



OrCAD Engineering Data Management (EDM)

Table of Contents: [Page 1](#)

First Things First: [Page 2](#)

Starting a new project in EDM and your first check in: [Page 8](#)

Check in and check out PCB Editor board files: [Page 15](#)

Packaging your PCB and schematic for release: [Page 18](#)

Creating a new project revision: [Page 25](#)

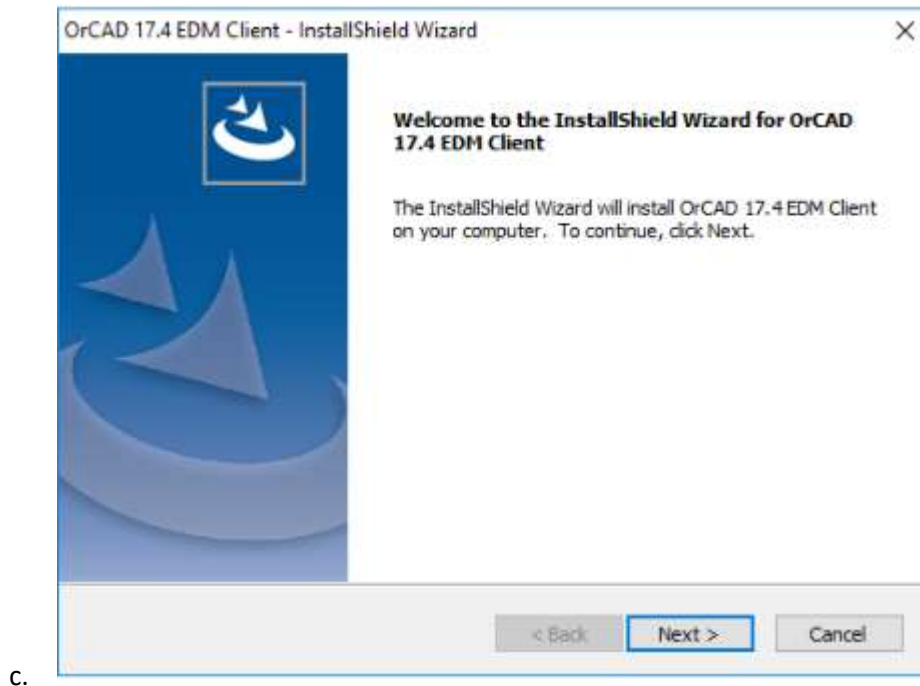
Do's and Don'ts: [Page 27](#)

First Things First:

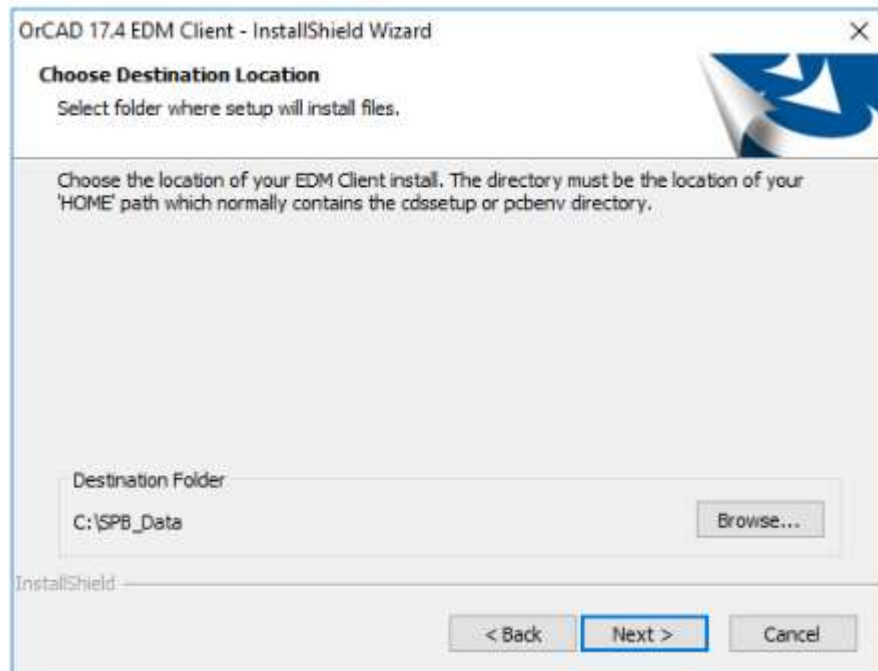
Installation:

EDM Install:

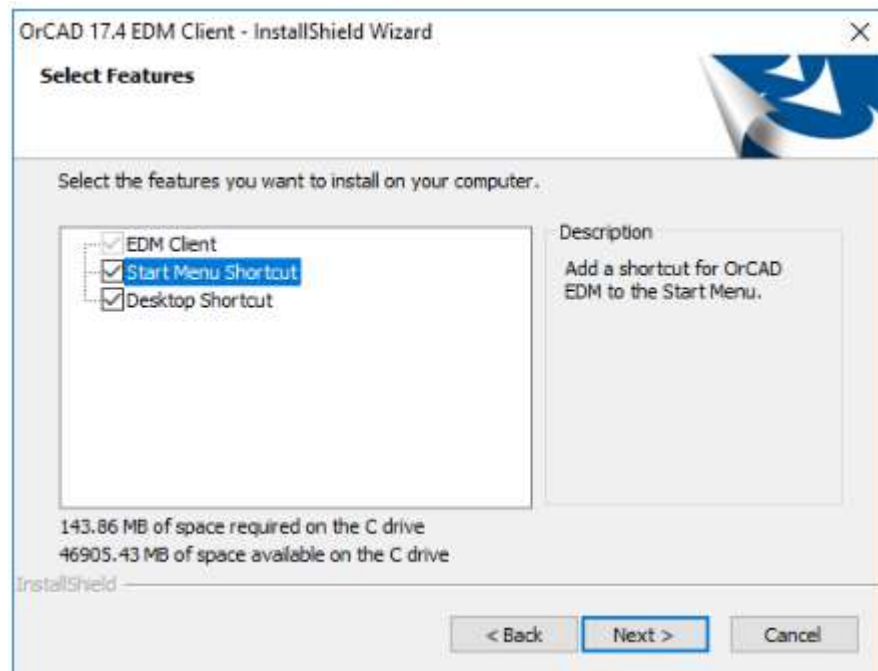
1. Make sure you are running the latest version of OrCAD before installing the EDM Client software. Go to the following address to run the OrCAD updater:
\\atlas\ProjectLibrary\TechServices\Software\ORCAD_UPDATES\Cadence_17-4\
2. Run the EDA Client EXE in from the following location:
\\atlas\ProjectLibrary\TechServices\Software\ORCAD_UPDATES\EDM174\EDM1740P1\
 - a. The installation home screen will open
 - b. Click **Next**



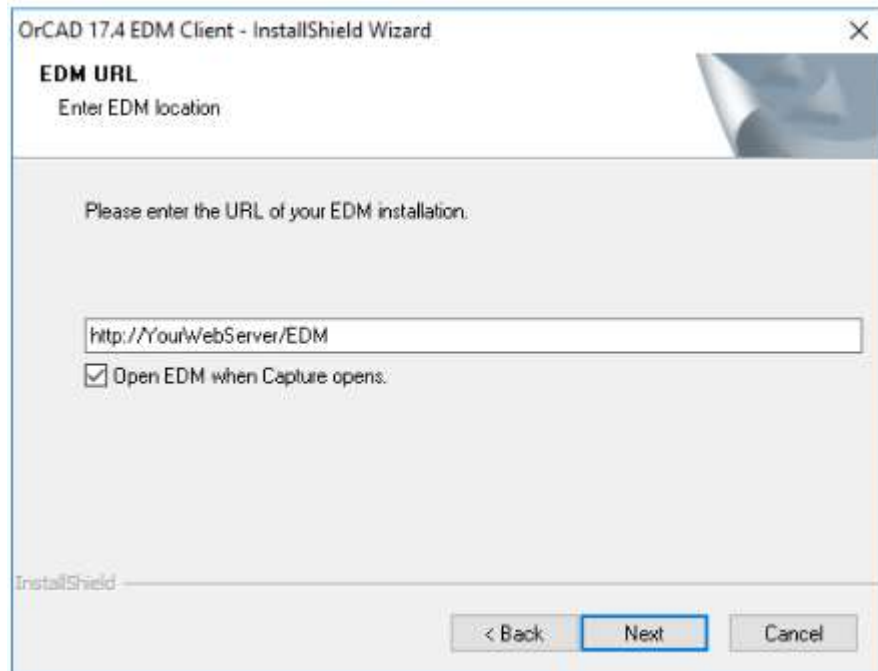
- d. Use the default destination location. Click **Next**.



- e. Select which shortcuts you want to add then click **Next**.



- f. Enter: <http://water2/edm> for the EDM URL then click **Next**.



OrCAD 17.4 EDM Client - InstallShield Wizard

EDM URL
Enter EDM location

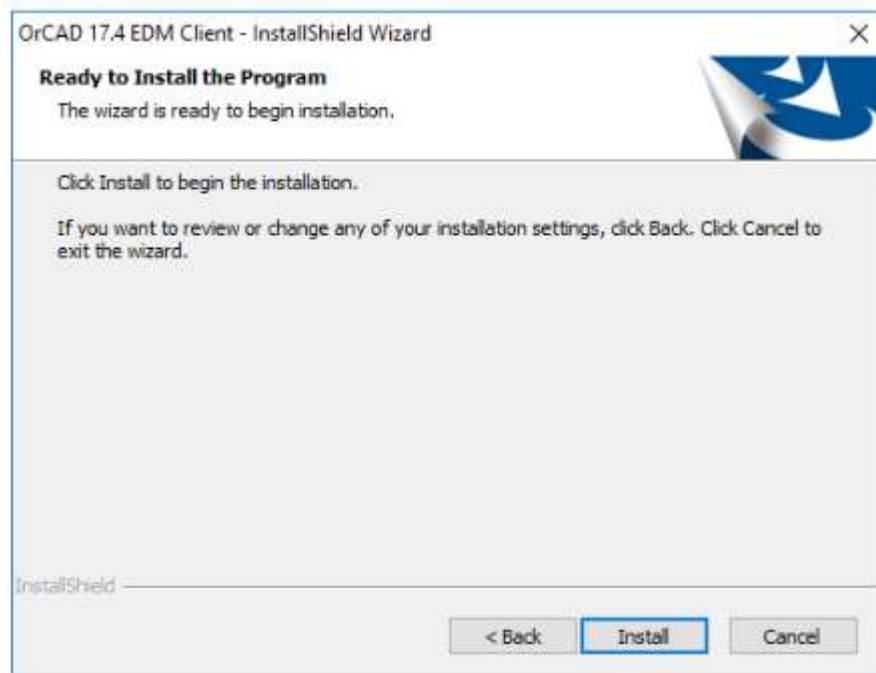
Please enter the URL of your EDM installation.

☒ Open EDM when Capture opens.

InstallShield

< Back Next Cancel

- g. Click Install



OrCAD 17.4 EDM Client - InstallShield Wizard

Ready to Install the Program
The wizard is ready to begin installation.

Click Install to begin the installation.

If you want to review or change any of your installation settings, click Back. Click Cancel to exit the wizard.

InstallShield

< Back Install Cancel

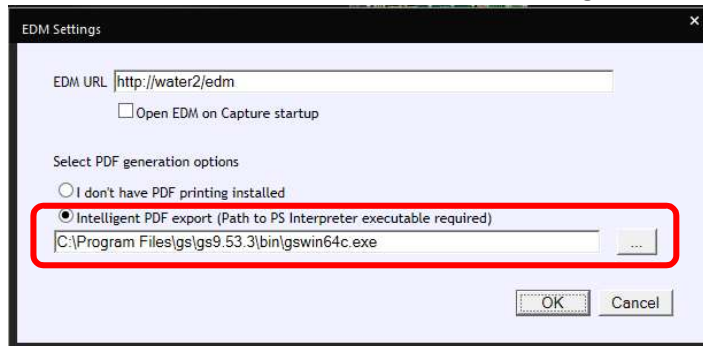
PDF print settings:

EDM can create PDFs of your Capture schematic and PCB Editor artwork/gerber files during every check in and when you use the Package for Release feature as long as you have some sort of PDF interpreter software installed in your computer.

1. Download and install a PDF interpreter such as Ghostscript's PDF interpreter.
 - a. <https://www.ghostscript.com/download/gsdnld.html>
2. Open OrCAD Capture CIS
 - a. Click **EDM > Settings** from the top menu bar.



- b. Enter the address of the executable in the Intelligent PDF Export address box.

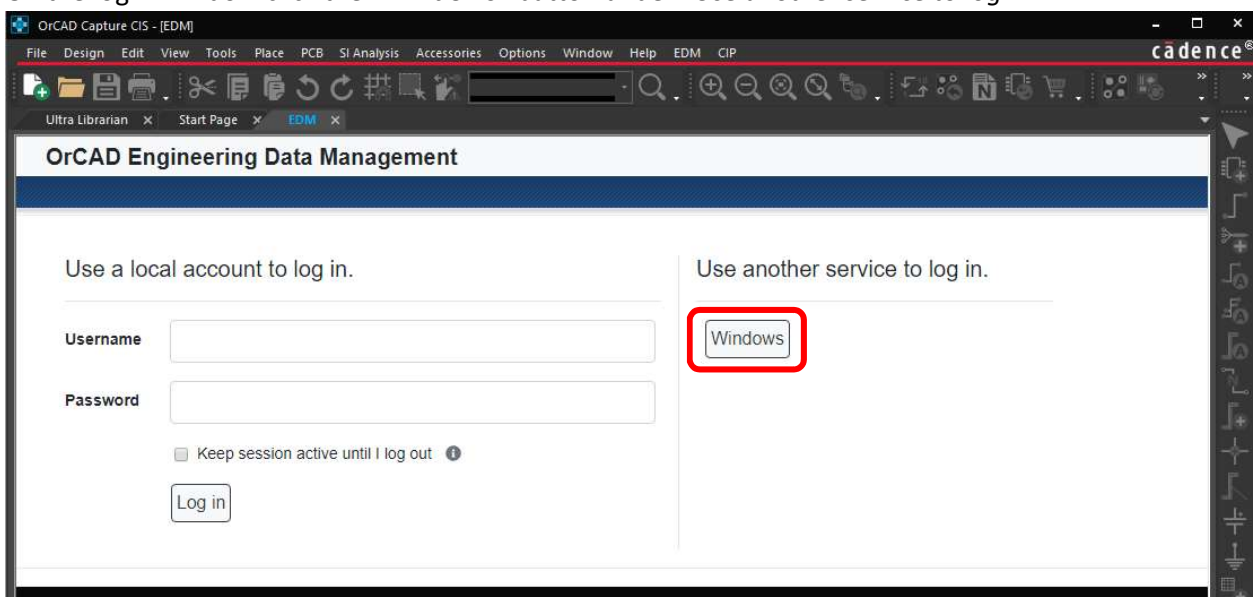


- c. Click **OK**.

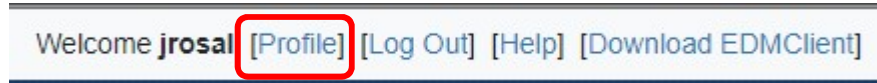
Setting up your home workspace folder:

EDM needs a location either on the network or on a local hard drive to act as a workspace for your projects. Once this is set, EDM will transfer the database files to this directory allowing you to review or edit your files until you are ready to check them back in.

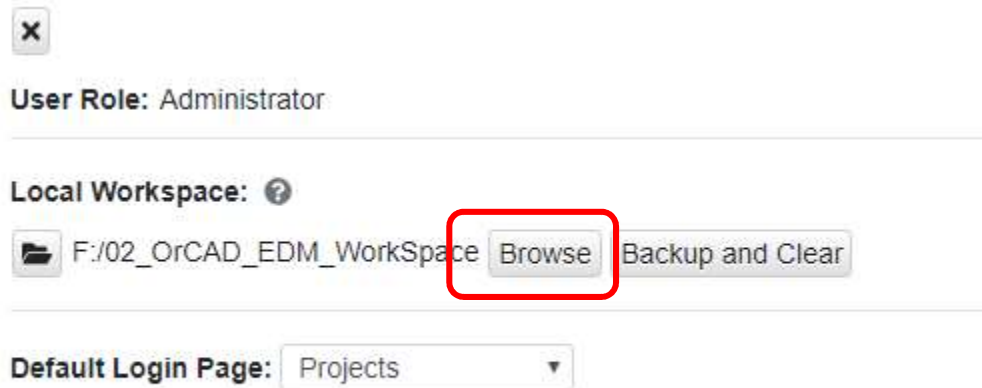
1. Open Capture CIS: **Start > CadencePCB 17.4-2019 > Capture CIS 17.4**
2. On the top menu click **EDM > Open EDM**
3. On the log in window click the **"Windows"** button under *"Use another service to log in."*



- a. Contact Jose Rosal or James McClure if you need help with logging in.
4. You will see several menu items on the top left of your screen. You need to click **Profile**.



5. The profile form opens. Click **Browse**. Either create or select a folder to act as your workspace

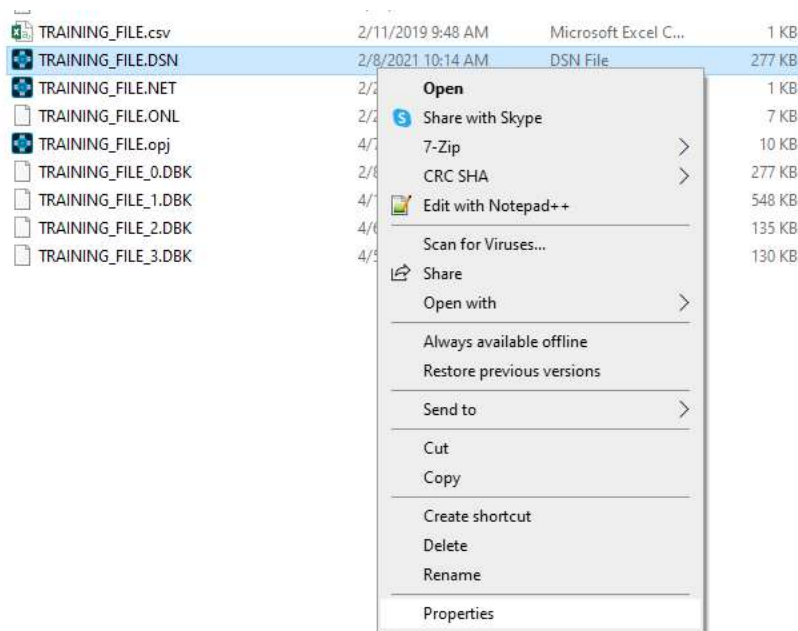


folder.

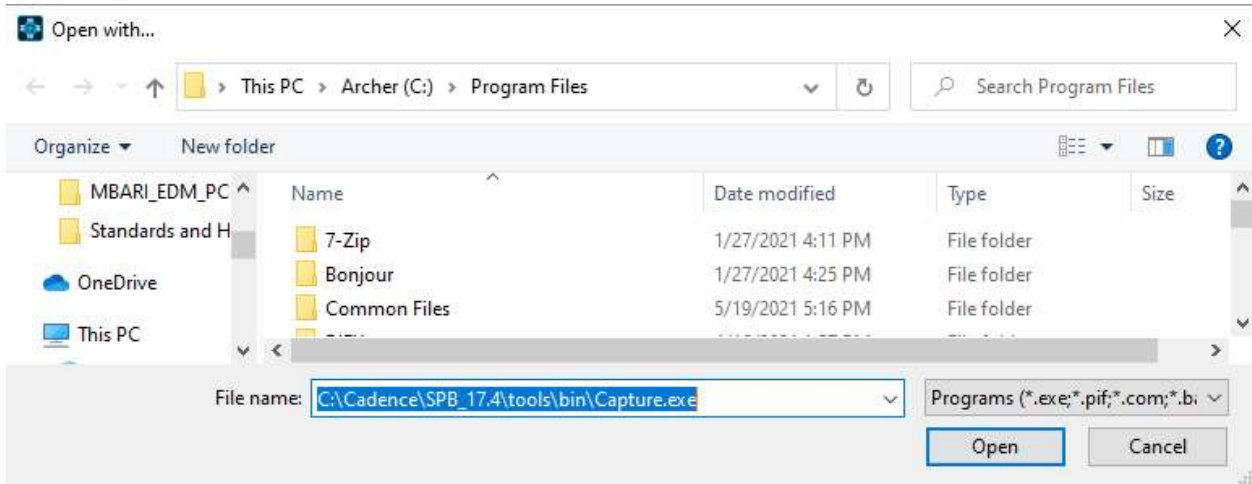
Associating DSN files with OrCAD Capture.

You may ignore this step if the DSN file extension on your computer is already associated with OrCAD Capture. If you're not sure, find an OrCAD DSN file and double click it. If a program other than OrCAD Capture opens, you'll need to complete the following steps to correct it.

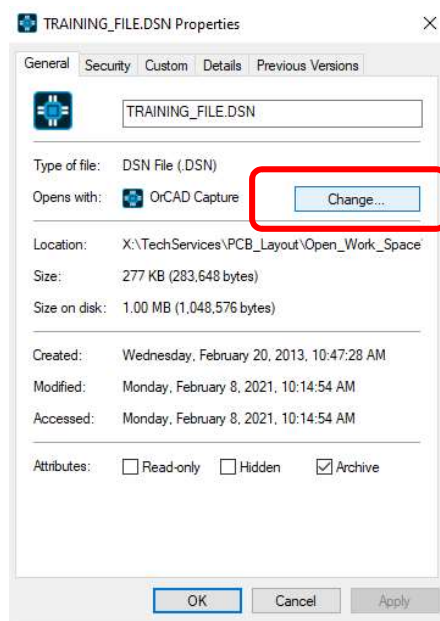
1. Find a OrCAD Capture DSN file. Right Click the file and select **Properties** in the sub menu.



2. The Properties window opens. Click the **Change** button.
3. A selection window opens. If you don't see OrCAD Capture on the list, click **"Look For Another App on this PC"**.
4. In the *Open With...* window, copy the following address and paste it into the File Name text field.
 - a. C:\Cadence\SPB_17.4\tools\bin\Capture.exe



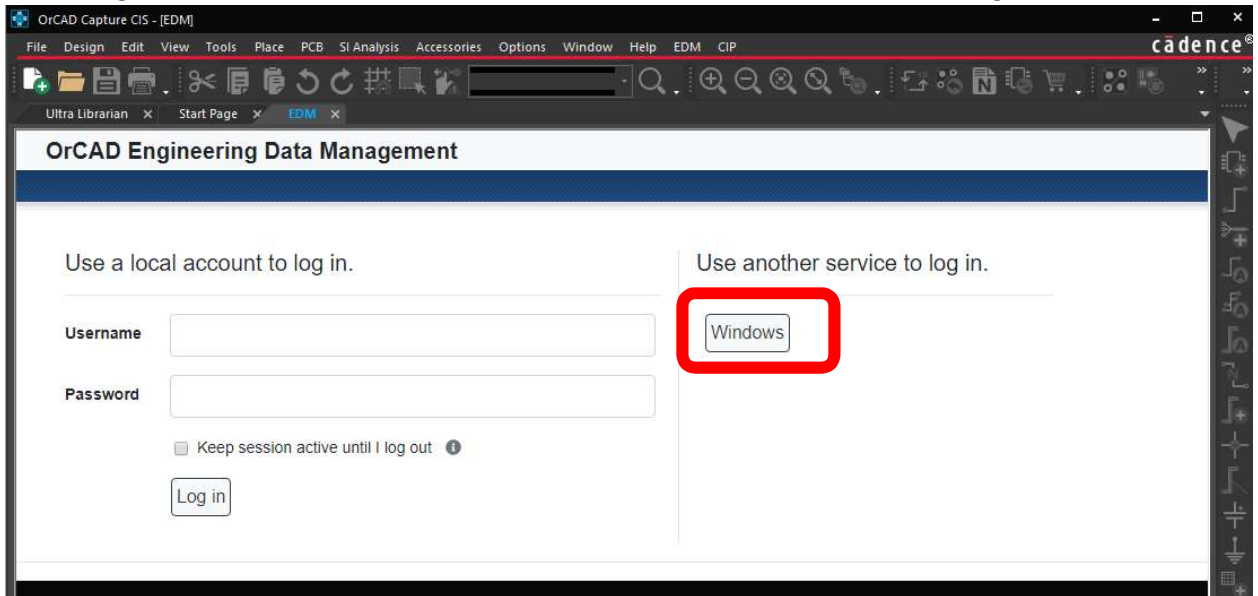
- b. Click **Open**.



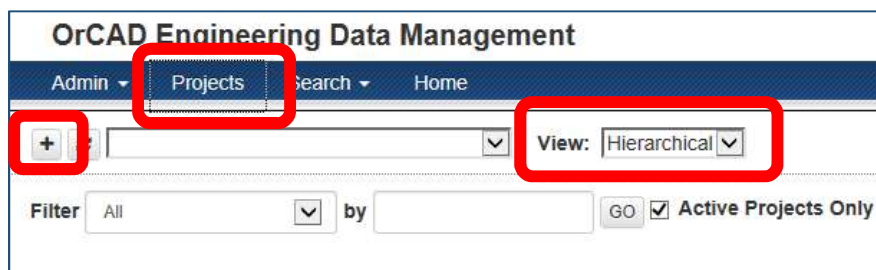
- c. Click **Apply** in the properties window then click **OK**

Starting a new project in EDM:

1. Go to Canyon Head and get a new unique MBARI drawing number.
<https://mwww.mbari.org/EngDraw/>
2. Open Capture CIS: **Start > CadencePCB 17.4-2019 > Capture CIS 17.4**
3. On the top menu click **EDM > Open EDM**
4. On the log in window click the “**Windows**” button under “Use another service to log in.”



- a. Contact Jose Rosal or James McClure if you need help with logging in.
5. Once logged in, change “**View**” to Hierarchical.
 6. Click “**Projects**” to get to the project window.
 7. Click the “**+**” button to create a new project.



8. Fill in the New Project window.

New Project

Project Name*

Revision*

Vault*

Project Group*

New ☒

Existing ☐ 123456789_SampleDrawingPackage

Project Type

Create from Template ☐ SimpleCaptureProjectTemplate

☒ Use Project Name

Design

Board

Allegro Config Browse

Allow Users to Check in New Folders ☒

Project Status

Project Leaders ☐ Assign All

Project Users

MBARI Project Name

Engineer

MBARI Drawing Number

MBARI Project Number

Designer

Subsystem

System

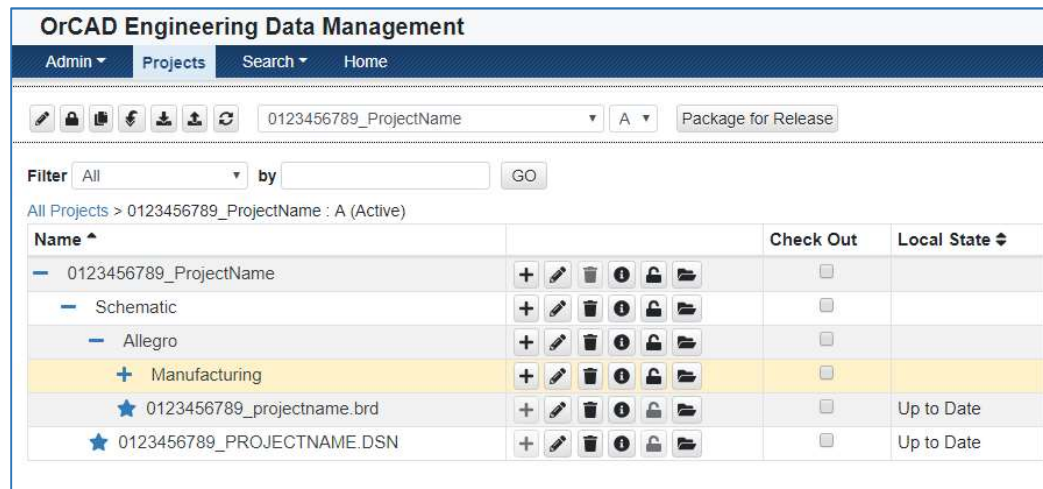
Comment*

+ x ☒ Activate on Add

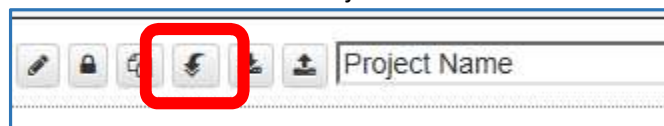
- a. Project Name:
 - i. This name must be unique. **Use the following naming convention:**
"DrawingNumber_DrawingName" Examples:
 1. 012345678_AnotherVicarBoard
 2. 987654321_SystemController
- b. Revision:
 - i. Enter "A" if this is will be the first release of this board.
- c. Vault:
 - i. Select **"Vault"**.
- d. Project Group:
 - i. Enter the name of your project group if it doesn't already exist in the drop down menu. Make sure the name you choose does not contain spaces. Examples:
 1. ESP3G
 2. LRAUVFontEndDome
 3. PowerBuoyBatteryBox
 4. NSFOCE_PowerBox
- e. Project Type:
 - i. Select Capture Design from the drop down menu.

1. Used for schematics, layouts, and PSpice models. This should be the default selection.
- f. Create from Template:
 - i. You **MUST** use one of the templates listed in this menu. Click the check button and select one of the options from the drop down menu.
 1. MBARI_EDM_TEMPLATE_2LYR: Creates files needed for 2 layer PCB.
 2. MBARI_EDM_TEMPLATE_4LYR: Creates files needed for 4 layer PCB.
 3. MBARI_EDM_TEMPLATE_WIRING: Does not contain folders and output files needed to create a PCB.
- g. Check “Use Project Name”
- h. Netlist Configuration Path:
 - i. Leave this blank to use the default configuration file.
 1. This changes the default configuration file, Allegro.cfg, used by Capture when a netlist is created.
- i. Allow Users to Check in New Folders.
 - i. This box as checked by default.
 1. Allows users to check in new folders for projects. If this option is not enabled, Project Leaders will need to create folders first before users can check in files to custom folders.
- j. Project Status:
 - i. Default selection for new projects should be “Active”. You can use “Released” or obsolete for archiving old projects that were not entered into the EDM workspace.
- k. Project Leaders and Project Users:
 - i. Select all project leaders and project users that apply.
 1. Only assigned Project Leaders are able to create folders and manage the project. Project leaders can check in/out projects, delete files, create folders, and assign new project users.
 2. Project users or user group members are only able to check out and check in files, pages and symbols that they have been assigned.
- l. MBARI Project Name:
 - i. Enter the MBARI Project Name. Example:
 1. ESP
 2. LRAUV
- m. Engineer:
 - i. Engineer’s name
- n. MBARI Drawing Number:
 - i. MBARI generated drawing number from Canyon Head. Use the drawing number you created from step 1.
- o. Designer:
 - i. Name of the PCB Layout designer.
- p. Subsystem:
 - i. Name of the subsystem this project belongs to.
- q. System:
 - i. Name of the System this project belongs to.

- r. Comment:
 - i. You must add a comment to proceed. Since this is a new project, “Initial project creation” is a good choice.
 - s. Activate on Add:
 - i. Check this button to activate the project during its creation.
 - t. Click the “+” button when done.
9. Opening your project:
- a. Click the Projects button on the top of the EDM user interface.
 - b. Click the “+” button next to the desired group name. This will expand the list of projects that belong to this group.
 - c. Click the “+” button next to the desired project name. This will expand the list of releases that belong to this project.
 - d. Select the desired release.
 - e. The following window should appear. Note that this was created using the "MBARI_EDM_TEMPLATE_2LYR" template.

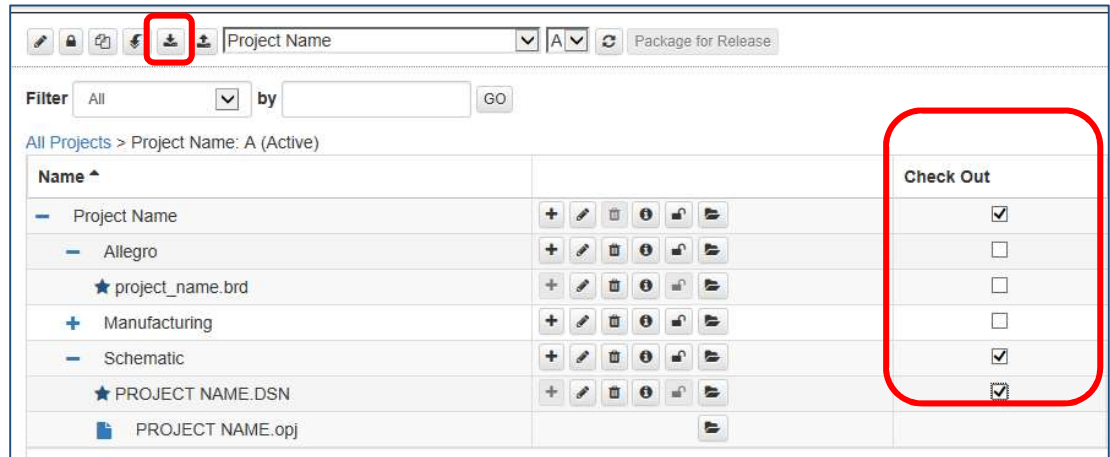


- f. Click the “Full Download Project” button.

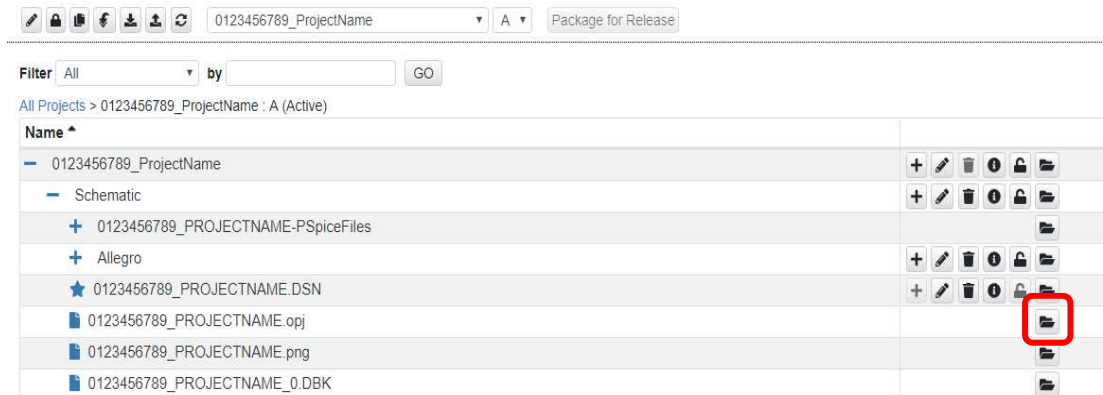


- i. This will autogenerate the BRD and DSN files and download them into your working directory.

- g. Click the **Check** button for the DSN File and click the **Checkout** button on top of the page.



- i. This will check out the design file so you can make edits.
- h. Click the **Open** button next to the OPJ file to open the project.



- i. Your drawing project will open. You will notice that the title block will NOT be filled correctly at first. **DO NOT** edit the title block at this time. This will be done automatically during each check in. After you are done with your first edits, save the project and DSN file and close the files.
- NOTE: In order to use EDM's version control features such as its automatic title block fill in feature, you must make some sort of edit to your schematic (add text, add a part, etc...). You must also use the **EDM client within Capture CIS** when checking your schematic back into the EDM database. EDM will not check in your work if the files in your workspace matches what is available in the database.
 - NOTE: The "Page Title" title block field will automatically take the name of your schematic page. Change the name of your schematic page in the OrCAD project window to modify this value.

- iii. Example of the title block before first check in:

 Monterey Bay Aquarium Research Institute 7700 Sandholdt Road, Moss Landing, CA 95039		Schematic Name: SCHEMATIC1		
		Page Title: PAGE1		
		Project Name: <Project Name>		
		Group Name: <Group>	System: <System>	Subsystem: <Subsystem>
Drawing Number: <MBARI DN>		Release: <Rev>	Version: <Version>	Status: <Status>
Page Modify Date: Thursday, October 29, 2020		Check In Date: <LCID>		Size: B
		Sheet <Page Number>		Page Count: <Page Count>

- j. **Log out** of EDM at this time to free up the network license seat for others to use. At this point, you can edit your schematic until you are ready to check it back in.

Welcome **jrosal** [Profile] [Log Out] [Help] [Download EDMClient]

About 6.0.2.7

- k. When you are ready to check your drawing in, go back to the EDM tab in the OrCAD Capture and log back in. If the EDM tab is not available, click **EMD > Open EDM** using the top menus in OrCAD Capture CIS. Note, you need to use the client EDM software **WITHIN** Capture to be able to update your Capture title block. Navigate back to your project using the steps **9a-9e**.
- l. Click the **“Check In”** button.

Project Name

- m. The check in window will appear.

OrCAD Engineering Data Management

Welcome **jrosal** [Profile] [Log Out] [Help] [Download EDMClient]

Admin Projects Search Home About 6.0.2.7

Check-in

Items to be checked in	State	CI*	Keep Checked Out	Status
NewGroup/Project Name/A/Schematic/PROJECT NAME.PSpiceFiles	Local Only	☐	☐	
NewGroup/Project Name/A/Schematic/PROJECT NAME.DSN	Up to Date (Mod)	☒	☐	

Select ... All None Modified Local Only Also Check Out ☐

Auto Generate Related Files

☒ BOM

☐ Netlist

☒ PDF

☐ Include Variant Files

Apply Status/Fields to Generated Related Files

☒ BOM

☐ Netlist

☒ PDF

☒ Update Title Block

Auto Comment

☒ Parts Changed

☐ Total # of Parts

☐ Total # of Nets

☒

- i. Check the DSN **“CI”** (Check In) button on the top of the screen.
- ii. Under **“Auto Generation Related Files”**, you may choose **PDF** and **BOM** now to start the database association process. This will create PDFs of all your schematic pages and your bill of material. You will have the option to update

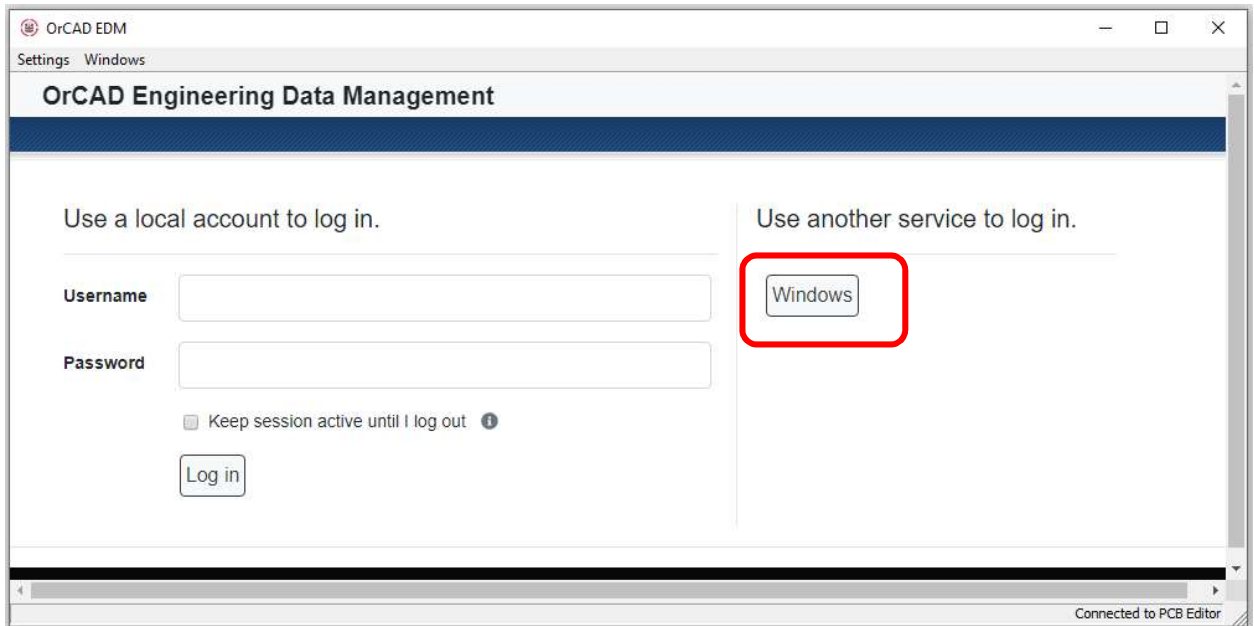
these files every time you check in your schematic. You may also choose to wait until you are further along with your design before creating them. Just keep in mind that these items must be created BEFORE using EDM's Package for Release feature. The EDM packager can only add these files and cannot create them.

- iii. **Update Title Block** must be checked.
- iv. Auto Comments create the described comment.
- v. You **MUST** add a comment in the Comment section if you don't choose one of the auto comments.
- vi. Check the **"Save"** button on the bottom of the screen when done.
- vii. The window will flicker as all the auto-generated files are created and the title block is updated with your project information.
- viii. The following is an example of the title block after the first check in.

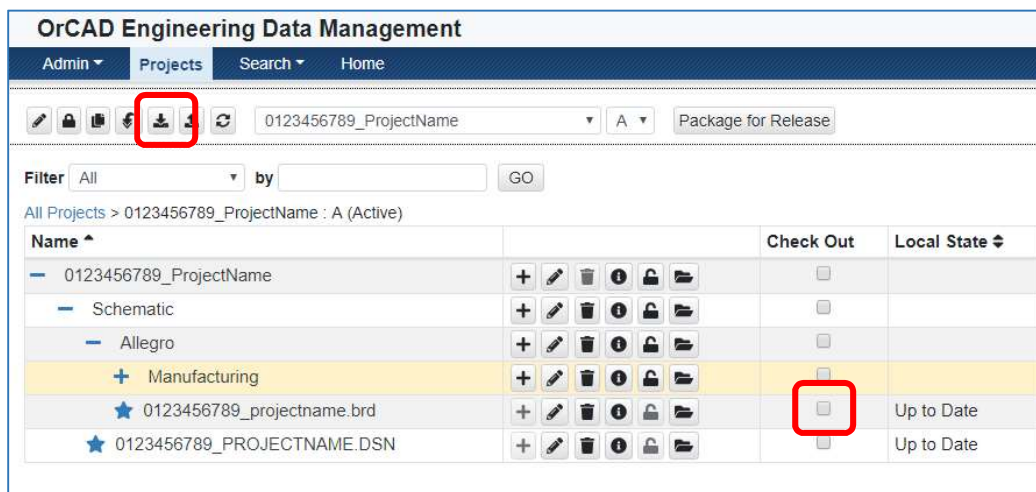
 Monterey Bay Aquarium Research Institute 7700 Sandholdt Road, Moss Landing, CA 95039		Schematic Name:			
		Name of Schematic			
		Page Title:		Engineer:	
		Schematic Page Name		Engineer	
		Project Name:		Designer:	
0123456789_ProjectName		LayoutDesigner			
Group Name:	System:	Subsystem:	Last Updated By:		
ProjectGroup	System	Subsystem	jrosal		
Drawing Number:	Release:	Version:	Status:	Size:	
0123456789	A	2	Active	B	
Page Modify Date:	Check In Date:	Sheet 1 of 1			
Friday, May 28, 2021	5/28/2021 10:58:50 AM				

How to check in and check out PCB board files

1. Open the EDM Client 17.4 Desktop application. Go to the windows search bar and type “EDM” to locate the application.
2. On the log in window click the “**Windows**” button under “Use another service to log in.”



- a. Contact Jose Rosal or James McClure if you need help with logging in.
3. Click the **Projects** button on the top of the EDM user interface.
 4. Click the “+” button next to the desired group name. This will expand the list of projects that belong to this group.
 5. Click the “+” button next to the desired project name. This will expand the list of revisions that belong to this project.
 6. Select the desired revision.
 7. The following window should appear. Note that this was created using the “MBARI_EDM_TEMPLATE_2LYR” template. Using a different template may change the project structure.
 8. Click the **Check Mark** button next to the BRD file then click checkout.



9. Your file will open in PCB Editor.

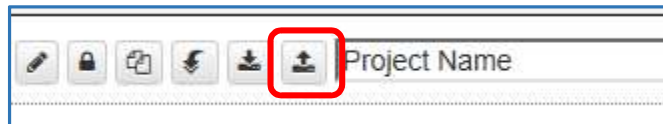
- a. You will notice that the title block will NOT be filled correctly at first. **DO NOT** edit the title block at this time. This will be done automatically during each check in.
 - i. NOTE: In order to use EDM's version control features such as its automatic title block fill in feature, you must check in your PCB Editor file using the **EDM desktop application**. You must make some sort of edit to your layout file (add text, add a shape, add a part, etc..) to be able to check in your drawing. EDM will not check in your work if the file in your workspace matches what is available in the database.
 - ii. Example of the title block before first check in:

UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN INCHES TOLERANCES ON: 2 PL DECIMALS +/- 0.02" 3 PL DECIMALS +/- 0.005" ANGLES +/- 1 1 Degree FRACTIONS *	DESIGNED BY: ENGINEER	TITLE			
	LAYOUT BY: LAYOUT				
	CHECKED IN BY: LCIN	MONTEREY BAY AQUARIUM RESEARCH INSTITUTE 7700 SANDHOLDT RD MOSS LANDING, CA 95039			
	CID: CheckInDate				
	SYSTEM: SYSTEM	DRAWING NUMBER: DRAWING NUMBER	RELEASE: RELEASE	VERSION: VERSION	
SUBSYSTEM: SUBSYSTEM					
PROJECT NAME: ProjectName		STATUS: DrawingStatus	SCALE 1-1	SIZE: B	SHEET 1 OF 1

- b. **Log out** of EDM at this time to free up the network license seat for others to use.



- c. When you are ready to check your drawing in, and log back in to EDM using the **EDM Desktop Application**. Navigate back to your project.
- d. Click the **Check In** button.



- e. The check in window opens:
 - i. Check the **CI** check mark for the BRD file.
 - ii. Click **Update the Title Block**
 - iii. Add a comment describing any changes you might have made.
 - iv. Under **"Auto Generation Related Files"**, you may choose **PDF** now to start the database association process. This will generate PDFs of all the gerber drawings listed in the Artwork Control Form in PCB Editor. You will have the option to update the PDFs every time you check in your drawing. You may also choose to wait until you are further along with your design before creating them. Just keep in mind that these items must be created **BEFORE** using EDM's Package for Release feature. The EDM packager can only add these files and cannot create them.

OrCAD Engineering Data Management

Admin Projects Search Home

Check-in

Items to be checked in	State	CI *	Keep Checked Out
ProjectGroup/0123456789_ProjectName/A/Schematic/0123456789_PROJECTNAME-PSpiceFiles	Local Only	☐	☐
ProjectGroup/0123456789_ProjectName/A/Schematic/0123456789_PROJECTNAME.png	Local Only	☒	☐
ProjectGroup/0123456789_ProjectName/A/Schematic/Allegro/0123456789_projectname.brd	Up to Date (Mod)	☒	☐
ProjectGroup/0123456789_ProjectName/A/Schematic/Allegro/Log	Local Only	☐	☐
ProjectGroup/0123456789_ProjectName/A/Schematic/Allegro/Log/ncustomization.log	Local Only	☐	☐
ProjectGroup/0123456789_ProjectName/A/Schematic/Allegro/Manufacturing/art_param.txt	Local Only	☐	☐
ProjectGroup/0123456789_ProjectName/A/Schematic/Allegro/Manufacturing/art_param.txt.1	Local Only	☐	☐
ProjectGroup/0123456789_ProjectName/A/Schematic/Allegro/Manufacturing/inc_param.txt	Local Only	☐	☐
ProjectGroup/0123456789_ProjectName/A/Schematic/Allegro/master.tag	Local Only	☐	☐

Select ... Also Check Out ☒

Auto Generate Related Files

☐ DB Doctor ☐ Apply Status/Fields to Generated Related Files ☒ Update Title Block

☒ PDF

Auto Comment

☐ Total # of Components ☐ Include DB Doctor Summary

☒ ☐

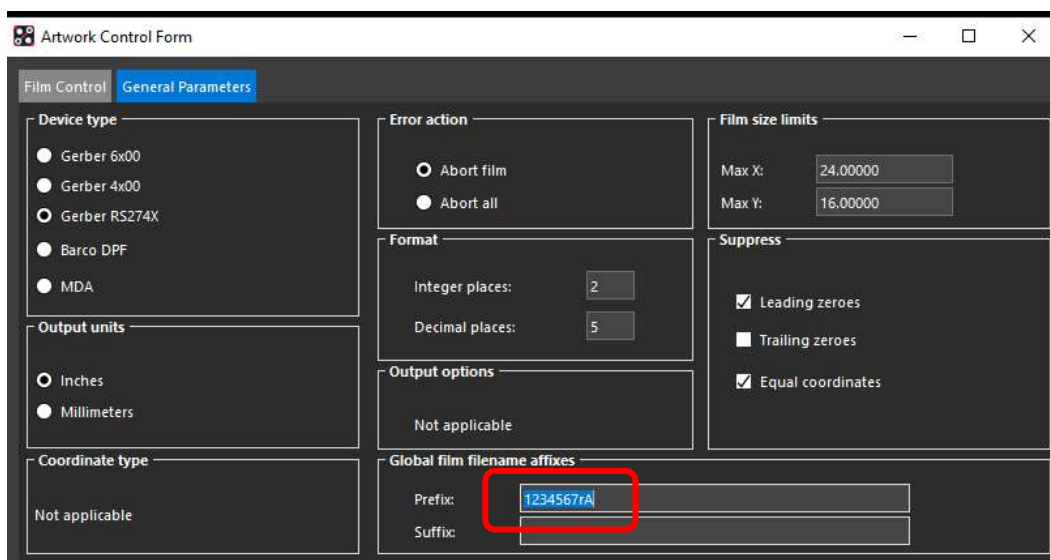
Comment *

- f. Click the **Save** button on the bottom left of the screen to complete check in.
- g. The following is an example of the title block after the first check in of your BRD file.

UNLESS OTHERWISE SPECIFIED	DESIGNED BY:Engineer	0123456789_ProjectName			
	LAYOUT BY:LayoutDesigner	MONTEREY BAY AQUARIUM RESEARCH INSTITUTE 7700 SANDHOLDT RD MOSS LANDING, CA 95039			
	CHECKED IN BY:jrosal				
	CID:5/28/2021 4:36:53 PM				
	SYSTEM: System				
DIMENSIONS ARE IN INCHES TOLERANCES ON: 2 PL DECIMALS +/- 0.002" 3 PL DECIMALS +/- 0.005" ANGLES +/- 1/1 Degree FRACTIONS +	SUBSYSTEM: Subsystem	DRAWING NUMBER: 0123456789	RELEASE: A	VERSION: 4	
	PROJECT NAME: 0123456789_ProjectName		STATUS: Active	SCALE 1-1	SIZE: B

Packaging a project for release.

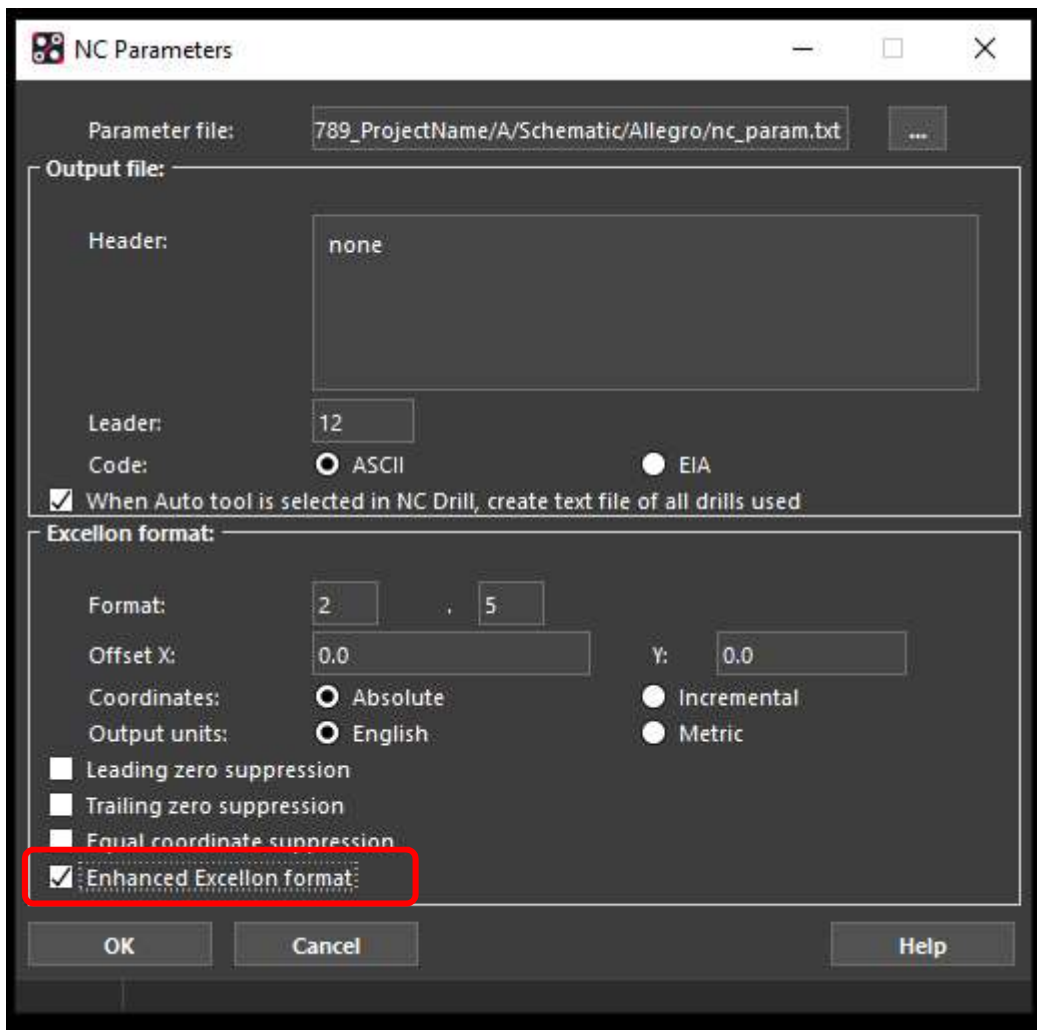
1. Check in your OrCAD schematic using the EDM client software within OrCAD Capture CIS. If you haven't done so already, make sure to pick the option to generate your bill of material if you plan on having the packager include it into your fabrication package.
2. Preparing PCB layout for packaging.
 - a. Open OrCAD PCB Editor.
 - b. Add a Prefix to your artwork/gerber files
 - i. Click **Manufacture > Artwork**



The screenshot shows the 'Artwork Control Form' dialog box with the 'General Parameters' tab selected. The form is divided into several sections: 'Device type' with radio buttons for Gerber 6x00, Gerber 4x00, Gerber RS274X, Barco DPF, and MDA; 'Output units' with radio buttons for Inches and Millimeters; 'Coordinate type' set to 'Not applicable'; 'Error action' with radio buttons for 'Abort film' and 'Abort all'; 'Format' with input fields for 'Integer places' (2) and 'Decimal places' (5); 'Output options' set to 'Not applicable'; 'Film size limits' with input fields for 'Max X:' (24.00000) and 'Max Y:' (16.00000); 'Suppress' with checkboxes for 'Leading zeroes' (checked), 'Trailing zeroes' (unchecked), and 'Equal coordinates' (checked); and 'Global film filename affixes' with input fields for 'Prefix' (containing '1234567rA' and highlighted with a red box) and 'Suffix'.

- ii. The Artwork Control Form opens. Click the **General Parameters** tab.
 - iii. Use the following prefix naming convention in the Artwork Control Form General Parameters.
 - iv. MBARIDrawingNumber + r + ReleaseLetter
 1. Example:
 - a. 01234567rA
 - b. 98765432rB
- c. Generate NC Drill File. Unfortunately, there is a bug in EDM that does not allow it to automatically create an Enhanced Excellon format drill file while using the project package feature. We need to create this file manually in order to add it to the manufacturing package.
 - i. In PCB Editor click **MANUFACTURE > NC > NC Parameters**

- ii. Make sure **Enhanced Excellon format** is checked. Click **OK**.



The image shows a software dialog box titled "NC Parameters". It contains several sections for configuring NC parameters. The "Parameter file" is set to "789_ProjectName/A/Schematic/Allegro/nc_param.txt". The "Output file" section includes a "Header" dropdown set to "none" and a "Leader" text box with "12". The "Code" section has two radio buttons: "ASCII" (selected) and "EIA". A checkbox "When Auto tool is selected in NC Drill, create text file of all drills used" is checked. The "Excellon format" section includes "Format" (2, 5), "Offset X" (0.0), "Y" (0.0), "Coordinates" (Absolute selected), "Output units" (English selected), and three unchecked checkboxes: "Leading zero suppression", "Trailing zero suppression", and "Equal coordinate suppression". The "Enhanced Excellon format" checkbox is checked and highlighted with a red rectangle. At the bottom are "OK", "Cancel", and "Help" buttons.

NC Parameters

Parameter file: 789_ProjectName/A/Schematic/Allegro/nc_param.txt

Output file:

Header: none

Leader: 12

Code: ☒ ASCII ☐ EIA

☒ When Auto tool is selected in NC Drill, create text file of all drills used

Excellon format:

Format: 2 . 5

Offset X: 0.0 Y: 0.0

Coordinates: ☒ Absolute ☐ Incremental

Output units: ☒ English ☐ Metric

☐ Leading zero suppression

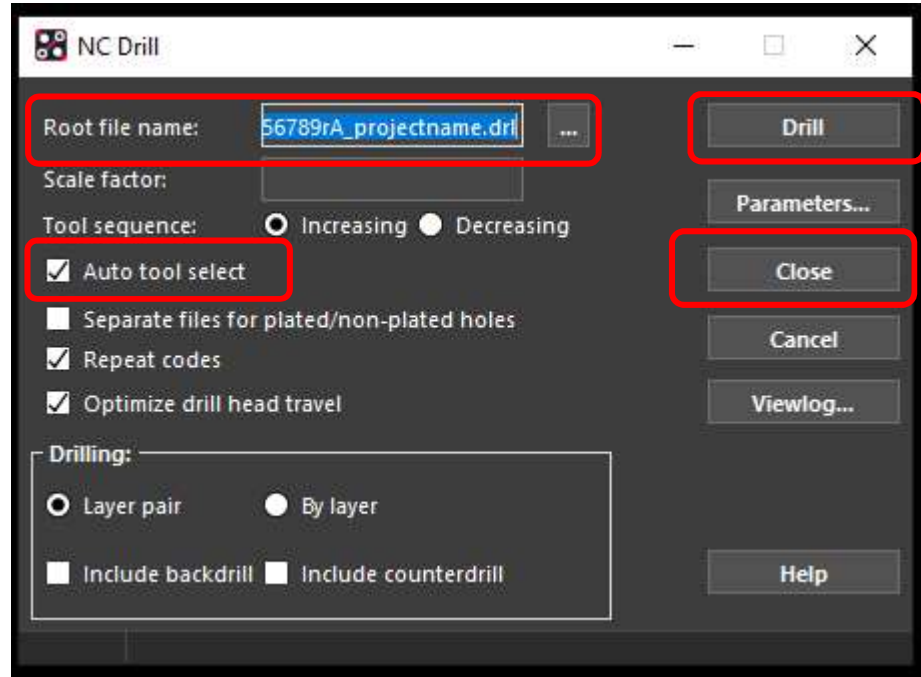
☐ Trailing zero suppression

☐ Equal coordinate suppression

☒ Enhanced Excellon format

OK Cancel Help

- iii. In the top menu bar of PCB Editor, click **MANUFACTURE > NC > NC Drill**



- iv. Make sure **Auto tool select** is checked.
- v. Next to the **Root file name** field click the “...” button to open the Save As window.
- vi. Navigate to the folder called “**AdditionalFiles**” within your project folder.
1. Note that the folder structure of your project should be the following:
 - a. ***Schematic > Allegro > Manufacturing > AdditionalFiles***
- vii. The name of your drill file should use the following naming convention:
1. ***MBARIDrawingNumber + r + ReleaseLetter + _ + ProjectName***
 - a. Example: 01234567rA_YourProjectNameHere
- viii. Click **Save** to close the Save As window.
- ix. Click **Drill**. PCB Editor creates a Drill file in the “**AdditionalFiles**” folder.
- x. Click **Close** to close the NC Drill window.
- d. Save and check in your PCB Editor design into EDM using the **EDM desktop application**.
- i. Make sure you select PDF output during the check in stage of your board file if you plan on including the PDFs of your gerbers in the fabrication package.
 - ii. Also check in the following items:
 1. Art_param.txt
 2. Nc_param.txt
 3. The DRL file you just created.
- e. Log on to EDM using the **desktop application** and navigate to your project. Click on the **Package for Release** button.

- i. Note: You must not have any associated project files open. All workspace files must be checked in and/or identical to the files located within the EDM database.

OrCAD EDM
Settings Windows

OrCAD Engineering Data Management

Admin ▾ Projects Search ▾ Home

0123456789_ProjectName A Package for Release

Filter All by GO

All Projects > 0123456789_ProjectName : A (Active)

Name ^		Check Out	Local State	Checked Out By
0123456789_ProjectName				
Schematic				
Allegro				
Manufacturing				
0123456789_PROJECTNAME.pdf			No Local Copy	
★ 0123456789_projectname.brd			No Local Copy	
0123456789_projectname.pdf			No Local Copy	
★ 0123456789_PROJECTNAME.DSN			No Local Copy	

3. The Package for Release form should open. Some options have been pre-selected using the Basic Release Package template. Next to “Include Additional Files”, click **Browse**.
4. Navigate to and select the **AdditionalFiles** folder mentioned in the last few steps. This will allow you to include the manually created drill file and any other file you might want to include in the manufacturing package.

Package for Release

Project: 0123456789_ProjectName **Revision:** A

Design: 0123456789_PROJECTNAME.DSN **Board:** 0123456789_projectname.brd

Template: Basic Release Package

Release Template

Include Additional Files: Browse

Variant Files:

	Part Number(s)	Type	Variant Name	Part Number
☒		DSN	Core_Design	
☒		BRD	Core_Design	

Create and Include into Package

- ☒ Artwork
- ☐ IPC-2581 -Rev: 1
- ☐ ODB++
- ☐ Fabmaster
- ☒ IPC-356
- ☐ Testprep
- ☐ NC Drill
- ☐ NC Route
- ☒ Pick and Place

Include Checked In Files from Vault

Capture Files

- ☐ DSN
- ☒ PDF(s)
- ☒ BOM(s)

PCB Editor Files

- ☐ BRD
- ☒ PDF

Read Me File:

- ☐ No Readme.txt
- ☒ Create Basic Readme.txt
- ☐ Use Template

Include [Release Options Summary] and [Error/Warning Files] ☐

Release Templates can be managed from Admin > Manage Templates > Release Templates.

Comment *

Packaged for Release using Release Template: Basic Release Package

☒

5. You may add to the comment field if necessary.
6. Press the **Package** check mark when you're ready to continue. Several windows will open and close automatically as the project is packaged.

7. A ZIP file with all the options you chose will be generated and saved into the EDM database. A copy of the ZIP file will also be in your workspace folder. This can be sent to PCB fabricators or assemblers.

Filter: All by GO

All Projects > 0123456789_ProjectName : A (Active)

Name ^		Check Out	Local State	Check
- 0123456789_ProjectName	+ [edit] [delete] [info] [lock] [unlock]	☐		
- Schematic	+ [edit] [delete] [info] [lock] [unlock]	☐		
+ 0123456789_PROJECTNAME-PSpiceFiles	[edit] [delete] [info] [lock] [unlock]			
- Allegro	+ [edit] [delete] [info] [lock] [unlock]	☐		
+ Log	[edit] [delete] [info] [lock] [unlock]			
- Manufacturing	+ [edit] [delete] [info] [lock] [unlock]	☐		
+ logs	+ [edit] [delete] [info] [lock] [unlock]	☐		
0123456789_PROJECTNAME.pdf	+ [edit] [delete] [info] [lock] [unlock]	☐	Up to Date	
0123456789_ProjectName_A_2021-06-03_1429.zip	+ [edit] [delete] [info] [lock] [unlock]	☐	Up to Date	
0123456789_projectname.brd	+ [edit] [delete] [info] [lock] [unlock]	☐	Up to Date	
0123456789_projectname.pdf	+ [edit] [delete] [info] [lock] [unlock]	☐	Up to Date	
netlist.log	[edit] [delete] [info] [lock] [unlock]		Local Only	
pstchip.dat	+ [edit] [delete] [info] [lock] [unlock]	☐	Up to Date	
pstxnet.dat	+ [edit] [delete] [info] [lock] [unlock]	☐	Up to Date	
pstxprt.dat	+ [edit] [delete] [info] [lock] [unlock]	☐	Up to Date	

8. At this point you can choose to mark this project as “Released” as a reminder to yourself or other users that this project is complete and was released to manufacturing.
 - a. Go to the project view and expand your project until you see the revision you want to edit.
 - b. Click the **Edit** button. It’s the one that looks like a pencil.

- ProjectGroup				
- 0123456789_ProjectName				
A	[edit] [delete] [info]	Active	Capture Design	☒

Edit Project

Project Name

JoseProject

Revision

A

Vault

Test_Vault

Project Group

JoseProject

Project Type

Capture Design

Allegro Config

Netlist Configuration Path...

Browse

Allow Users to Check in New Folders

☒

Project Status

Active
Active
Released
Obsolete

Project Leaders

Project Users

MBARI Project Name

Engineer

MBARI Drawing Number

MBARI Project Number

Designer

Subsystem

System

☒

- The Edit Project form opens. Select **Released** from the Project Status drop down menu.
- Click the **Save** button.
- Your project will now say **Released** in the status column.

Filter All by GO ☐ Active Projects Only

Group > Name > Revision ^		State	Project Type	User May Add Folder	Project Leaders	Assigned Users	Status	MBA
+ Coastal Biogeochemical Sensing								
+ Coastal Profiling Float								
+ CoastalBiogeochemicalSensing								
+ CoastalProfilingFloat								
+ JoseProject								
+ LRAUV								
+ Ocean Imaging								
- ProjectGroup								
- 0123456789_ProjectName								
A		Active	Capture Design	☒	jrosal		Released	0123
+ WEED LiFePO Battery Management System								
+ WEEDLifePOBatteryManagementSystem								

Creating a new Project Revision/Release

1. Open the project view window and navigate to your project.
2. Expand your project until you see its' revision information.
3. Click on the **Copy** button that is associated with the revision you want to upgrade.
4. The Copy to a New Revision form opens. In this example, the new revision letter will be "B" since the previous revision was "A". After adding a comment, click **Copy** button to complete the form.

Copy 0123456789_ProjectName: A to New Revision

Project Name	0123456789_ProjectName	Project Status	Active
Revision *	B	Project Leaders	jrosal
Vault	Test_Vault	Project Users	Select User
Project Group	ProjectGroup	MBARI Project Name	0123456789_ProjectName
Project Type	Capture Design	Engineer	Engineer
Allegro Config	Netlist Configuration Path... Browse	MBARI Drawing Number	0123456789
Allow Users to Check in New Folders	☒	MBARI Project Number	987654321
		Designer	LayoutDesigner
		Subsystem	Subsystem
		System	System

Options

☒ Activate

☒ Open after adding

Comment *

Comment goes here!



Your new revision should show up in the Projects window.

OrCAD Engineering Data Management

Admin ▾

Projects

Search ▾

Home

+

↺

▼

View: Hierarchical ▾

Filter

All ▾

by

GO

☐ Active Projects Only

Group > Name > Revision ^		State ⇅	Project Type ⇅	User May Add Folder ⇅
+ Coastal Biogeochemical Sensing				
+ Coastal Profiling Float				
+ CoastalBiogeochemicalSensing				
+ CoastalProfilingFloat				
+ JoseProject				
+ LRAUV				
+ Ocean Imaging				
- ProjectGroup				
- 0123456789_ProjectName				
A		Active	Capture Design	☒
B		Active	Capture Design	☒
+ WEED LiFePO Battery Management System				

Do's and Don'ts

1. **DO** use the EDM client software within OrCAD Capture to check in OrCAD schematic drawings. The EDM desktop application will allow you to check in schematics but it will not give you options to automatically generate other files like the bill of material, netlist, and PDFs. You will also not be able to update OrCAD Capture title blocks if you use the EDM desktop application.
2. **DO** use the EDM desktop application to check in OrCAD PCB Editor drawings. The EDM client software within OrCAD Capture will allow you to check in PCB drawings but it will not give you options to automatically generate other files like manufacturing files and PDFs of your gerbers. You will also not be able to update PCB Editor title blocks if you use the EDM client software within OrCAD Capture.
3. **DO** use the naming conventions provided in this manual. Doing so will help develop a standard for file/project naming. It will also help us avoid any file/project naming errors in the future as some data entry fields must be unique.
4. **DO** use the project folder structure used by the provided project templates. The project templates are set up to use as many of the default settings within OrCAD Capture and PCB Editor when creating output files.
5. **DON'T** edit the text within the schematic or PCB Editor title blocks on your own. Allow EDM to update the title blocks during the Check-In phase.
6. **DO** log out of EDM when you are not using it. Doing so will allow others to access the database.
7. **DO** use one of the project templates when creating your EDM project. Not doing so will cause great pain and suffering. On top of that, your schematic, board, and bill of materials templates will not have the super cool MBARI logo.
8. **DON'T** create DSN files that contain multiple different PCBs in them. There should only be one PCB design per project.