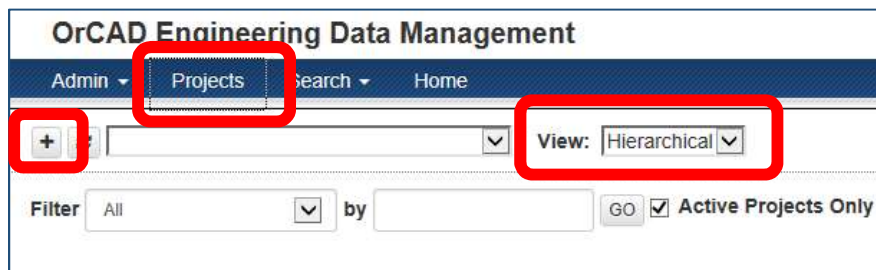


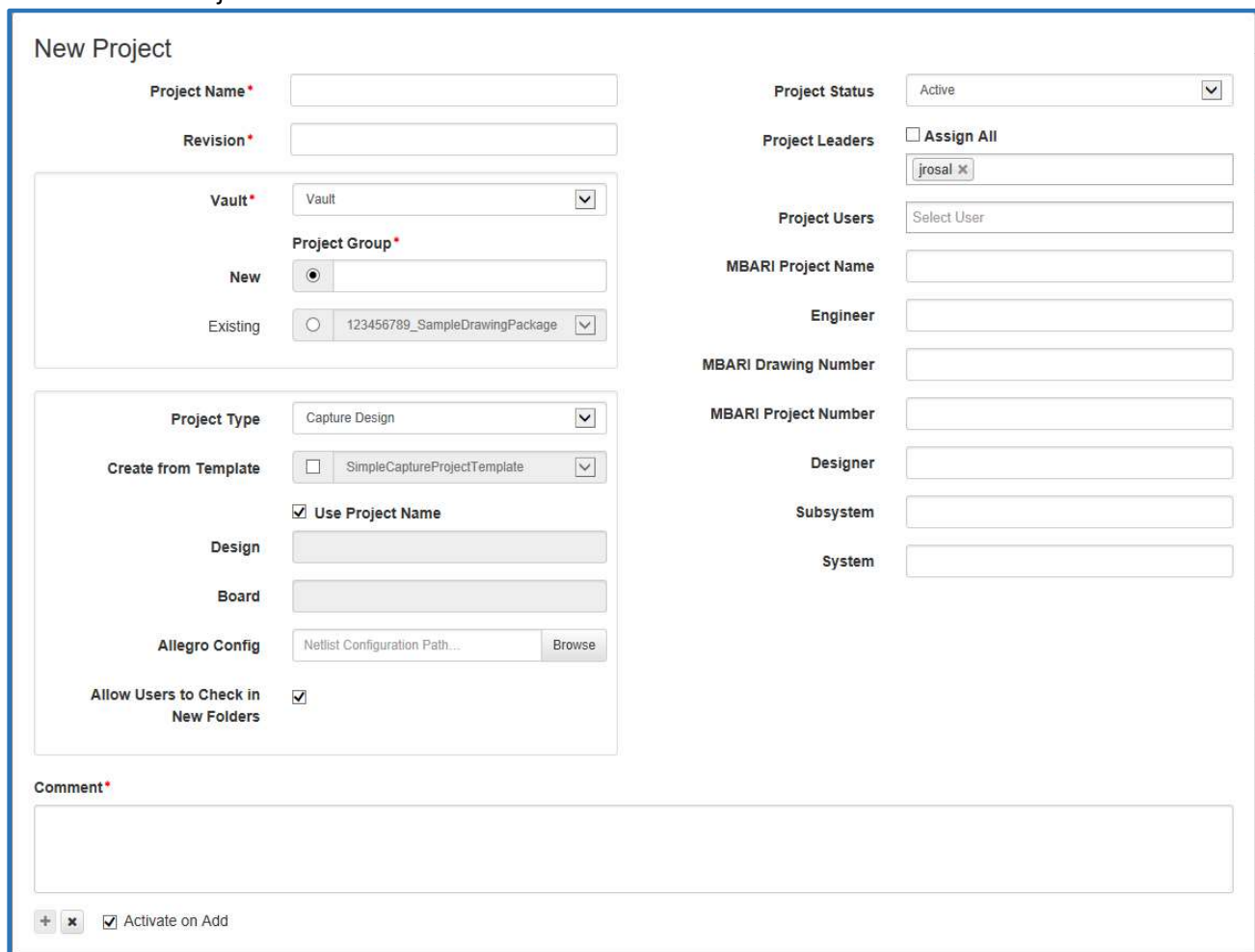
Version A (05/28/2021):

Starting a new schematic project in EDM:

1. Go to Canyon Head and get a new unique MBARI drawing number.
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.



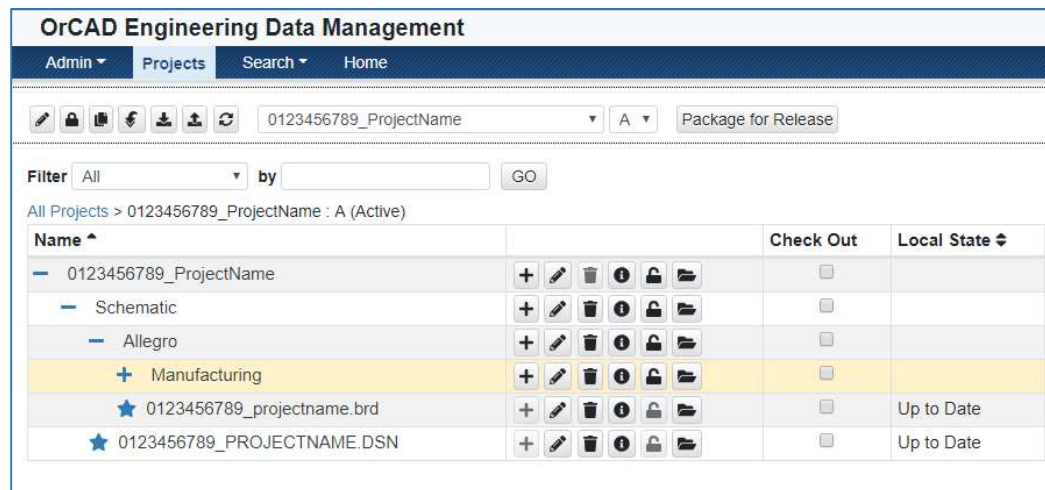
8. Fill in the New Project window.



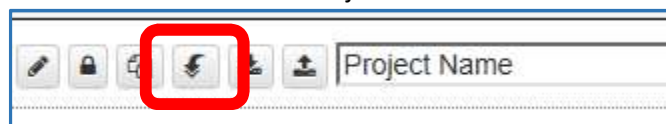
- a. Project Name:

- i. This name must be unique. Use the following naming convention:
“DrawingNumber_DrawingName” Examples:
 - 1. 012345678_AnotherVicorBoard
 - 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. 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 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, use by Capture when a netlist is created.
- i. Allow Users to Check in New Folders.
 - i. Check this box as 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 may check in files to the folder.
- 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.
 - 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" might be 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 you project for the first time:
 - 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.

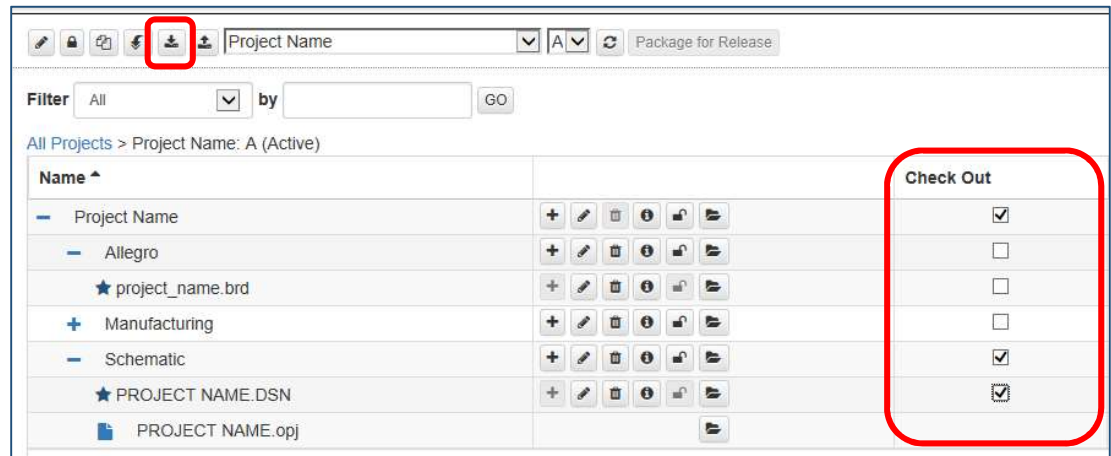


- f. Click the "Full Download Project" button.

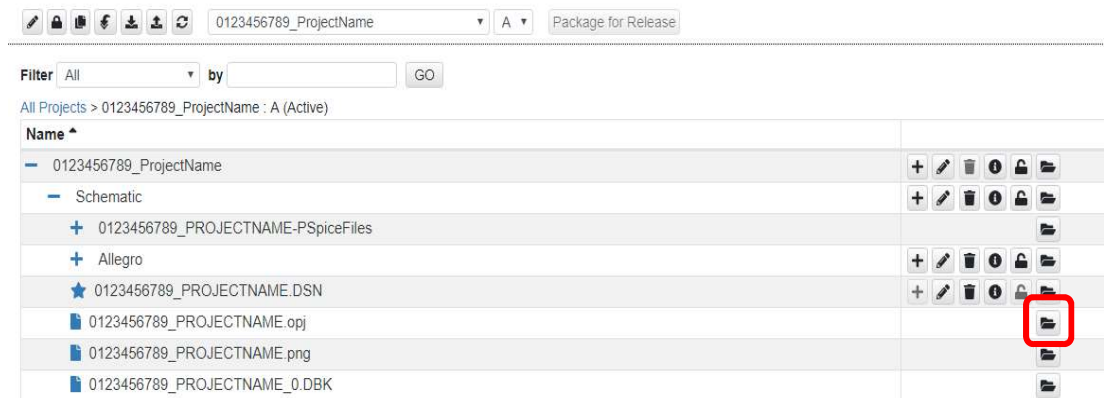


- i. This will autogenerate the BRD and DSN files.

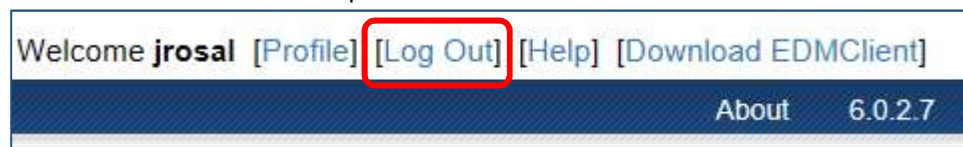
- g. Click the check button for the DSN File and click the checkout button on top of the page.



- 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. Go ahead and start adding parts and lines to your schematic.
j. **Log out** of EDM at this time to free up the network license seat for others to use.



- i.
k. 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 the first 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 check in your file using the EDM client within Capture. But in order for this to happen, you must make some sort of edit to be able to check in the drawing for the first time. You may add some text if you are not ready to create a circuit.
 - 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.

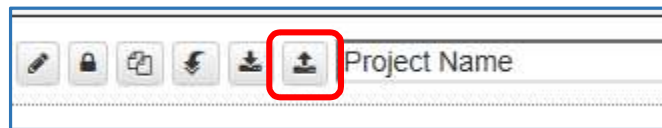
- iii. Example of the title block before first check in:

 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>	Last Updated By: <Updated By>
Drawing Number: <MBARI DN>		Release: <Rev>	Version: <Version>	Status: <Status>	Size: B
Page Modify Date: Thursday, October 29, 2020		Check In Date: <LCID>		Sheet <Page Number> of <Page Count>	

- iv. 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: Schematic Page Name			
		Project Name: 0123456789_ProjectName			
		Group Name: ProjectGroup	System: System	Subsystem: Subsystem	Last Updated By: jrosal
Drawing Number: 0123456789		Release: A	Version: 2	Status: Active	Size: B
Page Modify Date: Friday, May 28, 2021		Check In Date: 5/28/2021 10:58:50 AM		Sheet 1 of 1	

- l. When you are ready to check your drawing in, go back to the EDM tab and log back in.
 Navigate back to your project using the steps **9a-9e**.
 m. Click the **“Check In”** button.



- i. 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 ☒ Apply Status/Fields to Generated Related Files ☒ Update Title Block

☐ Netlist ☐ Netlist

☒ PDF ☒ PDF

☐ Include Variant Files

Auto Comment

☒ Parts Changed

☐ Total # of Parts

☐ Total # of Nets

Comment

☒ **Save**

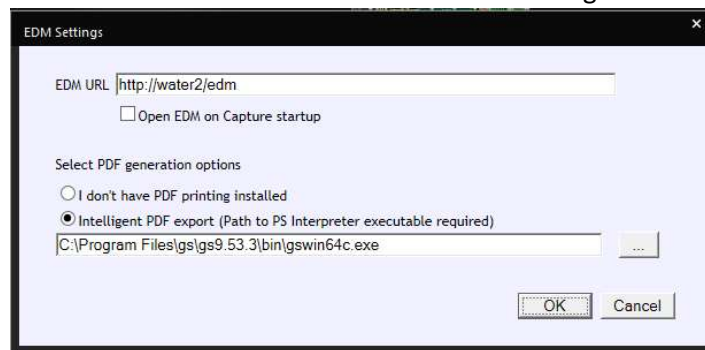
- i. Check the DSN **“CI”** (Check In) button on the top of the screen.
 ii. **“Auto Generation Related Files”** and **“Apply Status/Fields to Generated Files”** are optional.
 iii. **Update Title Block** must be checked.
 iv. Auto Comments create the described comment.
 v. You **MUST** add a comment 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.

Other Settings:

PDF print settings:

EDM is able to create PDFs of your Capture schematic and PCB Editor artwork/gerber files during every check in and when you use the Package for Release button.

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
 - 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.

OrCAD PCB Editor

NC Drill Parameter path:

When using EDM to automatically create drill and gerber files, it creates a temporary workspace to generate the files requested. If a gerber and ncd drill file is not specified, EDM uses the default art and ncd drill parameters that came with the software. Unfortunately, Enhanced Excellon format is not checked when using the default NC parameters. To force EDM to use specific parameter files, you must add the address to your art and drill parameters using ncdpath and artpath within the User Preferences Editor window in PCB Editor.

There's are corrected NC and ART parameter files located in the following directory if you choose not to create your own:

//water2/EDM/EDM_DesignTemplates/PCB_Editor/MBARI_EDM_PCB-Templates

