Altium Designer

Essentials Training with Altium 365

Module 28: Releasing a Project to Altium 365

Software, documentation and related materials:

Table of Contents

Module 28: Releasing a Project to Altium 365

1 Purpose

In this exercise, you will learn to open a predefined release process, check the configuration of the release process, and start the release process. You will see the flow for the release process and find the results of the release process, like Gerber files, NC-Drill file, PCB -, SCH documentation, saved at the Altium Cloud System, Altium 365.

Altium Designer provides powerful, high-integrity board design release management, courtesy of its Project Releaser. With an intuitive user interface, you are able to generate all manufacturing data for your project simultaneously – fabrication data, assembly data, design source, etc. The Project Releaser also provides the ability to generate the assembly data for multiple detected -variants of your board design at the same time. You don't even have to worry if you haven't created Output Job files – it'll do that for you if you ask it to!

With the Project Releaser, you'll be able to generate your manufacturing data with simplified ease, and with the highest integrity. And you'll also be able to survey the fruits of that generation before you commit to finalizing the release (viewing Gerbers/ODB++ data in the CAM Editor for example), ensuring that the data you have generated is exactly the data required to get your board manufactured on time, first time.

The prerequisites for releasing a PCB design to a Workspace are:

- Altium Designer is connected to the Workspace, which hosts the target repository for the release.
- The Altium Designer PCB project includes suitable Output Jobs – design data output generators (for outputs such as print, file or PDF). If no OutJobs are available, the Project Releaser will offer to create suitable Fabrication and Assembly OutJobs to drive the process.
- The project sourced by the Release process has all required elements in place: the source data is valid and up to date, the user has sufficient rights to release that project to the Workspace, and so on.

2 Important Information

Caution: The Training workspace will only be available during the time of the Training.
After the Training, you will lose the access to the Altium Training Workspace.
Do not upload personal files / projects /... to the Altium Training Workspace!

3 Shortcuts

Shortcuts used when working with Module 28: Releasing a Project to Altium 365

C	Project Releaser

4 Preparation

1. Close all existing projects and documents.
2. Next, we create a Copy / Clone of the Training Project Module `28 Releasing a Project to Altium 365`.
3. Select **File » Open Project...** to open the *Open Project* dialog.
4. Enable the folder view button  .
5. Navigate to the predefined Training Project `Module 28 Releasing a Project to Altium 365 (Top\Projects\Altium Designer Essentials Training Course\...)`.
6. Select **Open Project as Copy...** 

.
7. At the new dialog *Create Project Copy:*
   a) Add your name to the project name: `Module 28 Releasing a Project to Altium 365 - [Your Name]`.
   b) Add a description: `Altium Essential Training - Module 28 - [Your Name]`.
   c) Open the *Advanced* section.
   d) Select the Ellipsis Button  from the **Folder** configuration to open the *Choose Folder* Dialog.
      i) Select the folder with your name: `Project\For Attendees\[Your Name]`.
      ii) Select **OK.**
   e) Change the Local Storage path if needed.
   f) Select **OK** to create the copy.
8. Wait until Altium Designer creates the copy of the project and opens the project in the *Projects* panel; this can take up to 1 minute.

Hint: For details how to Copy / Clone the predefined training project see Module 9 Making the Connection, Step 3 Preparation.

5 Out Job Files for Release Process

9. Open all the schematic documents and `Module 28 Releasing a Project to Altium 365.PcbDoc`.
10. In the example Project there are two Out Job files. Open these files, the Output Job files should look similar to Figure 1.
    a) Fabrication.OutJob
    b) Assembly.OutJob

The two Output Job files are examples of possible configurations. The release process uses the Output Job files and its configured settings for the creation of Fabrication and Assembly Data.

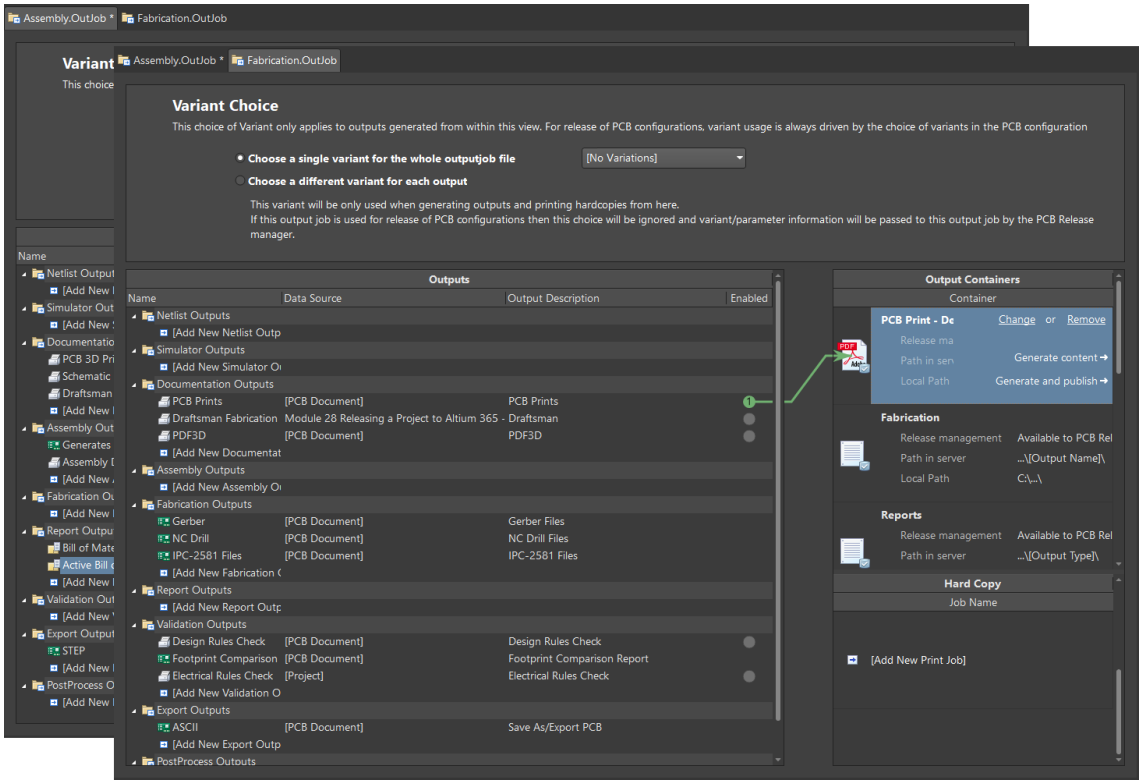


Figure 1. Out Job Configuration files

6 The Release Process

6.1 Open the Release Process

11. Select the command **Project » Project Releaser...** to open the release process, Figure 2.
    a) On the left-hand side, you can see the flow for the release process.
    b) On the right-hand side, you can see the integration of the Out Job files to the Release process.

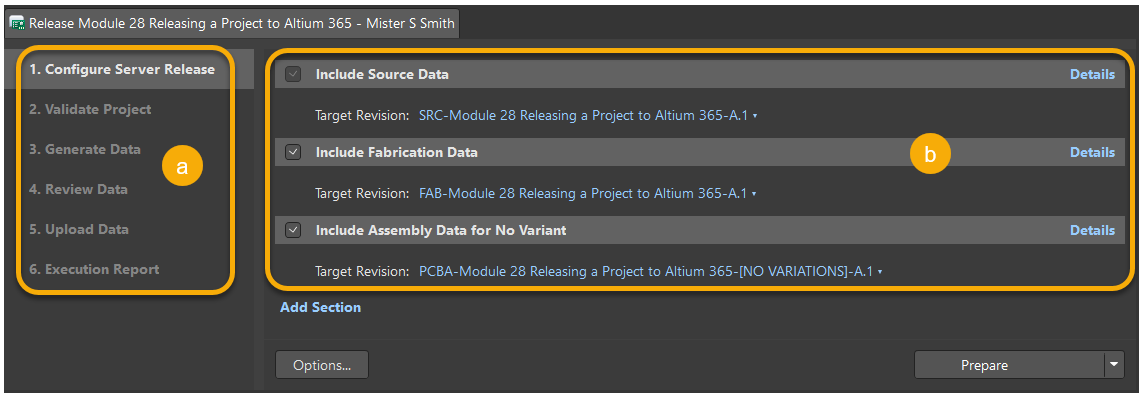


Figure 2. Release Process

6.2 Preparing the Release Process

12. Select the command **Prepare** at the lower right side of the release dialog.
13. If the Tools is ready, it will show you a dialog similar to the dialog you see at Figure 3. Select **Create Item** if you are ready to create the Items.

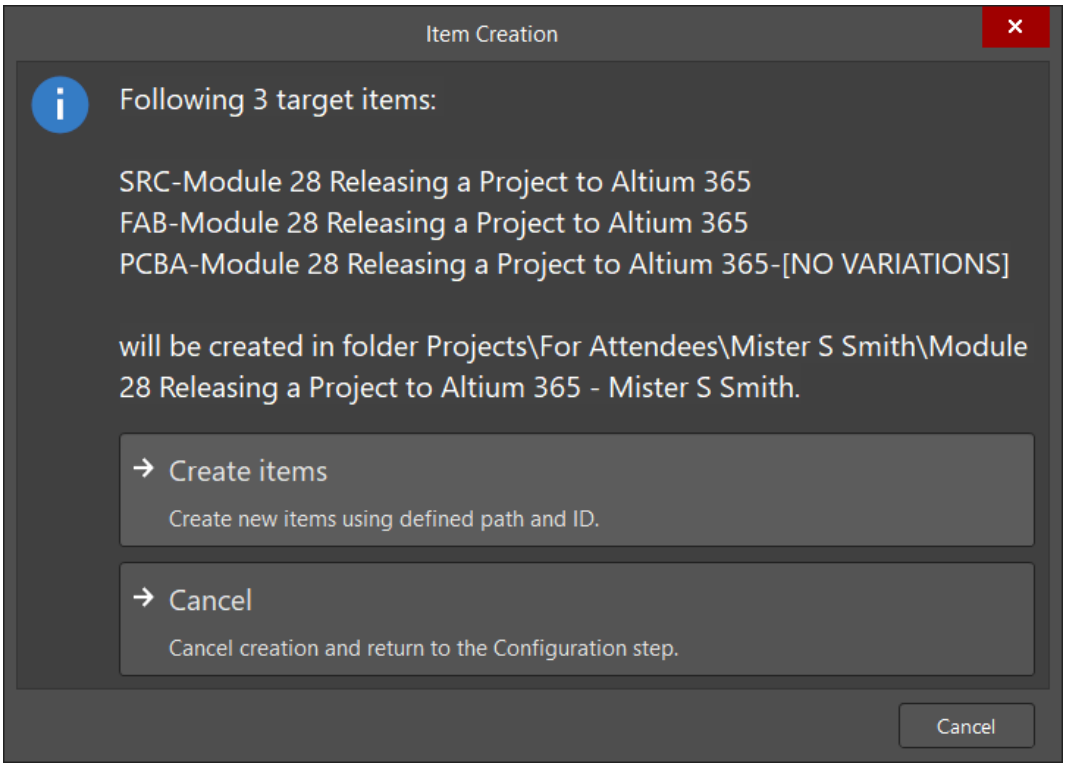


Figure 3. Item creation for release process

6.3 Project Not Committed

14. If you missed to commit modifications you have done so far (**Save to Server**) the tool will show you the *Pending Commits* dialog, Figure 4. If you see the message, select **Commit** to proceed.

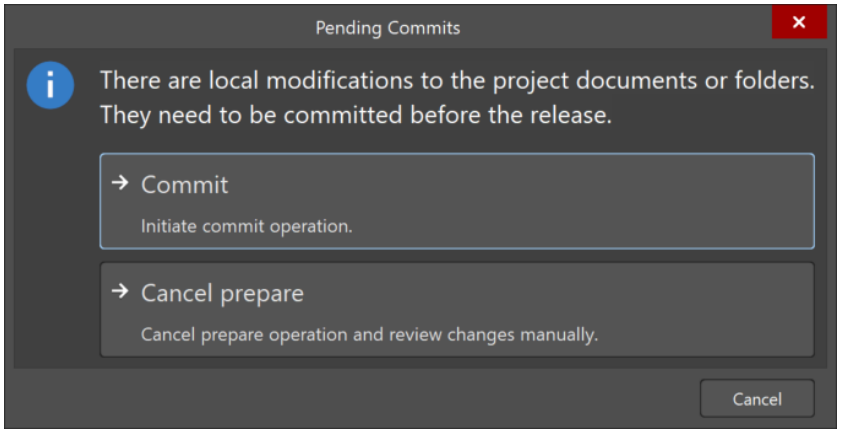


Figure 4. Reminder to commit modifications

15. Add a comment at the *Comment* field and select **Commit and Push** to proceed.

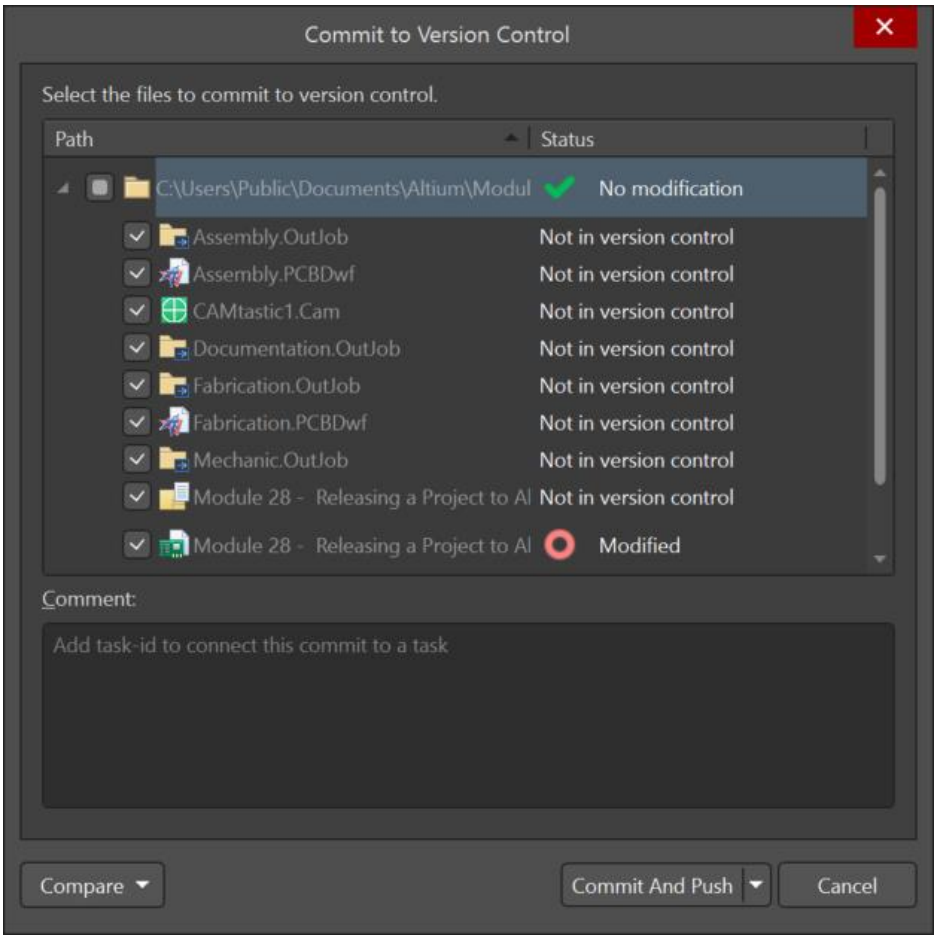


Figure 5. Example for Commit to Version Control

6.4 Release Step

16. The next step after **Prepare**, and if needed a commit, Altium creates the Items for the Workspace.
17. Included in the fabrication data of the OutputJob file is the generation of the DRC report. If any DRC errors are detected, the release process will stop.

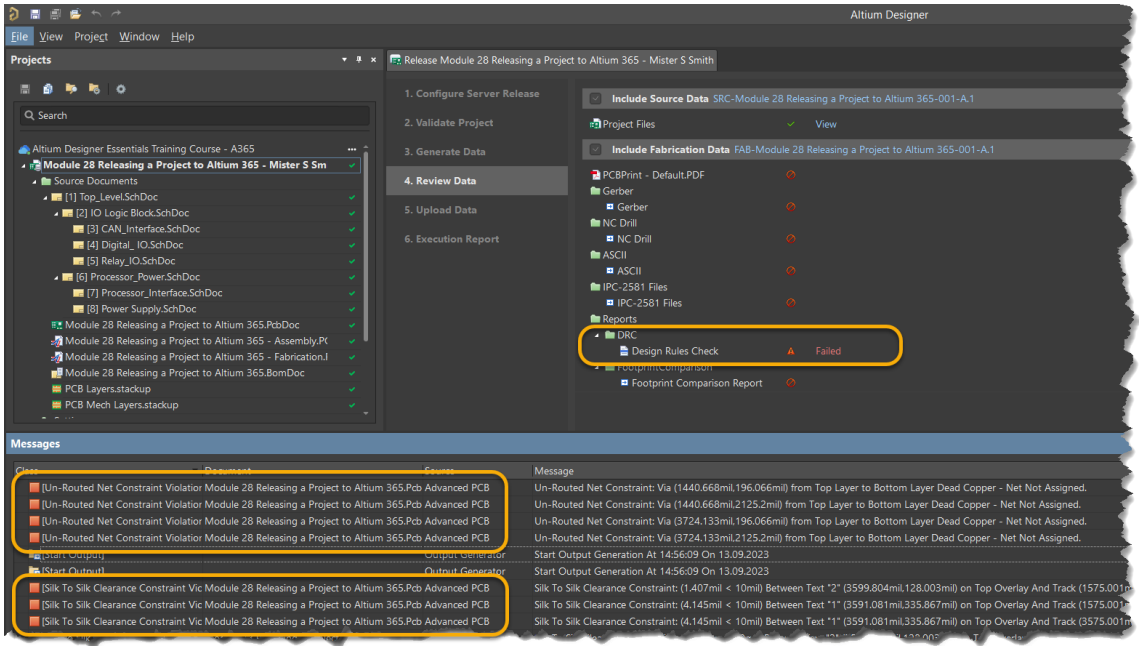


Figure 6. Release stopped - DRC errors detected

18. You have to correct the errors found at the PCB before you could continue with / restart the release.
19. Select **Cancel** for the Release Process and confirm with **OK** that all generated data will be lost.

6.5 Correct DRC Errors Detected by the Release

For the Training example we have built in 6 errors (that generate 8 error messages). Below you will find information how to fix these errors. The process to fix these is the same as already covered in the module PCB Design Rule Checks (DRCs).

20. Open the PCB document `Module 28 Releasing a Project to Altium 365.PcbDoc`.
21. Run the DRC.
    a) Execute from the main menu **Tools » Design Rule Check**
    b) Select **Run Design Rule Check...**, to start the DRC.
    c) The DRC reports 8 Errors.
22. Fix the 8 errors reported by the DRC.
    a) Open the *PCB Rules and Violation* panel to see, and jump to the errors.
    b) Change the position for the Pin 1 and Pin 2 markers of J1.
    c) Waive the four errors `Un-Routed Net Constraint Via`.
    d) Save the PCB.
    e) Return back to the Release Process.

Hint: Details about the DRC can be found at the module **PCB Design Rule Checks (DRCs)**

Note: Be aware of that changing the position of Pin 1 and Pin 2 markers are a local modification of the Footprint, the Library is not automatically updated. Commands Like **Tools » Update From PCB Libraries**... ,... will check for modifications and can undo local modification.
Updating the Library with the local modification, if it makes sense, is a separate step and a possible task for the Librarian of the Workspace.

6.6 Restart the Release Process

23. Select the command **Prepare** at the lower right side of the Release dialog.
24. If you missed to commit modifications for the PCB (**Save to Server**) the tool will show you the *Pending Commits* dialog, See Step 6.3. Select **Commit** and then **Commit and Push** to proceed.
25. During the Release Process, if no additional errors are detected, will now save the Source Data and create the different outputs, defined at the OutJob files.
26. Wait until the Prepare step finished. If the information is prepared select **Release.**
27. At the dialog *Confirm Release* add the Release Note `First release for Module 28 Releasing a Project to Altium 365 – [Your Name]` and select **OK.**
28. **In Step 5 of the Release Process, as shown in** Figure 7**, will now upload the data to the Server (Altium A365 Cloud).**
29. **Step 6 of the Release Process, as shown in** Figure 7**, the Execution report, indicates the release Process is now complete.**

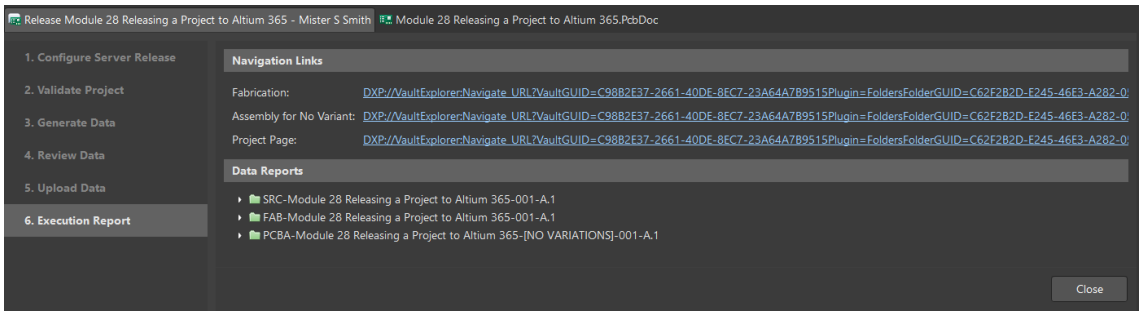


Figure 7. Execution report

30. The links from *Fabrication*, *Assembly for …,* and *Project Page* allow you to open the *Explorer* Panel and to jump to the specific set of files created during the release process. Select the Link for Fabrication to see the outputs generated, as shown in Figure 8.

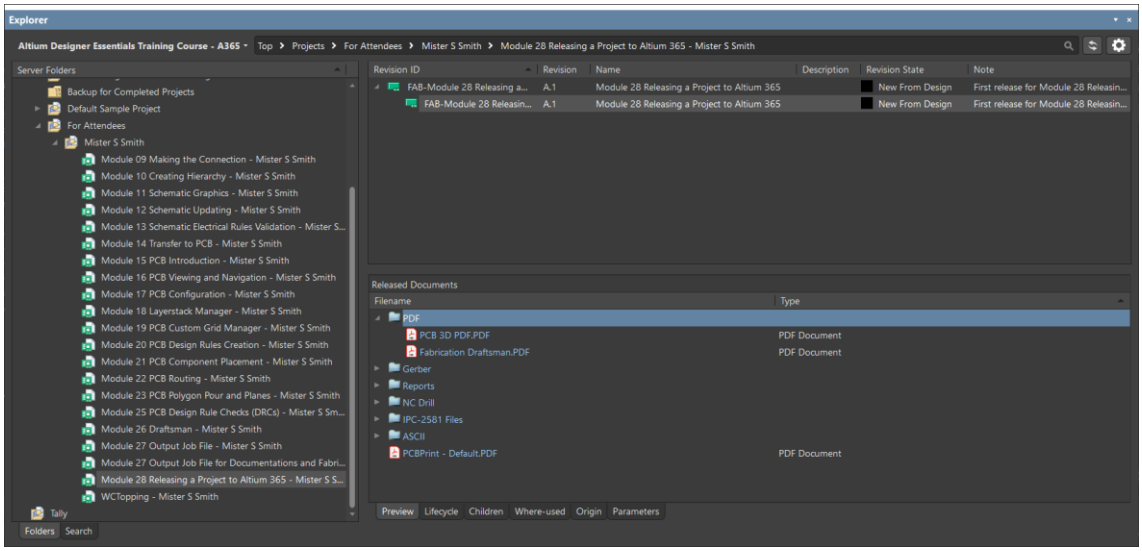


Figure 8. Explorer panel - first release of Fabrication Data

31. Save all documents using **File » Save All,** if you have done additional modifications after you checked the generated and saved Fabrication files.
32. When ready, close the release dialog and any open documents.

7 Start a Second Release

After a first release was done and a review, there may be design modifications that are required. After updates to the SCH, PCB, and further tests (ERC / DRC), you may start a second release. With the second release you may also want to change the Lifecycle from the state `New From Design` to the state `Prototype`.

33. Open a SCHDOC and modify the schematic, for example, drag a Power Port.
34. Save the modification (Local and Save to Server).
35. Select the command **Project » Project Releaser...** to open the release process.
36. The release Process starts again with Step *1. Configure Server Release*, Figure 9.
    a) The Target Revision for Source Data, Fabrication and Assembly Data is updated to Revision A.2.
    b) Select the Drop-Down list to see the options that are available:
        i) A2
        ii) B1
        iii) Select the option A2

Hint: The options that are available are based on the Naming Scheme / Revision configuration from the Workspace, that is managed from the Workspace Administrators.

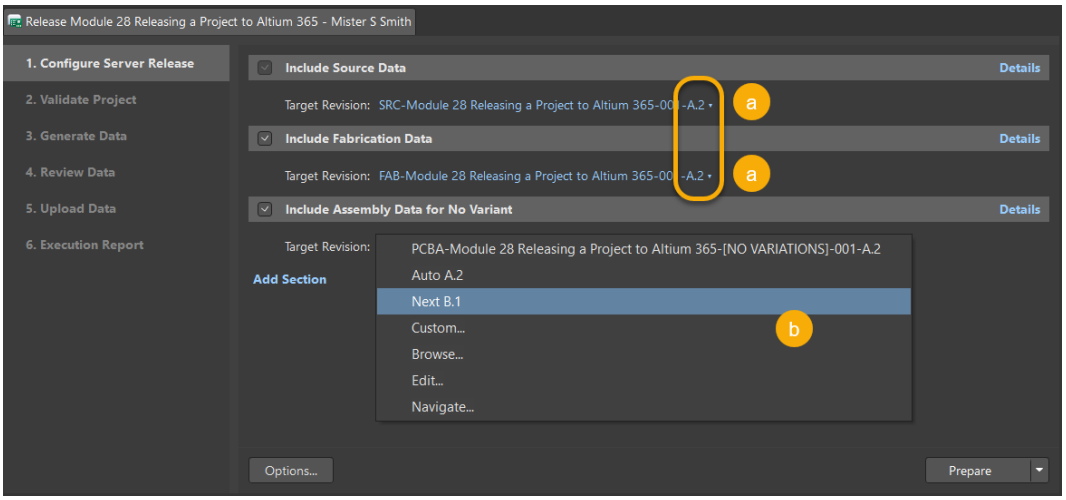


Figure 9. Second Release

37. Create a second release by selecting **Prepare and Release**

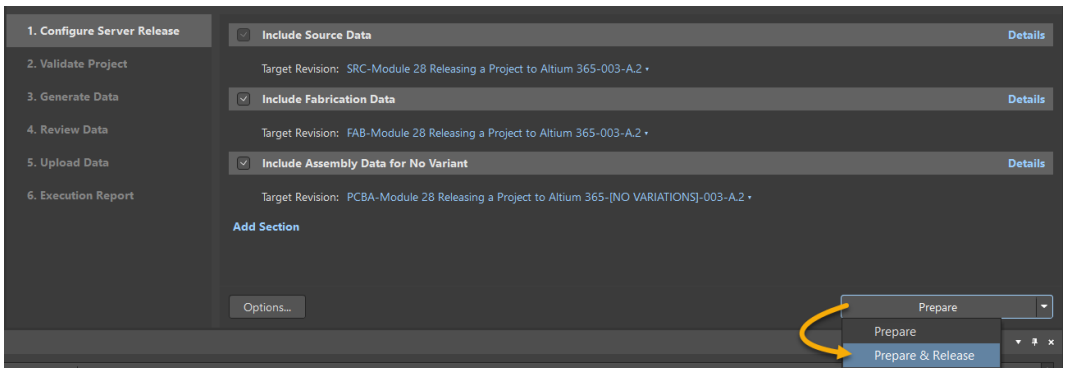


Figure 10. Second Release - Prepare and Release

38. After the release finished, select **Close** to close the Release view.

7.1 History

Next, we will see the History for the Project.

39. At the *Project* panel, select the Project, right-click and execute the command **History and Version Control** - **Show Project History**, Figure 11.

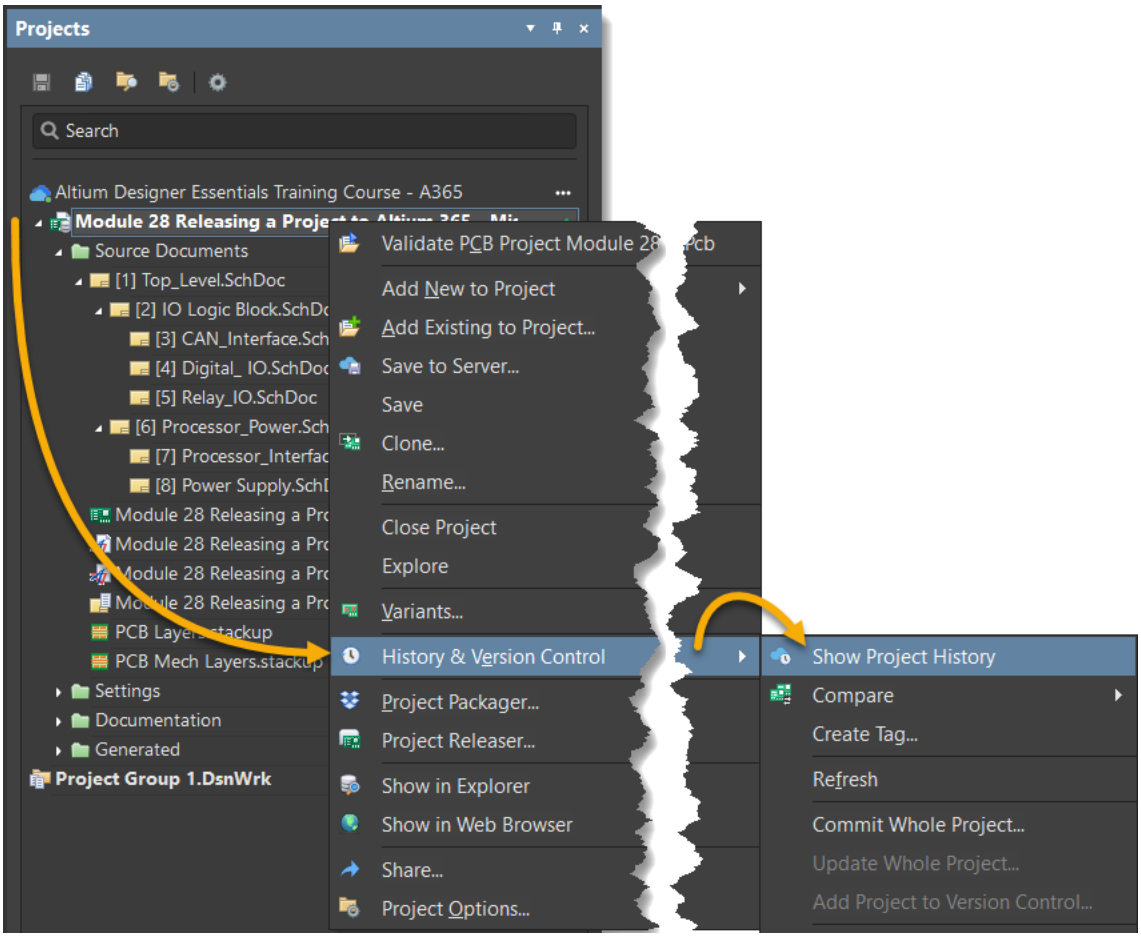


Figure 11. Open the Project History view

40. The *History* shows you the different Commit and Release steps for your Project. It also shows you if a clone from the project was created. You can see an example history at Figure 12.

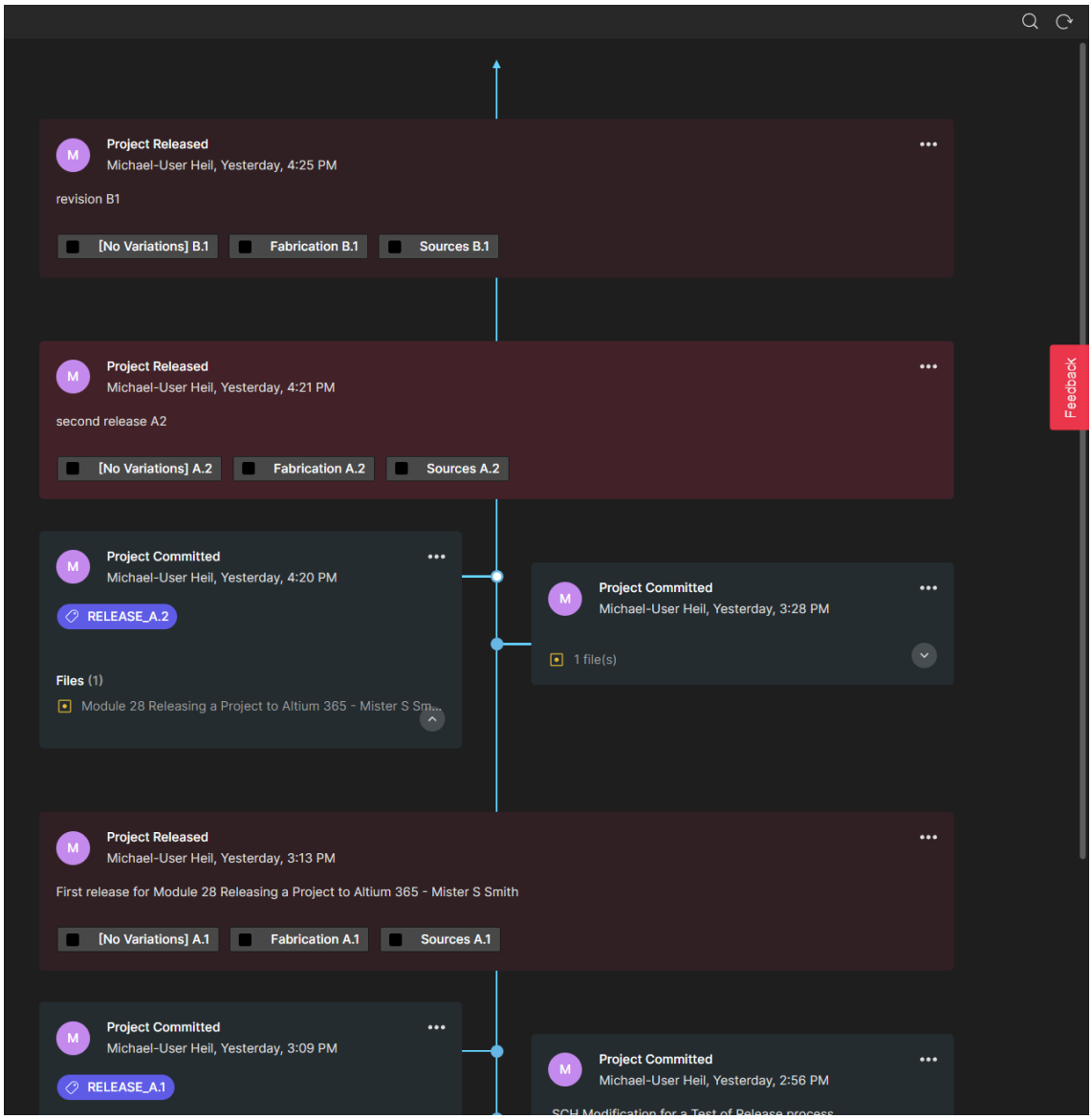


Figure 12. Example for a Project History

7.2 Change Lifecycle Information

Next, we open the *Explorer* panel for your Project `Module 28: Releasing a Project to Altium 365 - [Your Name]`.

41. At the *Project* panel, select the Project, right-click and execute the command **Show in Explorer,** Figure 13.

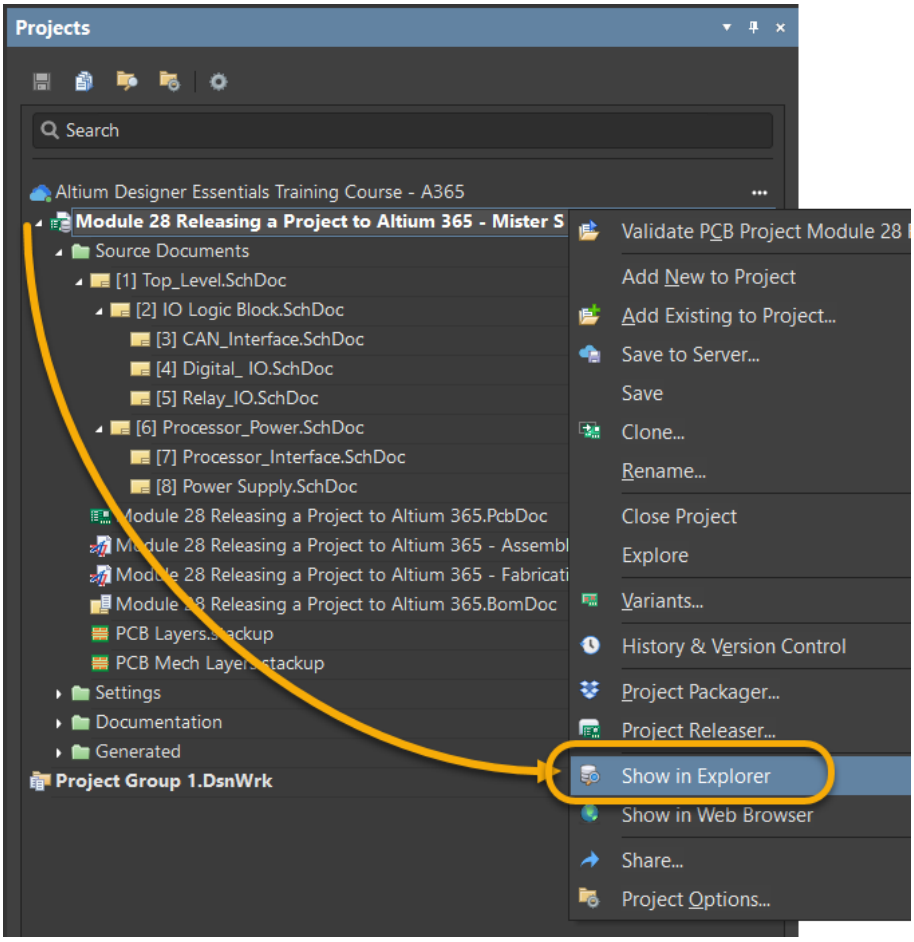
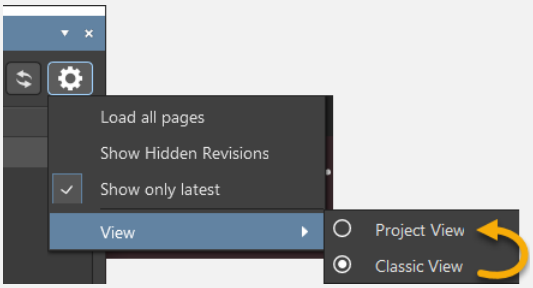


Figure 13. Open the Explorer Panel

42. The *Explorer* panel is showing you the Project with the release information below of the PCB preview.

Note: The Explorer Panel has two views the Project View and the Classic View. It might be necessary that you change the View by selecting the Gear icon and change to the Project View.



43. Select the Ellipsis button ••• and select **Change State**, Figure 14.

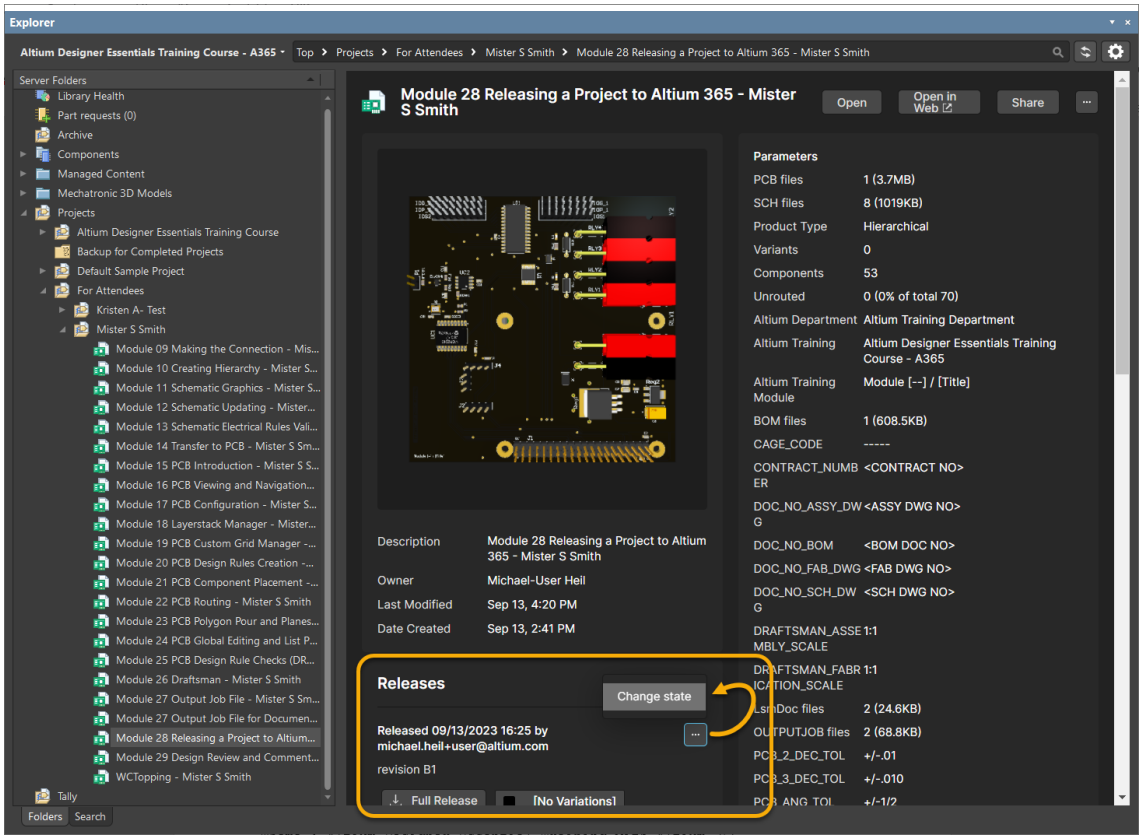


Figure 14. Example View for Explorer Panel with released Project

44. Next you see the *Batch State Change* dialog. The current state is `New From Design`, the next state is `Prototype`.
    The different Lifecycle steps that are possible, are configured from the Workspace Administrator as Lifecycle Definitions.
45. Select **Process**, add a Confirm Information and select **Yes** to change the lifecycle, Figure 15.

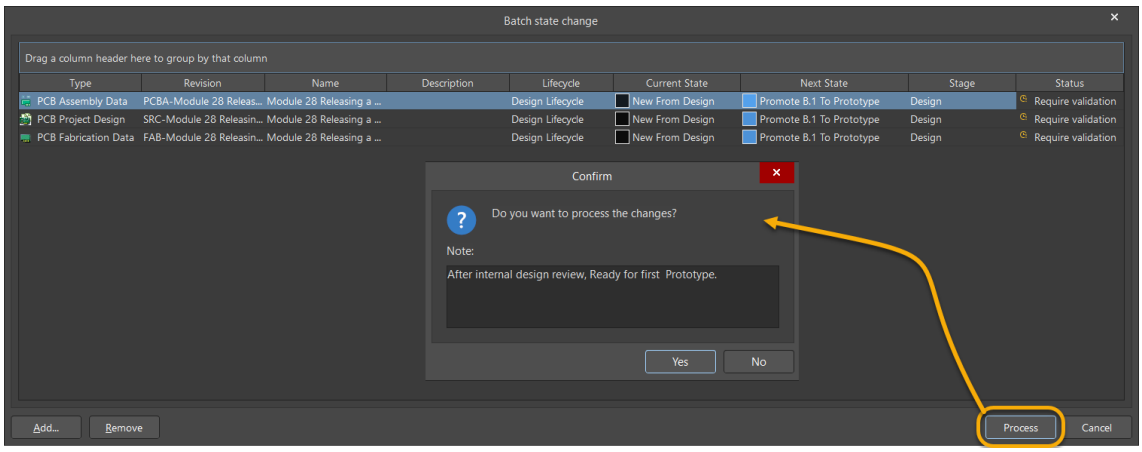


Figure 15. Confirm Change of Lifecycle state

46. Back to the *Explorer* panel you see the new Lifecycle for the last release, Figure 16.

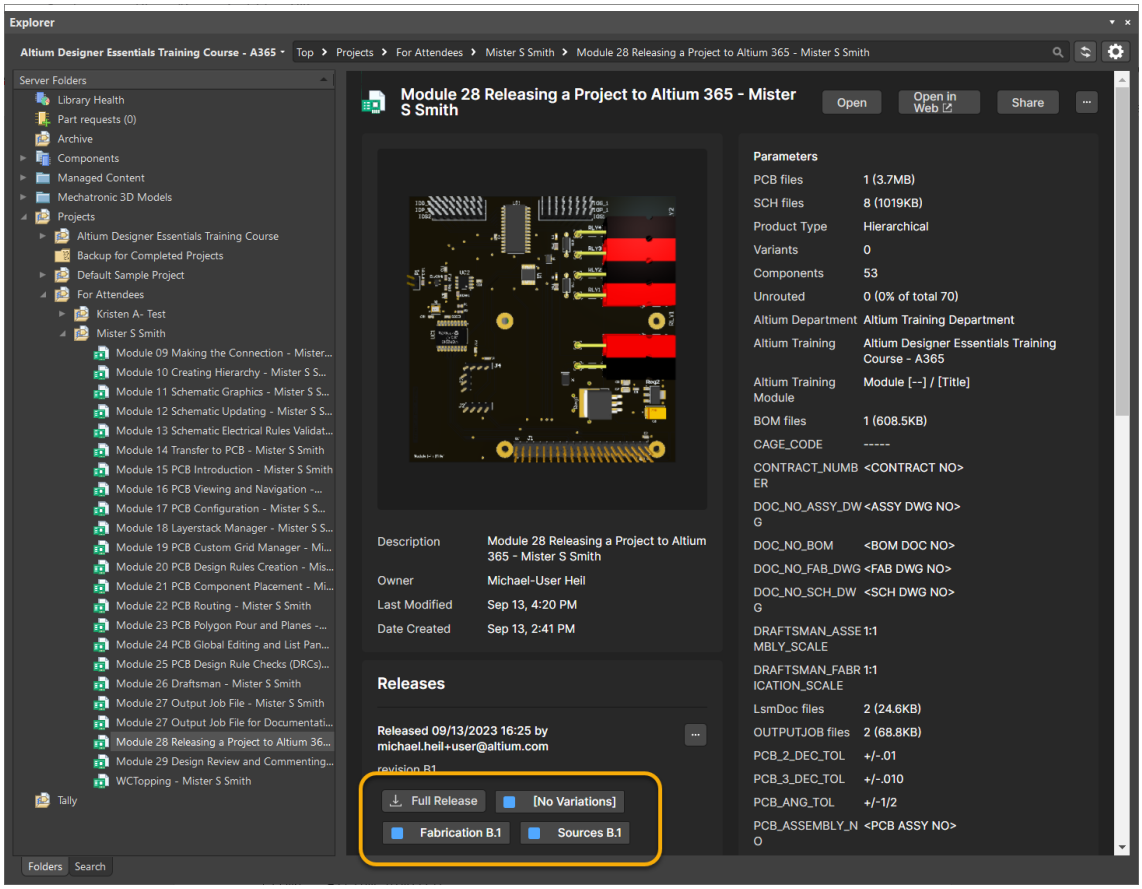
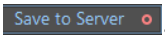


Figure 16. Lifecycle State Prototype

47. Save the modifications to the server if needed.
    a) At the *Project* panel, next to the Project name you find the command **Save to Server**  .
    b) Select **Save to Server.**
    c) At the dialog *Save [Project Name],*
        i) Add the comment `Module 28: Releasing a Project to Altium 365 - [Add Your Name].`
        ii) Select **OK.**
48. When ready, close the project and any open documents, **Window » Close All**.

Congratulations on completing the Module!

Module 28: Releasing a Project to Altium 365

from

Altium Designer Essentials Training with Altium 365

Thank you for choosing **Altium Designer**