Overview

OE projects hosted on the Altilum 365 “Workspace” are saved in Altlium’s cloud server. All of our library components are also saved here. This poses 3 problems.

1. We trust Altium to never lose our files
2. Our component library is completely dependent on having access to our cloud workspace (note that Altium “caches” your library and files locally - via GIT
3. Understanding what files you actually fabricated a PCB with and ensuring those never change even with future revisions of the PCB.

This document is a quick introduction to Altium’s **Project Releaser** and **Project Integration Library** exporter.

Project Release provides a static “snapshot” of the project and includes all source/output files.

Project Integrated Library exporter provides an .IntLib library of all project components in a single file (symbols & footprints) for safekeeping and use outside of *our workspace library*.

End Result

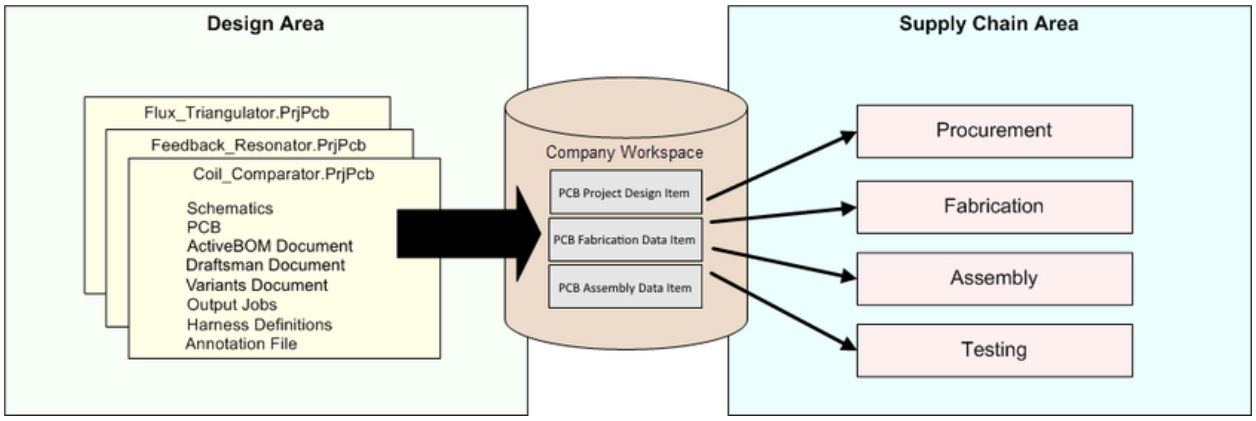
I recommend you export and archive these files external to the Altium 365 cloud workspace.

- Project Release .zip folder
    - *Source files (Project, schematic, PCB design, and IntLib)*
    - *Fabrication Files (Gerbers, etc…)*
    - *Assembly Files (Pick and Place, etc…)*
- Project Component Library (.intlib)
    - *This is optional if the above released ZIP file contains the most recent IntLib file*

Project Releaser

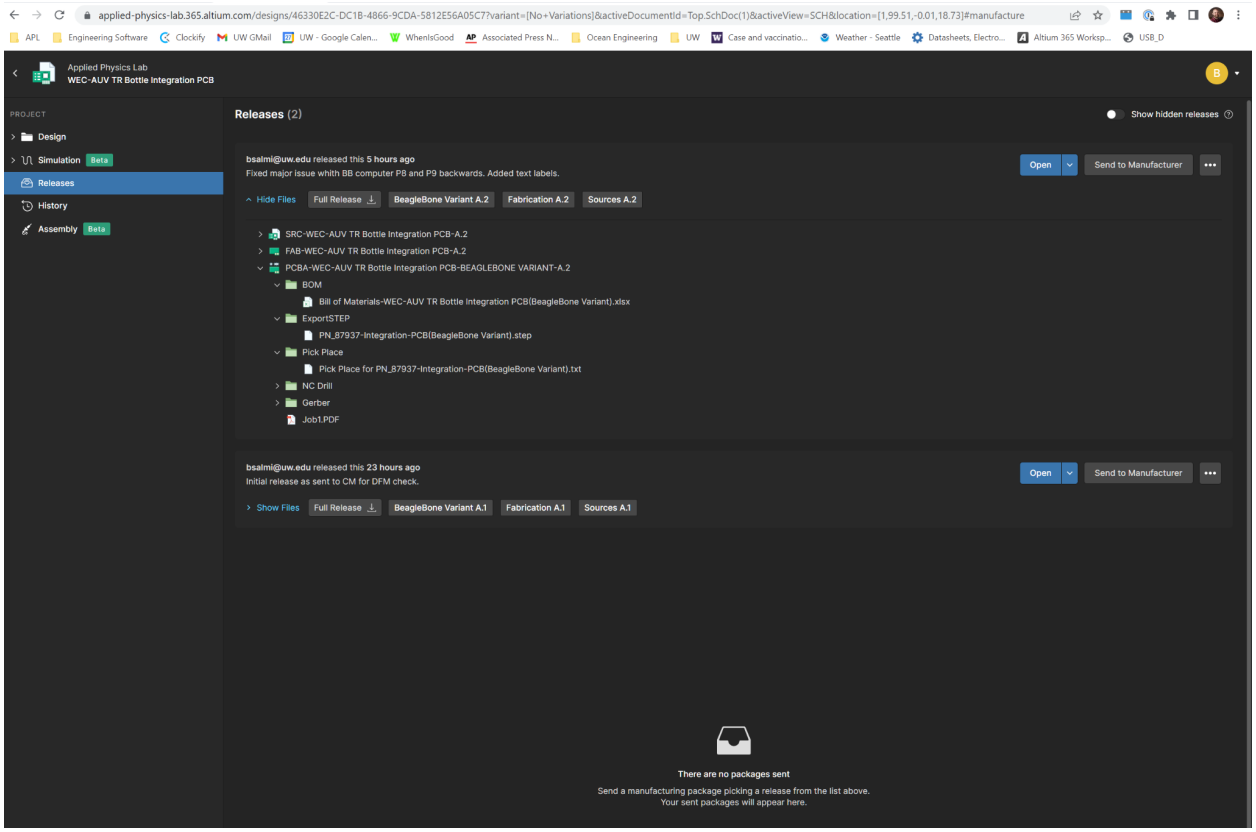
Altiums documentation is quite good albeit lengthy.

The project releaser tool provides a revision control “packaging” of all the data AS-IS within the project. It is akin to a software version release within a rolling update schedule. The output will be a packaged ZIP file containing the source (design files), fabrication files, and assembly files with a revision stamp that cannot be changed in the Altium 365 workspace. You can always go back and get the files from THAT release.



We have Altium 365 Standard so we can only release REVA.x where x is a number.

This is extremely helpful since output files will always be available immediately and pre-generated. All of the files are available in Altium or in the web dashboard.


applied-physics-lab.365.altium.com/designs/46330E2C-DC1B-4866-9CDA-5812E56A05C7?variant=[No+Variations]&activeDocumentId=Top.SchDoc(1)&activeView=SCH&location=[1,99.51,-0.01,18.73]#manufacture
Applied Physics Lab
WEC-AUV TR Bottle Integration PCB
PROJECT
Design
Simulation Beta
Releases
History
Assembly Beta
Releases (2)
Show hidden releases
bsalmi@uw.edu released this 5 hours ago
Fixed major issue whith BB computer P8 and P9 backwards. Added text labels.
Open
Send to Manufacturer
Hide Files
Full Release
BeagleBone Variant A.2
Fabrication A.2
Sources A.2
SRC-WEC-AUV TR Bottle Integration PCB-A.2
FAB-WEC-AUV TR Bottle Integration PCB-A.2
PCBA-WEC-AUV TR Bottle Integration PCB-BEAGLEBONE VARIANT-A.2
BOM
Bill of Materials-WEC-AUV TR Bottle Integration PCB(BeagleBone Variant).xlsx
ExportSTEP
PN_87937-Integration-PCB(BeagleBone Variant).step
Pick Place
Pick Place for PN_87937-Integration-PCB(BeagleBone Variant).txt
NC Drill
Gerber
Job1.PDF
bsalmi@uw.edu released this 23 hours ago
Initial release as sent to CM for DFM check.
Open
Send to Manufacturer
Show Files
Full Release
BeagleBone Variant A.1
Fabrication A.1
Sources A.1
There are no packages sent
Send a manufacturing package picking a release from the list above.
Your sent packages will appear here.

Release Process

It is most appropriate to use the release process at the END of a design when ready to submit to a PCB fabricator. Use the normal Altium 365 workspace GIT based commit system prior to project releasing.

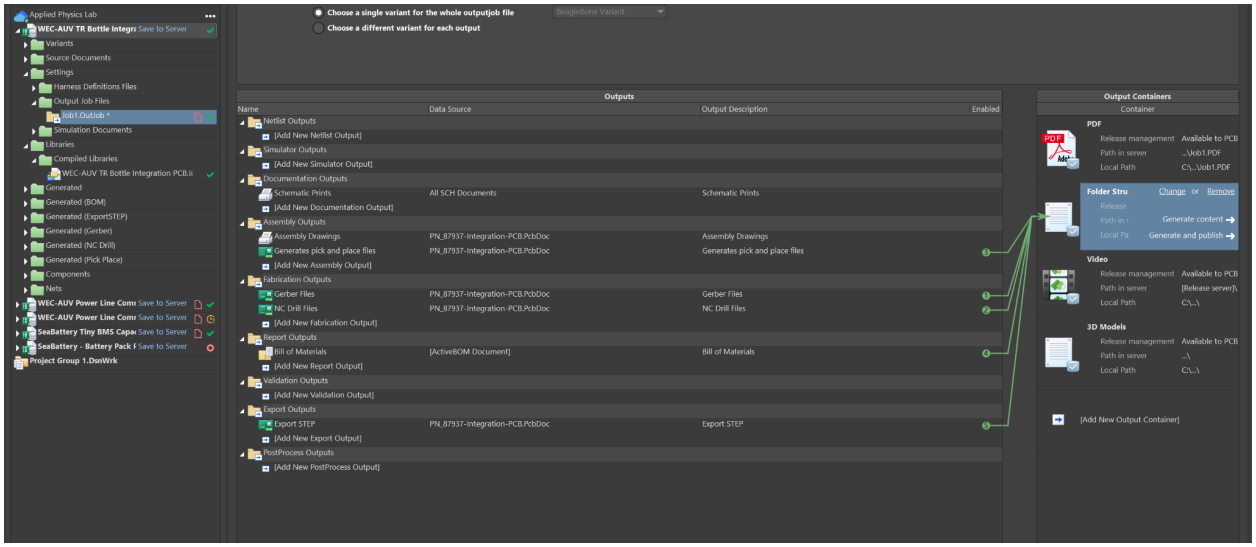
- Complete the entire design process
    - No Schematic ERC errors
    - No PCB DRC errors
- Finalize/Create an output job file
    - Ensure the files output correctly
- Prepare the project release
- Release the project & upload to server

Note that if you need to modify the project after releasing (i.e. scope change, DFM change, error found, etc…) then you simply re-release the project. A new release will increment the revision number.

Because we only have Altium 365 Standard - I recommend cloning the project files to create a “Rev 2” of a PCB for major board revisions and use the release process for only a single board fabrication effort.

Output Job File

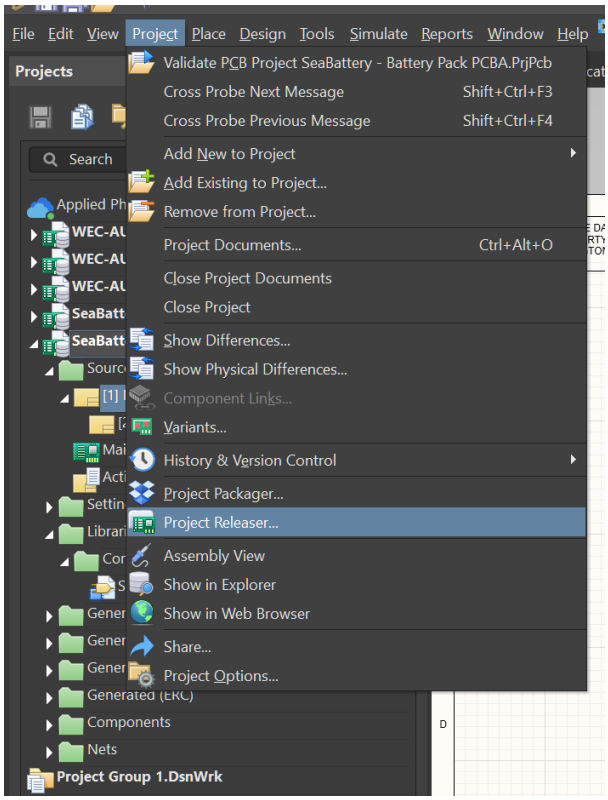
To effectively use the project release tool you need to utilize “Output Job” files for creating output files such as GERBER’s and schematics. Output Job files are simply Altium scripts that when run automate the output file creation process.



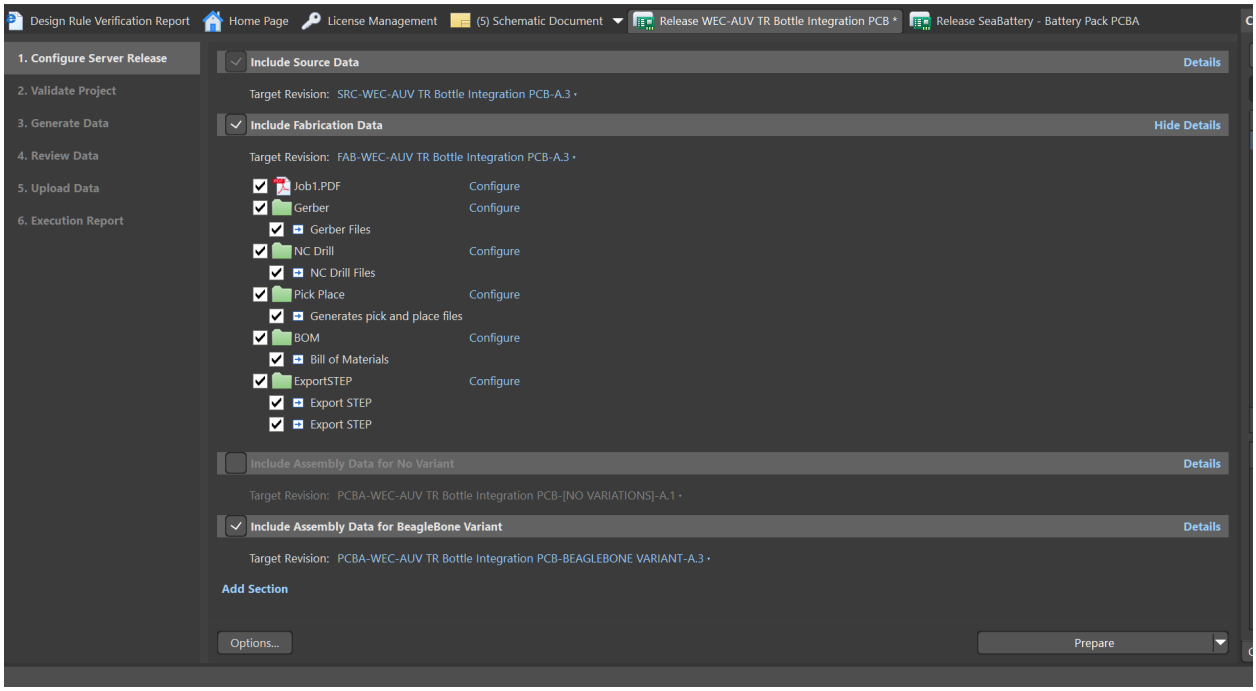
Output Job files are located in the “Settings” folder of your project. If it is not there then right click and add one to the project.

Releasing A Design

To utilize Project Releaser you need to select the menu Design>>Project Releaser.

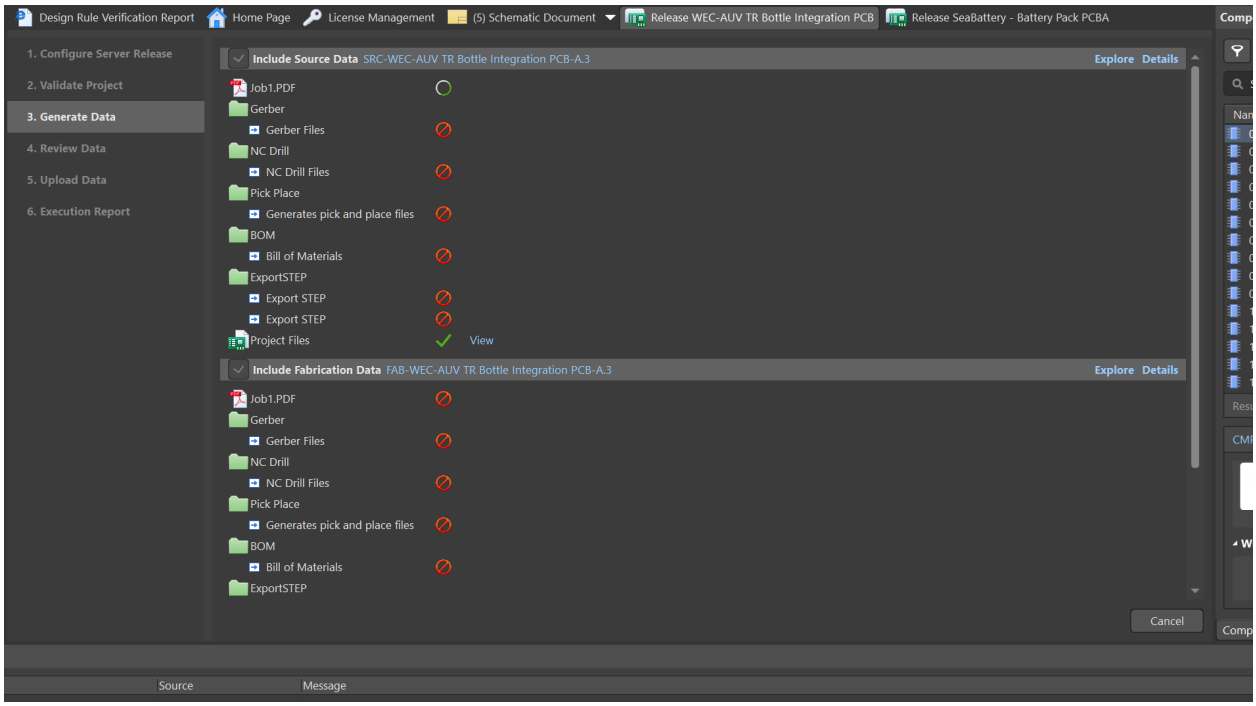


This will bring up the release tool and you can select the outputs to generate and include in the release (i.e. Source files, FAB, ASSY, etc....). The initial release will be REVA.1 but following releases will increment to REV A.2 and so on.



Click the “Prepare” button and you will be asked to save and commit (always, even if already committed).

The project will start generating all of the outputs, if there are no failures then you can release it and it will upload it to our workspace and the release is completed. If there are failures in the generation (i.e. design rule errors, etc…) you will need to fix or “mute” them and try again.

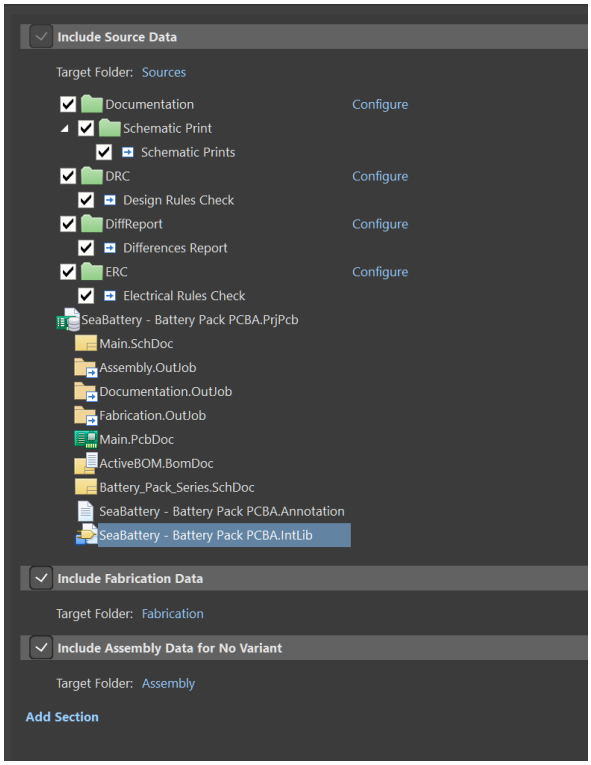

Design Rule Verification Report
Home Page
License Management
(5) Schematic Document
Release WEC-AUV TR Bottle Integration PCB
Release SeaBattery - Battery Pack PCBA
1. Configure Server Release
2. Validate Project
3. Generate Data
4. Review Data
5. Upload Data
6. Execution Report
Include Source Data SRC-WEC-AUV TR Bottle Integration PCB-A.3
Explore Details
Job1.PDF
Gerber
Gerber Files
NC Drill
NC Drill Files
Pick Place
Generates pick and place files
BOM
Bill of Materials
ExportSTEP
Export STEP
Export STEP
Project Files
View
Include Fabrication Data FAB-WEC-AUV TR Bottle Integration PCB-A.3
Explore Details
Job1.PDF
Gerber
Gerber Files
NC Drill
NC Drill Files
Pick Place
Generates pick and place files
BOM
Bill of Materials
ExportSTEP
Cancel
Source
Message

Project Integrated Library Export

A .IntLib file is the most basic form of an Altium component library. An IntLib contains symbols, footprints, and components that link the symbol/footprints and provide other parameters. The export tool can be used to output a .IntLib file for all components in a single project.

Managed Library - This refers to our cloud based Altium 365 workspace library
Unmanaged Library - This is primarily local IntLib files

Note that if you export a library prior to project release you can include the library IntLib within the “Source” folder of the release!



Export Project Integrated Library

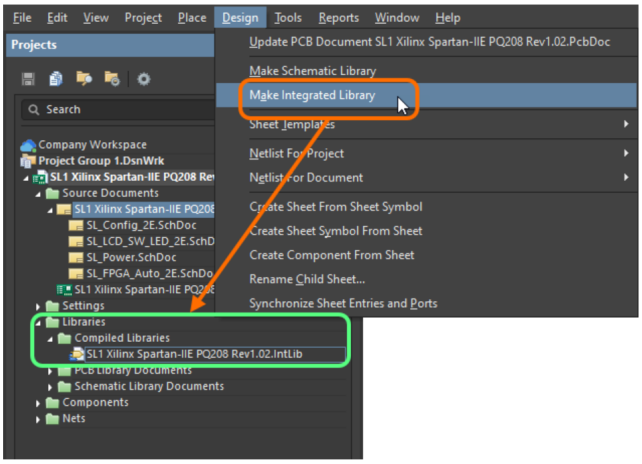
Follow these directions.

Creating an IntLib from Project Documents

Contents

1. Opening all source schematic documents and making a schematic library.
2. Making a PCB library from the PCB document.
3. Compiling these libraries into an integrated library, named after the project (`<ProjectName>.IntLib`).

The IntLib is added to the project (under `Libraries\Compiled Libraries` in the Projects panel), added to the Installed libraries (as part of the *Available File-based Libraries* available to any project), and made available through the Components panel.

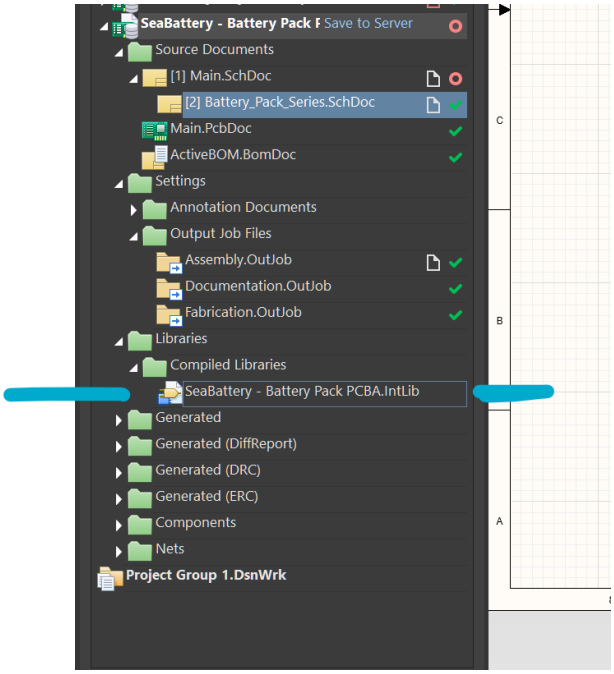


Streamlined creation of an integrated library, directly from the schematic and PCB documents in the active design project.

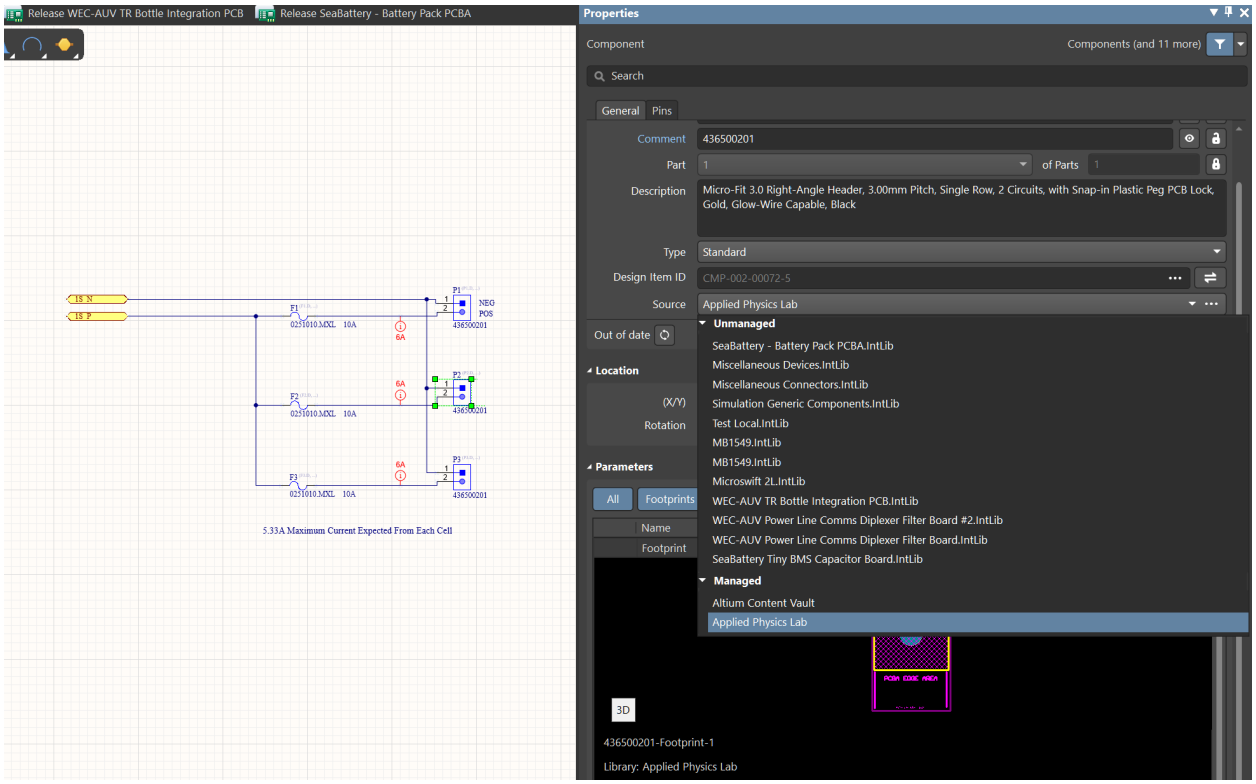
Link Components From Workspace to Local IntLib

If a designer doesn't have access to the workspace (i.e. another team, a contractor, etc…) you can link the components in a project to the local IntLib file.

Ensure the IntLib is generated in the project or open within Altium. Best practice is to ensure it is included in the project:



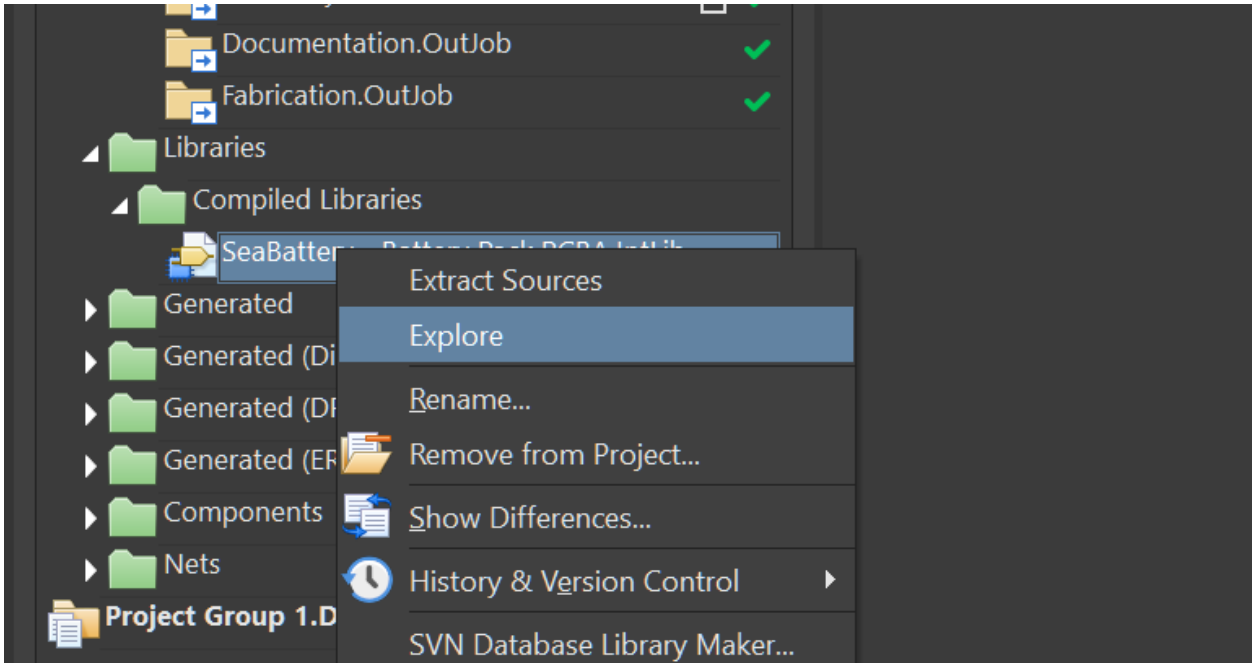
Then click on a component and change the “Source” from the **Managed** server library to the **Unmanaged** local IntLib that is appropriate for the project. Note that you can select multiple/ALL components at once and make this selection.

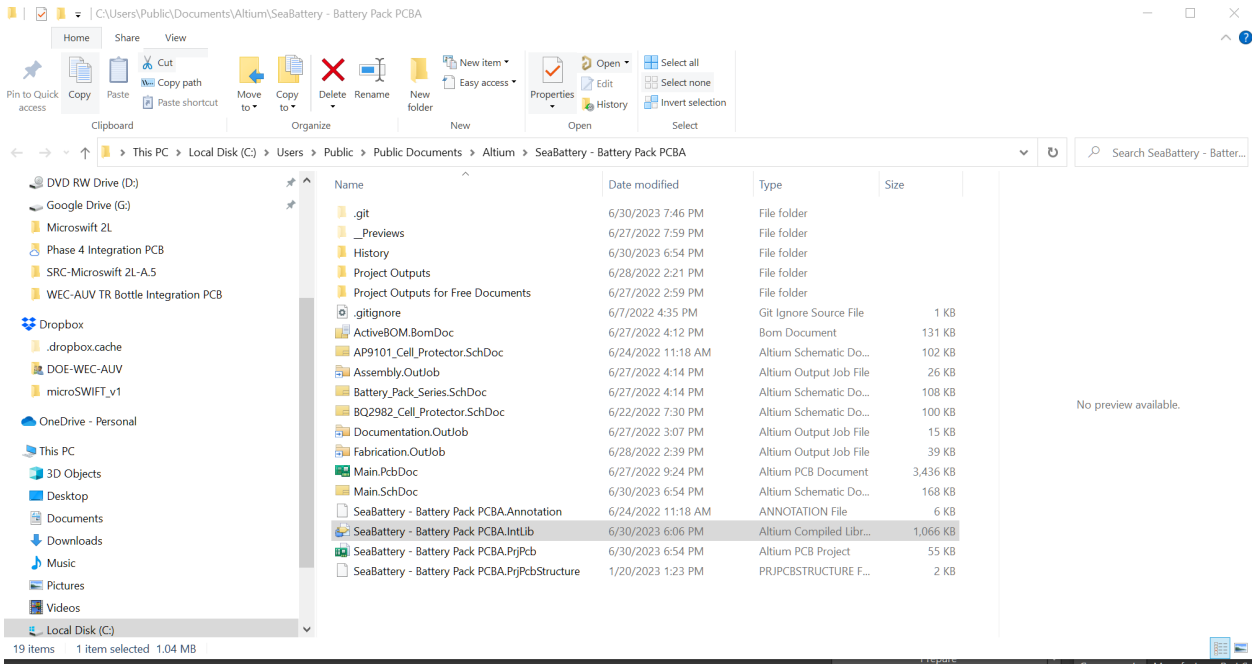


The component is completely referenced locally via the IntLib.

Windows Explorer - IntLib

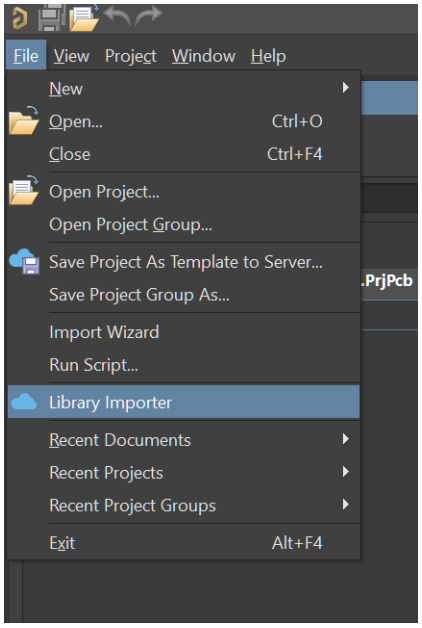
Right click on the IntLib and select “Explore” and you’ll open the folder containing the IntLib file you can archive outside of Altium.

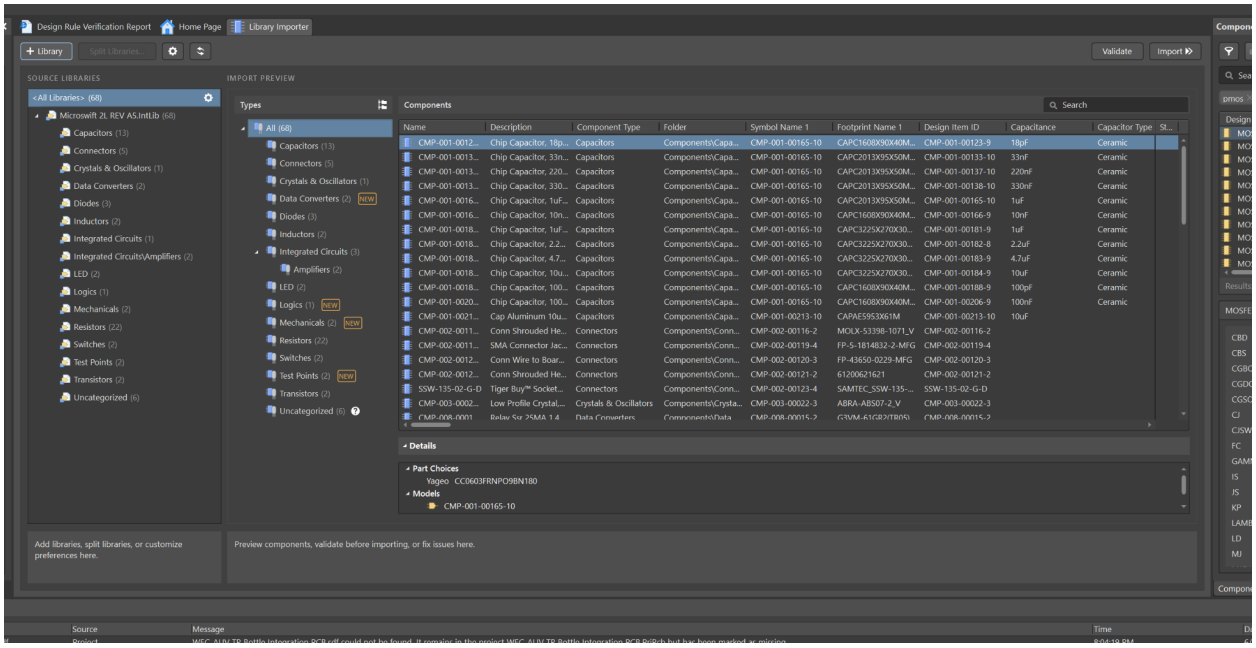




Library Importer

Library importing is useful to import any IntLib into our **Managed** Altium 365 cloud library. This is useful if the project was made outside of our 365 workspace or if you are importing .IntLib files from external sources such as manufacturers or SnapEDA. Another example is transferring library components between multiple workspaces such as OE and OPD.





This is pretty self explanatory, just make sure you link the component type correctly if it auto-selects the wrong category.