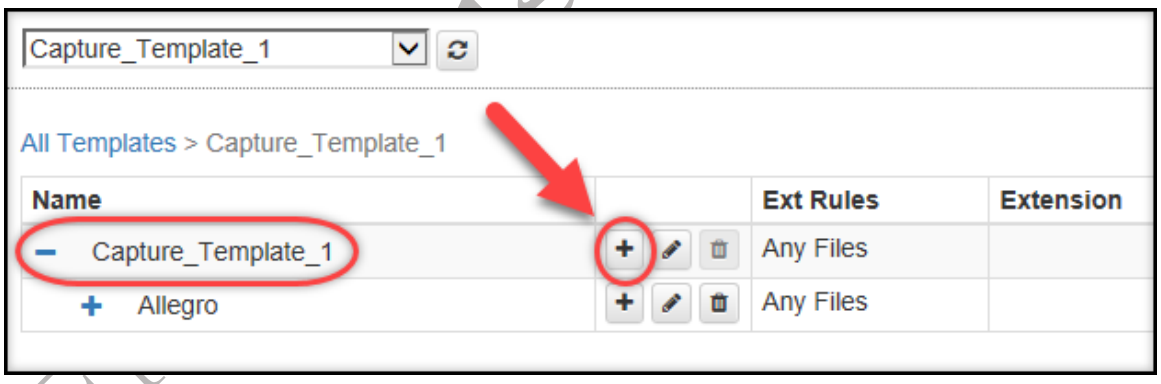
EMA Design Automation, Inc ®© 2020 All rights reserved



3. Click the + plus sign to add the new sub-folder.

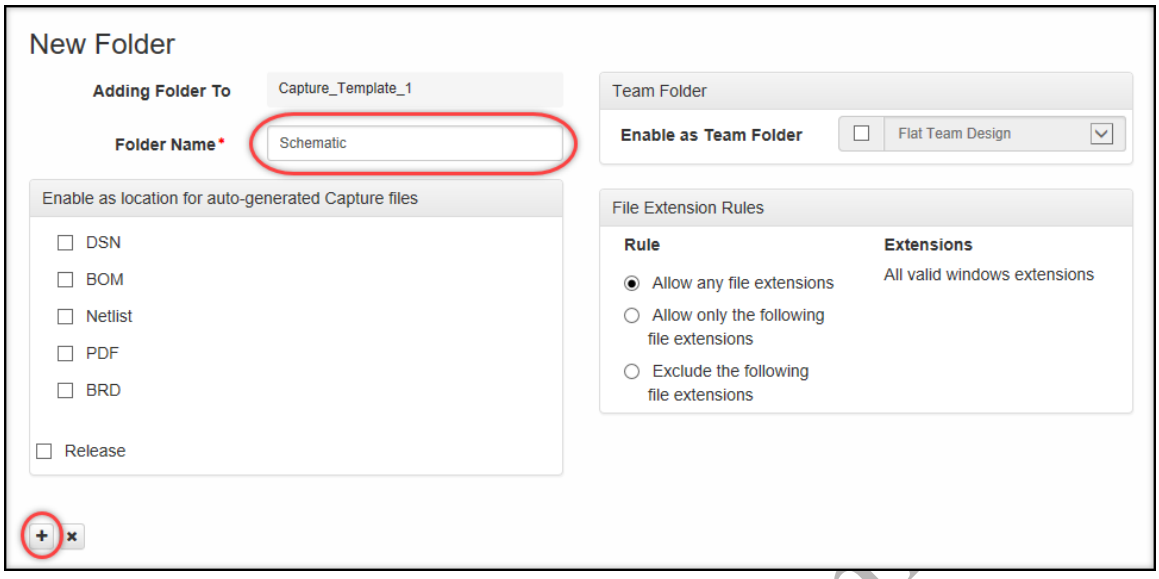
Add Schematic Sub-Folder

1. Click the + plus sign next to the Capture_Template_1 name to add more folders as shown in the following illustration.



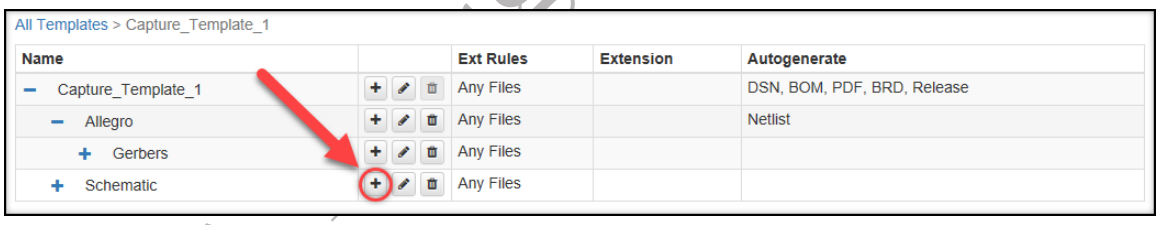
2. When the New Folder page opens, add the Folder Name: **Schematic**. Click the plus + sign to add the folder. See the following illustration.

EMA Design Automation, Inc ®© 2020 All rights reserved



Add PDF Sub-Folder

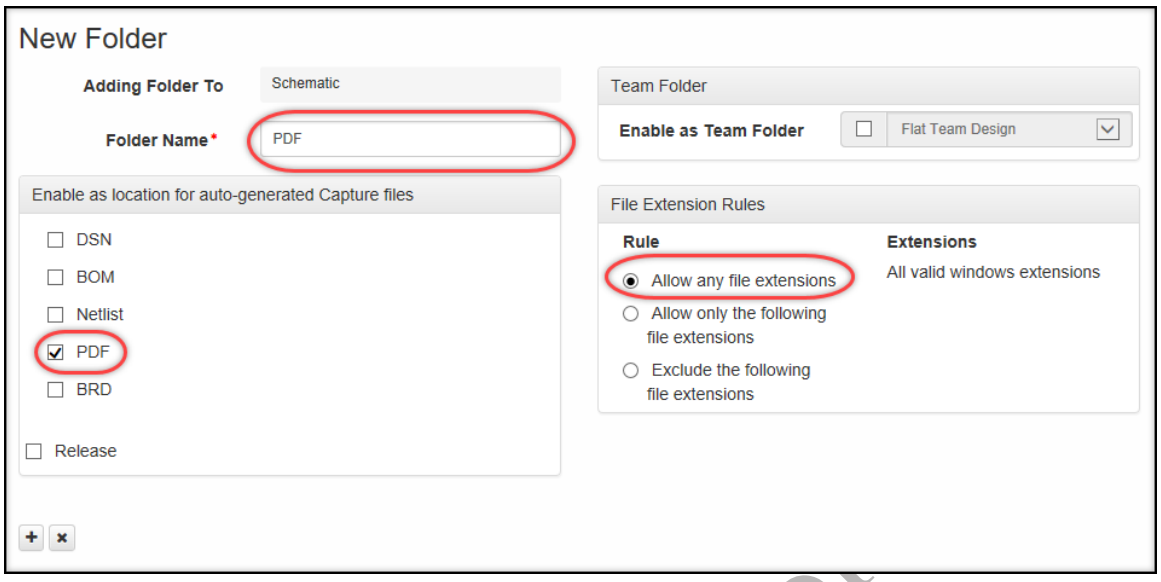
1. Click the + plus sign next to the **Schematic** name to add one more folder as shown below.



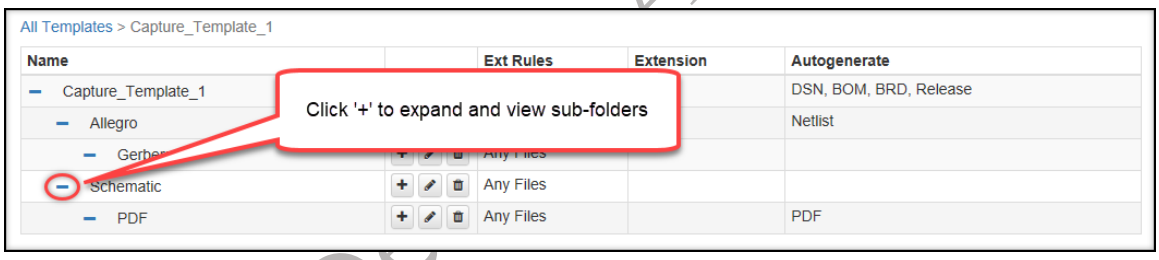
2. Name the new folder **PDF**, and select PDF to Enable Auto-generation. Click the plus + sign to add the new folder. See the illustration below.

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

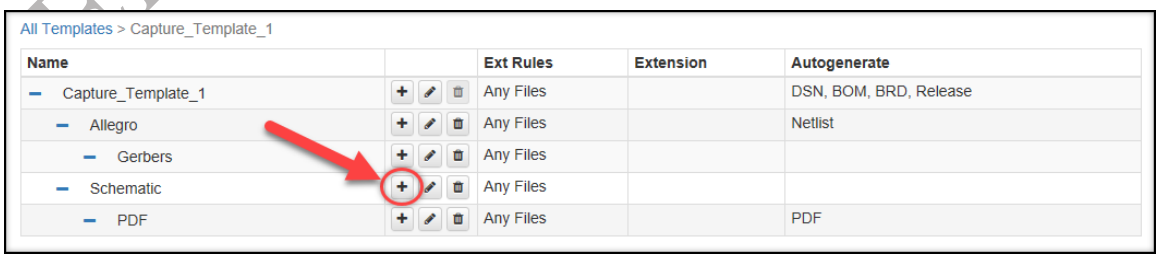


3. Once the folder is added click the plus + sign just to the left of the name "**Schematic"** to expand it so you can see its sub-folders. View the illustration below for reference.



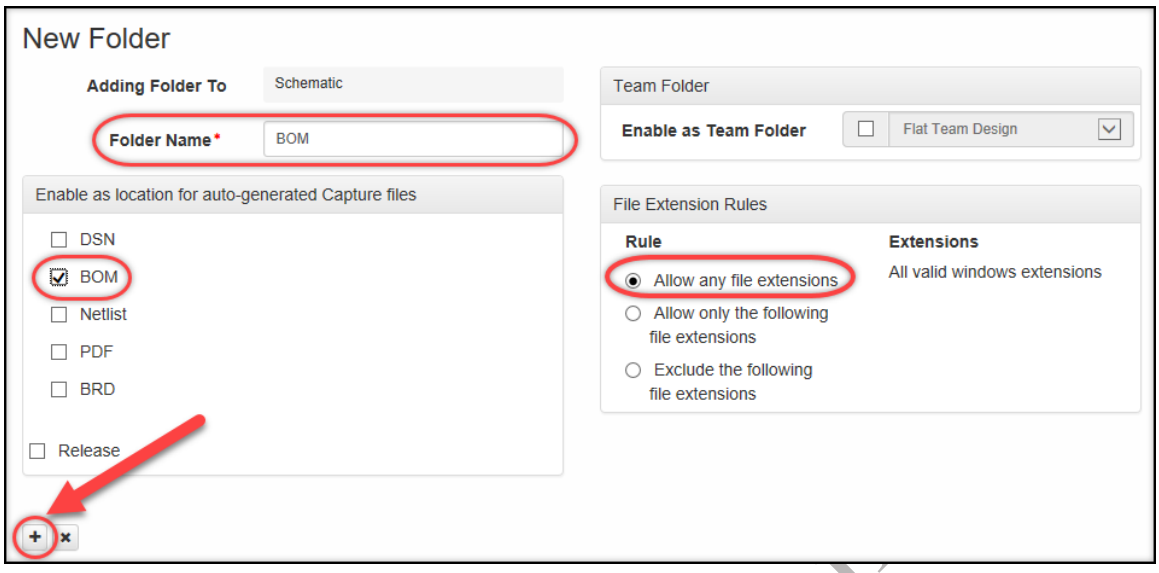
Add BOM Sub-Folder

1. Click the + plus sign to the right of the **Schematic** name to add one more folder. See below.



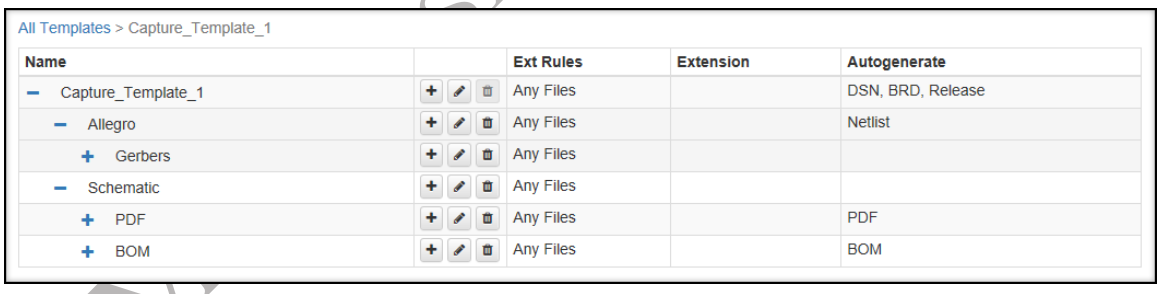
2. When the New Folder page opens add the new folder name as **BOM**. See the illustration below.

 EMA Design Automation, Inc ®© 2020 All rights reserved



3. Choose BOM in the "Enable as location for auto-generated Capture files"
4. Select the + plus sign to add the new folder.

Your finished template should look like the following illustration. Note the Autogenerate column is indicating the files that would be autogenerated upon check-in of design work.



Name	Ext Rules	Extension	Autogenerate
Capture_Template_1	Any Files		DSN, BRD, Release
Allegro	Any Files		Netlist
Gerbers	Any Files		
Schematic	Any Files		
PDF	Any Files		PDF
BOM	Any Files		BOM

Later in this training we will examine templates that have been generated to accommodate team-design folders.

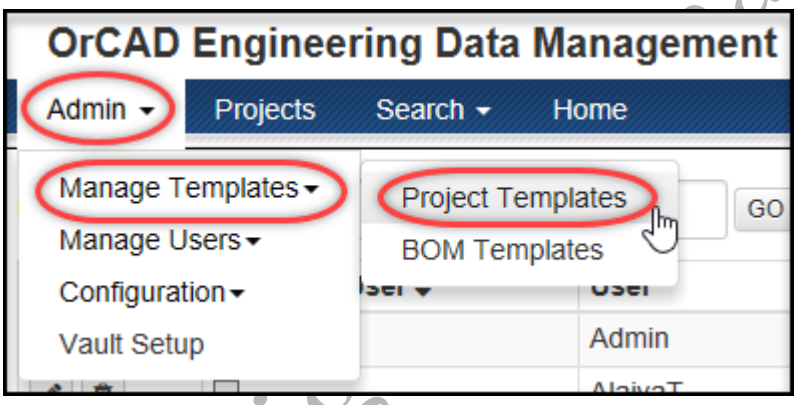
 EMA Design Automation, Inc ®© 2020 All rights reserved

Lab 1-2: Creating a Team Template

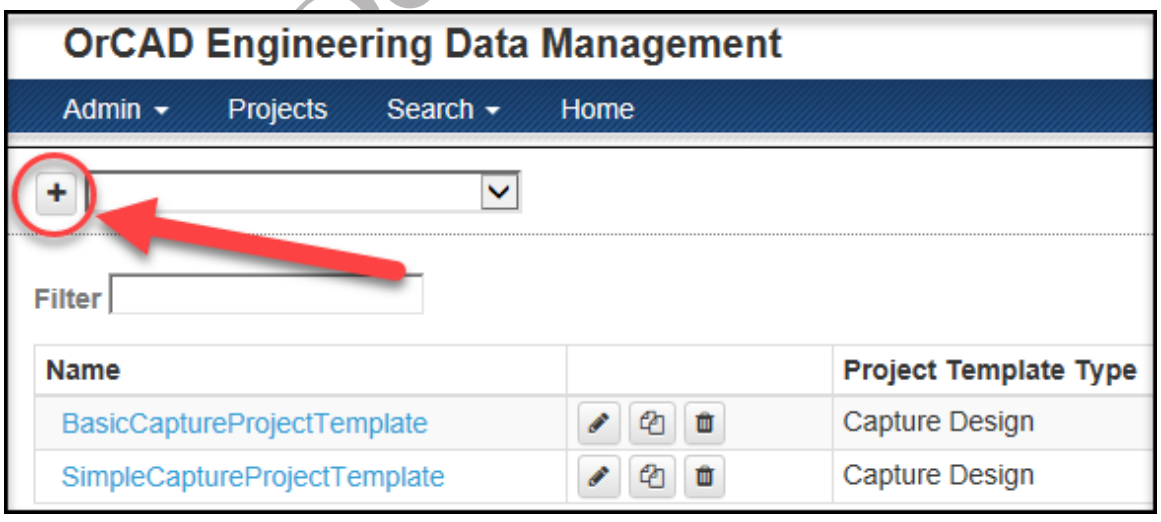
Next you will create a template that can be used for team design. Team design enables multiple users to work concurrently on a design. Having a template for this ahead of time can make developing these types of Capture projects easier.

Creating a Template used for Team Design

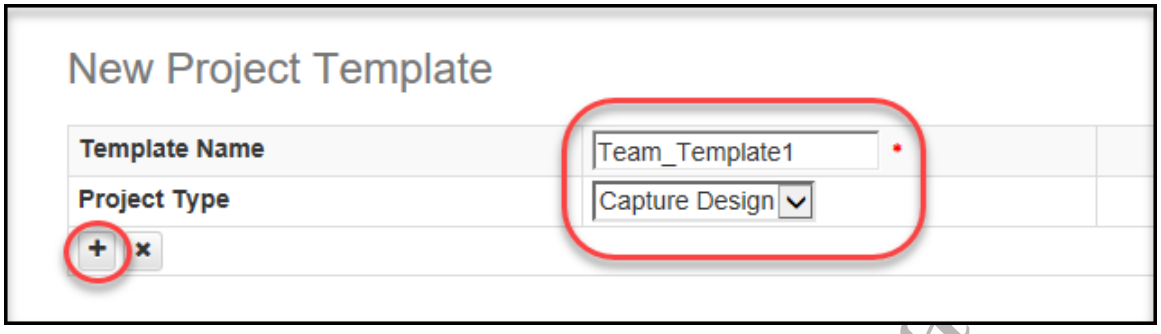
1. From the Admin menu choose **Manage Templates > Project Templates**



2. Next, choose the Add Template button.



3. When the New Project Template window appears, name the template **Team_Template1**, and leave the Project Type as Capture Design. Then click on the plus sign + to create the new template (as shown in the following graphic).



4. Choose the new Team_Template1 from the list of templates by selecting the template name.

Add Sub-folders

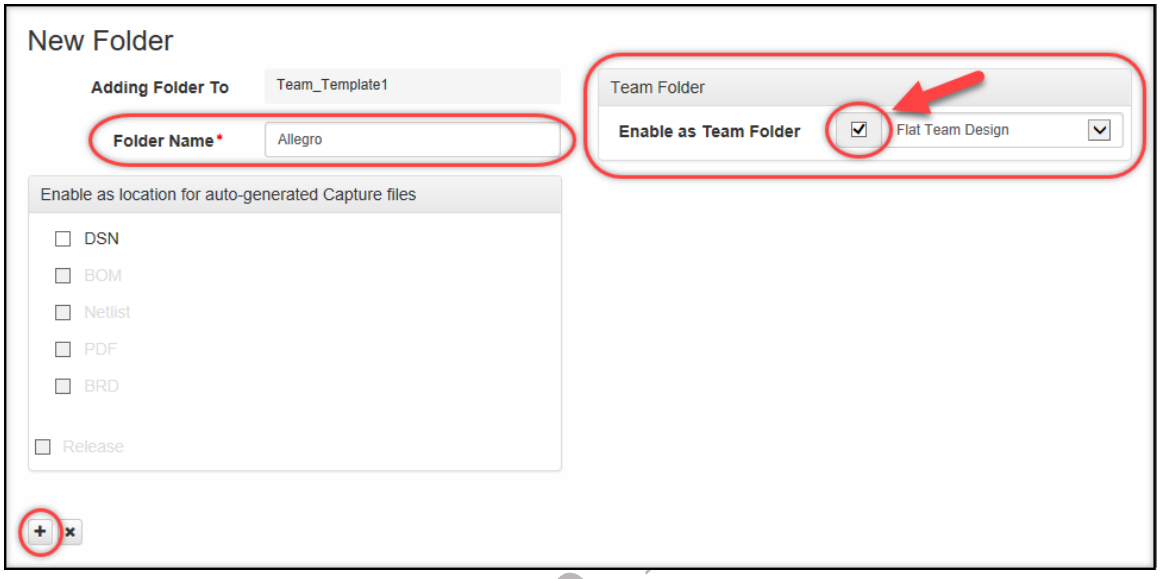
1. Now that the template is created and ready for building the structure, choose the **Add Folder** button, as shown in the following graphic.

2. For the first sub-folder make the Folder Name **Allegro** and choose the **Enable as Team Folder** option.

EMA Design Automation, Inc ®© 2020 All rights reserved

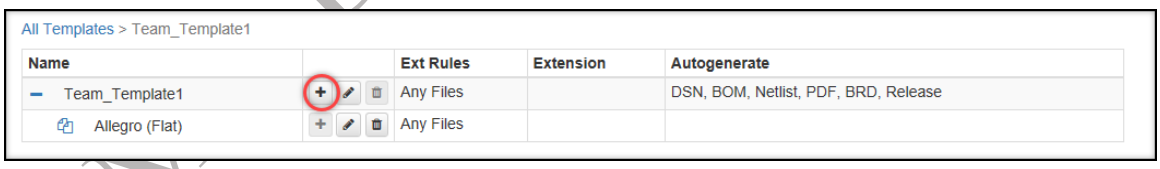
Notice that there is a drop-down that offers a Flat Team Design, a Hierarchical Team Design, or a Team Library.

3. Leave the option enabled that is: **Flat Team Design** and choose the plus sign + to create the sub-folder. See the following graphic.



Next you will add another sub-folder to house the schematic files.

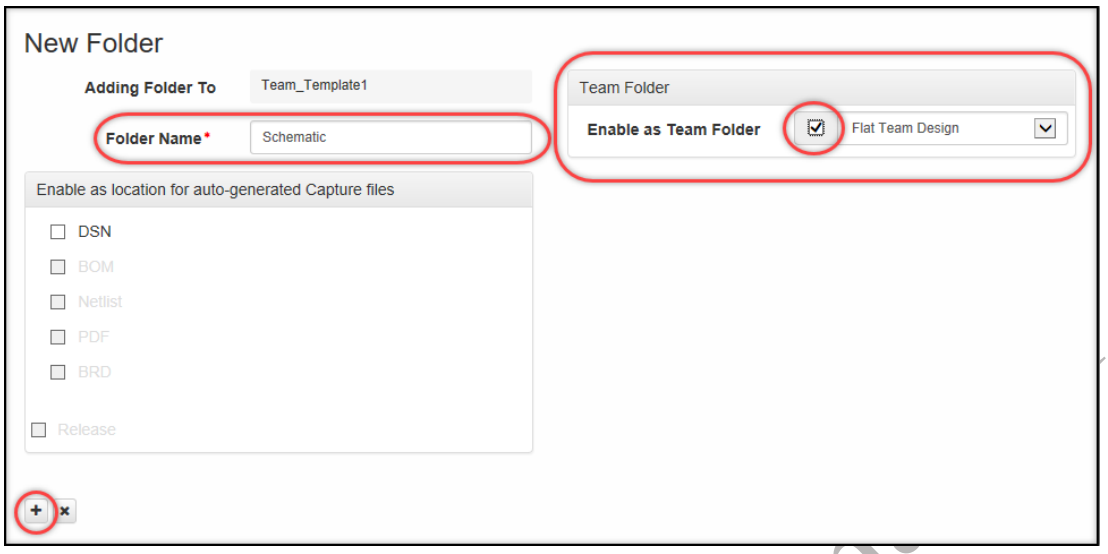
4. From the Team_Template1 top level, choose the **Add Folder** button, as shown in the following graphic.



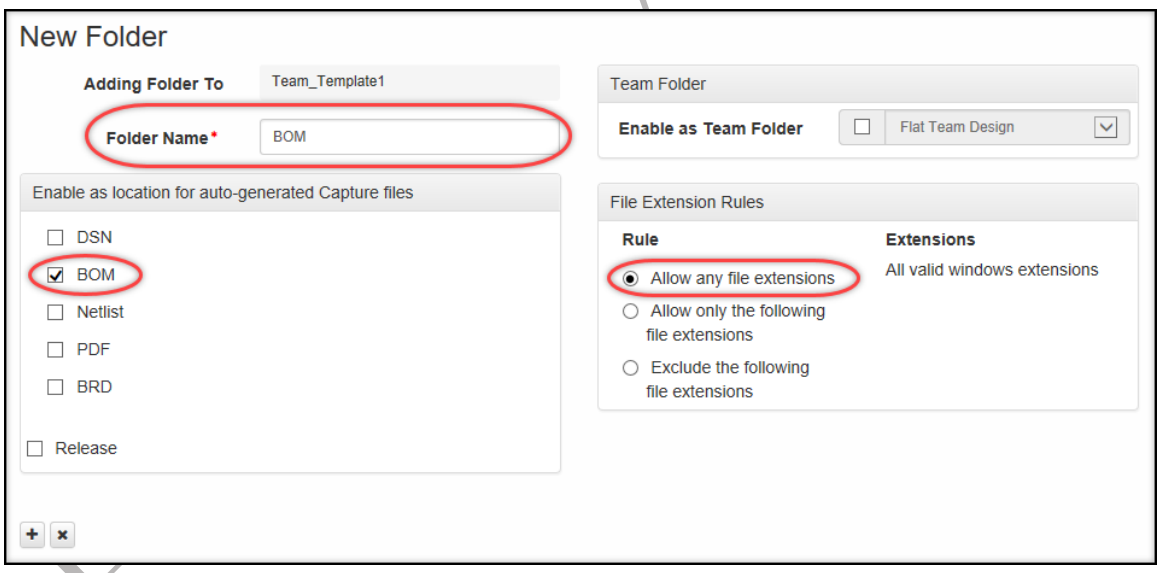
5. In the New Folder window, make the Folder Name **Schematic**, and Enable it as Team Folder. See the following graphic.

EMA Design

 EMA Design Automation, Inc ®© 2020 All rights reserved



6. Click the plus sign + to add the Schematic sub-folder.
7. At the top level of Team_Template1, click to add another sub-folder. Give the Folder Name as BOM and check the option for BOM (for auto-generating the file). Click the plus sign + to add the new folder.

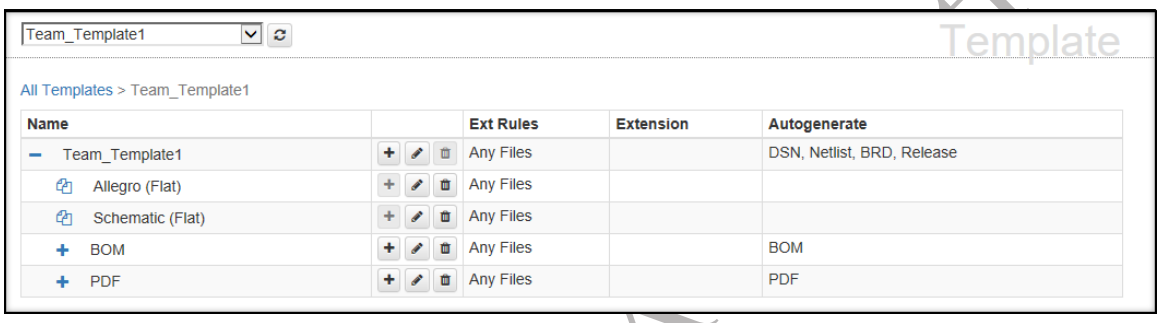


 EMA Design Automation, Inc ®© 2020 All rights reserved

Add PDF Sub-Folder

1. Click the plus sign + to add the Schematic sub-folder.
2. At the top level of Team_Template1, click to add another sub-folder. Give the Folder Name as PDF and check the option for PDF (for auto-generating the file). Click the plus sign + to add the new folder.

Your new template should resemble the following graphic.



Remember, the templates in this training guide are just samples. In your own work environment, you may have a more preferred way to build your templates.

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

Lesson 2: Working with Schematic Projects

In Lesson 2 you will be working in tasks that are typically assigned to Project Leaders, such as creating new projects and assigning users.

Lesson Objectives

After you complete this lesson you will be able to:

- Create a new project
- Assign users to a project
- Create a new project using team-design folders
- Assign users to a team design project
- Perform work and check work in

EDM Terminology

There are several terms within EDM that need to be defined before you begin working. It is important to conceptually understand them, but also because it will help clarify how certain facilities within EDM will behave. Some of them will be covered again during the appropriate lesson and lab, but let us begin by looking at a few of them now.

Team Design

The definition of team design within EDM refers to "*…the management of designs that require multiple engineers to work in a team environment. Team design functionality enables teams of engineers to work on a single design file, allowing for concurrent development on a single design*." (See Appendix A of the EDM User Guide).

If a project has been set up with non-team-design folders, it means that multiple members can be assigned to a project but ***only one assigned member can have it checked out for work at a time***.

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

This can apply to both **flat** schematic designs and **hierarchical** schematic designs. The common principle being that you can break out pages of the schematic and assign them to specific users. While only those users would be able to “check out” the pages for actual work-in-progress, the other team users would be able to download copies of those pages for reference only.

Later we will see how we can designate a project as a team design project.

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

EMA Design Automation

Lab 2-1: Creating a Capture Schematic Project

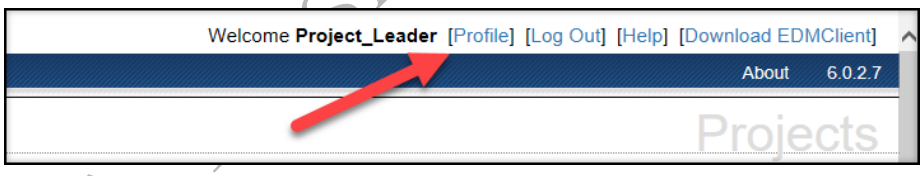
In the upcoming lab exercises you will create a *Capture_Design* project using a pre-generated template.

Creating a New Project

To perform any editing on schematic designs, the user who is logged in must identify a "Workspace", or directory location where the design gets checked-into and out of for editing purposes. The SVN Server ultimately manages the check-in and check-out of data and therefore requires this location to be set prior to performing any design work within EDM. You can see or set the path to your preferred workspace by selecting your User Profile once you log into EDM.

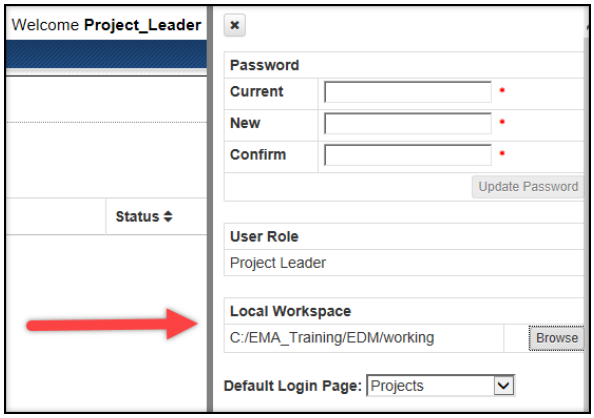
Setting Up the Profile

1. Click on the "**Profile**" link to check your local workspace for this training. See the following illustration.



NOTE: If you are taking the EMA classroom based EDM course using EMA laptop machines, or you are taking the EMA Virtual EDM training using EMA virtual training machines, the profile has already been set. If you are taking the EMA E-Learning EDM course you will need to set your own preferred workspace.

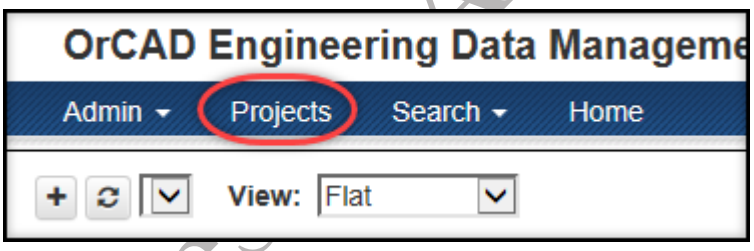
EMA Design Automation



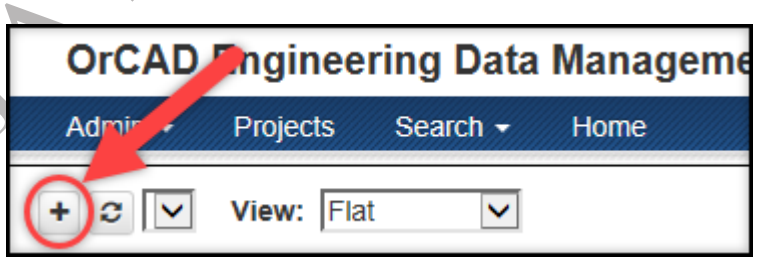
2. Close the Profile page.

Beginning the New Project

1. To begin a new project, click on Projects as in the following illustration



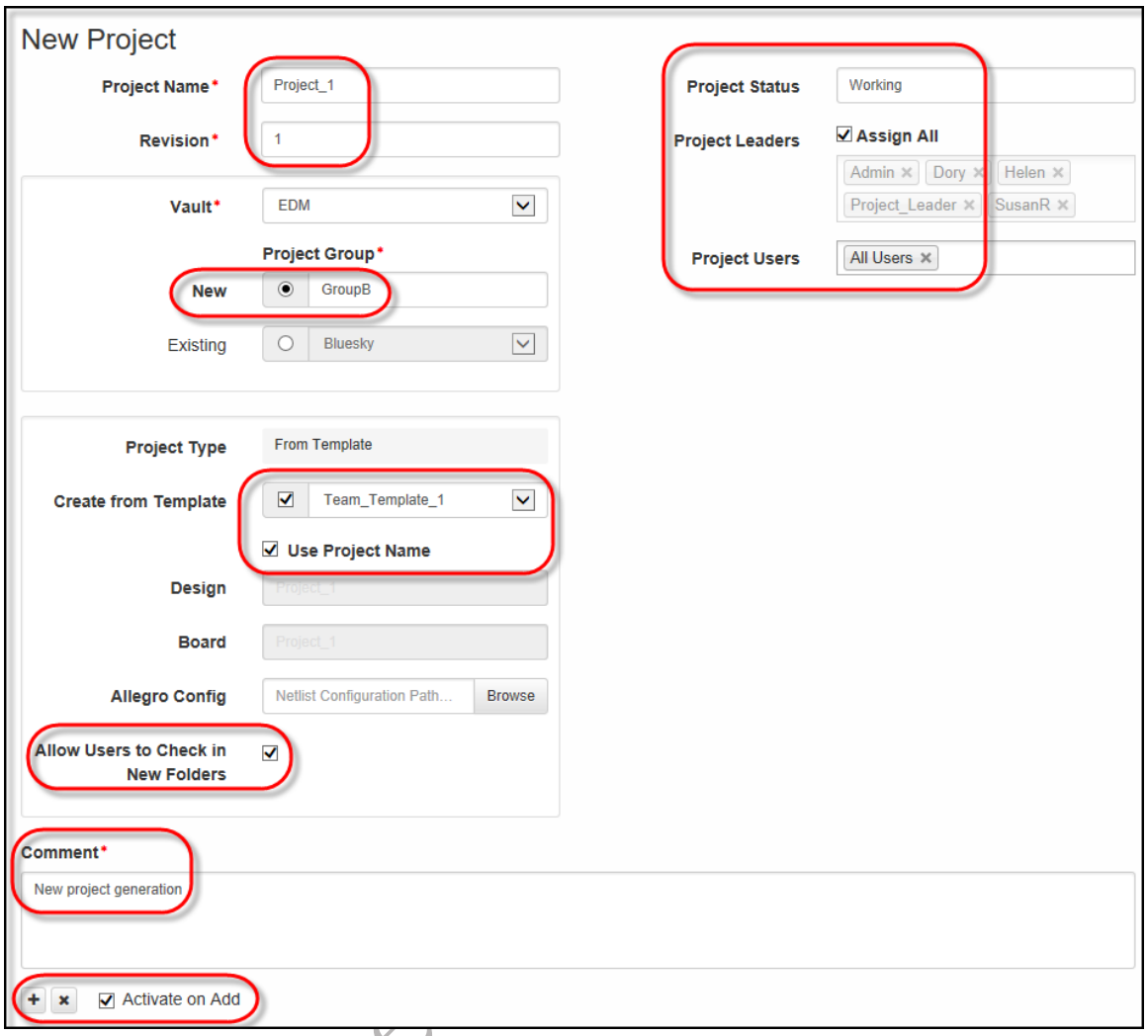
2. Click on the Add Project button to begin generating a new project as shown in the following illustration.



3. When the New Project page opens enter the settings as shown in the graphic below. If the named project already exists, assign the new project a unique name. If the Group name already exists you may create a new Group name (GroupB, GroupC, etc.)

EMA Design Automation

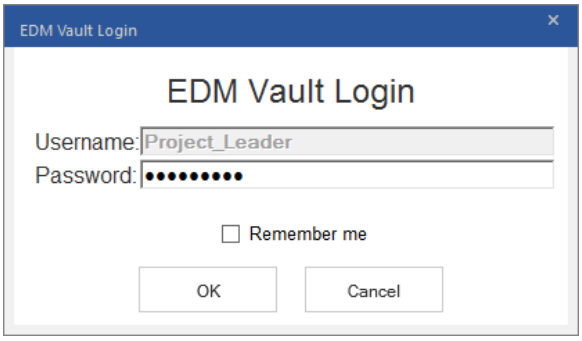
 EMA Design Automation, Inc ®© 2020 All rights reserved



4. Be sure and add the **Project Status** (Working); **Assign All Project Leaders**; and **Assign Project Users**.
5. Add a **Comment** and choose to **Activate on Add**
6. After all the information has been added click the plus + sign to add the project.
7. You may be asked to login to the EDM Vault. Be sure the Username is set to the login you are using (**Project_Leader**). Enter the password for the Project_Leader (**Pleader_01**). Click the "Remember me" box and click OK.

EMA De

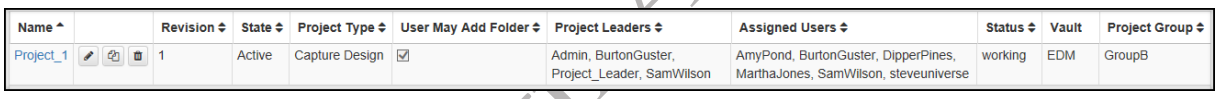
 EMA Design Automation, Inc ®© 2020 All rights reserved



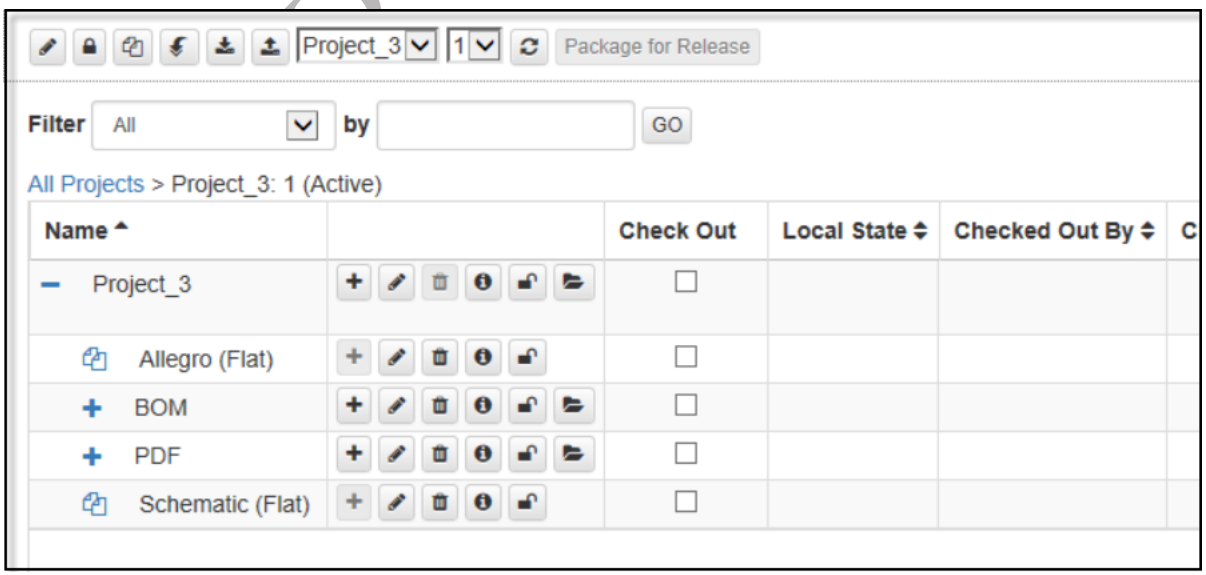
For this training there are multiple workspaces due to the multiple people using the student accounts. In your own local environment you would normally just have your own local workspace.

Review Settings

Once the project has been added review the settings in the Project Dashboard.



1. Click on the project name that you just created to open it. The new project should look like the following graphic. You may need to click the plus signs to expand the Allegro and Schematic folders.



Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

Lab 2-2: Adding a Schematic Design to the Project

Ways to Add a Design to a Project

There are essentially two ways to "add" a schematic design to a project.

- **Generating a New Design**
  - Create the new project
  - Do a Full project Download which will generate a new design named after the project
- **Import an Existing Design**
  - Use the Import Design facility to locate and import the desired design file

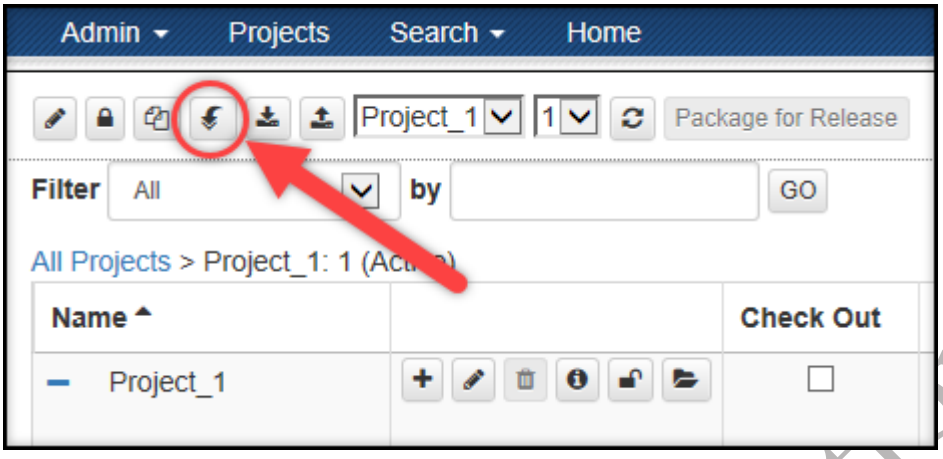
For this lab we will be using the method for creating a new design within the project. Following the steps you will create a new design associated with the newly generated project.

Using Full Project Download

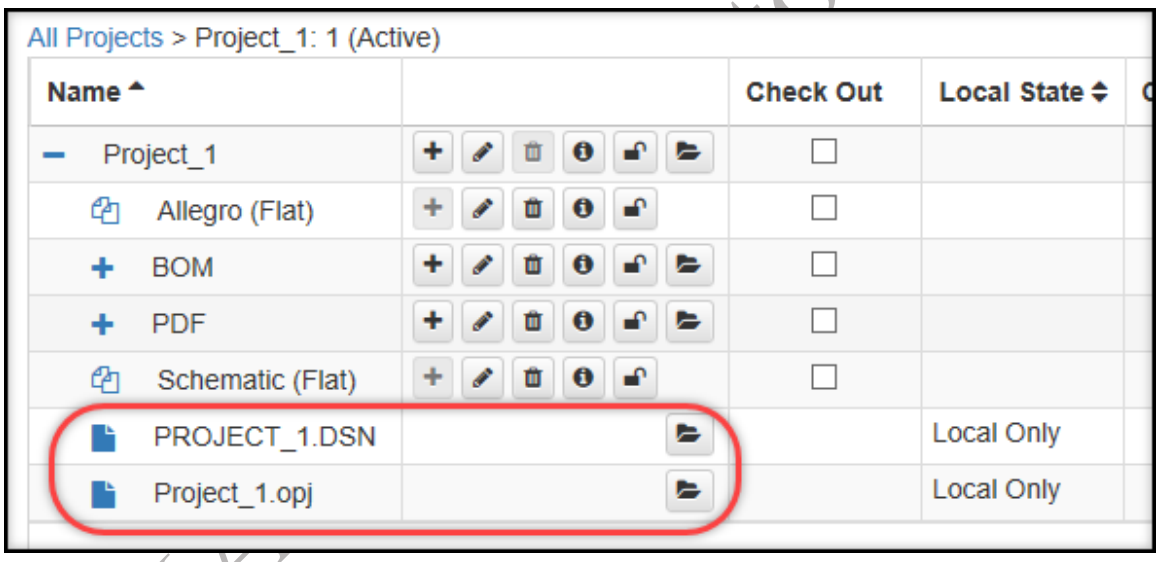
Although the project structure has been generated based on a template, it still needs a schematic design in order to perform work. In the next steps you will need to add a design for this project.

1. Select the project by clicking on the project name.
2. Next, click on the **Full Project Download** button as shown in the following graphic.

EMA Design Automation



Notice that a design has been generated and it is named after the project name.



If you do not see the Project-1.DSN or the Project_1.opj make sure to scroll to the bottom of the page.

EMA Design Automation

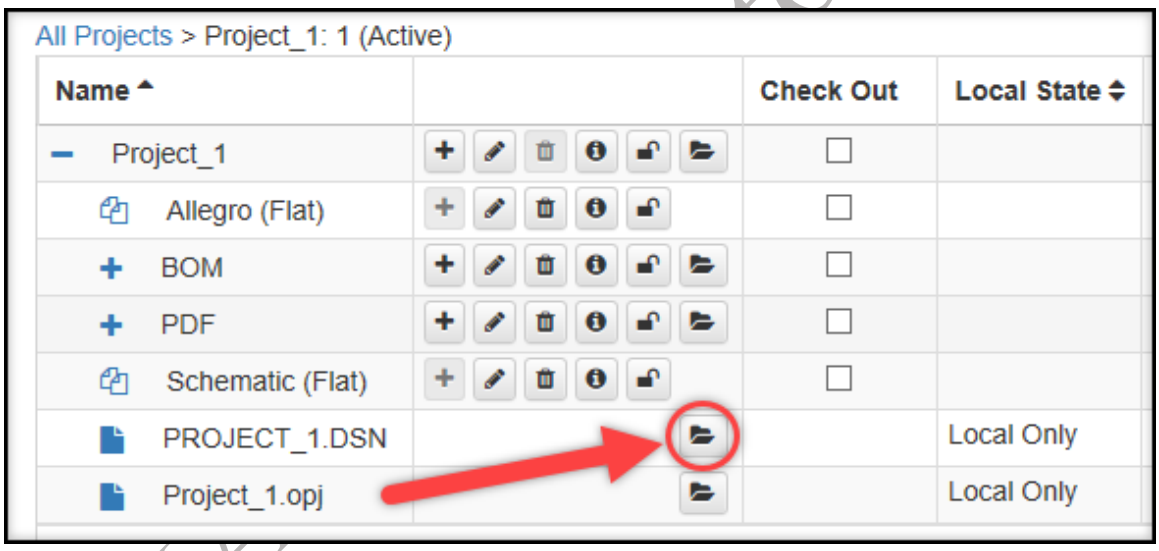
 EMA Design Automation, Inc ®© 2020 All rights reserved

Lab 2-3: Performing Work on the Schematic Project

Opening the New Design

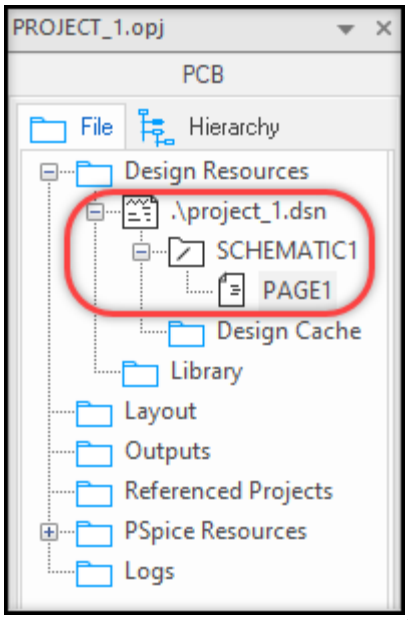
To have the new design become active, you will need to open it in Capture and save the design. This activates the versioning for files that would normally be checked in and out during routine changes.

1. Open the design by clicking on the Open Folder button next to the PROJECT_1.DSN, as shown in the graphic below.



Notice that Capture is launched, the new project has been created, and a new schematic page has been added.

EMA Automation

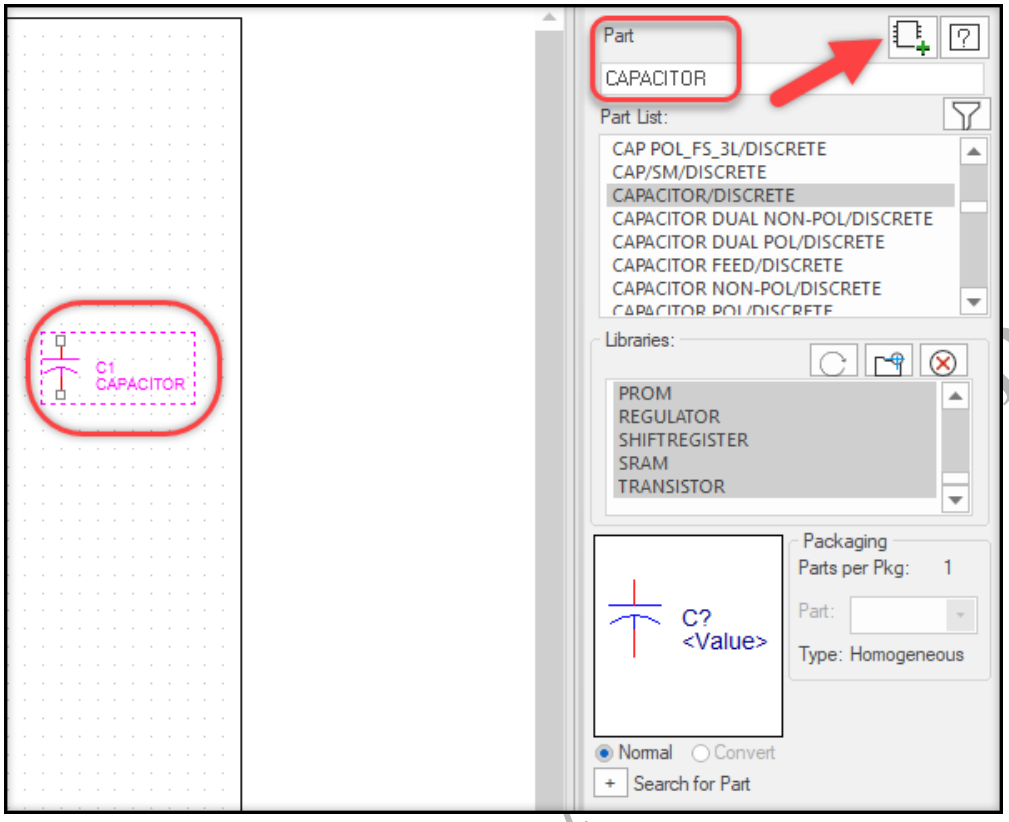


Placing a Part on the Schematic Page

1. Open the schematic page, then open the **Place > Part** menu and place a part on the schematic.

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved



NOTE: If you are using an OrCAD Capture CIS license you could also place a part from the database by choosing **Place > Database** Part.

Save and Check In

1. Save and Close the new design project.

The design will close, and you will be redirected back to the EDM project dashboard.

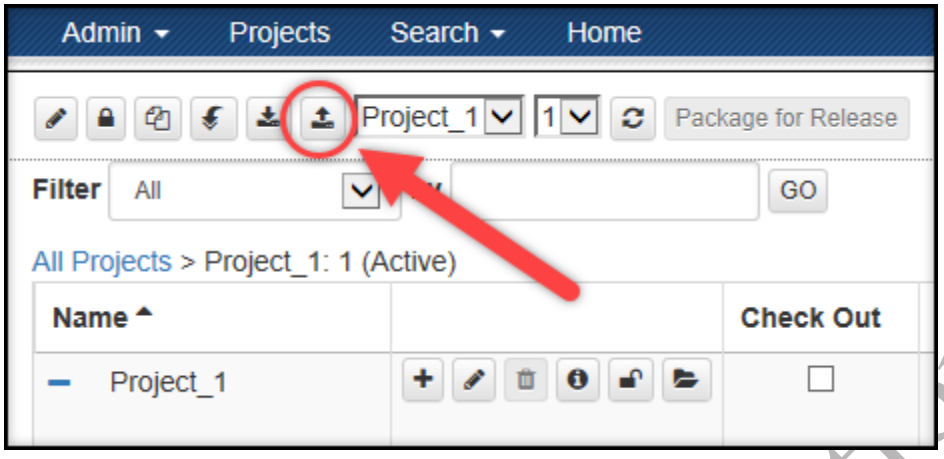
Checking the Design In

Now, you may begin the Check In process so that the design will be revision managed.

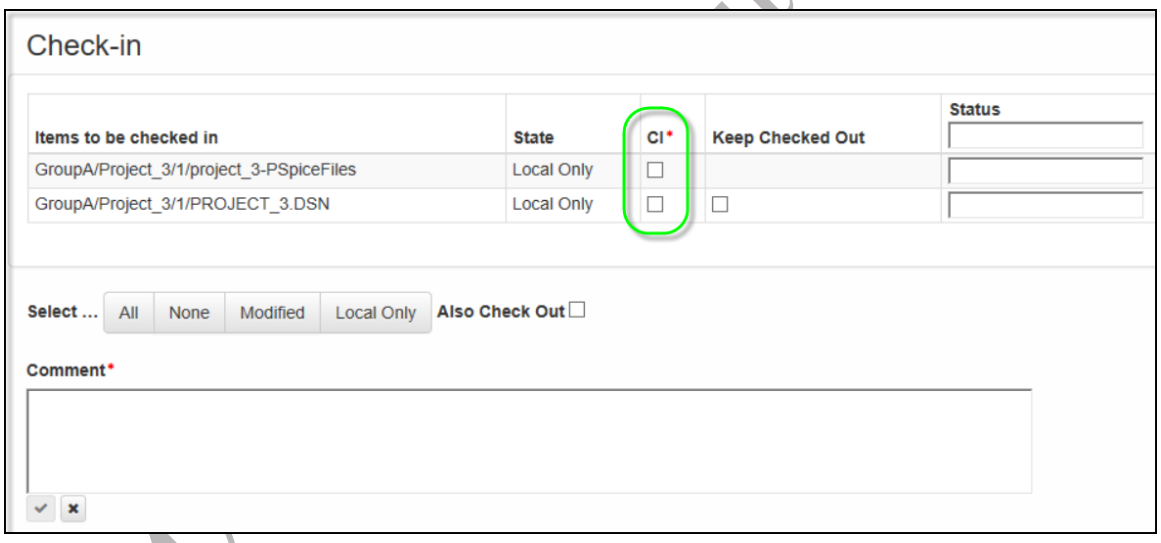
1. Click on the Check In button to begin the check in process.

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved



Since this is a new, ***unversioned*** project you will need to fill in the **Items to be Checked In** form. See the graphic below.



The Check In form only shows the list of files that can be checked in. This includes new files (shown as unversioned) and checked out files that are modified. Checked out files that were modified are pre-selected for Check In (CI*). If there are new or modified files that you do not want to check into the vault, do not select those items. You will need to complete this form before check-in can proceed.

In the **Select** area notice there are several buttons. The options are as follows:

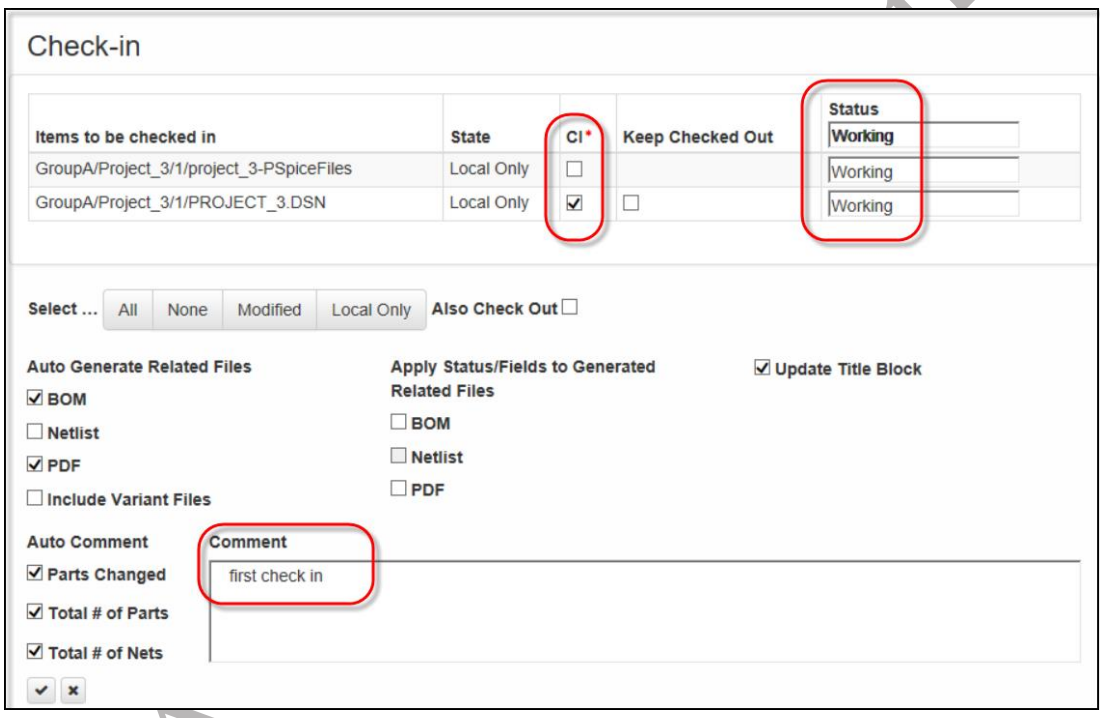
- **All** – Select files in the list for check in
- **None** – Deselect all files on the list for check in

EMA Design Automation, Inc ®© 2020 All rights reserved

- **Modified** – select only the files shown as modified under the State column
- **Unversioned** – Select only the files shown as unversioned under the State column

For our first project **we only have one file that we are checking in** (the design file).

2. Click to select the **CI*** (Check In) button. Notice the form expands to include more items.

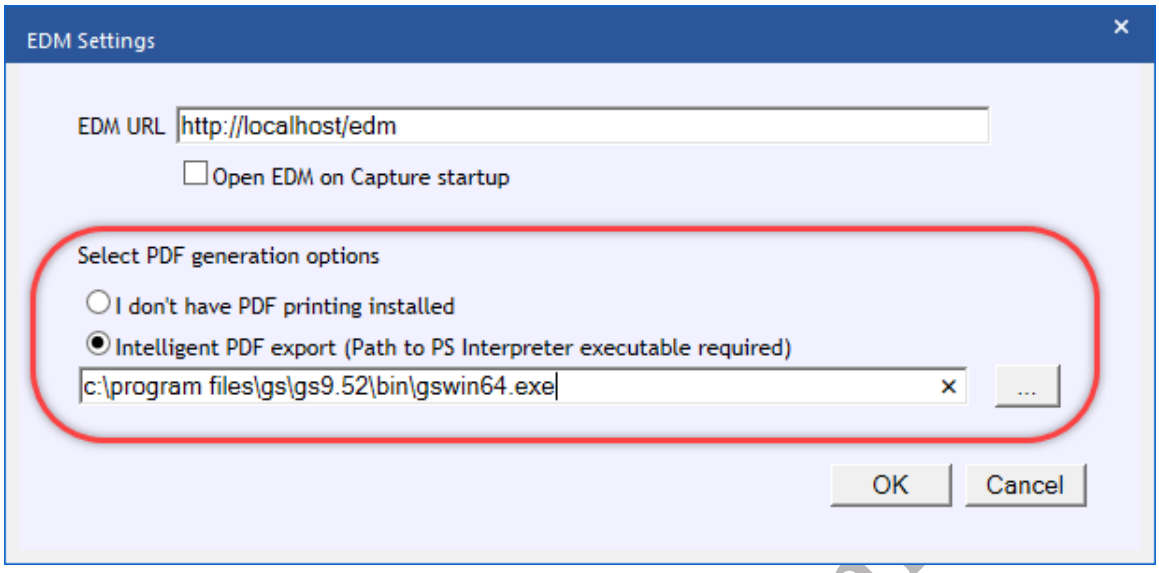


3. Fill the **Status** as: Working (as shown in the above graphic)
4. Add a comment (required)
   - Notice that other items have been defaulted to be "Auto-Generated" such as the BOM, PDF, and also the Auto Comment area (bottom left area)
5. Select the **Auto Generate Related Files** for the BOM and the PDF
6. Select the **Auto Comment** options for ***Parts Changed***, ***Total # of Parts***, and ***Total # of Nets***
7. Click on the check button (bottom left) to check the files back in.

NOTE: You must have the Intelligent PDF export path setup to generate a PDF document. This can be found in the **EDM > Settings** menu.

EMA

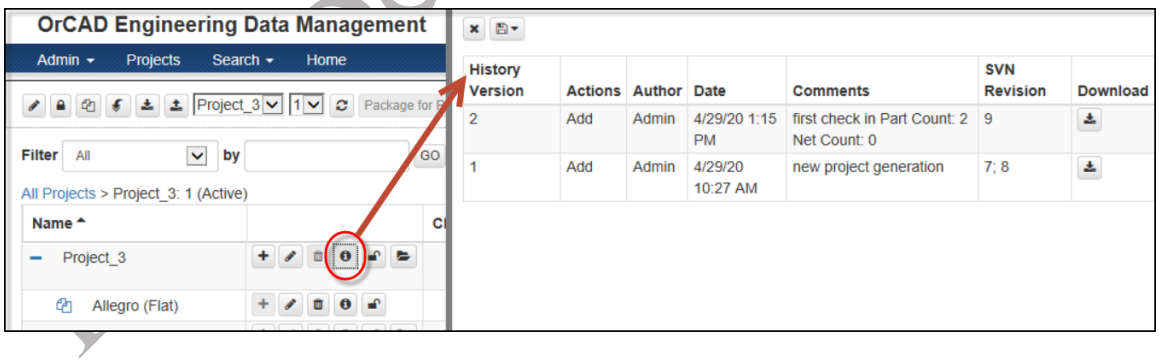
 EMA Design Automation, Inc ®© 2020 All rights reserved



Checking Project History

Once the files have been versioned and checked in you can access the history to see how the project has progressed. Use this option to view history of the project.

1. Click on the **History** button as shown below. The button looks like a small in a circle. Choosing the history button at the **Project_1** level will show the entire project history. See the graphic below:



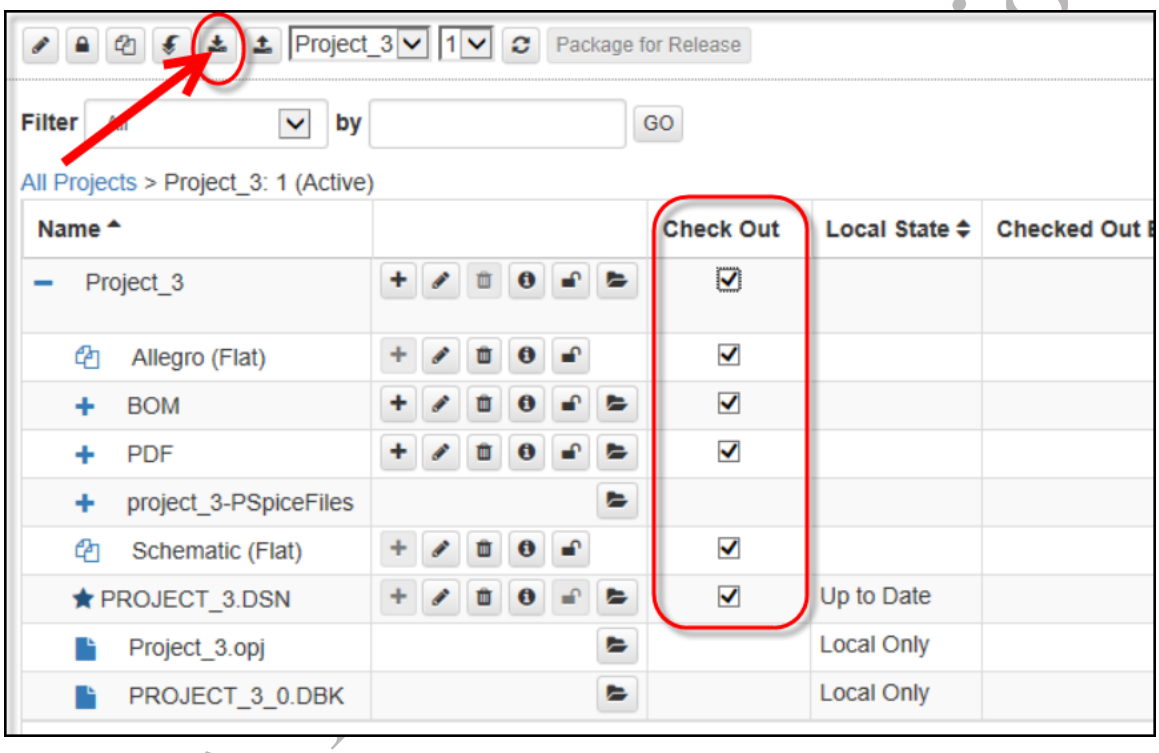
Previous revisions may be downloaded. History may be exported to an Excel file or to a CSV file. After reviewing the project history, click the "X" in the top left corner to close the History form.

EMA Design Automation, Inc ®© 2020 All rights reserved

Checking Files Out

Now that the files have been entered into the Vault system for version management you will need to select the files and check them out for work.

1. Check the **Check Out** boxes next to the Project (name). Selecting this box at the top level will auto select all the items for checkout.
2. Next click on the **Check Out** button above the project name.

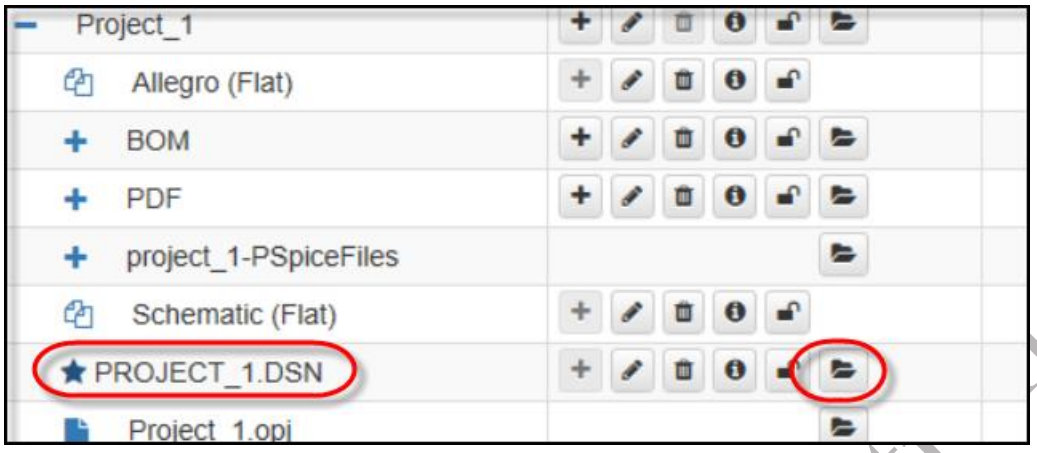


Notice that, once the items have been checked out, the **Checked Out Date** column will indicate the time and date stamp.

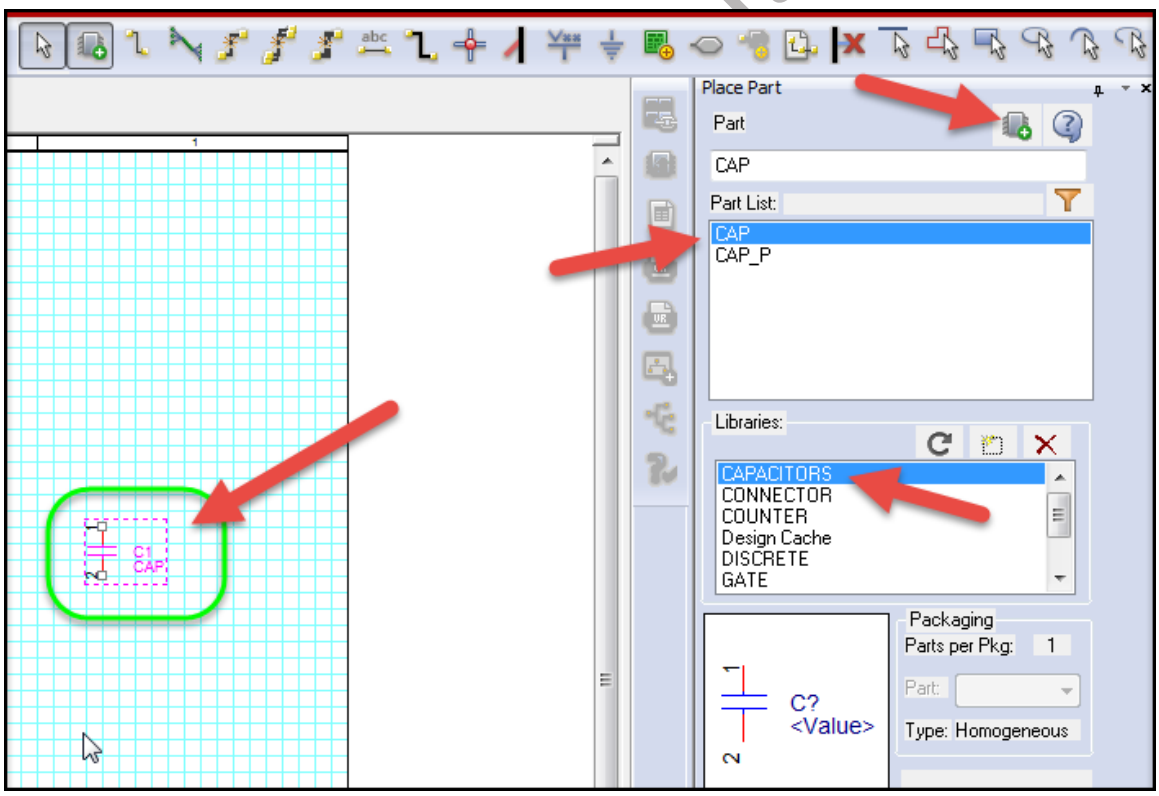
Open Design in Capture & Place Part

1. Next, click on the **Open Folder** button next to the **PROJECT_(name).DSN** name to open the design in Capture.

EMA Design Automation, Inc ®© 2020 All rights reserved

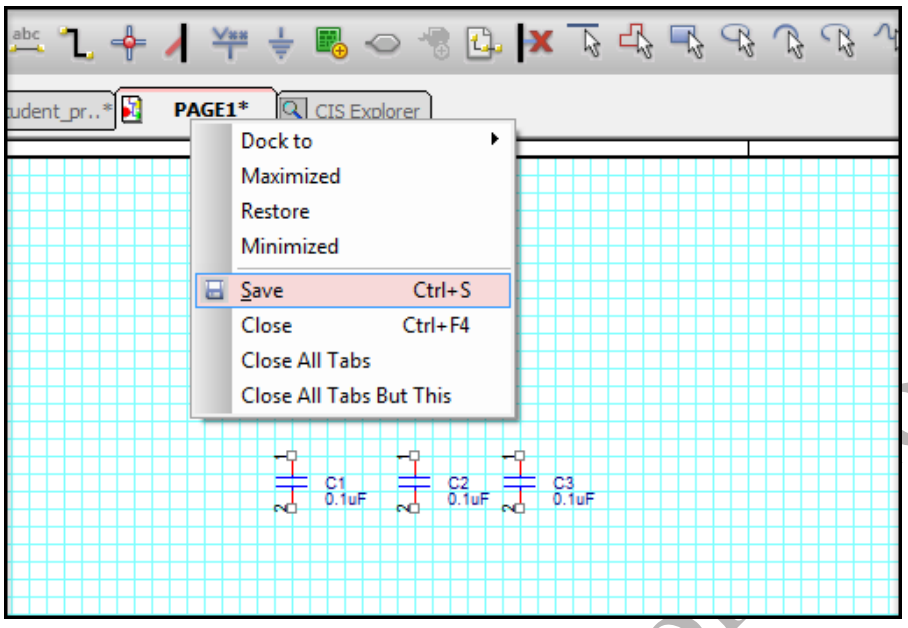


2. When Capture CIS opens, expand the schematic folder and open PAGE1 of the schematic.
3. Choose **Place > Part** (or if you are using an OrCAD Capture CIS license you can choose **Place > Database Part**), find any part and place it on the schematic page.



4. Right click on the PAGE1 tab and choose Save.

EMA Design Automation, Inc ®© 2020 All rights reserved

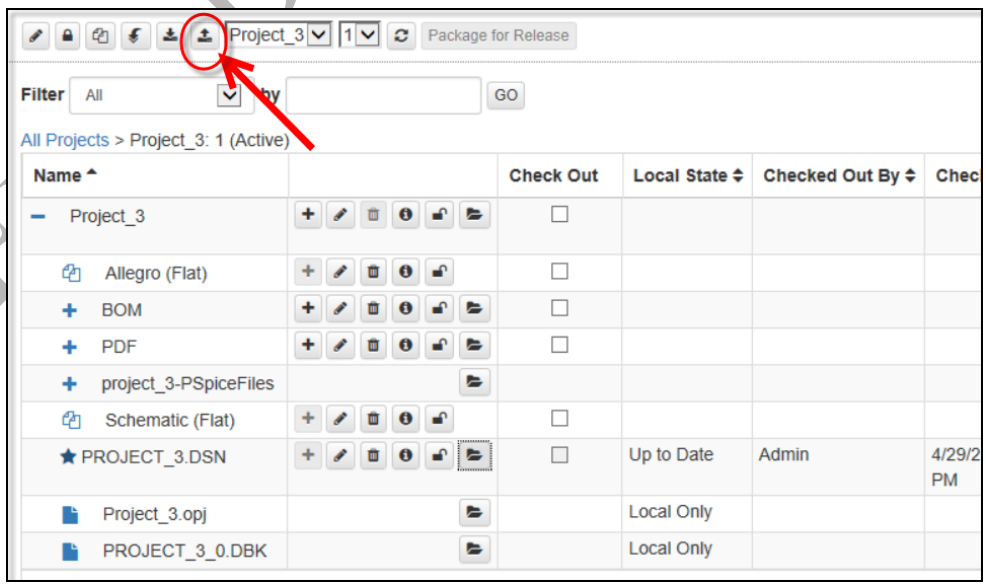


5. In the Project Manager select the design and save it. Then close the project.

Checking In the Design After Editing

After the design has been checked out and edited, the design may be saved and checked back into EDM.

1. In the EDM window select the **Check In** button to check the design back in.

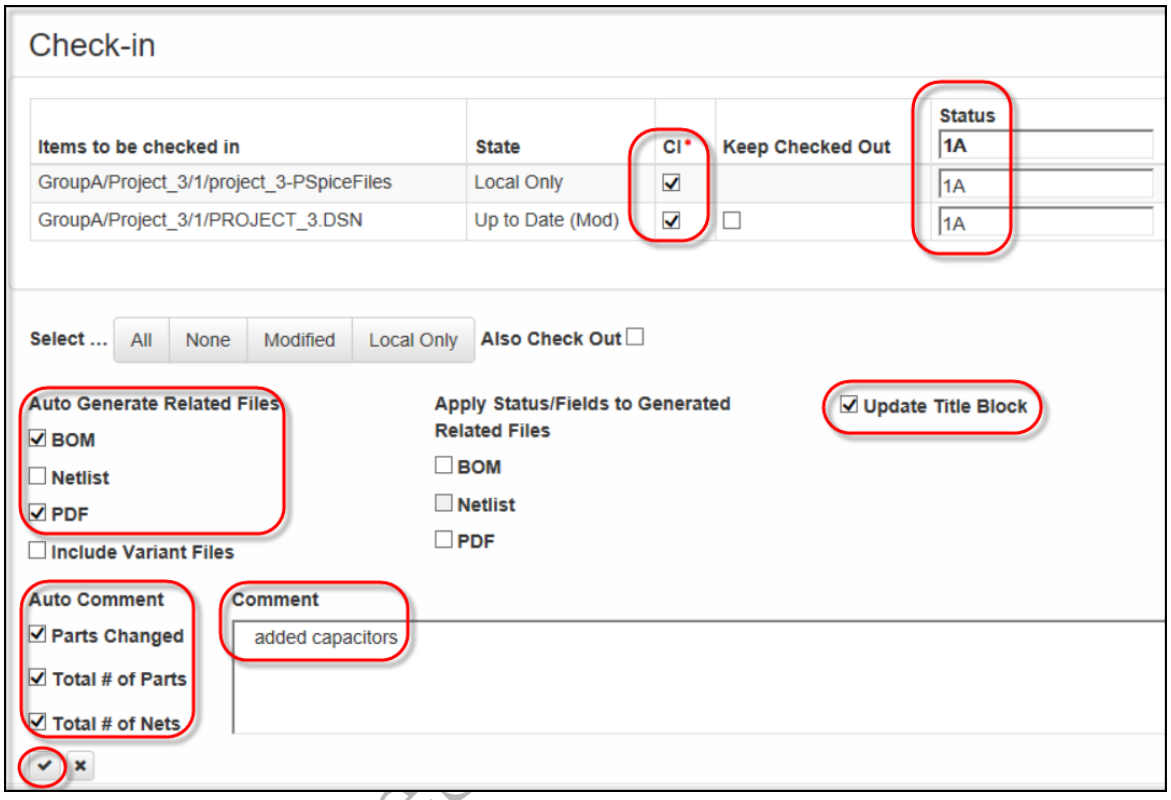


For Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

When checking the design back in you will be able to identify the design status as well as auto-generating the related files like PDF, etc.

2. Fill out the check in dialogue as represented in the following graphic.



3. Once the information is filled out click the check button at the bottom of the page to complete check in.

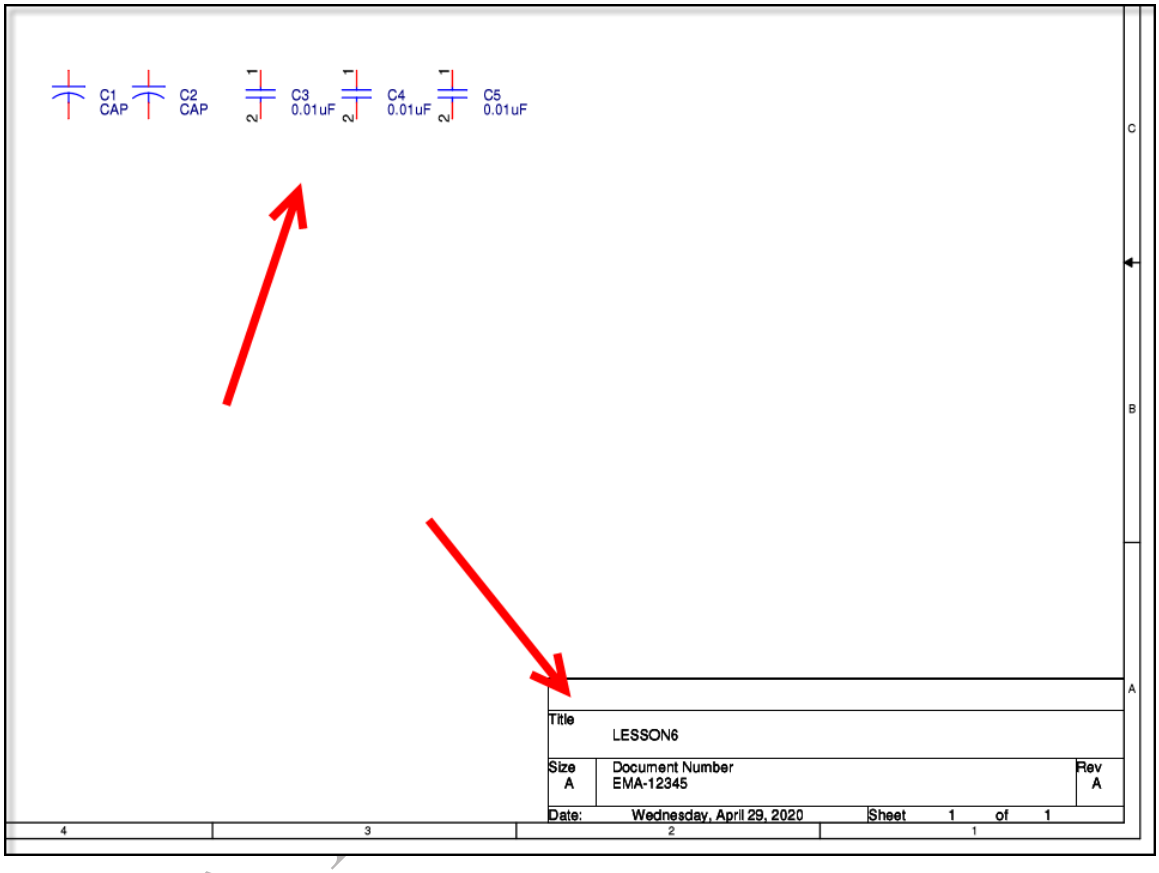
Viewing the Auto-generated PDF

1. Expand the PDF folder and click on the Open Folder button to open the auto-generated PDF document as shown in the following illustration.

EMA Design

 EMA Design Automation, Inc ®© 2020 All rights reserved

Notice an auto generated PDF opens and you can see the components that were added previously to PAGE1. Since you chose to update the title block upon check-in observe that the schematic name and status have been updated as well.



EMA

 EMA Design Automation, Inc ®© 2020 All rights reserved

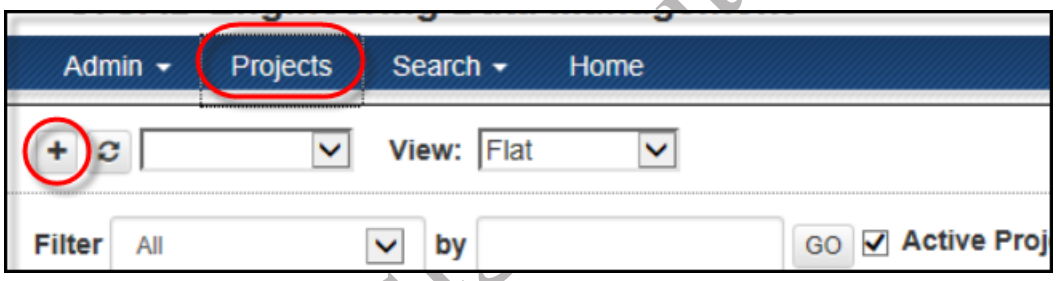
Lab 2-4: Creating a New Project

For this exercise you will create a new project for a team design. This is like the previous exercise except you will make the schematic pages available to other team members. Additionally, you will be importing an existing schematic design into this project instead of creating a new schematic.

Remember, this task is only available to Project Leaders (and Admins) but not to General Users.

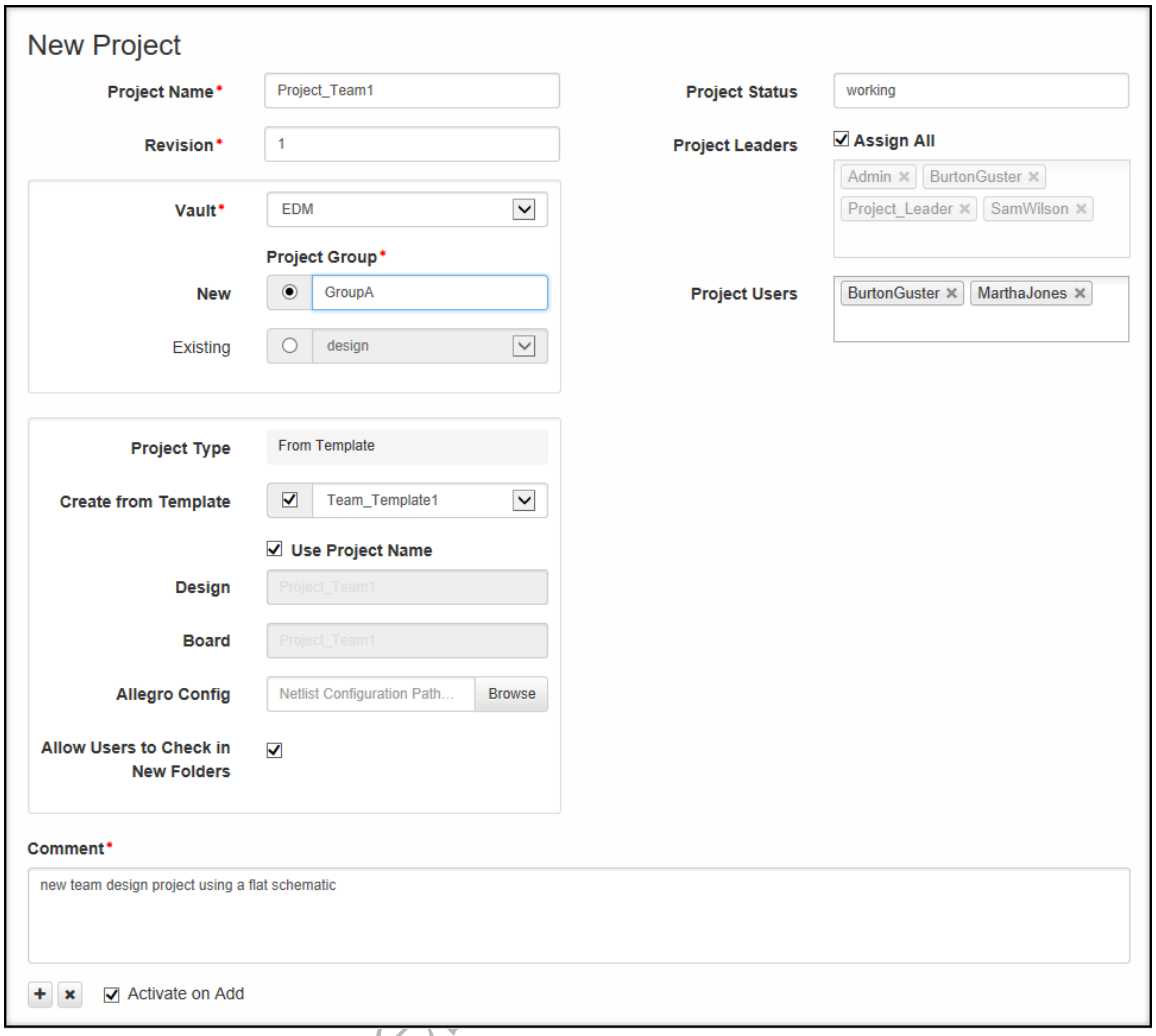
Create the New Project

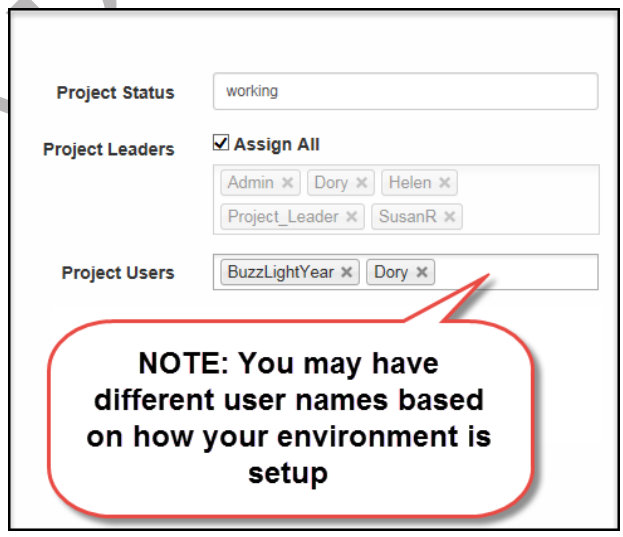
1. Click on the Projects menu, then click on the plus + sign to begin a new project.



2. Fill the project information out as in the following list.
    - **Project Name:** Project_Team1 (or another name if this already exists)
    - **Revision:** 1
    - **Vault:** EDM
    - **Project Group**: GroupA (or whatever group is available)
    - **Project Type**: From Template
    - **Create From Template**: check, use Team_Template1
    - **Allow Users to Check In New Folders**: check
    - **Project Status**: Working
    - **Project Leaders**: Choose Assign All
    - **Project Users**: select 2 from the list
    - **Comment**: “new team design project using a flat schematic”
    - **Activate on Add**: check

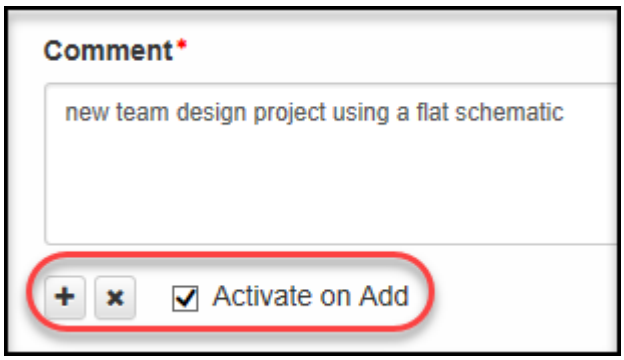
EMA Design Automation



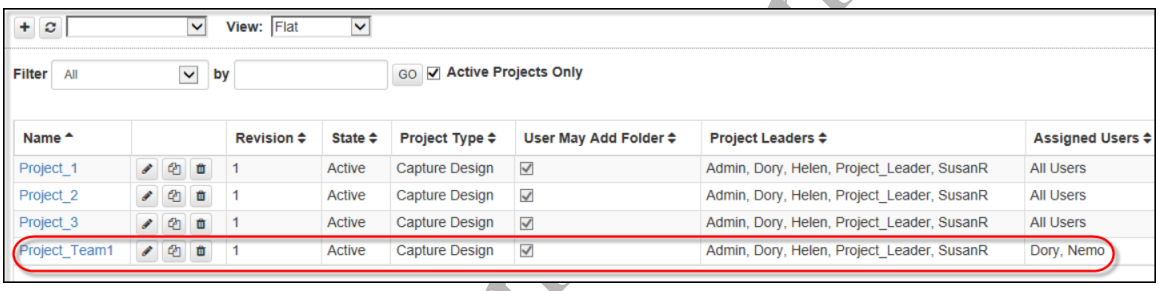


EMA Design

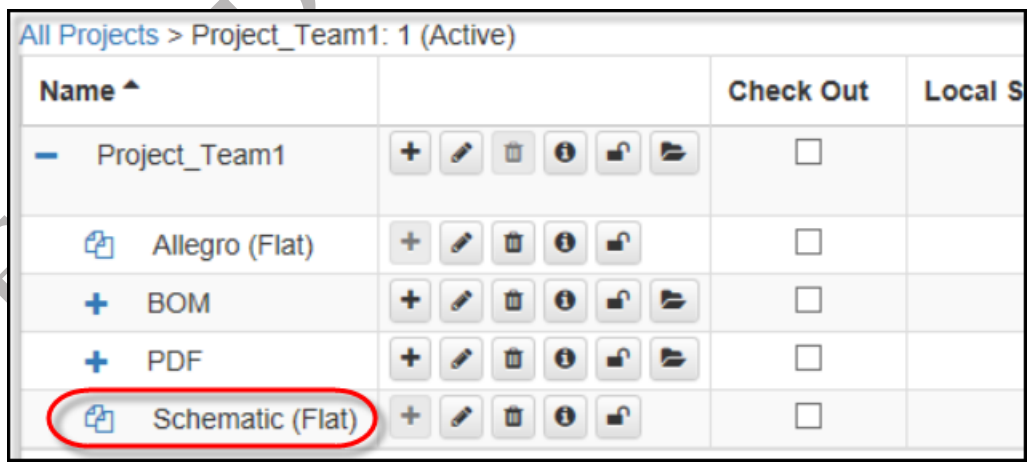
 EMA Design Automation, Inc ®© 2020 All rights reserved



3. Once the information has been added click the plus sign to generate the project.
4. From the dashboard, click on the project name to open the project. See the following illustration. (Note the project name in the illustration may be different than your project but the structure should be the same).



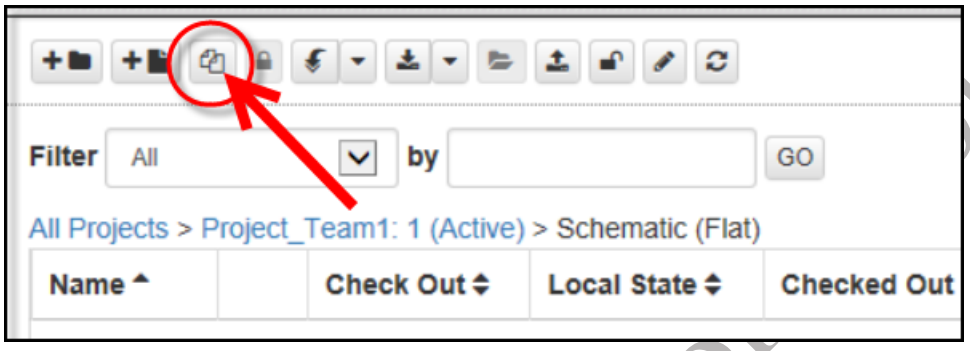
5. Select the new project to open it.
6. Click on the schematic to open as shown in the following graphic. This will open a menu of options so you can import the desired schematic.



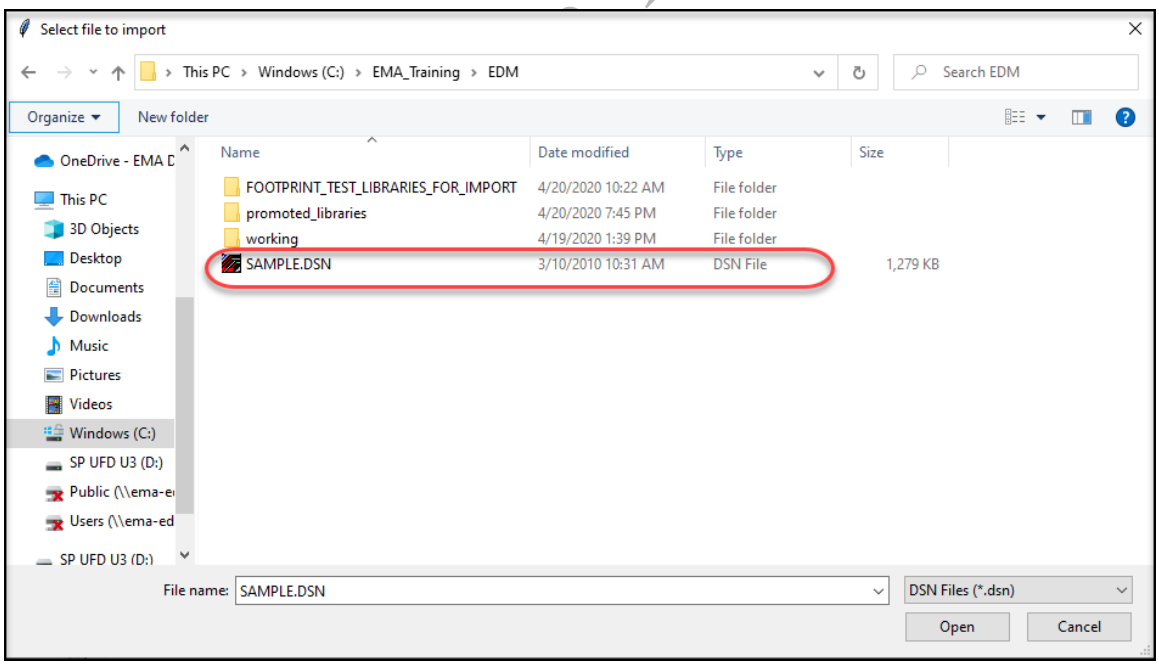
 EMA Design Automation, Inc ®© 2020 All rights reserved

Importing the Design

1. Once the menu options appear for Project_Team1, click on the **Import** button as shown below.

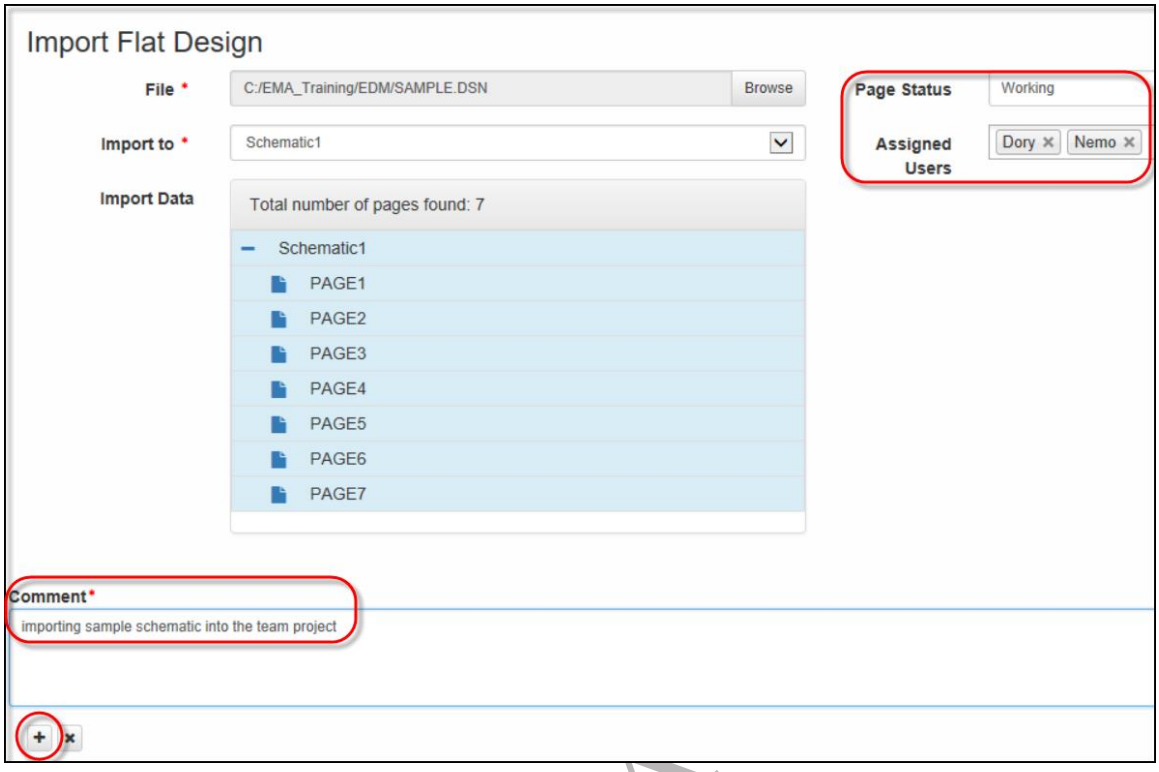


2. The ***Select File to Import*** window will appear. Navigate to the location where you have stored your training files. On the EMA training machines this is: C:\EMA_Training\OrCAD_EDM_17.4\SAMPLE_DSN and select the **Sample.dsn** file.



3. Next you will need to fill out the ***Import Flat Design*** window. Fill it out as shown below. (Note your users may be different).

 EMA Design Automation, Inc ®© 2020 All rights reserved



4. Add a Comment and click on the plus **+** sign (bottom left of the window) to import the schematic.

In the next lab you will work with locking the Reference Designators. This is a typical task that a Project Leader would do as part of preparing a design for team use.

EMA Design A

 EMA Design Automation, Inc ®© 2020 All rights reserved

Lab 2-5: Applying Lock Ref Des

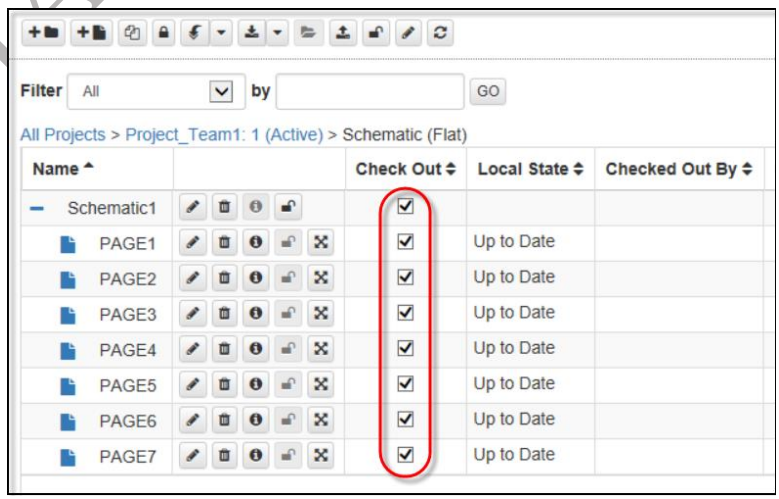
With multiple users assigned to work on schematic pages of the same design, duplication of reference designators must be avoided. EDM can aid in the management of reference designators ensuring uniqueness as well as allowing the project manager to restore saved references. This step would most often be performed in advance by the Project Leader who has generated the project, before making it accessible to the team members assigned to the project.

In the next lab you will be locking the reference designators on the schematic. 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 project leader manage references across all the pages of a design. If 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.

Preparing the Project for Multiple Users

In this step you will be checking out the design (as Project Leader) and then applying **Lock Ref Des**.

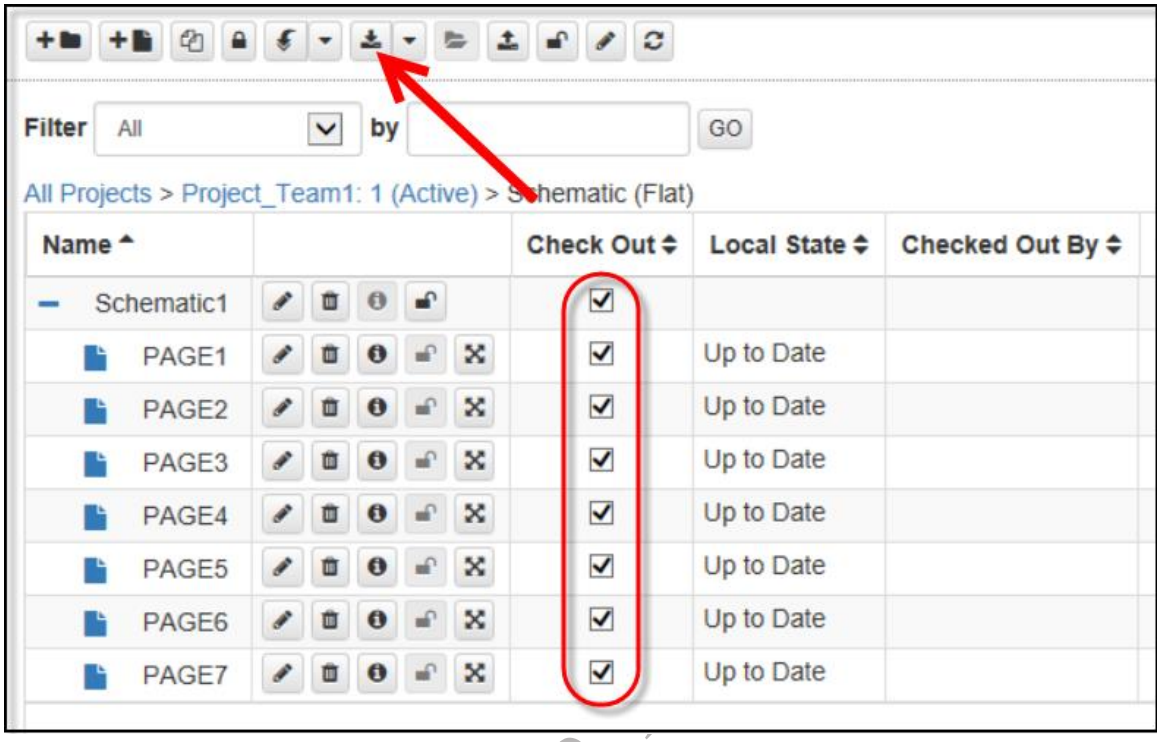
1. With the project still open at the schematic level from the previous exercise, select the **Check Out** button to begin checking out. See the following illustration for reference.



EMA Design Automation

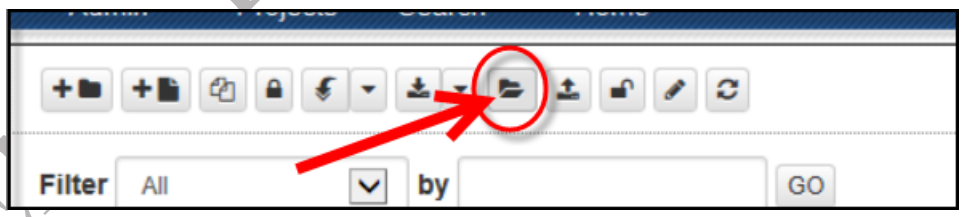
 EMA Design Automation, Inc ®© 2020 All rights reserved

2. Next, click on the Check Out option at the top menu as shown below.



Notice that all the pages have been checked out (note the date/time stamp) and the user who checked them out. Now you are ready to open the design.

3. Click on the **Open Design** folder from the menu icons across the top of the project, as shown below.

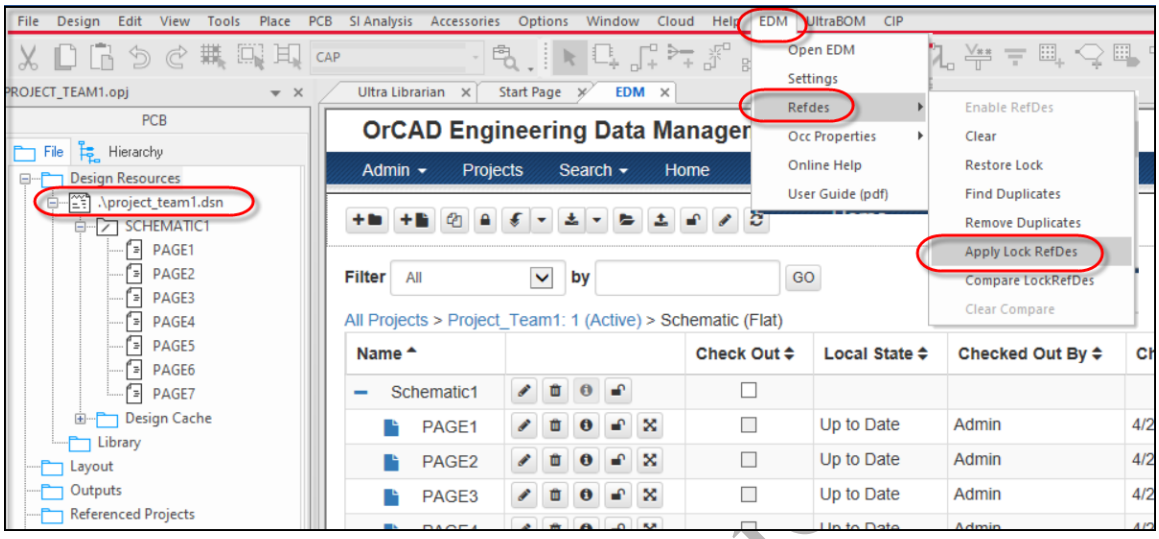


The primary reason for checking out all the pages in this exercise is so you can set **LockRefDes** for all of the schematic pages. **LockRefDes** is explained further on in this document. More information on handling reference designators can be found in the EDM User Guide. As a Project Leader, you will need to set **LockRefDes** prior to allowing any user to perform edits on the design. This will assist in making sure there are no duplicate reference designators introduced if users are working concurrently.

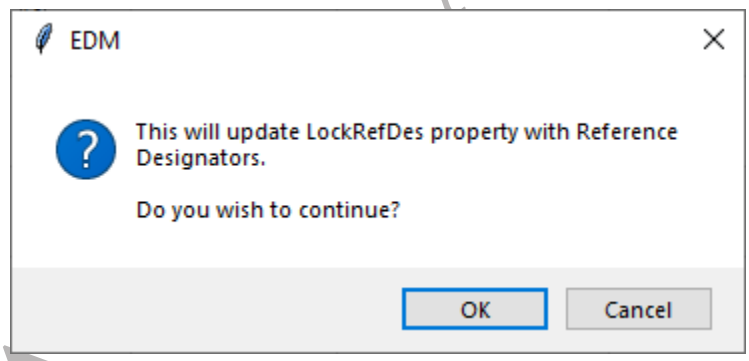
4. Once the design opens expand the schematic folder.

 EMA Design Automation, Inc ®© 2020 All rights reserved

5. Select the dsn in the Project Manager.
6. Then, choose **EDM > Refdes > Apply LockRefDes**



7. Click OK to the message "*This will update LockRefDes……*"



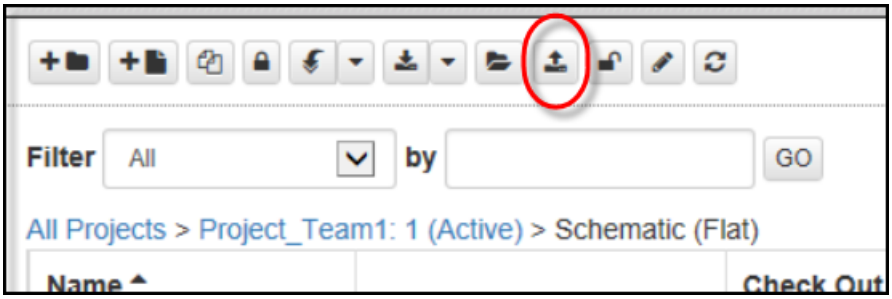
8. Save and Close the design.

Check In After Applying Lock Ref Des

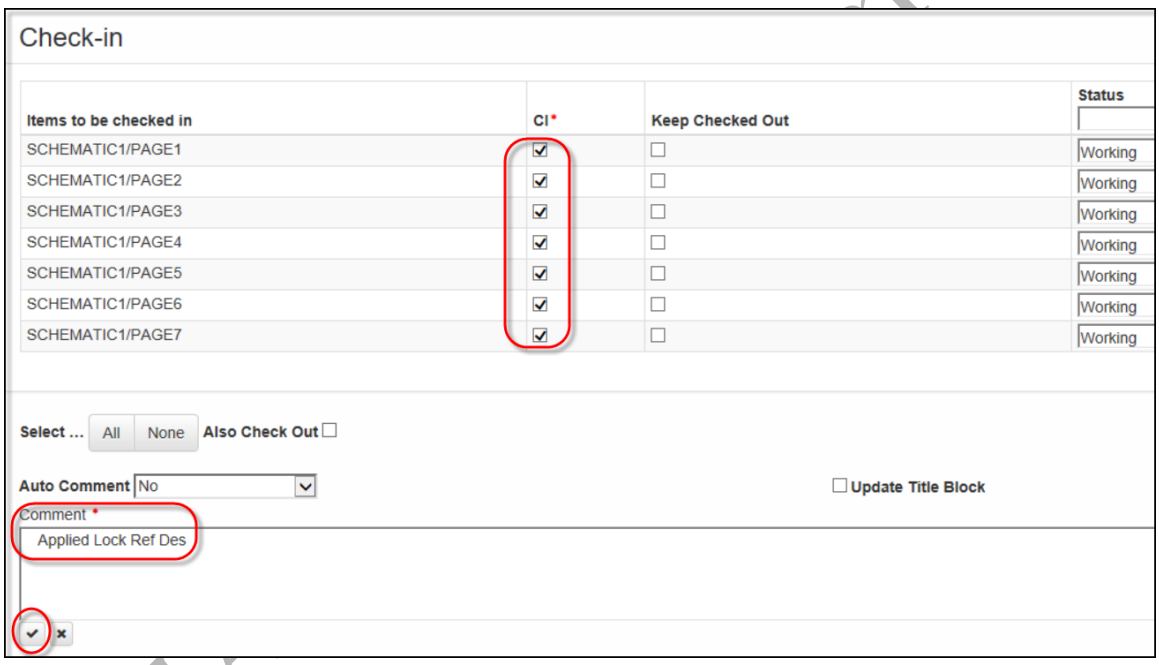
1. Next, select the **Check In** button to check the design back in. See the following illustration.

EMA Auto

 EMA Design Automation, Inc ®© 2020 All rights reserved



2. Be sure to mark the CI (Check In) column for each item. In the Comments area indicate that **Apply LockRefDes** was implemented. See the following illustration.
3. Then click the Check button to check in the design. Now the design is ready to assign the users.

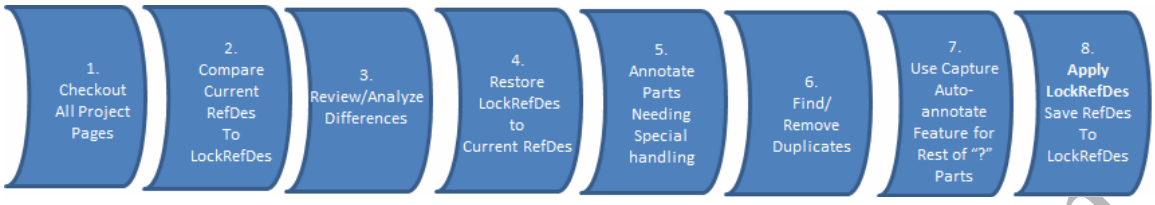


More About Locking Reference Designators

Before you perform any work it is important to understand that for some designs and hierarchical designs in particular if they are set up for team design work that the reference designators be locked. With multiple users working on various schematic pages duplication of reference designators may occur. EDM can aid in the management of reference designators ensuring uniqueness as well as allowing the project manager to

EMA Design Automation, Inc ®© 2020 All rights reserved

restore saved references. Typical steps for effective reference designator management are shown in the following figure.



An explanation of the above steps is as follows. For this training, we will only be setting Lock Ref Des to illustrate the primary step that a Project Leader would take to prepare the design. Review the items listed below to understand the remaining steps to be followed after the design has been completed.

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.

- 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** has not 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.
- Reviewing and understanding differences is important so that work for users who need certain references is not 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.
- 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 "?".
- After the Restore, you may hand edit parts that need specific references. If you do not have any, you can skip this step. You will need to remove all duplicates next. You can use Find Duplicates to identify all parts that have a corresponding duplicate. After finding duplicates, you may resolve this by hand editing them or use the Remove Duplicate command. The Remove Duplicate command will set all references with duplicate values to "?".

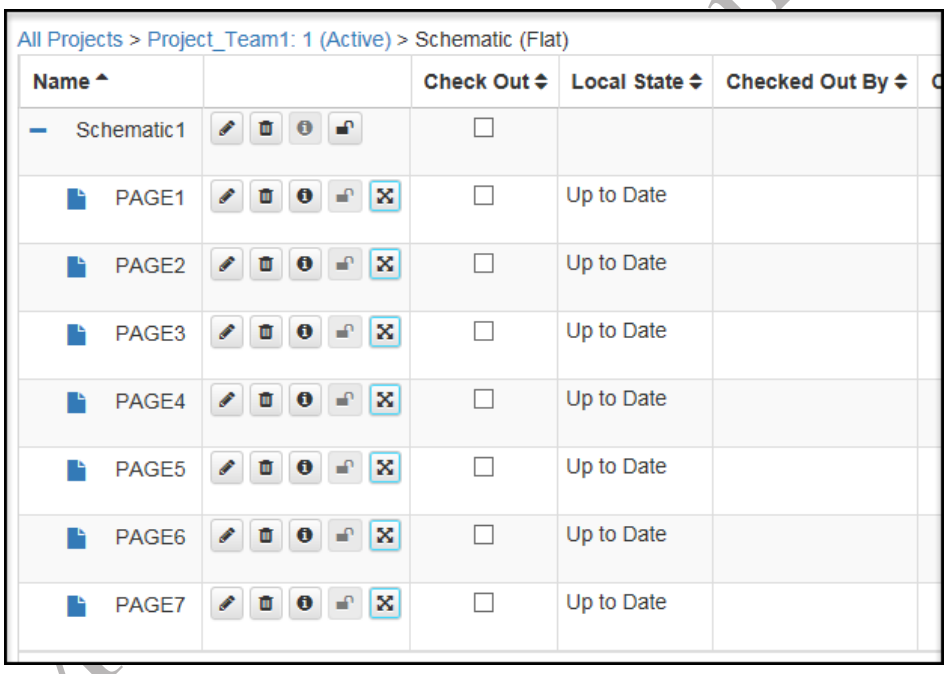
EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

- 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.
- After all duplicates have been removed, you can **Apply LockRefDes** to save the references.

Assigning Users to Schematic Pages

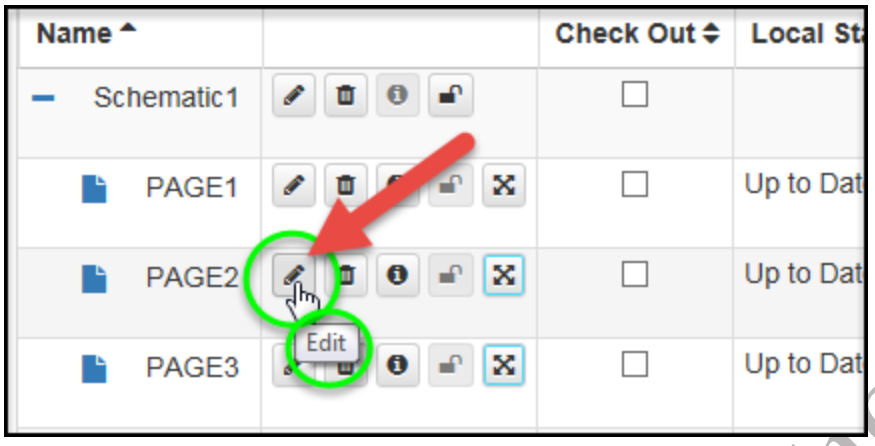
Once you have imported the schematic and, as Project Leader, locked the reference designators, you can now assign specific pages to users. Because you generated this as a "Team" project you can have multiple users working concurrently based on their page assignments.



In order to assign a user to a page you will edit the page from the list.

1. Next to the PAGE2 (just to the right), click on the pencil icon to edit the page as shown below.

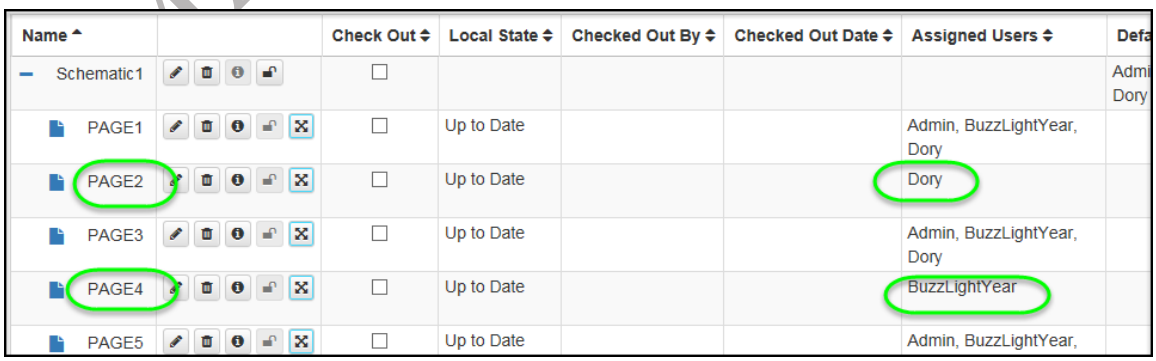
EMA Design Automation, Inc ®© 2020 All rights reserved



This will open the Edit Page window where you will assign users to the page.

2. In the Assigned Users area remove all but one of the names. An example is shown below. Then, click the check mark (bottom left) to finish editing the page.

3. Repeat steps 1 & 2 for PAGE4, using a different username. The result will look like the following graphic.



 EMA Design Automation, Inc ®© 2020 All rights reserved

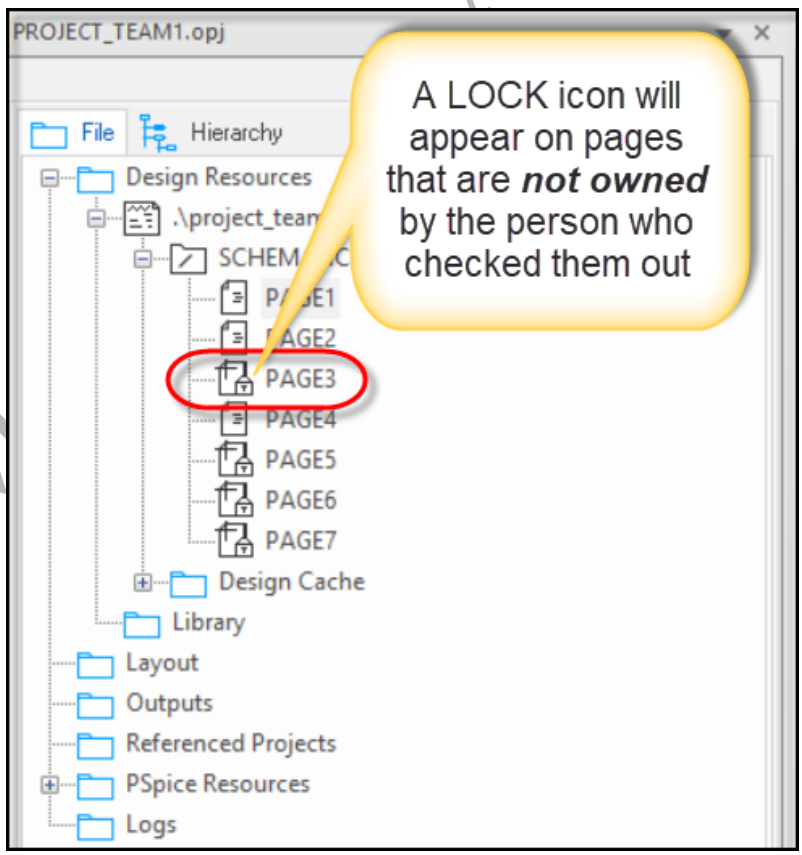
Viewing Team Design Pages in Capture

In this section we will show you how the schematic pages appear for someone who is not assigned to them for editing.

For example, in the project you just created there were a few users assigned to various pages. Any one of the assigned users can check out all the pages, but only the page(s) assigned to them for editing will be enabled for editing.

If one of the users is a Project Leader they will have access to *all* the pages for editing, but if a used is assigned as a General User, they will *only* have edit privileges for the pages they are assigned.

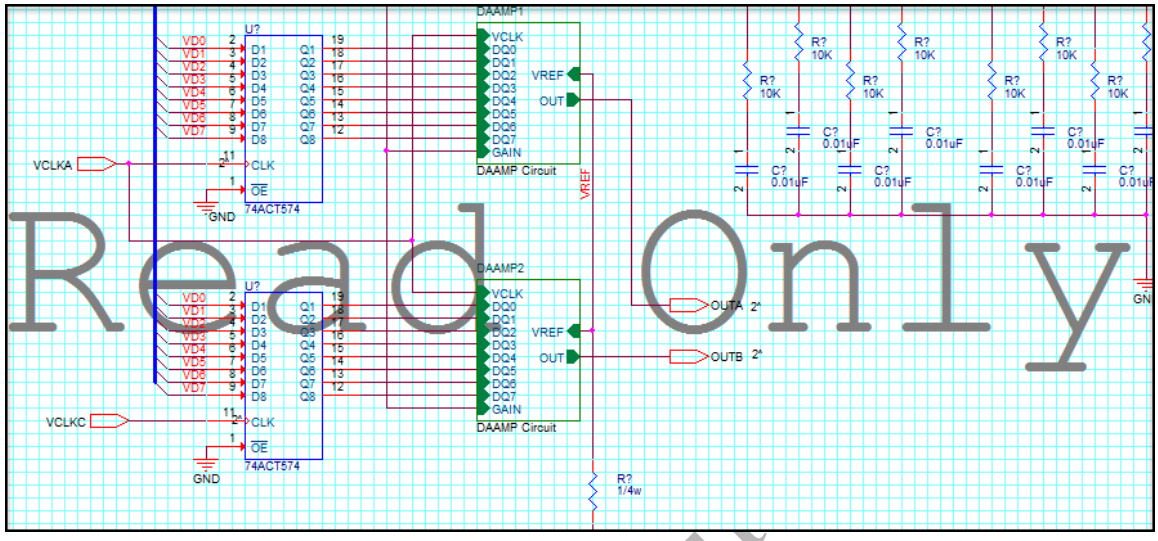
In the graphic below, the project pages have been checked out by a user, who was granted editing privileges to only some of the pages. Notice the "lock" on some of the pages in the Capture project manager.



EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

When editing a schematic, it may be necessary to have all the pages available as references. For the user who is locked out of certain pages, when they open pages they do not control they will find a READ ONLY watermark on the page, as shown below.



This is the end of Lesson 2.

EMA Design Au

 EMA Design Automation, Inc ®© 2020 All rights reserved

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

Lesson 3: Working with Schematic Symbol Libraries

Lesson Objectives

After you complete this lesson you will be able to:

- Create a new project for Capture symbol libraries
- Assign users to the library project
- Import a library into the project
- Perform work and check work in

It may be necessary to control schematic symbols that could be associated with a given project. This can be handled within EDM as a library project in almost the same way you can control schematics and pages in a project.

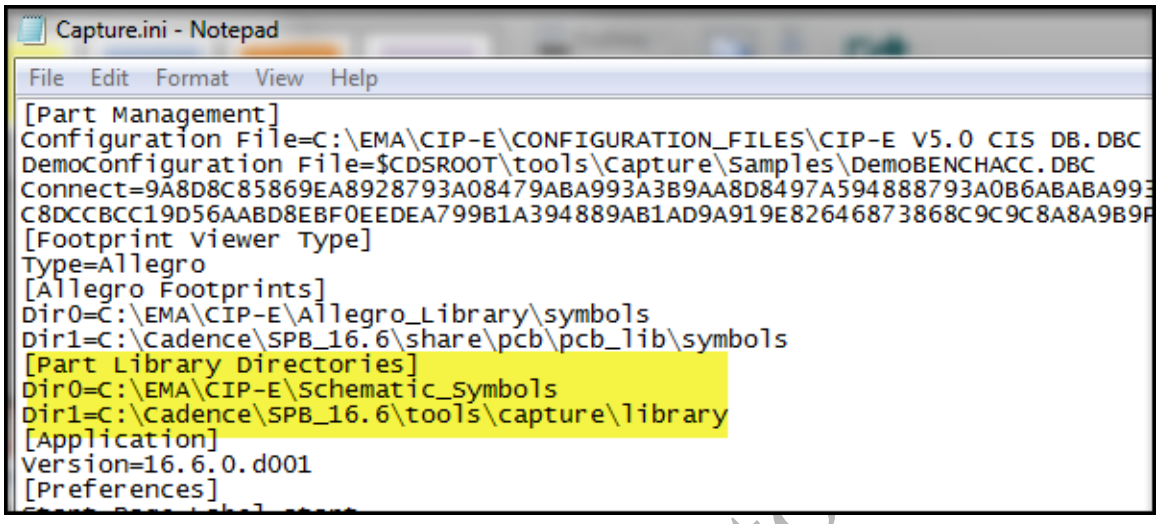
In this lesson you will generate a library project and import a set of schematic symbols into it for editing. You will also be able, as Project Leader, to assign users to certain parts. Once the parts have been successfully edited and approved, they can be "promoted" to the master libraries.

The **master libraries** would be the location that all team members use when *placing parts* onto their schematics. If you are using OrCAD Capture CIS with CIP this would most likely be the path where the master database part libraries are located. These paths are reflected in the *Capture.ini* file, in the Part Library Directories section (see the following illustration).

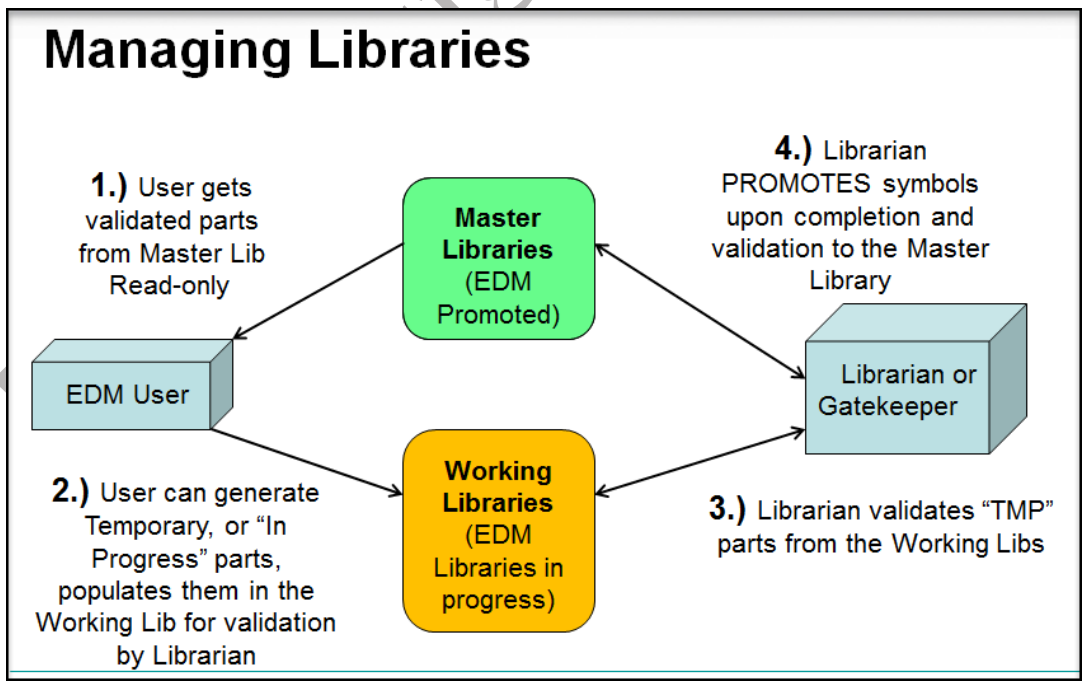
EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

Library Management Flow



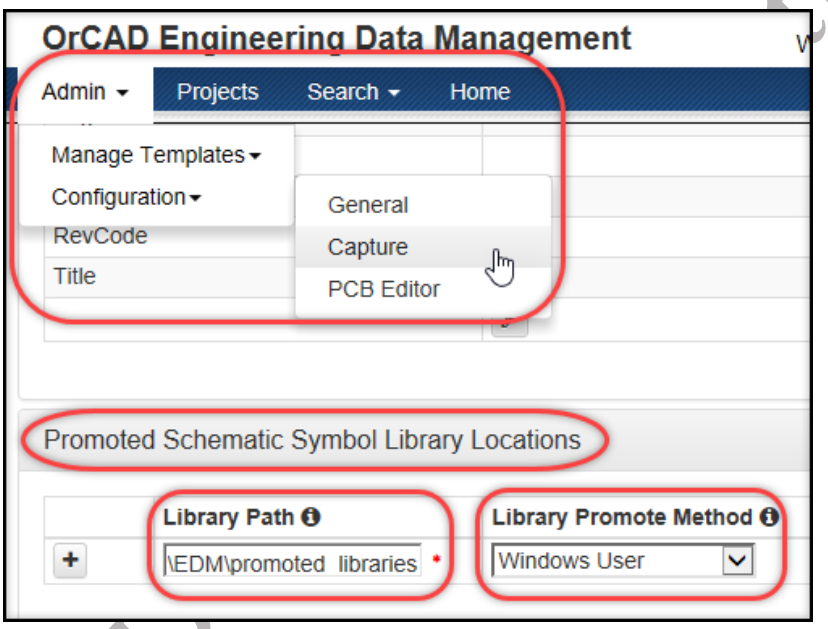
In a typical *library symbol flow* for a part request, a user may do one of two things. They may make a formal part request from someone who handles Librarian tasks, or, if **OrCAD CIP** is used, they may generate the TMP part themselves within CIP. The following illustration shows an example of how a typical part request/generation may occur within EDM.



 EMA Design Automation, Inc ®© 2020 All rights reserved

Libraries that are managed within EDM would be considered “Working” libraries until they are completely validated according to a company’s internal policy. Users who may be assigned to a working library within EDM would have access to those parts and be able to generate new parts within the working library. Once a part meets the criteria and is approved (usually by a Gatekeeper, or Librarian) it can be ***promoted*** from the working library to the master library locations. This final step of promoting the library symbol is handled by the Project Leader or by an Administrator.

The promoted libraries location is set up as either an Administrative function or by the Project Leader in the **Admin>Configuration>Capture** area.



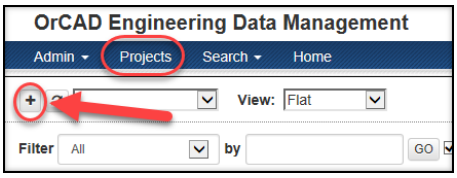
EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

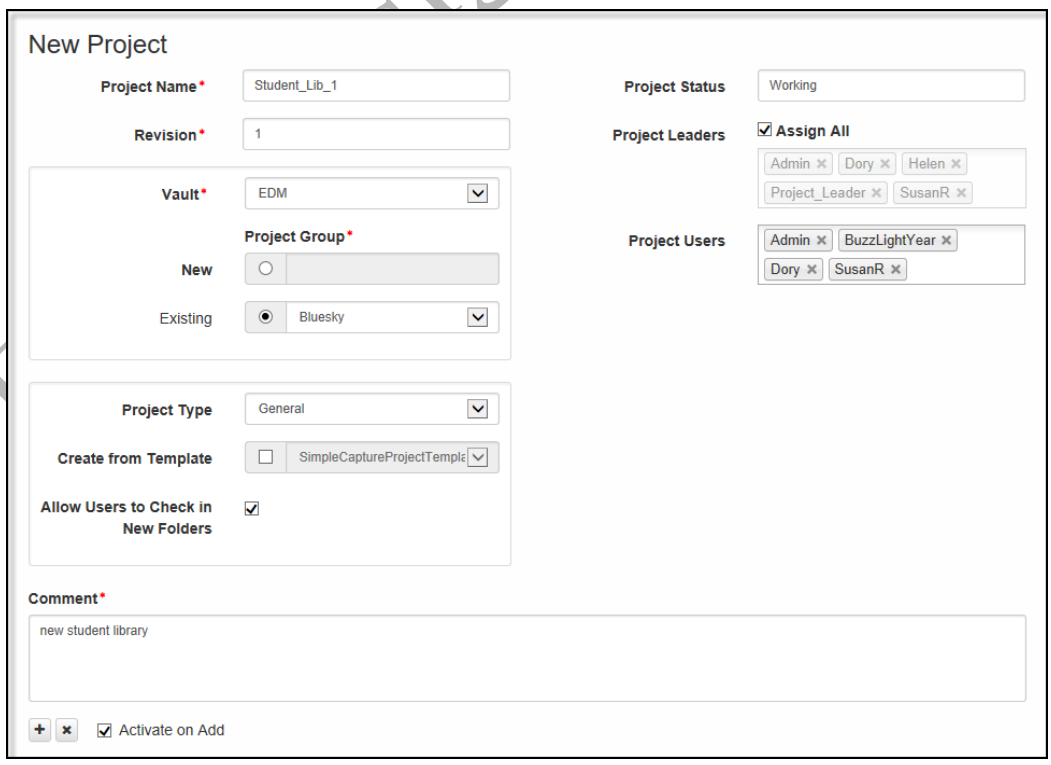
Lab 3-1: Creating a Library Project

In this lab you will create a library project.

1. Click on the Projects tab in EDM. Next, click on the Add Project button to begin generating a new library project. See the following illustration.



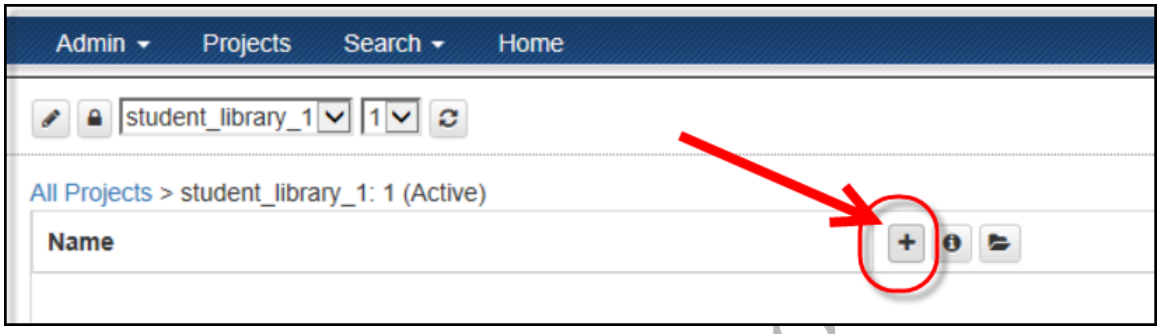
2. Fill in the project information as it appears in the following graphic. You can name the project what you prefer. Be sure and add project leaders and project users and be sure to include yourself (as Project_Leader) in the list of users. Select the Project Type as General and Allow Users to Check in New Folders. Note we are not using a template for this project.
3. Be sure and add a comment and select to Activate on Add. Then click on the plus sign + to create the project. See the graphics below.



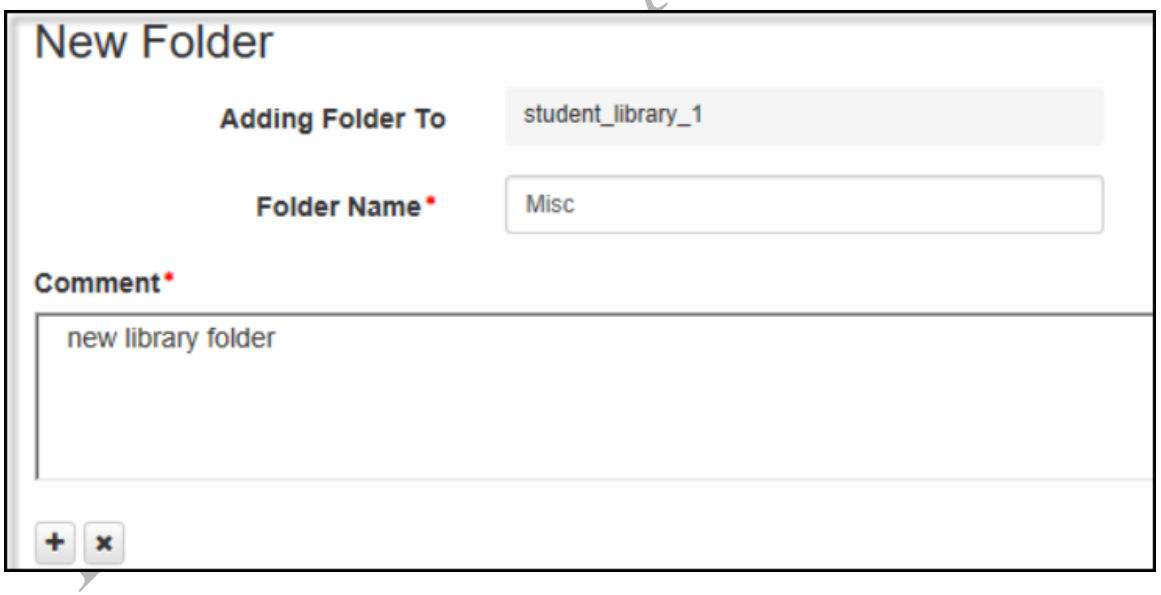
Cadence Design Automation

Adding a Folder to the Project

1. Once the project is created select it from the project list.
2. Click on the Add Folder button to begin adding folders. (See the following illustration).



3. Fill out the information in the New Folder page as follows. Since you will be importing a library of miscellaneous type symbols the Folder name can be Misc.



4. Add a Comment and click the plus sign + to add the library folder.

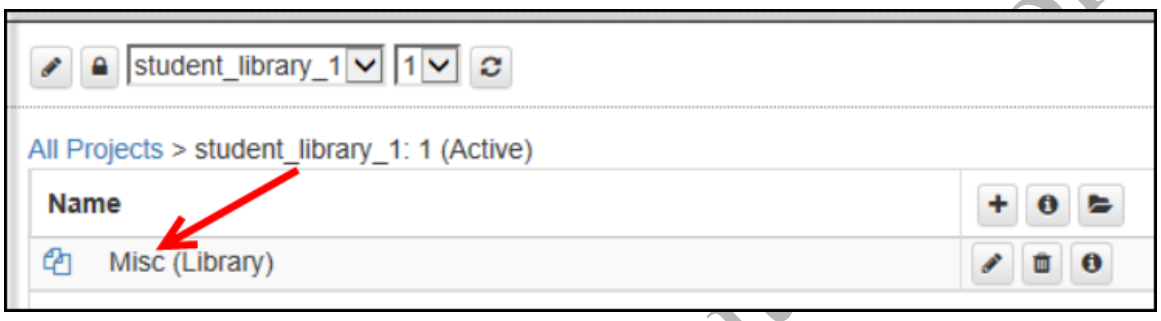
In the next lab you will Import a library of symbols.

EMA Design Automation, Inc ®© 2020 All rights reserved

Lab 3-2: Importing a Library of Symbols into the Project

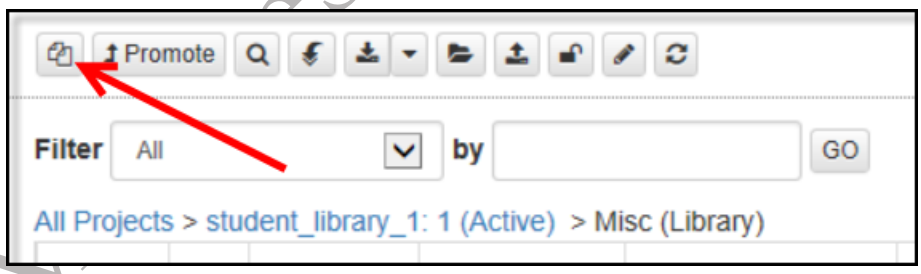
Now that the folder is created you'll be able to import the symbols.

1. Click on the Misc (Library) name.

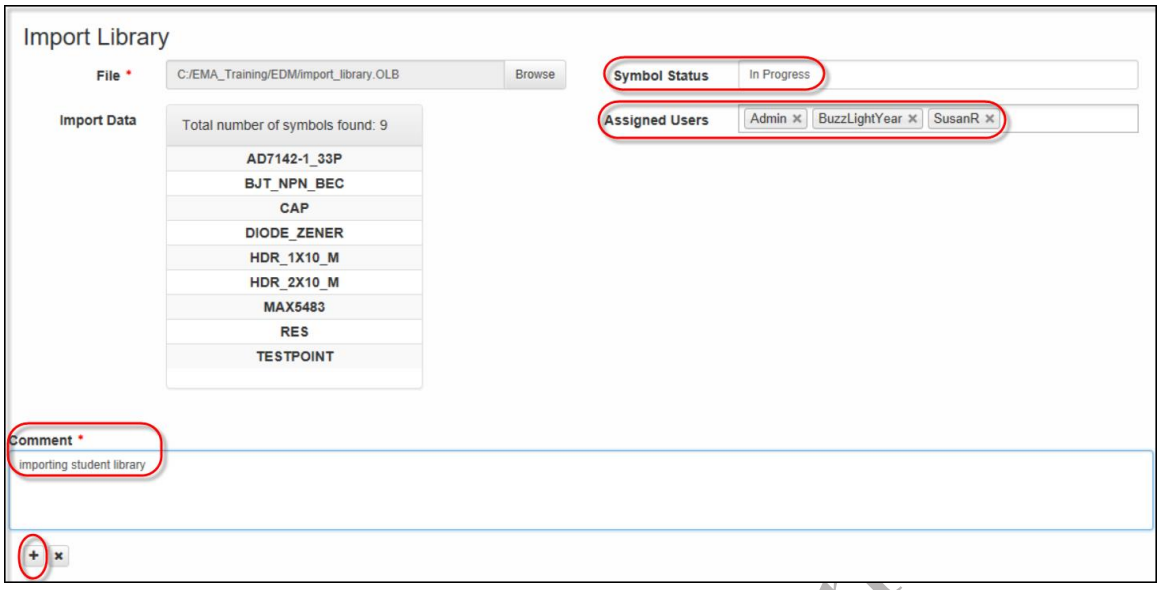


Notice that there is nothing yet in the library but you now have access to the Import and Promote symbol buttons.

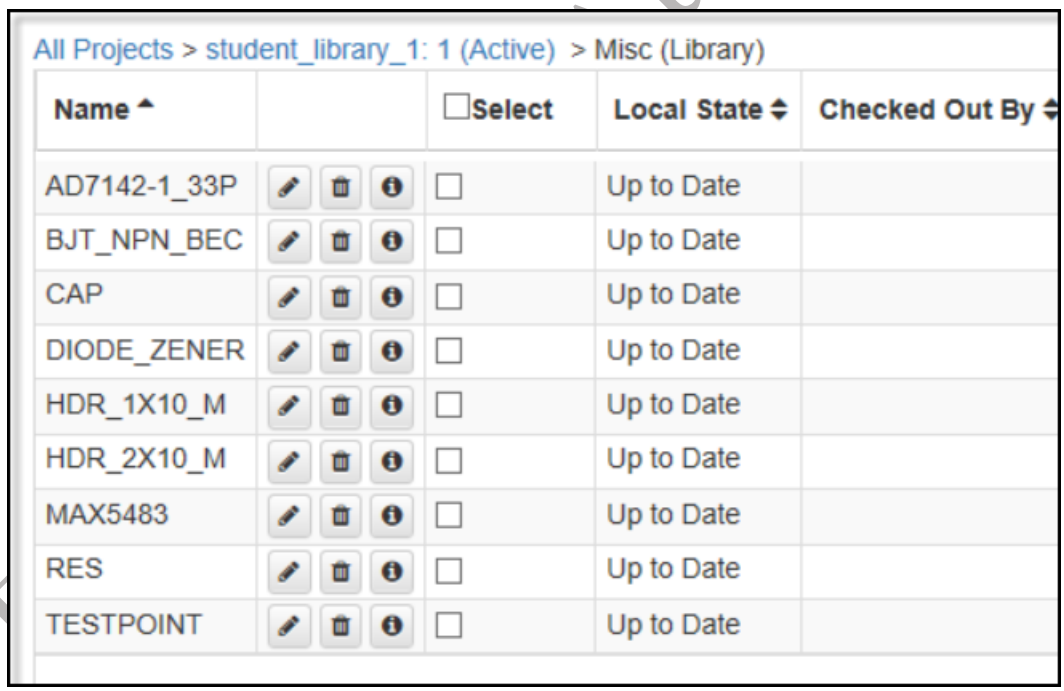
2. Click on the **Import** button to begin importing symbols. See the following illustration.



3. When the Windows Explorer opens navigate to the C:\EMA_Training\OrCAD_EDM_17.4 and select the **IMPORT_LIBRARY.OLB**
4. Fill out the information as follows. Be sure to assign users (including yourself) and the symbol status. Since the parts are not yet approved you can mark them as In Progress.



The finished library should look like the following.

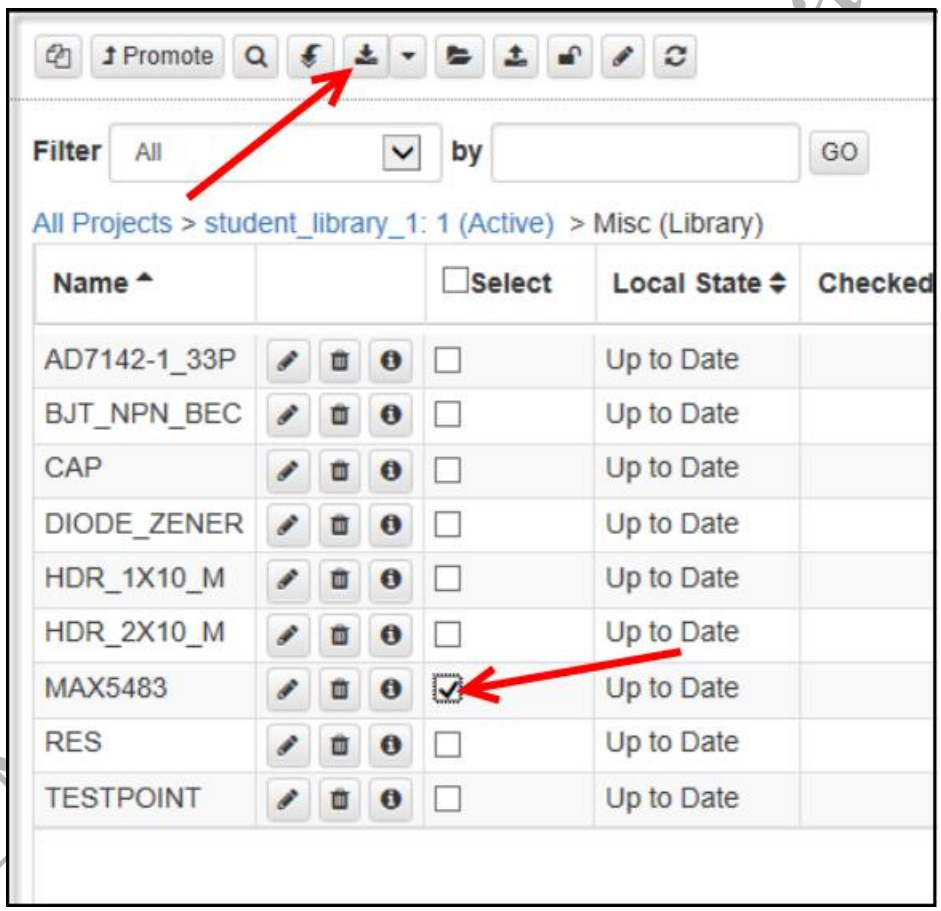


 EMA Design Automation, Inc ®© 2020 All rights reserved

Lab 3-3: Performing Design Work on a Symbol

In this lab you will be checking out a symbol, performing minor edits, and checking the part back in to EDM. This is a common task that a user assigned to this type of project would accomplish.

1. In the Select column click on the select box next to the MAX5483 part. Then click on the **Check Out** button as shown in the following illustration.

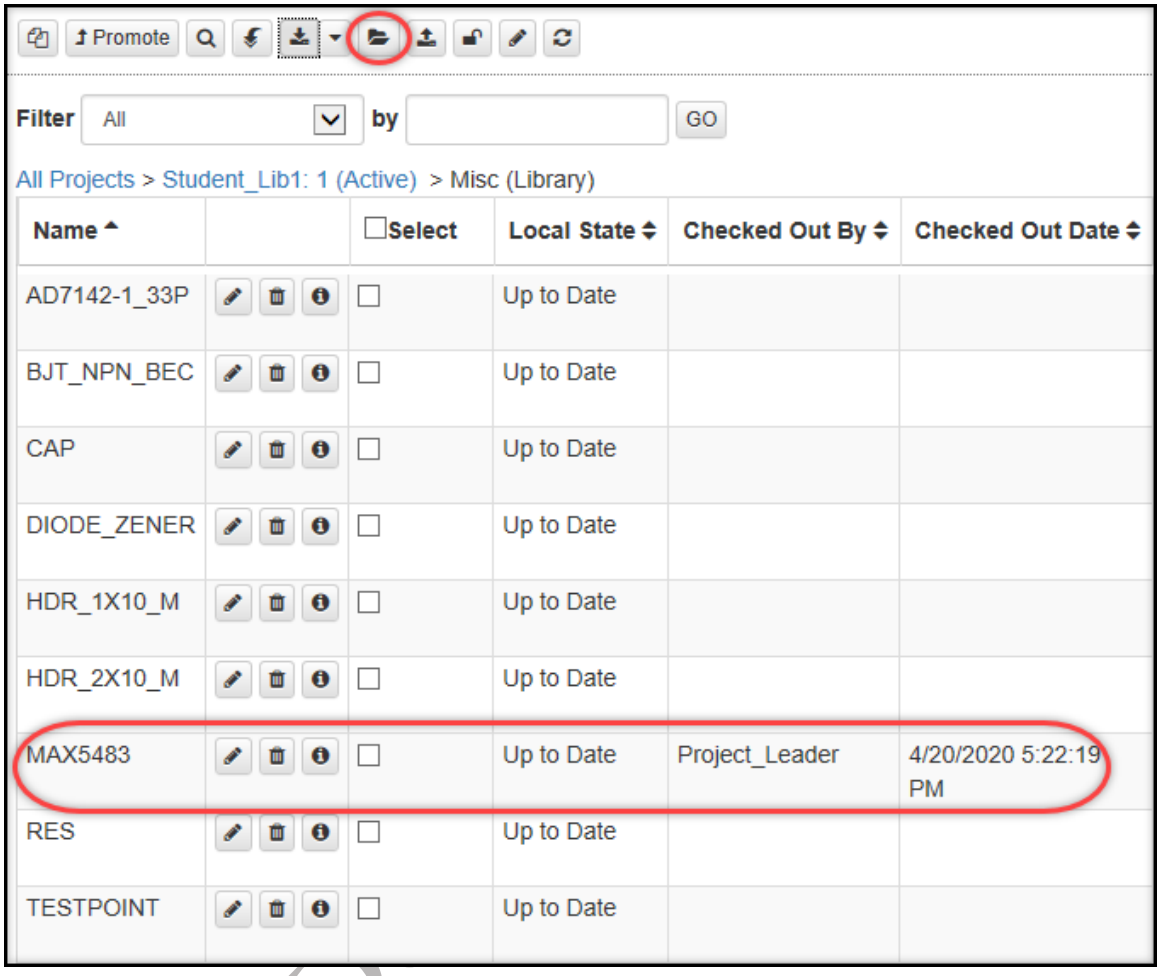


Notice that the part is now checked out. Observe the status: Check Out By and Checked Out Date.

At this point you can now open the library for editing in Capture.

 EMA Design Automation, Inc ®© 2020 All rights reserved

2. Select the Open Library button. See the following illustration.

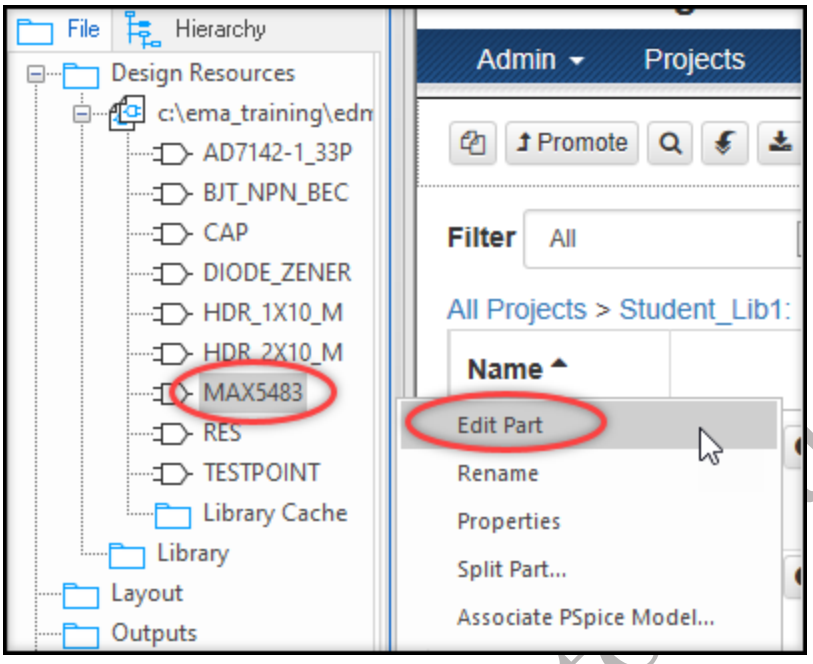


Notice the library is now open in Capture CIS.

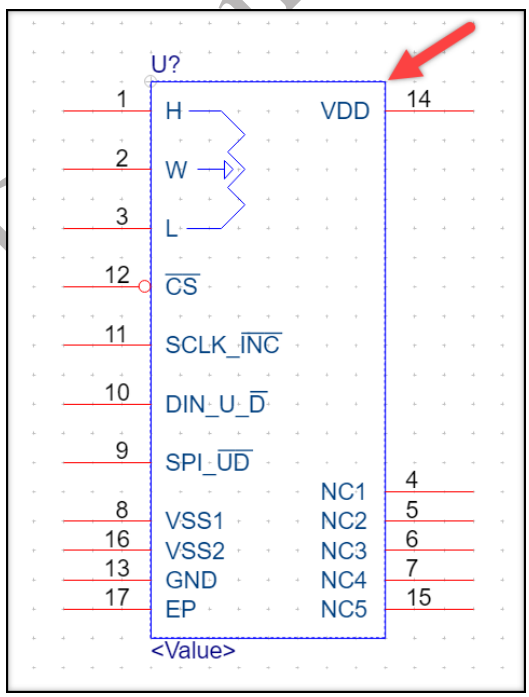
3. When Capture CIS opens, right click on the MAX part and choose Edit Part.

EMA D

 EMA Design Automation, Inc ®© 2020 All rights reserved



4. When the part opens, adjust the body size to be 2 grids wider in width. The purpose in this step is not to make any major change to the part, but a minor one to simulate the activity of check-out/check-in and part promotion. See the following illustration.



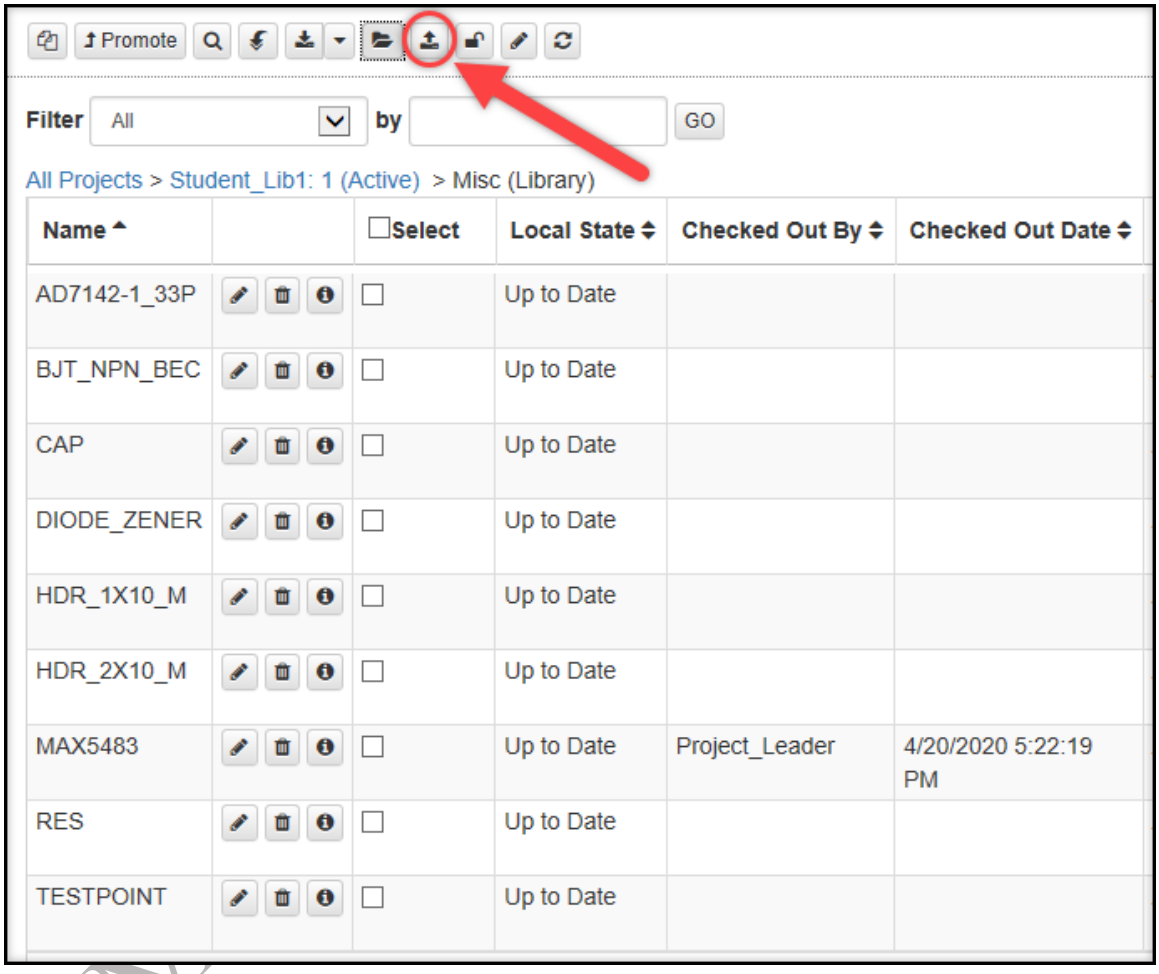
5. After the edit is complete save and close the part.

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

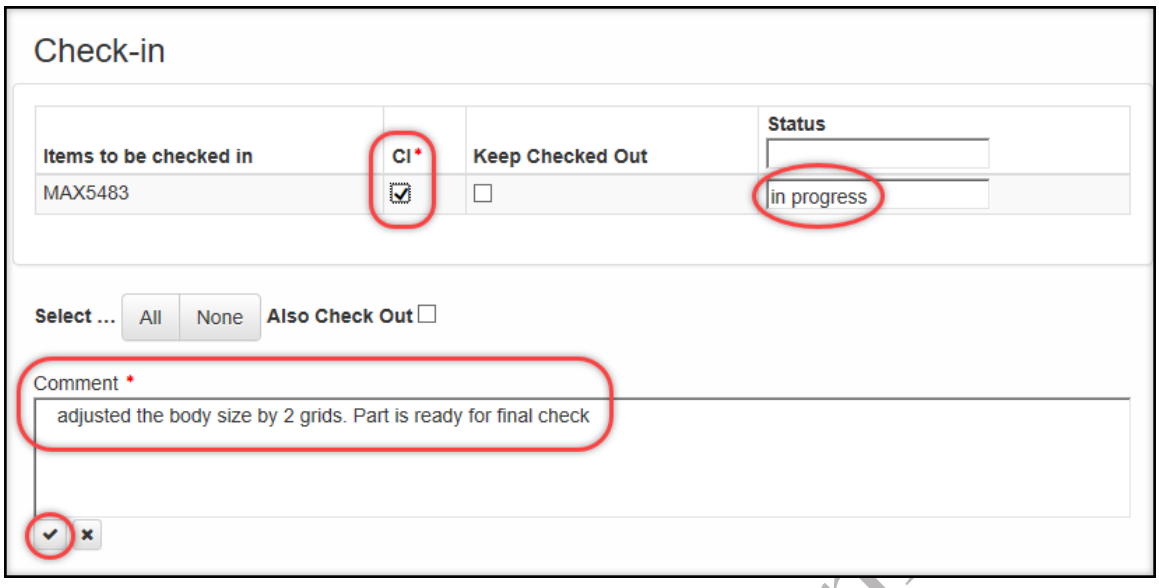
Checking In the Edited Library Part

1. Save and close the library and the Capture project manager window.
2. When the EDM window returns select the Check In command button as in the following illustration.



3. Fill out the information as illustrated in the following graphic. Since this part is complete you can mark the Status as Released. Be sure to fill in the comment.

EMA Design Automation, Inc ®© 2020 All rights reserved



4. Click the check sign to check in the part. Note the Status of the part after it is checked in.

EMA Design Automation

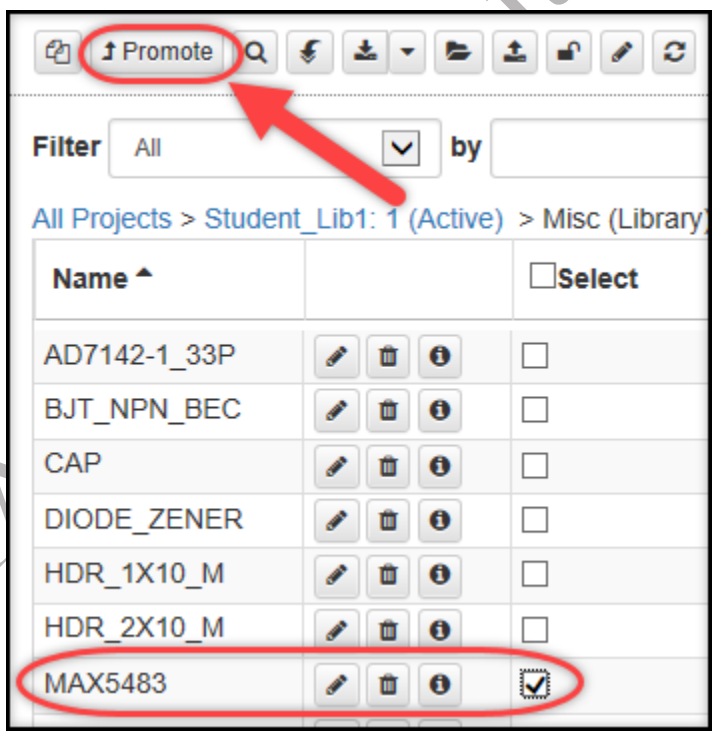
 EMA Design Automation, Inc ®© 2020 All rights reserved

Lab 3-4: Promoting the Symbol

The final step in this process would be promoting the part. The library that the part gets promoted to would be the Master library that was defined in the EDM Configuration. Promoting a part is a privilege that the Project Leader has and they would be the one to approve the part(s) and finally promote them to the master libraries.

The following graphics show the steps to be taken to promote a part once it has been finally approved by the Project Leader. You will not be performing the Promote step in this lab.

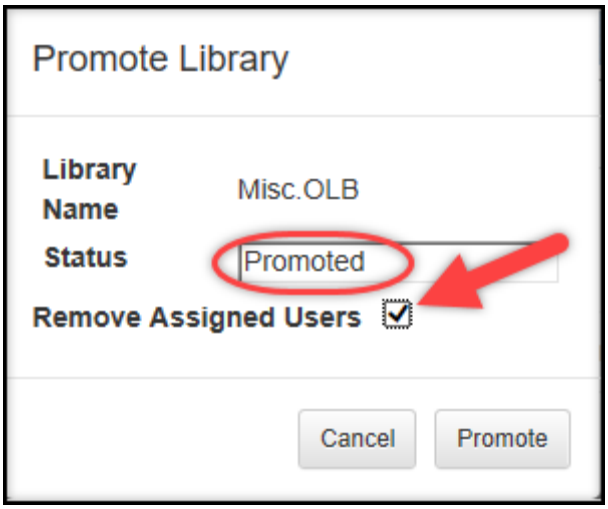
Promote Verified Part



When the Promote Library window opens it indicates that the Status can be edited as Promoted. The Project Leader could also Remove the Assigned Users at this time.

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved



This is the end of Lesson 3

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

Lesson 4: Working with PCB Projects

Lesson Objectives

After you complete this lesson you will be able to:

- Open an existing project from EDM within the OrCAD PCB Editor
- Import a board to the project

In this lesson we'll explore the steps to generate a board file that is associated with a schematic project file.

Generating the Board File

The Project Leader will most often be the one who will generate the projects before assigning users to work on them. Schematic projects are generated from EDM within the Capture environment as we have seen in earlier Lessons.

Generating a printed circuit board file to accompany the schematic in a project must be done via the EDM Desktop application. This is because of the associations between the Desktop application and the OrCAD PCB Editor that enables changes to be recognized during work.

Following are the high level steps a Project Leader would take to generate a complete design package.

- Project Leader opens EDM from within the Capture environment.
- Project Leader generates a new schematic project
- Project Leader opens EDM Desktop version and opens the schematic project
- Project Leader imports either an existing board or an existing board template

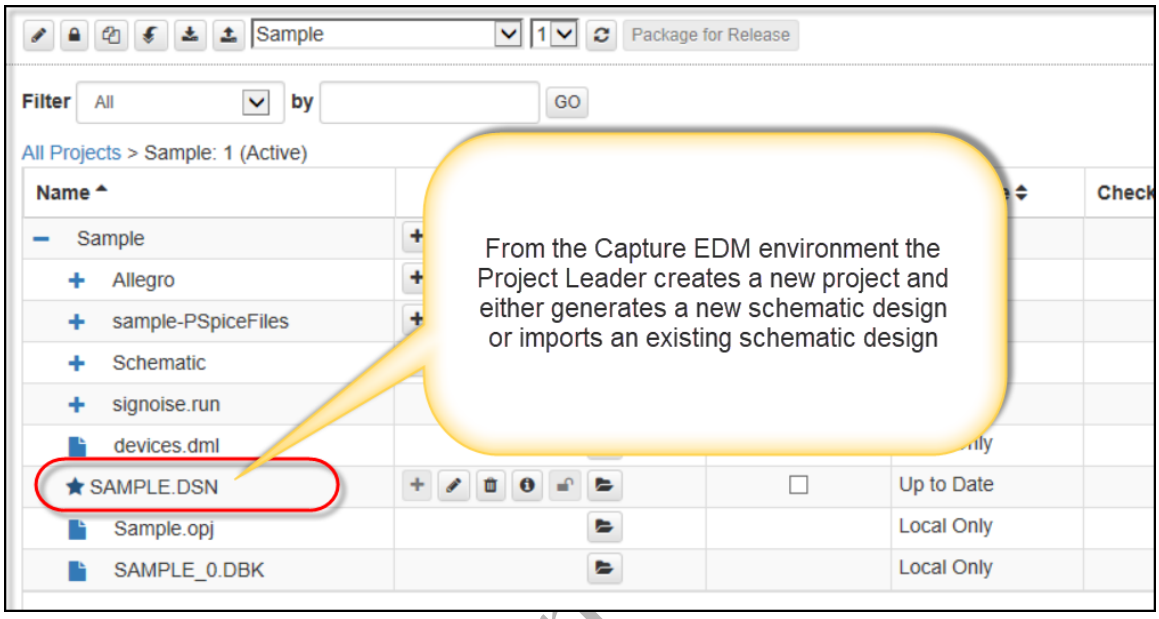
See the graphics below.

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

Project Leader Creates the Schematic Project

From the Capture EDM environment the Project Leader creates a new project and either generates a new schematic design or imports and existing schematic design.

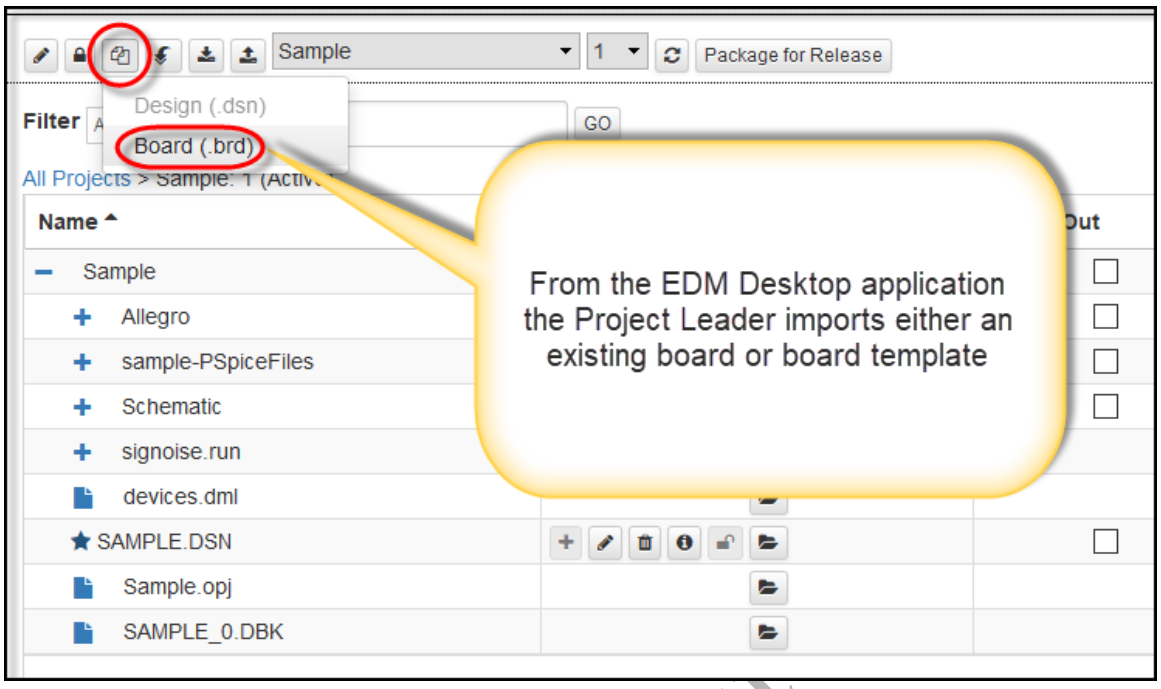


Project Leader Adds the Board to the Schematic Project

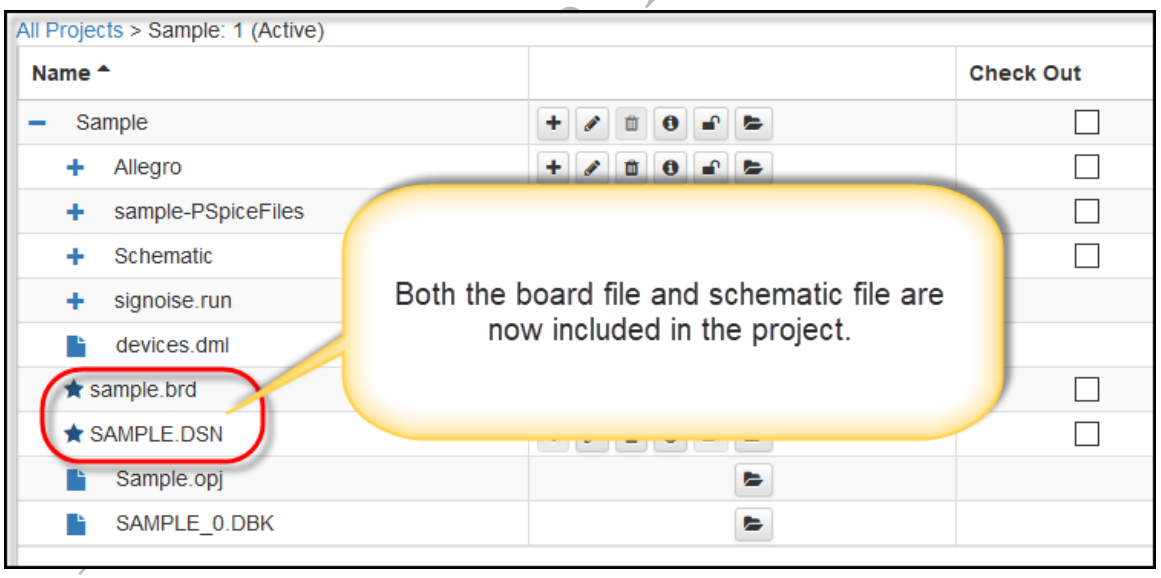
From the EDM Desktop application the Project Leader imports either an existing board or a board template.

EMA Design

 EMA Design Automation, Inc ®© 2020 All rights reserved



Both the board file and the schematic file are now included in the project.



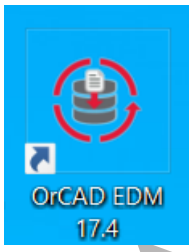
Assigned users to the schematic would work on the schematic using EDM from the Capture environment, while users assigned to the board would work on the board from EDM within OrCAD PCB Editor.

 EMA Design Automation, Inc ®© 2020 All rights reserved

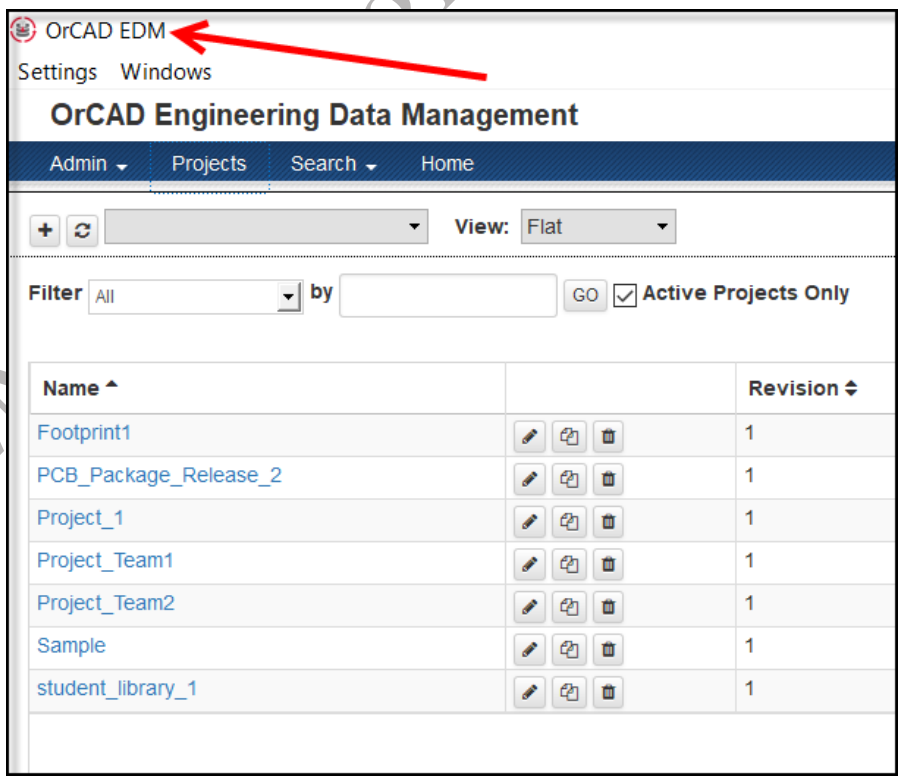
Lab 4-1: Opening the EDM Desktop Application

In this lab you will open EDM from the Desktop application and open a project that you generated in a precious lab exercise.

1. Open OrCAD PCB Editor (Standard or higher may be used).
2. If EDM is open in Capture log out of it now.
3. Next, open EDM using the desktop application. This will be accessible by using the icon on the desktop:



Notice that EDM shows that it is open in Desktop mode from the icon in the upper left.



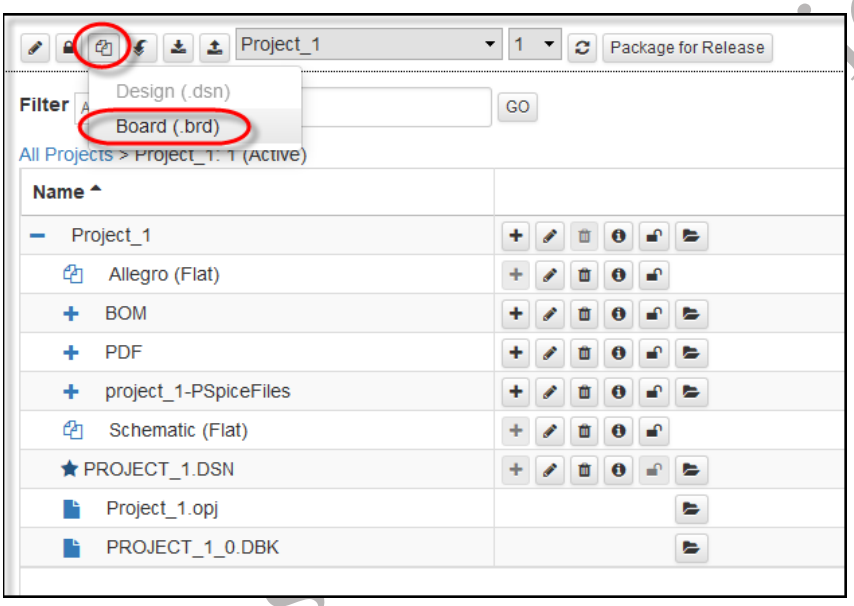
EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

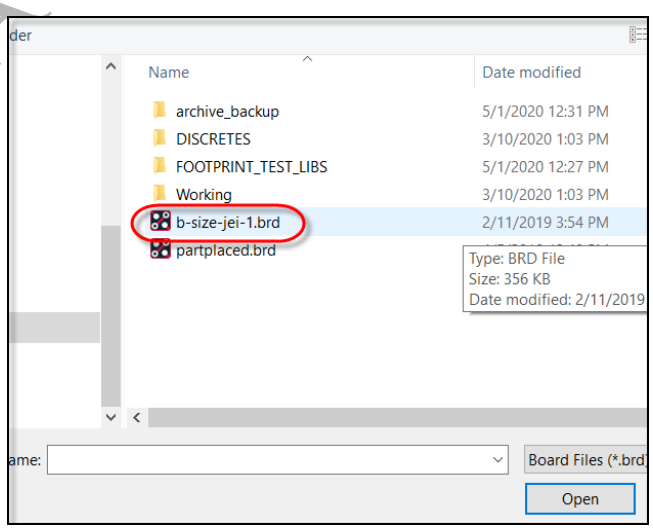
Lab 4-2: Importing a Board to a Project

Next you will select a project that you previously generated from an earlier exercise and import a board file into it.

1. From the Project list select the first project that you created (Lab 2-1)
2. Next, choose ***Import >> Board (.brd)***



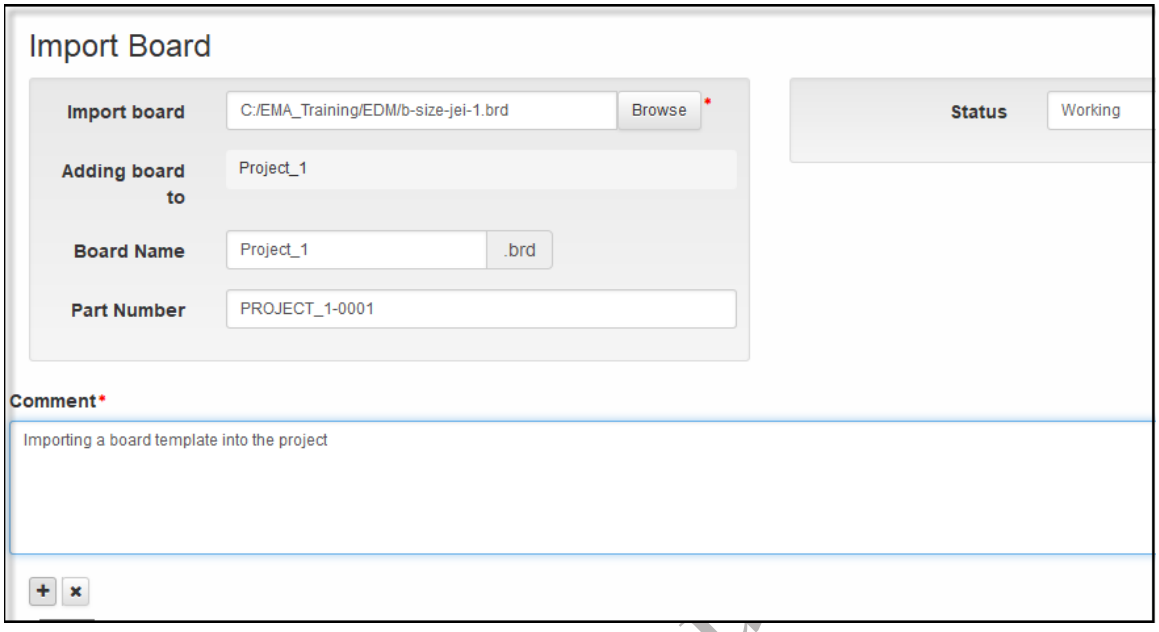
3. Navigate to the folder where your training files are located and choose the file named `b-size-jei-1.brd`



EMA Design Automation

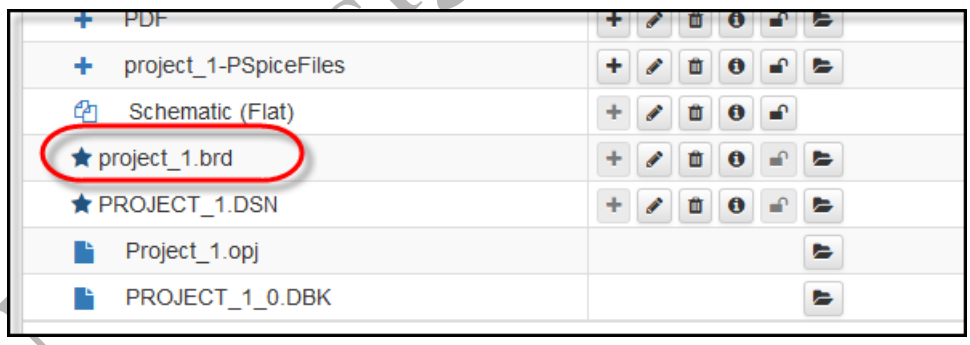
 EMA Design Automation, Inc ®© 2020 All rights reserved

4. In the Import Board window fill out the information. See the following graphic.



5. Click the plus sign to Add the board to the Project.

The board is now added to the project.



 EMA Design Automation, Inc ®© 2020 All rights reserved

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

Lesson 5: Working with PCB Footprint Symbols

Lesson Objectives

After you complete this lesson you will be able to:

- Create a project for PCB Footprints
- Import a footprint library to the project
- Navigate views in the footprint project
- Check-out a footprint for editing
- Check-in the footprint and Promote it to the master library

Projects created as footprint projects can only be used for footprints. Footprint projects opened *in Capture* have limited functionality. To enable full functionality, you need to open a Footprint project from the **EDM Desktop** application. If you want to import or check in footprints, you also need to have your **PCB Editor** open. When footprints are brought into EDM, PCB Editor is used to determine file relationships.

Footprint files may be checked in and checked out as with other files by the general user. Additionally, a Project leader may also import footprints from another directory, promote footprints, delete footprint, and edit properties.

Working with PCB Footprints

In this lesson and lab, you will generate a footprint project. To gain full functionality you will be working with the **EDM Desktop** and with **OrCAD PCB Editor** open. As expressed above, both the **EDM Desktop** application and **OrCAD PCB Editor** must be open in order to be able to import a footprint library, open a footprint for editing, and be able to check the footprint and associated files back in to EDM.

Working with pcb footprints in EDM is much like working with Capture symbols, in that users may be assigned to work on footprints, and when the work is complete, the file may then be promoted to a master location by the Project Leader.

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

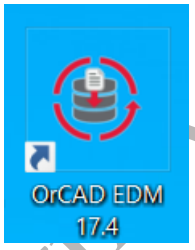
EMA Design Automation

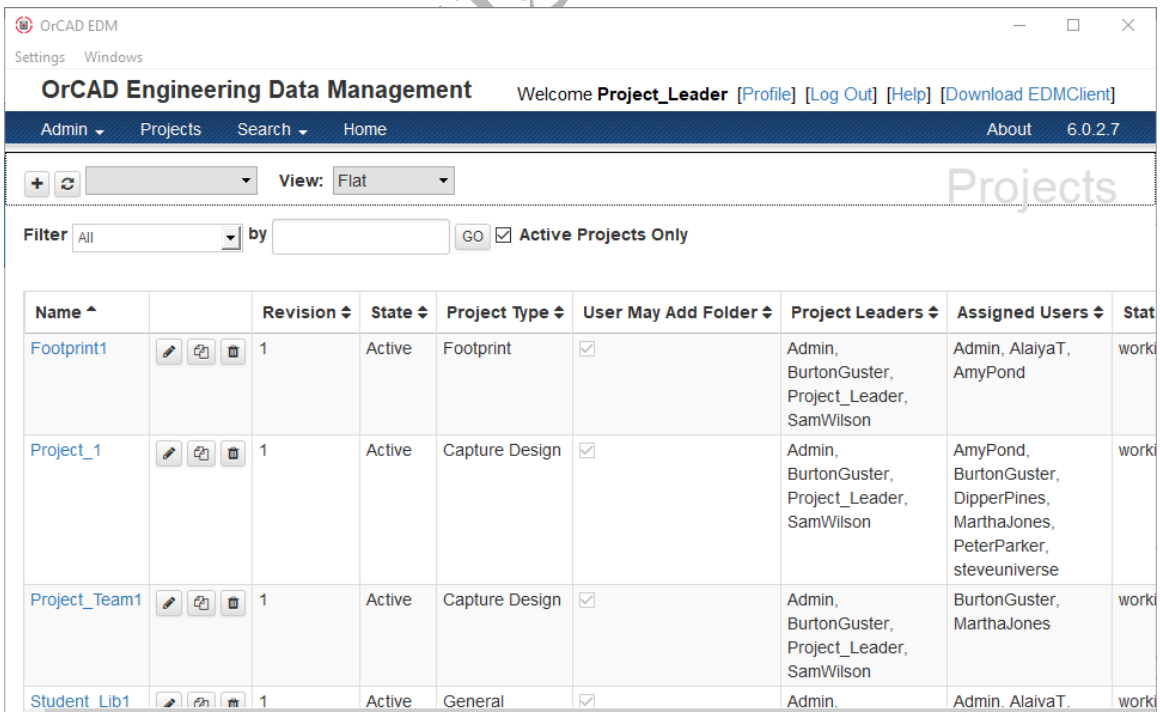
 EMA Design Automation, Inc ®© 2020 All rights reserved

Lab 5-1: Generating a New Footprint Project

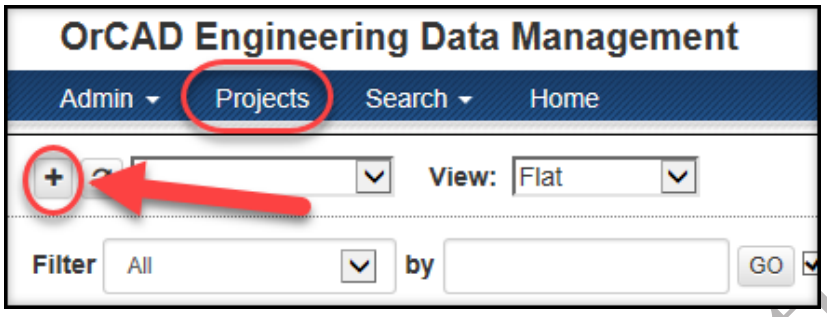
In this lab you will be generating and working with a new footprint project. To fully access the editing capabilities inside the PCB Editor you will need to have OrCAD PCB Editor open. Additionally, you will be working on this lab from the EDM Desktop mode.

1. Open OrCAD PCB Editor (Standard or higher may be used).
2. If EDM is open in Capture log out of it now.
3. Next, open EDM using the desktop application. This will be accessible by using the icon on the desktop:





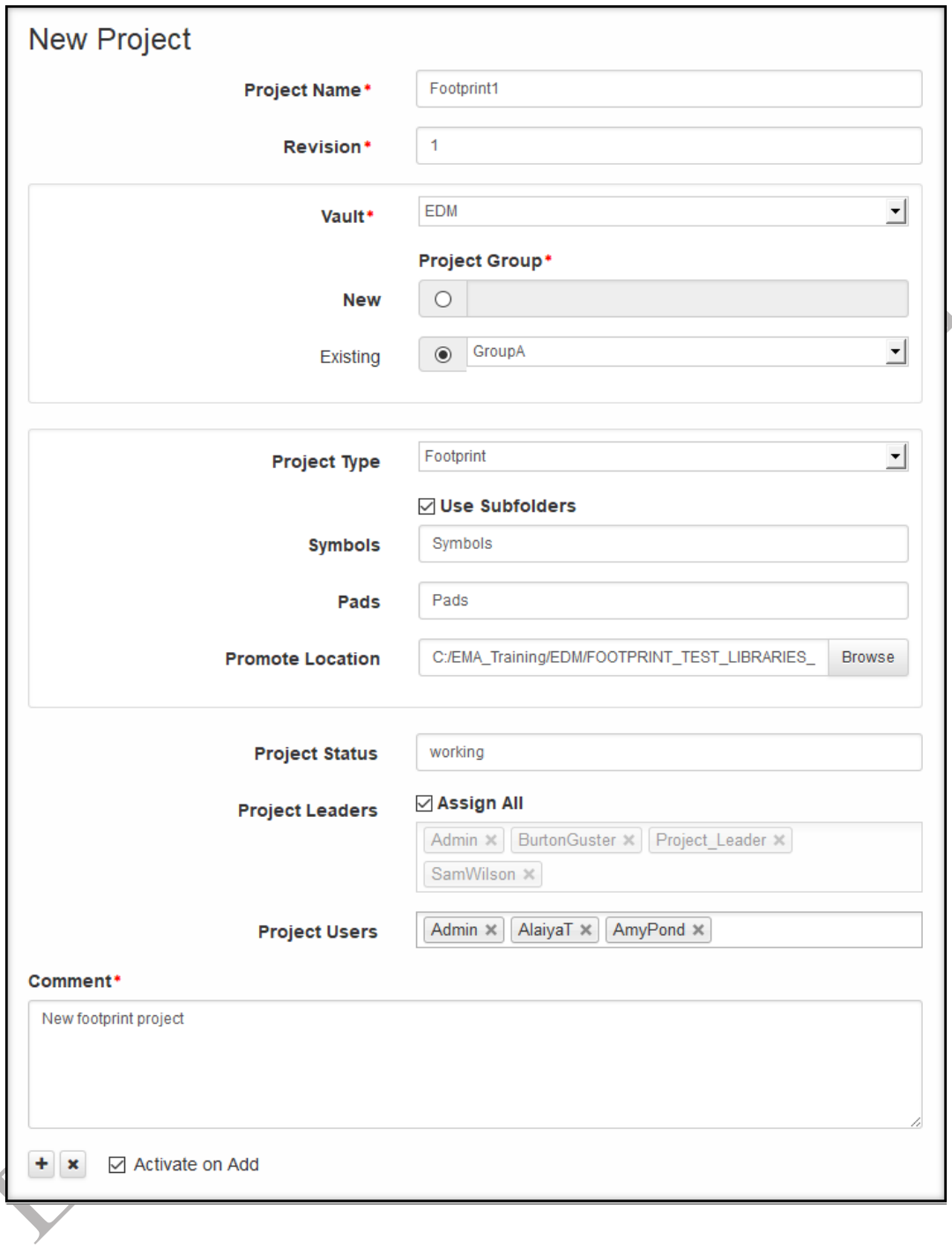
4. Make sure you are on the Projects page and select the plus sign to generate a new project.



5. Fill out the New Project form as in the following graphic.

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved



Observe that when you select the **Project Type** as **Footprint**, that the option to **Use Subfolders** appears. It is a good idea to separate the pads and symbols to maintain easy organization of files.

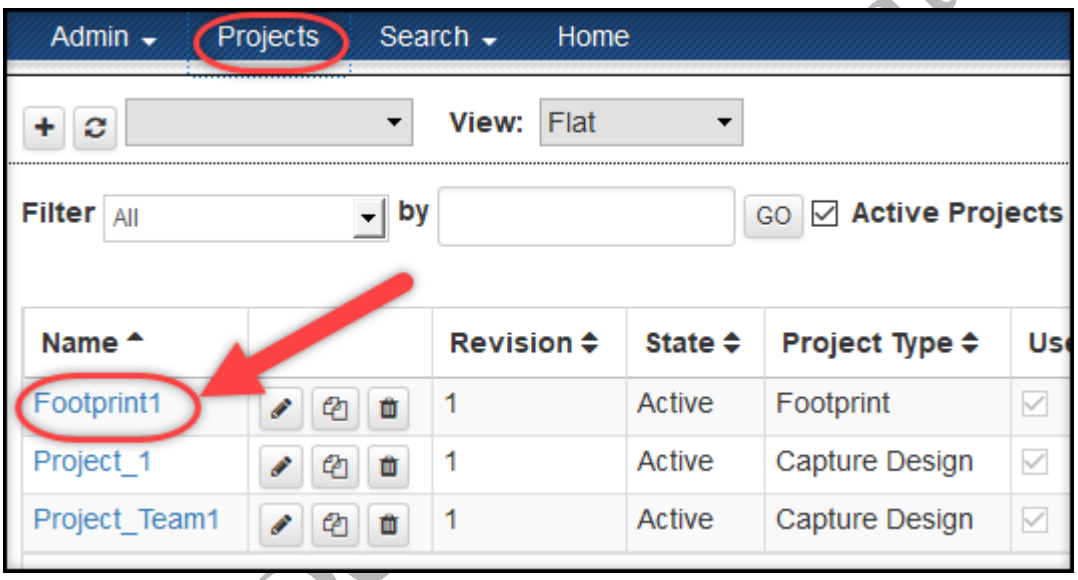
 EMA Design Automation, Inc ®© 2020 All rights reserved

Lab 5-2: Importing a Footprint Library

Next, you will need to import a library of pcb footprints. To import, the EDM desktop application must be open.

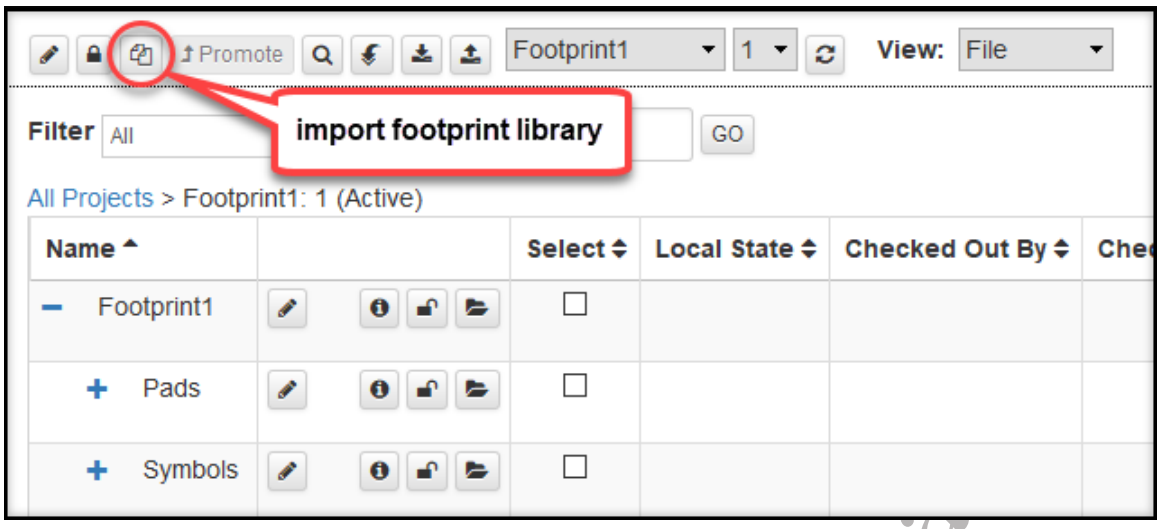
Import Footprint Library

1. From the Projects page, select the Footprint1 project you just created.



2. Next, click on **Import Footprint Library** button located in the row of icons toward the top right area. See the graphic below.

EMA Documentation



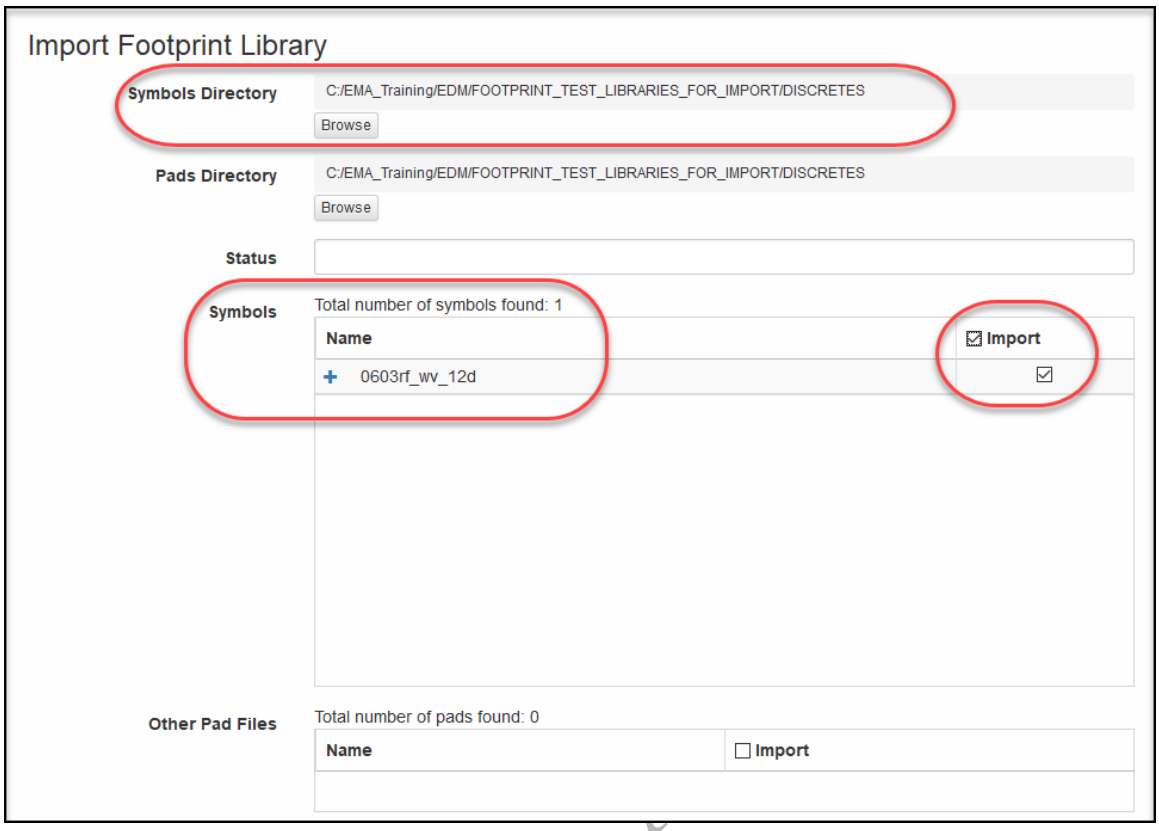
3. Navigate to the location where you have stored your EDM training files to locate the footprint library folder named **DISCRETES**. On EMA Training computers, this location is: **C:\EMA_Training\EDM\FOOTPRINT_LIBRARY_FOR_IMPORT\DISCRETES**

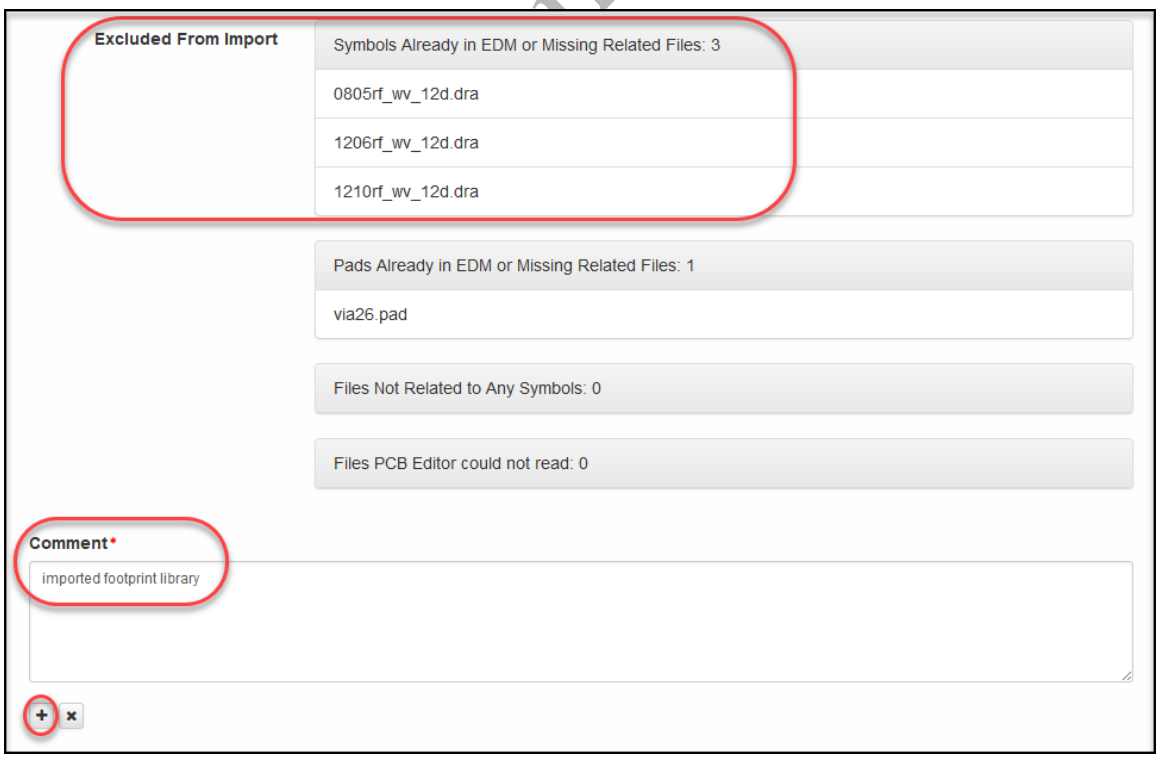
It will ask you the location for both symbols and footsprints; both are located in the DISCRETES folder. The Import Footprint Library window will appear. It will list the total number of symbols found as well as any other pad files and files excluded from the import that already exist in EDM.

4. Expand the **Import** button in the **Symbols** section to view the files that will be imported. Scroll down, add a comment then select the plus sign + to import the footprint library. See the graphics below.

EMA Design Automa

 EMA Design Automation, Inc ®© 2020 All rights reserved



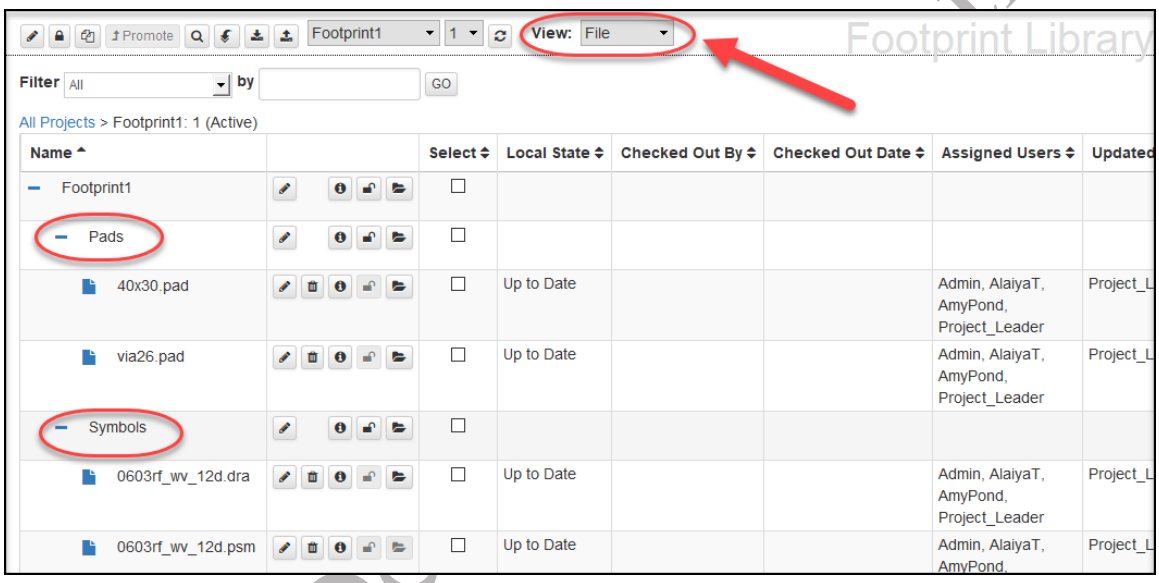


 EMA Design Automation, Inc ®© 2020 All rights reserved

Notice that some symbols were excluded from the import because they were missing files or already existed in EDM.

Navigating the Footprint Library Views

Notice when the footprint library has been imported you can see the pads and symbols, and you can expand the pads and symbols folders. The **View** is defaulted to "**File**" which enables you to see the pads and symbols but also any other files that may be generated during editing.

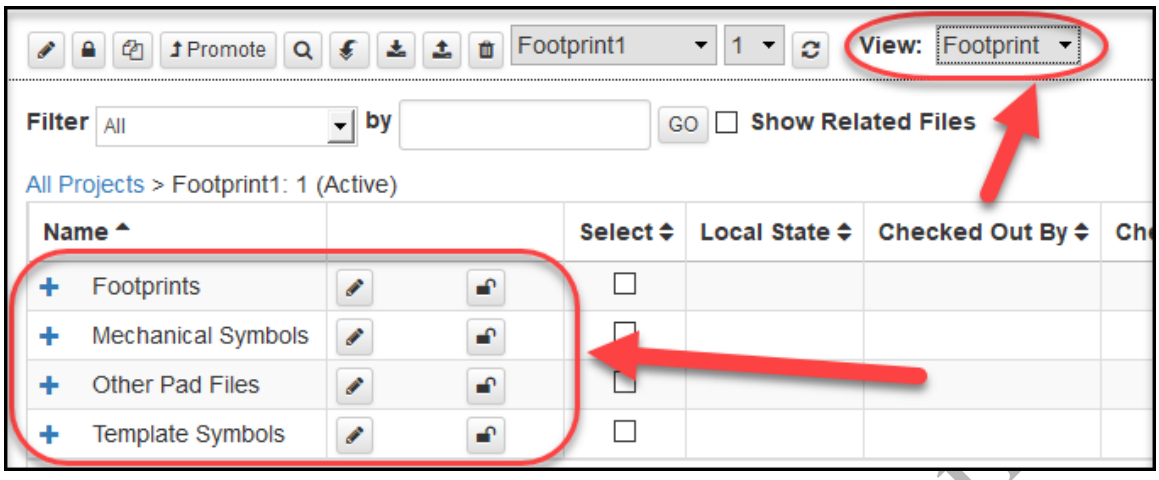


But it is sometimes necessary to understand which pad(s) is associated with a footprint, and it is easier to see this from the Footprint view. Complete the following steps to navigate the views.

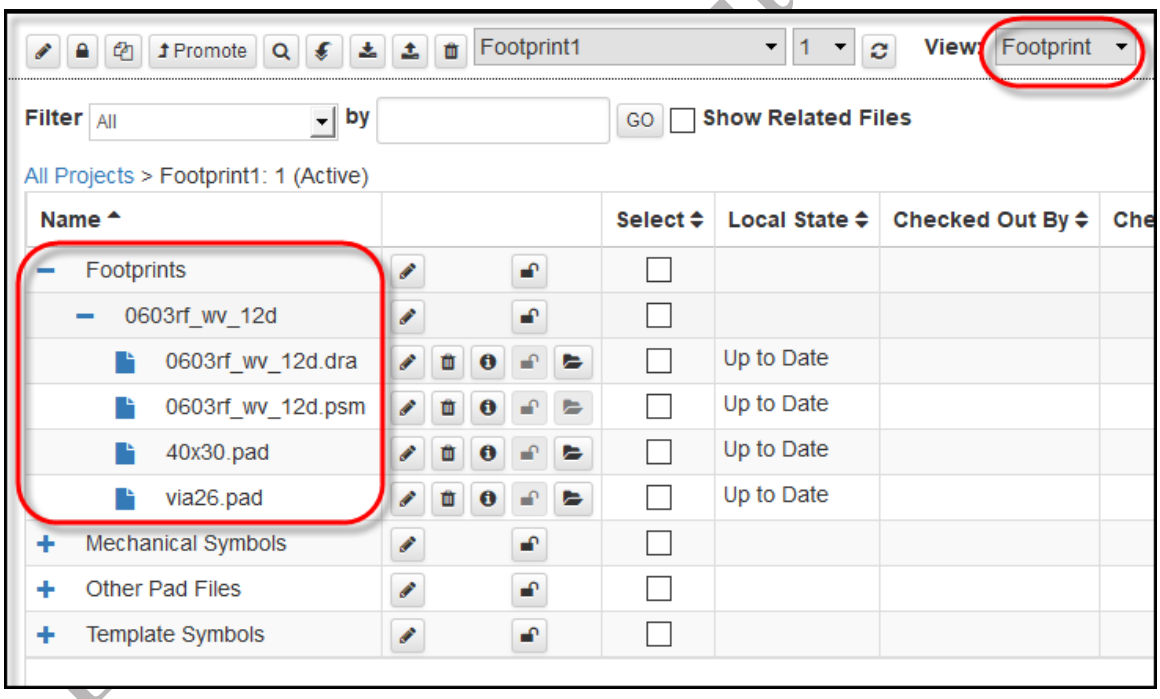
1. In the **View** dropdown, select **Footprint** as shown below. Notice the view will change and the structure will appear slightly different.

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved



2. Expand the **Footprints** folder and the **0603rf_mv_12d** folder beneath it. Notice that now you can see which pads are associated with the ***.dra*** and ***.psm*** files. See the following graphic.



 EMA Design Automation, Inc ®© 2020 All rights reserved

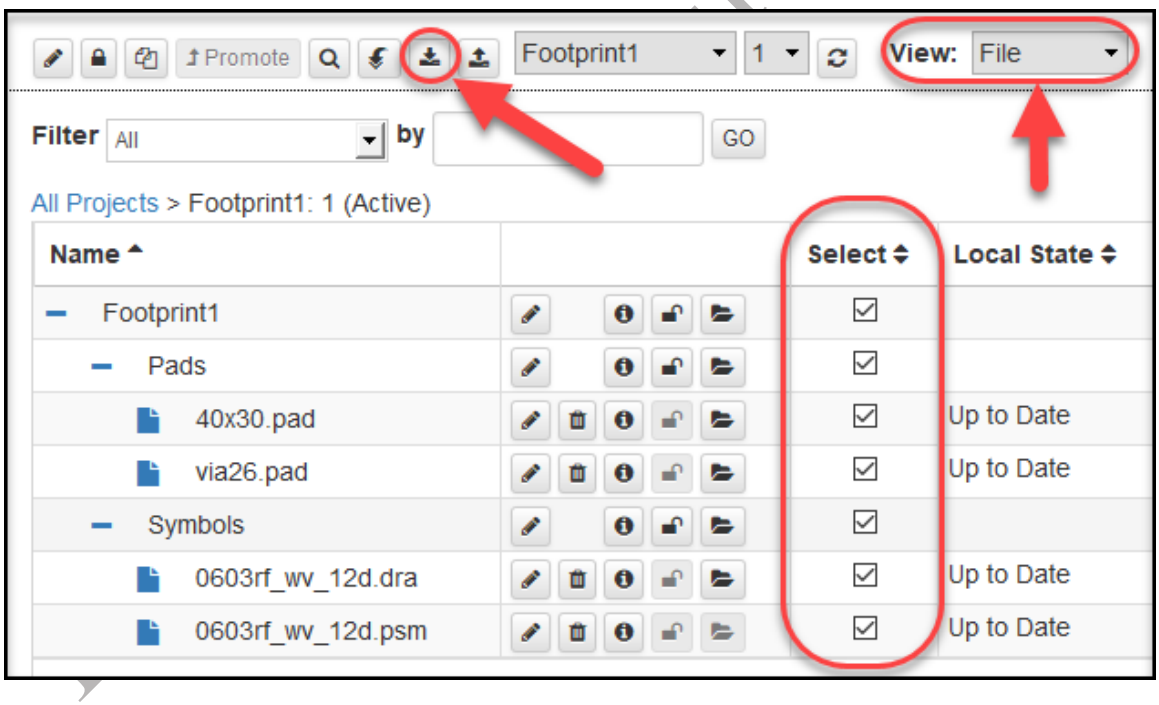
Lab 5-3: Editing the Footprint

In this lab you will open the OrCAD PCB Editor, check out and make a slight edit to the pcb footprint.

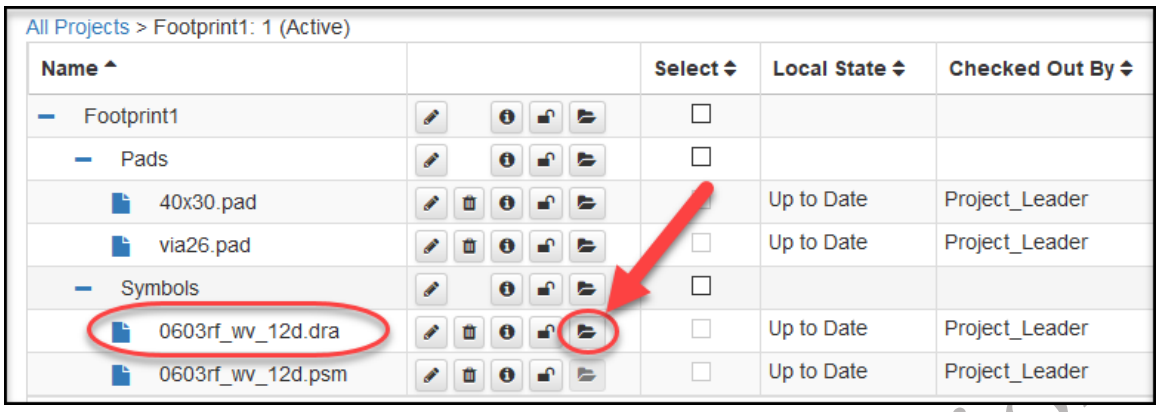
Checking Out and Editing the File

1. Switch back to the **File** view, then expand the folders to view the files.
2. Select the files, as shown in the following graphic, and then click on the Check Out button located across the row of icons towards the top right area.

Notice when you select at the top level, “Footprint1”, that all the other files in the sub-folders are selected.



3. In the **Symbols** folder, select the **Open** button located next to the 0603rf_wv_12d.dra file (just to the left as shown in the graphic below).

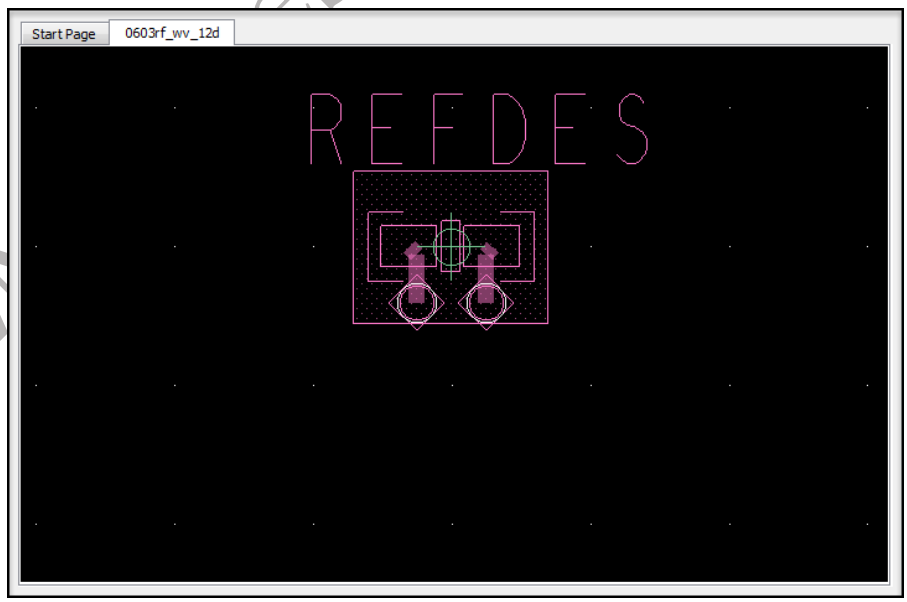


Now you will need to make the PCB Editor window active. It will show the file you have just checked out and opened.

If you are fluent in PCB Editor you may want to adjust the Visibility to show just the pins, vias, etch, Package Geometry- Assembly Top and Assembly Bottom, Silkscreen Top and Silkscreen Bottom.

4. Make a slight change to the footprint, such as moving the Ref Des to the top of the symbol.

   Type move in the command window, then choose your origin and move the REFDES to the top.



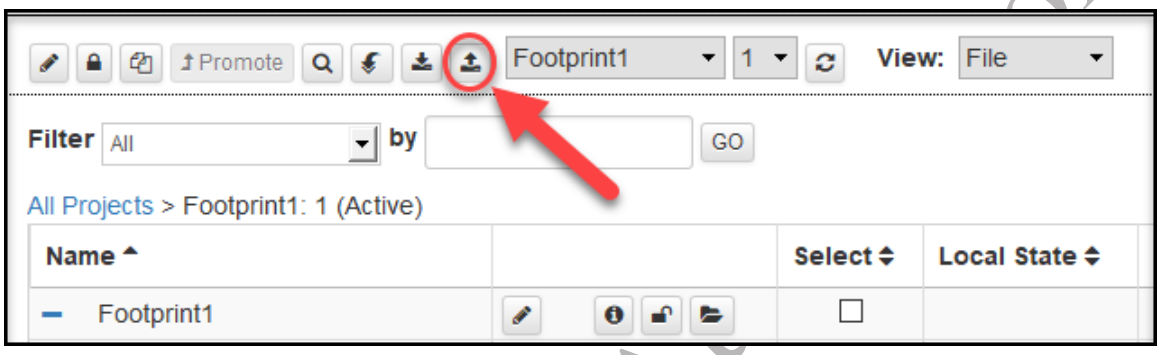
5. Save the file. Click Yes to the Overwrite message if it appears.

EMA Design Automation, Inc ®© 2020 All rights reserved

Checking the File Back In

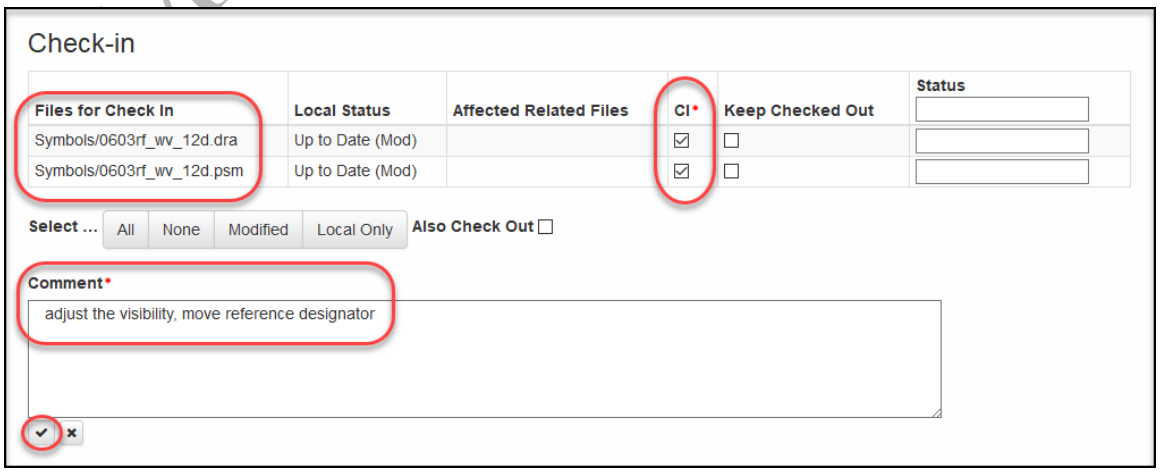
Now you will need to activate the EDM window to begin the Check In process.

1. Activate the EDM window.
2. Click the Check In button located in the top row of icons, as shown in the following graphic.



Note that you can check in footprint files from either the File or Footprint view. EDM will obtain file relationships of all unversioned files. A **Check In** form will appear. You will see a list of checked out files that have been modified, new (unversioned) symbol files in the symbols sub-directory or new (unversioned) padstack files in the pads sub-directory. Files that do not belong in the designated Pads or Symbols sub-directory will not be made available for check in. For example, log files will not be allowed to be checked in.

3. Select the files in the list for **CI*** (Check In). The status can be entered as “working”. Add a **Comment** and select the check mark to check the files in. see the following graphic.

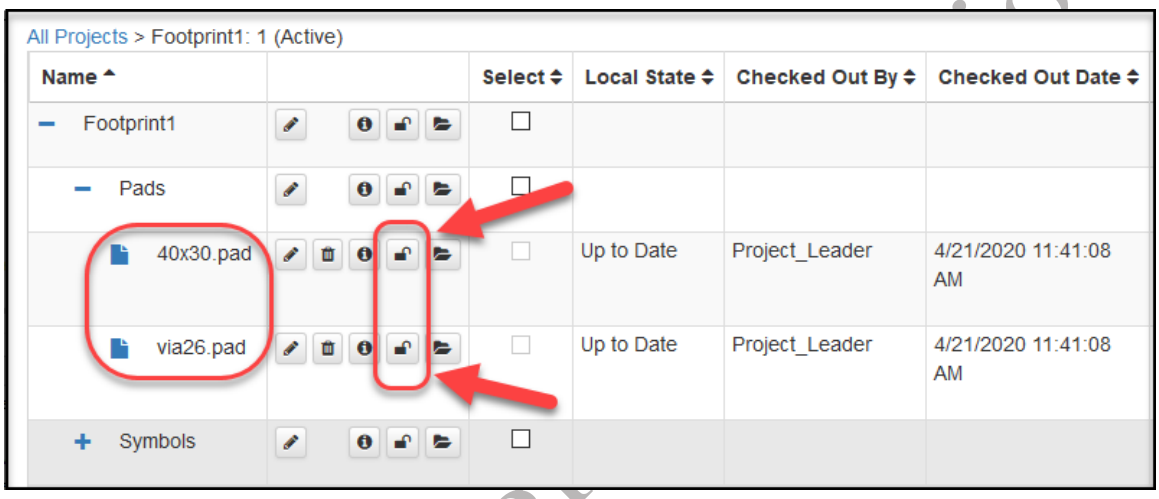


 EMA Design Automation, Inc ®© 2020 All rights reserved

The list may show two files that are still checked out. These pad files did not have any work performed on them and therefore did not appear in the list for check in.

As Project Leader, you can cancel the check out for these 2 files so they will be available for future editing.

4. In the File view, click on the **Cancel Check Out** button to the right of the 40x30.pad and the via26.pad, as shown in the following graphic.



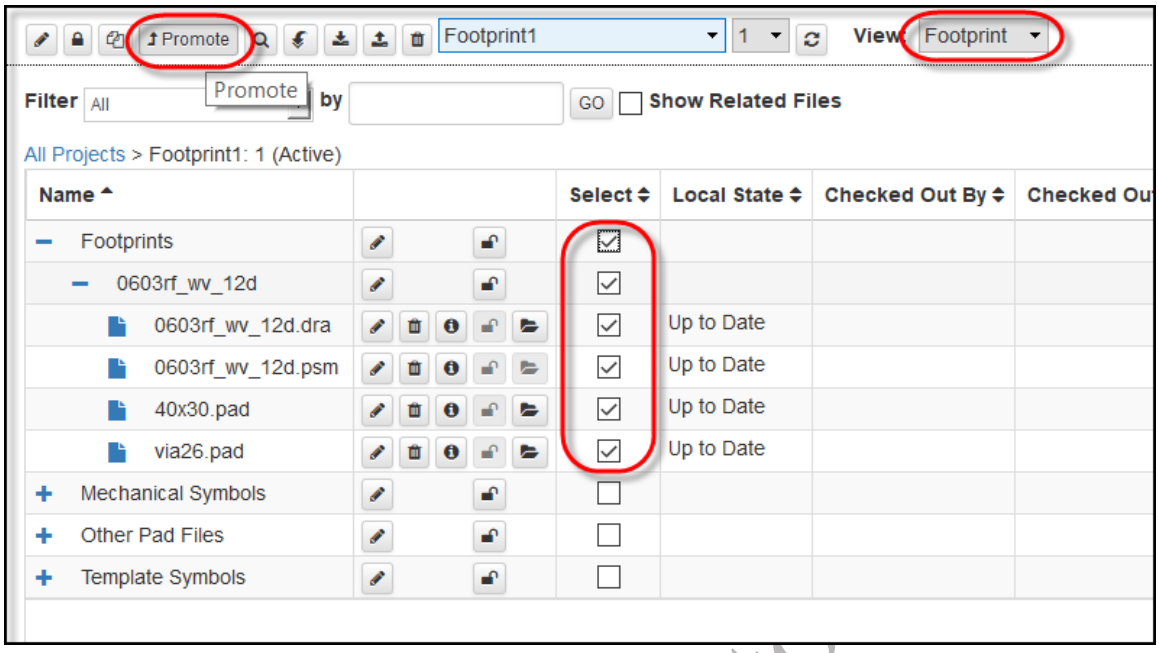
Promoting the Completed Footprint

Once all the edits are completed on a footprint, and the Project Leader has verified that the changes are acceptable, the footprint may now be promoted to the master library location where it can be used by the entire team.

To do this, the view needs to be set to Footprint, and the top-level footprint selected from the list. Next, the Promote button is selected.

EMA Design

 EMA Design Automation, Inc ®© 2020 All rights reserved



For a footprint to be promoted, the Promote Library path must have been established. You will not be promoting a footprint at this time.

EMA Design Auto

 EMA Design Automation, Inc ®© 2020 All rights reserved

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

Lesson 6: Final Package Release

Lesson Objectives

After you complete this lesson you will be able to:

- Create a Package Release zipped file for a project

In this lesson we'll explore the steps to generate a board file that is associated with a schematic project file. We'll also look at releasing the final package when work has been completed.

Releasing the Package

When you have completed a development phase, you can use EDM to create a package for release. This package may be used as an archive or sent to contract manufacturing.

Note: This feature is ***only*** available from the **EDM Desktop** application and is unavailable when you open EDM from Capture or an EDM supported browser.

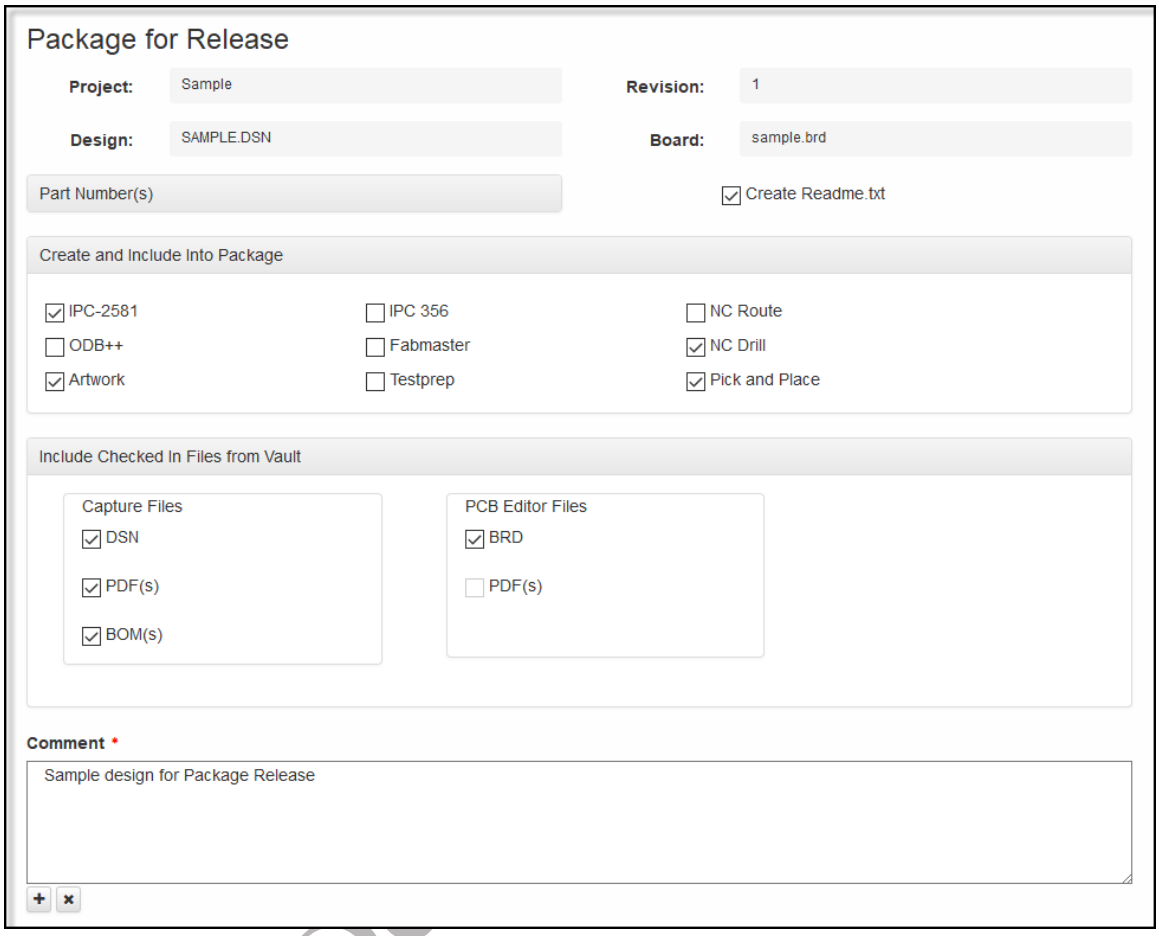
The following steps would be used to generate a Package Release

- Open the EDM Desktop Application. Open your project from the Project Dashboard.
- Select the version of your project that you want to package for release.
- Click the Package for Release button. This will open the Package for Release form.
  - If your managed design includes a part number, click the Part Number(s) heading to expand or collapse the display and verify the part number(s).
- Select the Create Readme.txt check box to include a file that identifies the content included in the Package for Release zipped folder you are creating.
- Select the check boxes in the "Create and Include into Package" section of the page to indicate the items you want to automatically generate when the Package for Release is created. These items will be placed into a zipped file that will be created in your project directory and automatically checked in.

EMA Design Automation

 EMA Design Automation, Inc ®© 2020 All rights reserved

The Package for Release Form

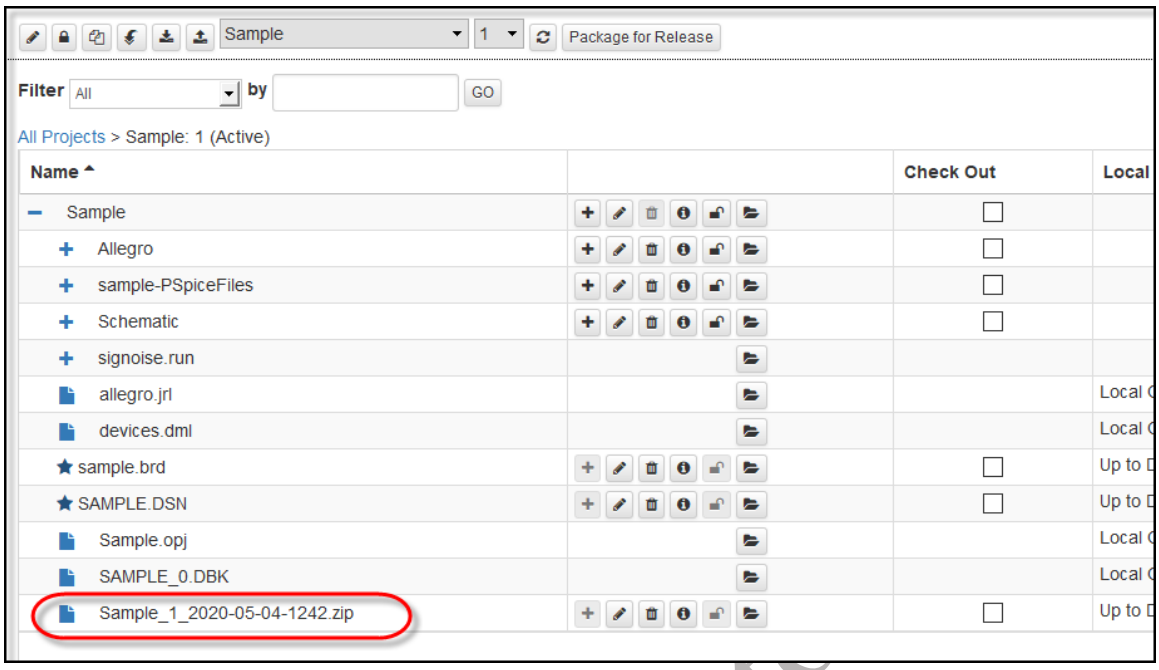


EMA D

The Release Package Zip File

Clicking the Add button will generate the released package in a zipped file.

 EMA Design Automation, Inc ®© 2020 All rights reserved



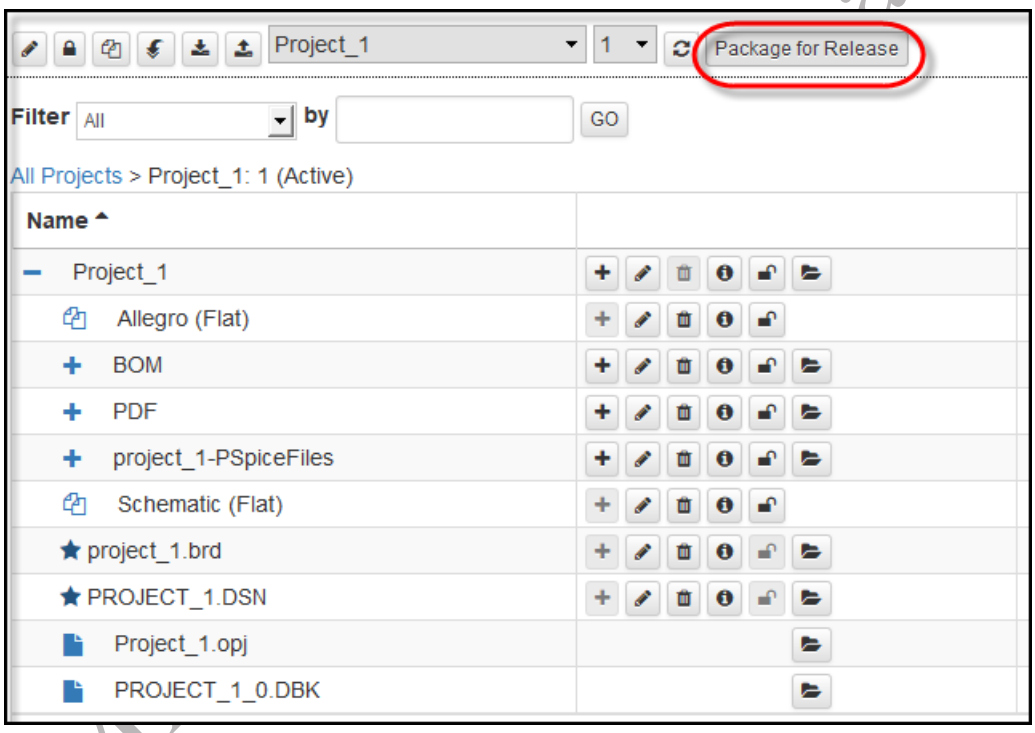
EMA Design Auto

 EMA Design Automation, Inc ®© 2020 All rights reserved

Lesson 6-1: Generating a Package Release File for a Project

In this lab you will generate a Package Release using a previously generated project.

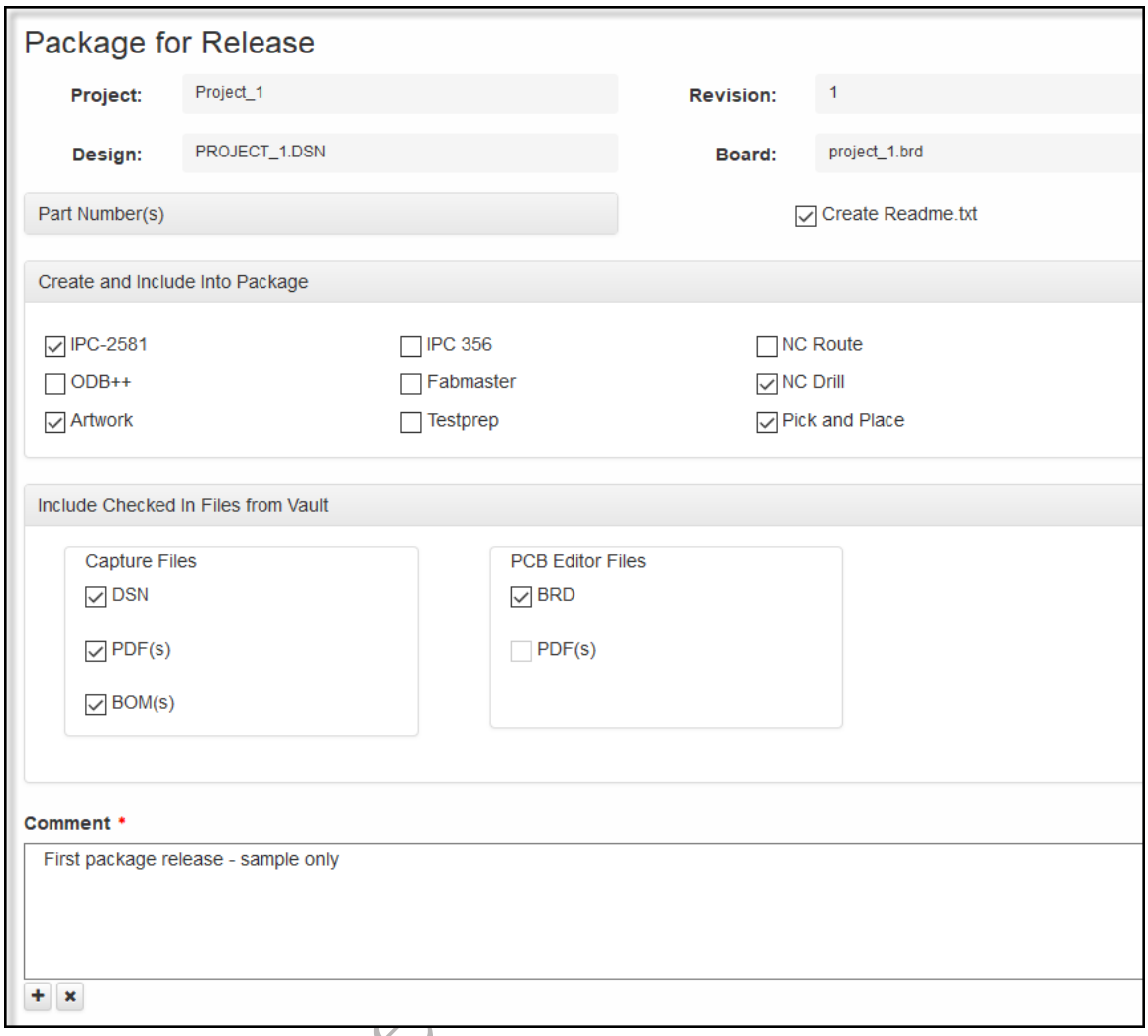
1. From the Projects list select the Project you recently imported a board file into (Lab 4-2)
2. Select **Package for Release**



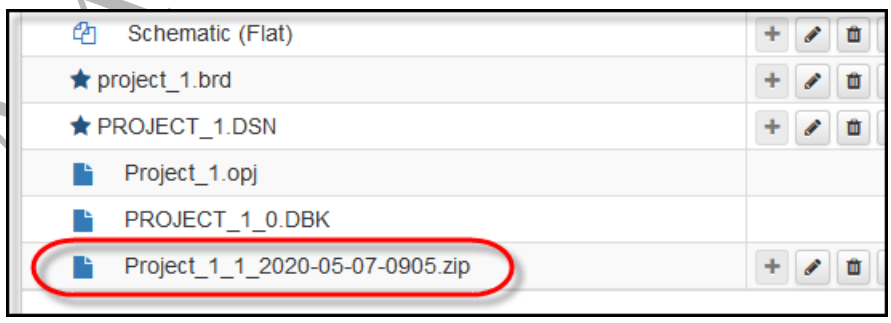
3. When the Package for Release form opens fill out the information as shown.

EMA Education

 EMA Design Automation, Inc ®© 2020 All rights reserved



4. Click the plus sign to create the package release zip file.



This completes EDM training.

 EMA Design Automation, Inc ®© 2020 All rights reserved