

Altium Designer

Advanced Course

Module: Project Templates

Software, documentation and related materials:

Copyright © 2022 Altium LLC

All rights reserved. You are permitted to use this document provided that (1) the use of such is for personal use only and will not be copied or posted on any network computer or broadcast in any media, and (2) no modifications of the document are made. Unauthorized duplication, in the whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Altium LLC. Unauthorized duplication of this work may also be prohibited by local statute. Violators may be subject to both criminal and civil penalties.

ACTIVEBOM®, ActiveRoute®, Altium 365™, Altium Concord Pro™, Altium Designer®, Altium Vault®, Altium NEXUS™, Autotrax®, Camtastic®, Ciiva™, CIIVA SMARTPARTS®, CircuitMaker®, CircuitStudio®, Codemaker™, Common Parts Library™, Draftsman®, DXP™, Easytrax®, EE Concierge™, xSignals®, NanoBoard®, NATIVE 3D™, OCTOMYZE®, Octopart®, P-CAD®, PCBWORKS®, PDN Analyzer™, Protel®, Situs®, SmartParts™, Upverter™, X2®, xSignals® and their respective logos are trademarks or registered trademarks of Altium LLC or its subsidiaries. All other registered or unregistered trademarks referenced herein are the property of their respective owners and no trademark rights to the same are claimed.

Table of Contents

1.1 Purpose	3
1.2 Shortcuts	3
1.3 Preparation	3
1.4 Create a New Project Template	3
1.4.1 Add Schematic Templates	4
1.4.2 Add PCB Template.....	6
1.4.3 Adding Draftsman Templates	10
1.4.4 Add OutJob Templates.....	12
1.4.5 Using the Project Template.....	19
1.4.5.1 Using New Templates with New Document Defaults.	19
1.4.5.2 Create a New Project from the Template	20

Project Templates

1.1 Purpose

Project templates are a powerful tool to automate much of the documentation and file generation process of each new design. The project template becomes the starting place for each new design and has an extensive amount of preformatting in place to streamline the documentation and final file generation process. Assembly and fabrication drawing packages can be predefined, enabling the user to generate these documents with a few clicks in the OutJob file, creating drawings consistently, correctly the first time every time. A well-organized file directory structure will be preconfigured so that the outputs generated are easily referenced and understood. In this exercise, you will create a size B document project template.

1.2 Shortcuts



Shortcuts when working with Project Templates

F1:	Help
C-O:	Project Options
L:	View Configuration Panel (PCB)
T-P:	Preferences
CTRL+S:	Save Document

1.3 Preparation

1. **Close all existing projects and documents.**
2. Open the `Size B Template.PrjPCB` project found in the `Project Templates` folder of the Advanced Training.

1.4 Create a New Project Template

3. In the *Projects* panel right click on the `Size B Template.PrjPCB` and choose **Project Options...**
4. In the *Project Options* dialog, select the *Parameters* tab at the top-right of the dialog.
5. Note the various preconfigured Project level parameters.
6. Click the **Add** button and add a new parameter.
7. In the *Parameter Properties* dialog set the *Name* field to `PCB_REV` and *Value* field to `A`, as in Figure 1.

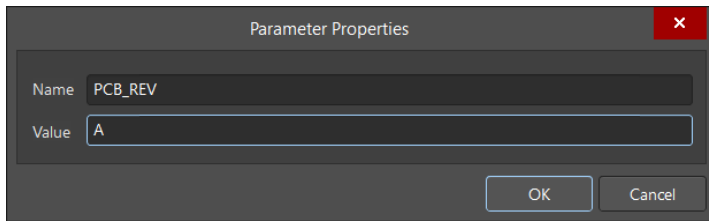


Figure 1. Adding a new Project Parameter

8. Click **OK** to close the dialog.
9. Click **OK** again to close and accept the changes.
10. Select **File » Save Project**.

1.4.1 Add Schematic Templates

11. Use **File » Open...** to open the two schematic template (.SchDot) files from ..\Project Template\Schematic Templates:

- a) TopSheet_Template_B.SchDot.
- b) NextSheets_Template_B.SchDot.

Note: These will open as Free Documents and not part of the current project.

12. Inspect each of the document's title block.

- a) The TopSheet_Template_B.SchDot has a full title block.
- b) The NextSheet_Template_B.SchDot has a subset of the title block commonly referred to as a continuation block.

The title blocks are populated with special text strings referred to as special strings. These special strings allow the title blocks to reference either the Project or Document parameter names and display the associated value. If a value is assigned to the parameter, it will display. If no parameter value is entered, the name of the special string will display, e.g., =DOC_NO_BOM. The equal sign designates a special string.



Visibility for Special String is controlled by the Schematic Preference setting under Graphical Editing - *Display Names of Special Strings that have No Values Defined* and *Display Name of Special String*.

13. Open TopSheet_Template_B and double-click on the Altium logo in the title block to open its properties in the *Properties* panel

- a) Click the three dots **...** to the right of the *File Name* field to select a new graphic (your company logo), see Figure 2.



Figure 2. File location of placed graphic specified in the Properties panel

- b) The **Embedded** checkbox is enabled. This will cause the image to be embedded in the schematic document and no longer references the external source.
- c) The **X:Y Ratio 1:1** option is enabled (Figure 3). Try resizing the image on the schematic by dragging one of the highlighted vertices.

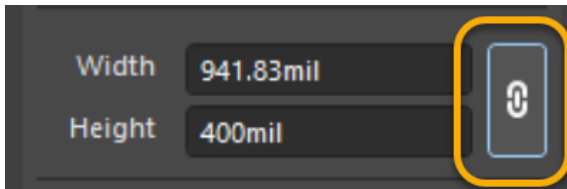


Figure 3. Image Size with active X:Y Ratio option

14. Repeat the previous step for the `NextSheet_Template_B.SchDot`.
15. Save your modifications.
16. Add two schematics to the open project:
 - a) Right-Click on the project `Size B Template.PrjPCB` and select **Add New to Project » Schematic**.
 - b) Perform this command again to add the second schematic.
17. Adding a template to the schematic:
 - a) With the first schematic, `Sheet1.SchDoc`, as the focused document, add the `TopSheet_Template_B.SchDot` to it as a template using **Design » Sheet Templates » Local » Load From File...**
 - b) In the *Open* dialog browse to `..\Project Templates\Schematic Templates\TopSheet_Template_B.SchDot`.
 - c) In the *Update Template* dialog that opens, under the *Choose Parameter Actions* section, select the **Replace all matching parameters** option.
 - d) Select **OK** to attach the template, next select **OK** in the *Information* pop up box that will appear.

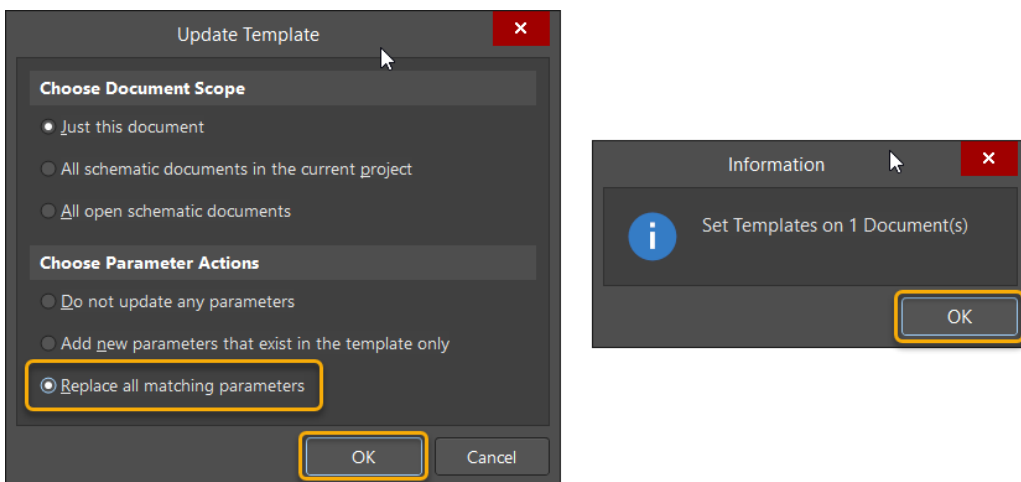


Figure 4. Update template settings

- e) Similarly, select the second schematic, `Sheet2.SchDoc`, and attach the `NextSheet_Template_B.SchDot` template using **Design » Sheet Templates » Local » Load From File...**
- f) In the *Open* dialog browse to `Advanced Design Files\Project Templates\Schematic Templates\NextSheet_Template_B.SchDot`.

- g) Use the same settings as before and click **OK**.
- 18. Save `Sheet1.SchDoc` schematic with the new name `TopSheet_Template_B.SchDoc`.
- 19. Save `Sheet2.SchDoc` schematic with the new name `NextSheet_Template_B.SchDoc`.
- 20. Try selecting the parameters in the title block for either of the project schematics. Note that the title block of each of these files are no longer selectable. The values will be populated parametrically by the Document and Project parameters values.
- 21. Close all Schematics.
- 22. Save the Project, **File » Save Project**.
- 23. Close all document in the *Free Documents* section, if still open.

1.4.2 Add PCB Template

- 24. Right click on the `Size B_Template.PrjPCB` project and choose **Add Existing to Project...**
- 25. In the *Choose Document to Add to Project* dialog browse to `..\Project Templates\B_Template.PcbDoc`.
- 26. Right click on the `Size B_Template.PrjPCB` project and choose **Add Existing to Project...**
- 27. In the *Choose Document to Add to Project* dialog browse to `\Advanced Design Files\Project Template\Template.PcbLib`.
- 28. Open the `B_Template.PcbDoc` PCB document.
- 29. Open the `Template.PcbLib` footprint library. It can be found under Libraries - PCB Library Documents in the *Projects* panel.
- 30. In the `B_Template.PcbDoc` open the *View Configurations* to view the mechanical layer allocations by pressing **L**, or with **View » Panels » View Configuration**.
- 31. Scroll down to the Mechanical Layers (M) section, see Figure 5.

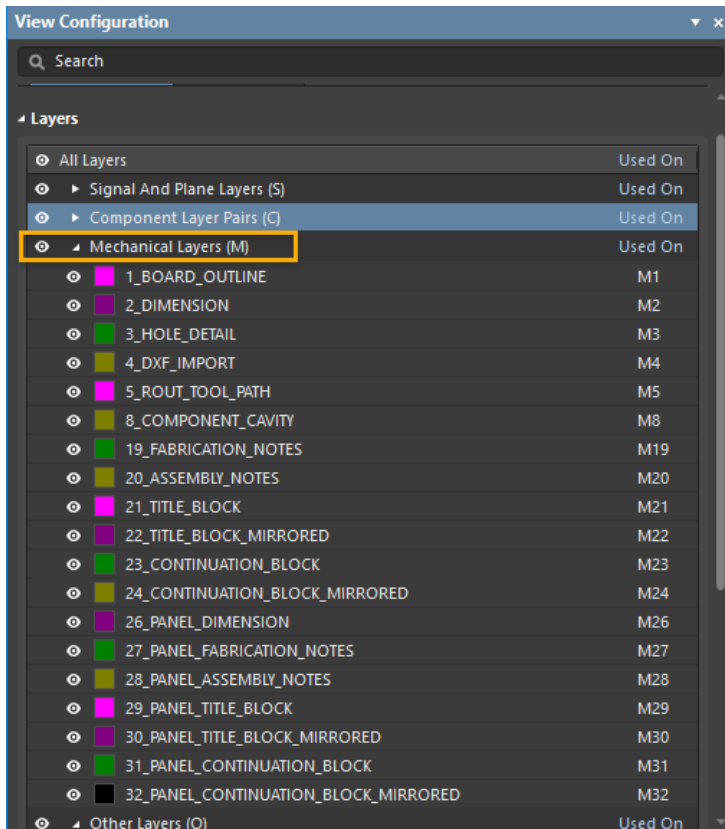



Figure 5. Mechanical layers

32. Open the footprint library `Template.PcbLib` from the *Projects* panel.
33. Once the library is open, open the *View Configuration* panel by pressing **L**.
 - a) Note the mechanical layer allocations are different from the PCB, for template standardization it is advisable to maintain the same mechanical layer setup in both the PCB library and the PCB documents. This can be done manually, or once the mechanical layer setup has been defined in the PCB, it can be imported into the PCB library.



The information placed on mechanical layers in the footprint library will reside on the same mechanical layers in the PCB document.

34. Return to the PCB document `B_Template.PcbDoc`.
35. Open the *View Configuration* panel.
 - a) Scroll to the *Component Layer Pairs* section.
 - b) Click the arrow , if necessary, to expand the section.
 - c) Paired Mechanical layers will not display in the *Mechanical Layers* section but will be listed in the *Component Layer Pairs* section. Mechanical layers 9 thru 16 have been paired as shown in below in Figure 6.

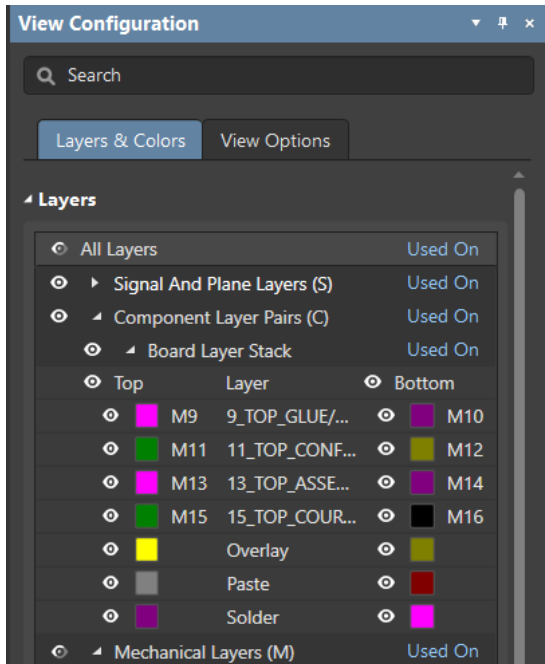


Figure 6. Component Layer Pairs. Paired Mechanical Layers showing Top (left column) and Bottom (right column) associated pairs

- d) The paired mechanical layers form a Component Layer Pair which associates component information between the top and bottom of the PCB. Component primitives existing on one of the Component Layer Pairs will be transferred to its associated paired layer when the component is flipped from the top side to the bottom side of the PCB. Component layer pairs function much the same way as the top and bottom solder mask layers, overlay layers and paste mask layers.
36. Next, we will export the PCB Mechanical Layer setup and import it into the PCB Library.
37. To export the PCB Mechanical Layer setup, select menu **Tools » Export Mechanical Layers...**
38. When prompted, name the stackup file `Mechanical.stackup`, and save it to the `...\Project Template` folder.
39. Change focus to the PCB Library, import the newly created stackup into the PCB library by selecting menu **Tools » Import Mechanical Layers...**; from file explorer window select the file `Mechanical.stackup` from the `...\Project Template` folder.
40. Open the *View Configuration* panel and observe the PCB Library Mechanical Layers are now the same as the PCB template. If needed, the import and export can also be done from the library to the PCB.
41. Save all files, **File » Save All**.
42. Switch back to the PCB, in the PCB editor workspace select mechanical layer `19_FABRICATION_NOTES` from the tabs at the bottom of the workspace. Click the arrow button to scroll the layer tabs until it is visible for selection, see Figure 7.



Figure 7. Selecting `19_FABRICATION_NOTES` layer tab

43. Text and other information can be placed into the PCB editor from Windows OLE compliant files. Place fabrication notes to the right side of the PCB, **Place » Object From File**, in the *Choose file* dialog, browse to the file `...\Project Templates\Other Templates\NOTES UNLESS OTHERWISE SPECIFIED.docx` and click **Open**.

44. Position the text on the mouse cursor to the right side of the PCB, like Figure 8 below.
45. Save the PCB file.

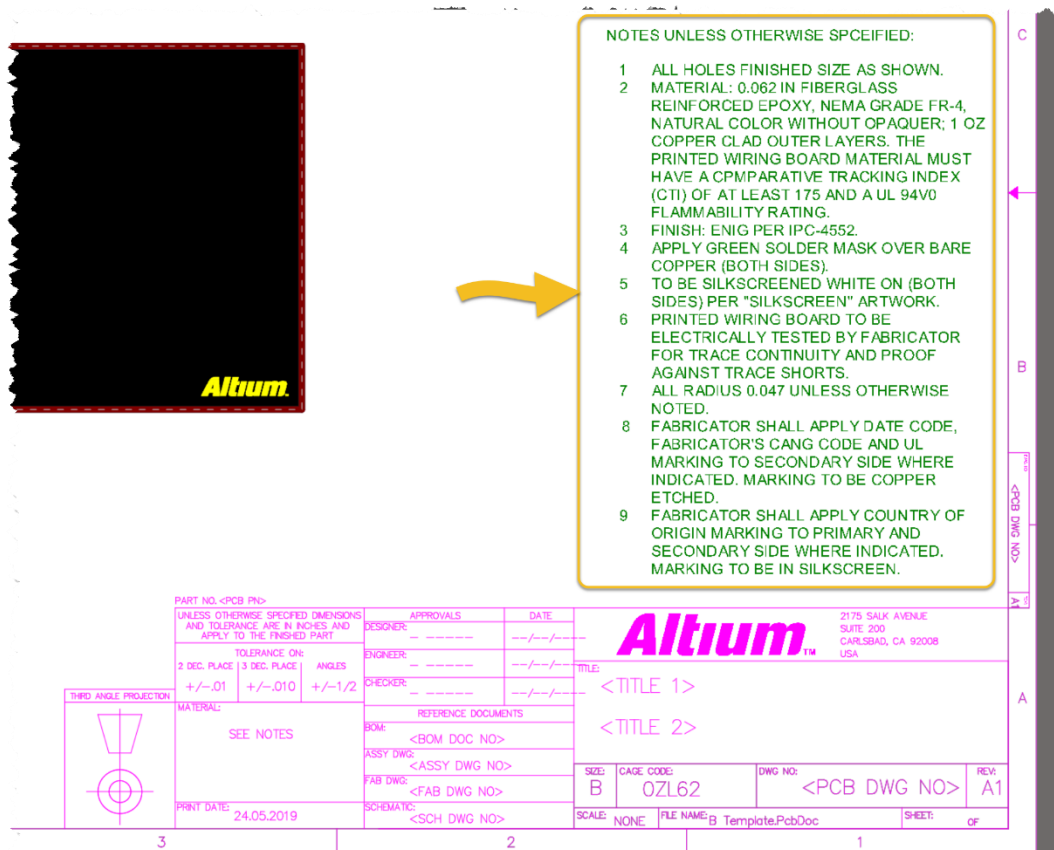


Figure 8. Place fabrication notes from file to the right side of the PCB



The feature to place notes and other information directly from a file is very powerful, allowing various notes sections for different fabrication requirements to be preconfigured and stored as a library of informational files.

46. Zoom into the lower left corner of the title block/reference zone region and Double Click on the text, *Printout_Name is not interpreted until output* (choose the text on the 21_TITLE_BLOCK layer when the option pops up). Clicking on this text, view its properties in the *Properties* dialog. Note the special string. *Printout_Name* is a special string which exists in the PCB document by default. Close the Properties dialog.

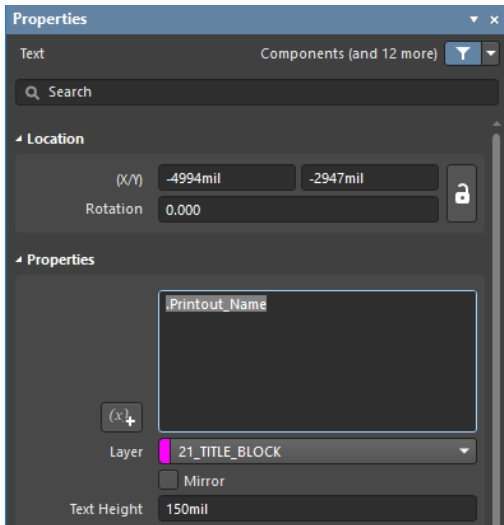


Figure 9. *Printout_Name* special string



The special string *.Printout_Name* is populated when documents are generated utilizing the associated layer where the string is placed. *.Printout_Name* is populated with the sheet name as configured in the Output Job file for fabrication and assembly drawings. The usage of the special string *.Printout_Name* will be investigated in the next section.

47. At the lower right of the PCB, zoom in to the title block. Note the *Project* level parameters placed in the Reference Documents section. This allows documents to be named in one place, the Project Options Parameters tab, and those names will propagate down into the title block for reference. They will also name the files during output generation.

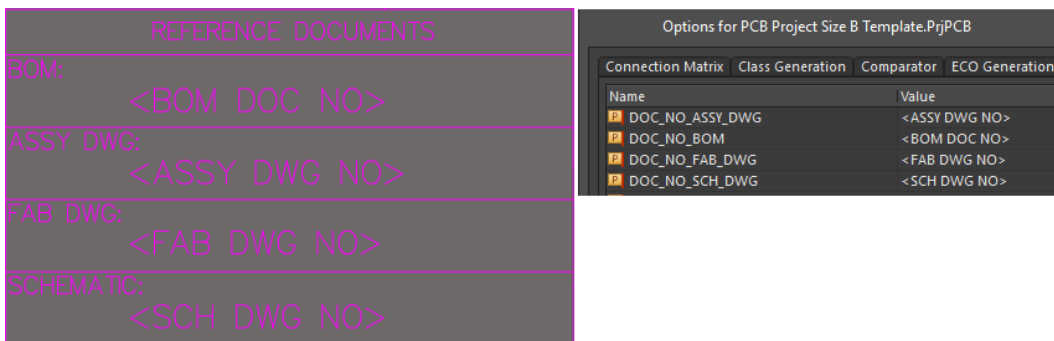



Figure 10. Reference Documents section of title block

1.4.3 Adding Draftsman Templates

48. Locate the Draftsman template directory on the client machine.
49. Open  **Preferences – Data Management - Templates**.

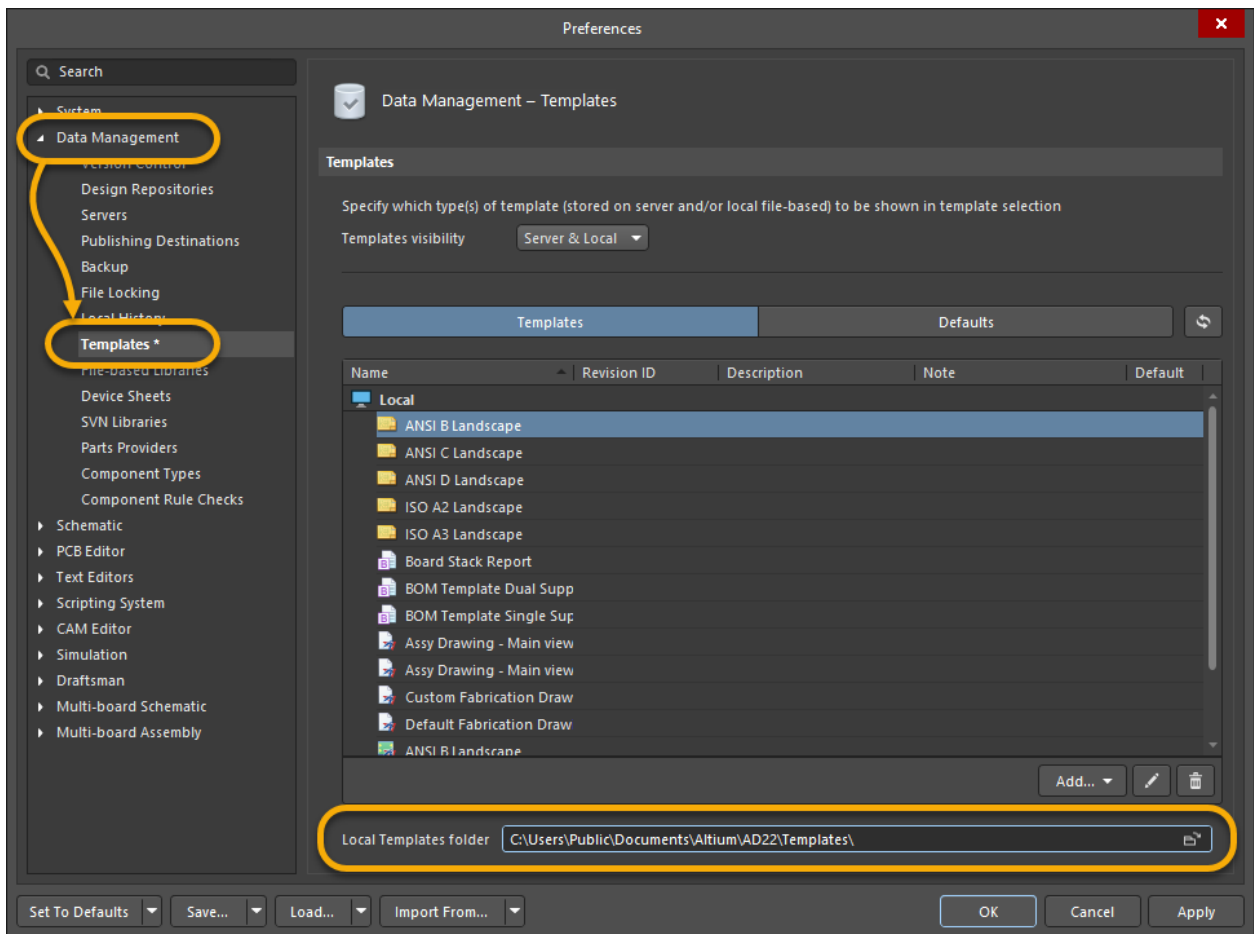


Figure 11. Location of Draftsman templates on client machine

50. Copy the *Local Templates Folder* path and paste it into Windows File Explorer.
51. Copy the Draftsman templates from `.. \Project Templates \Draftsman Templates` and paste them into the Draftsman templates path location in Windows Explorer, Figure 12.

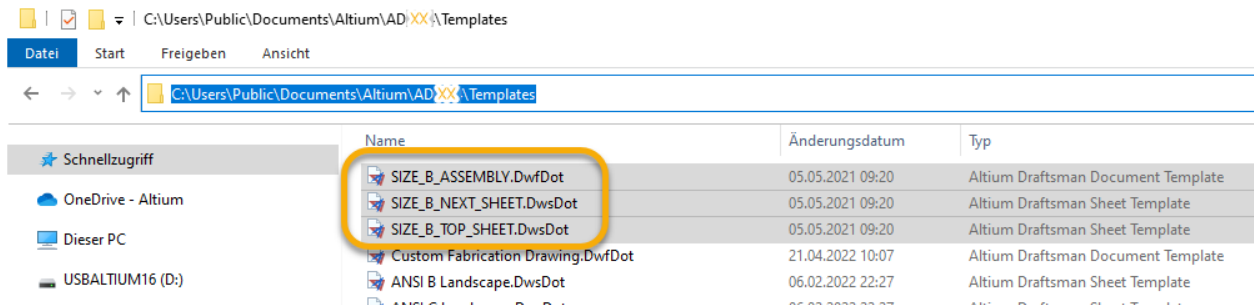


Figure 12 Training Draftsman Template Folder

52. Close the *Preferences*.
53. Add a Draftsman document to the Project Template, **File » New » Draftsman Document**.
54. In the New Draftsman Document dialog, choose the `SIZE_B_ASSEMBLY` template from the local section and click **OK** (see Figure 13, which may look different based on any servers you're connected to).

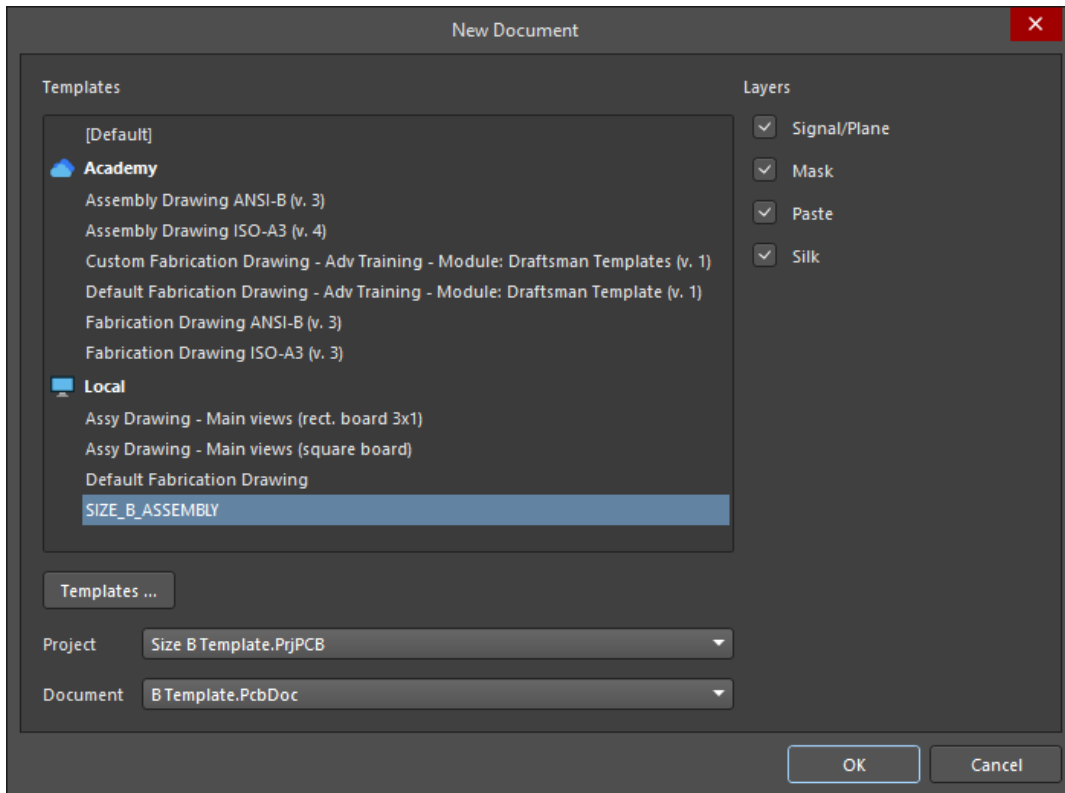


Figure 13. Choose the Draftsman template

55. Select **File » Save** to save the new Draftsman document (PCBDwf). Leave the default name.
56. Save the Project.

1.4.4 Add OutJob Templates

57. Right click on the `Size B Template.PrjPcb` file in the *Projects* panel and choose **Add Existing to Project...**
58. Browse to `... \Project Templates \` and choose all three Output Job files, `Assembly.OutJob`, `Documentation.OutJob`, `Fabrication.OutJob` and click **Open**.
59. Open the `Assembly.OutJob` file by double-clicking on it in the *Projects* panel. It will be located under the project's *Settings - Output Job Files* folder.
60. On the right side of the OutJob editor there are, but not limited to, four Output Containers. A PDF generator for `Assembly Drawing`, and a PDF generator for `Bill Of Materials`. Two file generators, one for `Assembly Files` and one for generating a `Bill Of Materials` as an Excel file, see Figure 14.

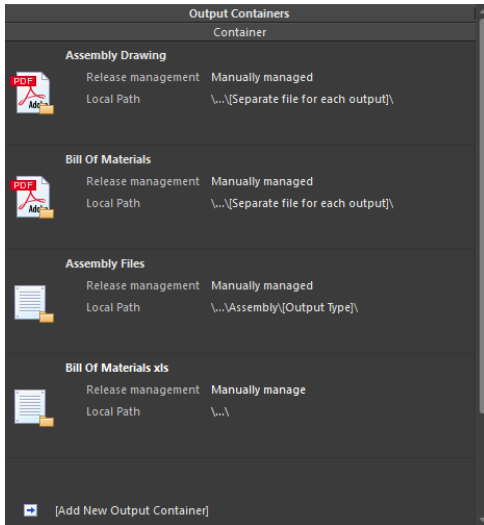


Figure 14. OutJob Output Generators

61. Click on each Output Container, Figure 15. and notice what files will be generated. Note the Assembly Drawing Output Container doesn't have any files pointing at it.

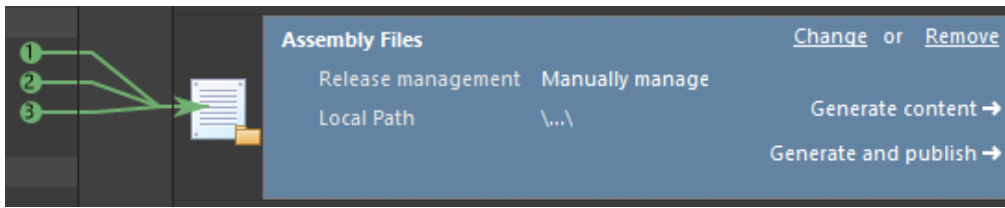


Figure 15. Targeted Output Containers

62. Double-click on the Bill Of Materials Output Container or select it and click on **Change** to see how the directory structure is setup and how the output file naming is configured. The area just above the Preview area allows configuration of the final output path, Figure 16.
- Note the final bill of materials document will be named by the project level parameter DOC_NO_BOM with the text _BOM appended to the end of the file name. This is done with the =DOC_NO_BOM+'_'+'BOM' string.
 - Find the *Preview* area. Here you will see the file name and directory structure and location of where the file will be generated within the project, see Figure 16.

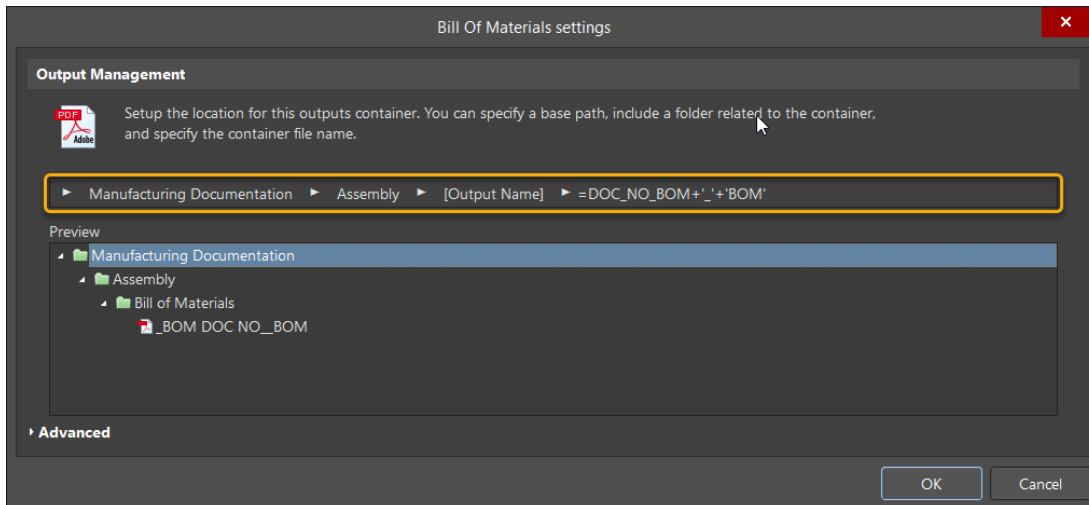


Figure 16. Bill of Materials Output Container settings dialog

63. Close the *Bill of Materials settings* dialog.



Document names can be changed in one place, which can be found in the **Project Options, Parameters** tab. Those names will propagate down through the rest of the project. They can even be used to name output files.

64. Next, we will add and configure an Assembly Drawing output.

65. In *Assembly.OutJob*, create a new *Assembly Drawing* output.

- Click **Add New Assembly Output** under the *Assembly Outputs* category in the *OutJob* editor.
- Select *Assembly Drawings - [PCB Document]* and double click, or use right mouse button and select **Configure...** as shown in Figure 17 to open the *Preview PCB* properties window, as shown in Figure 18.

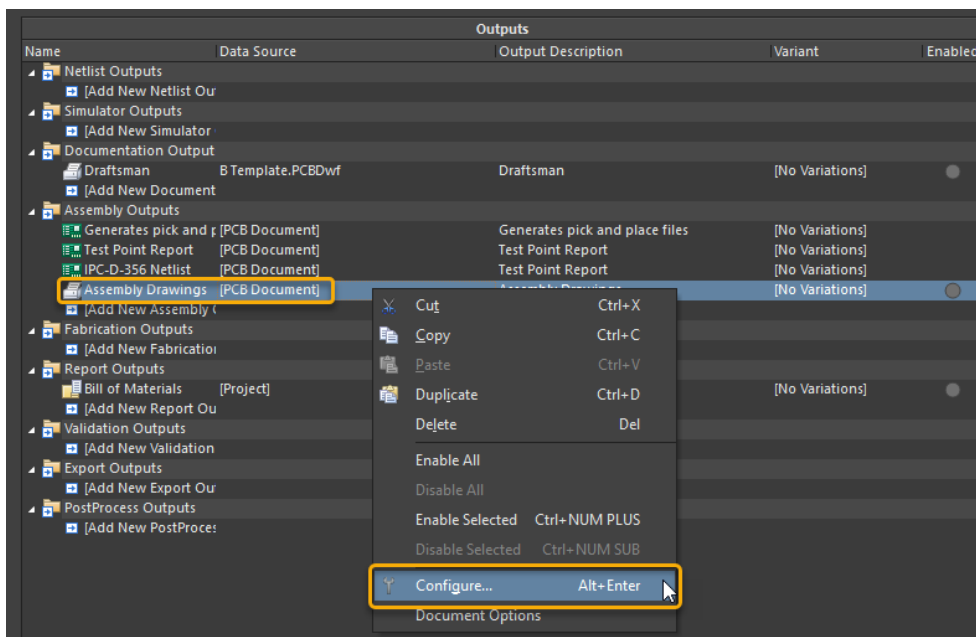


Figure 17. Create new Assembly Drawing output

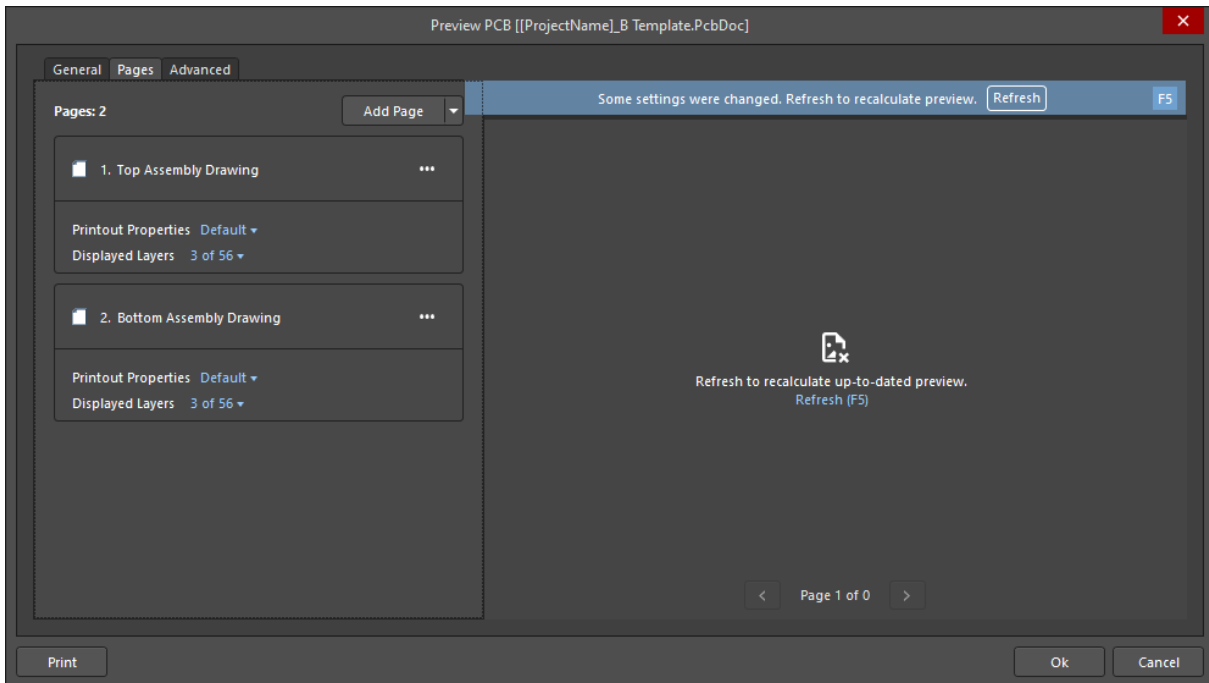


Figure 18. The default drawing sheets and layer sets

66. Select the *Page* tab and click on *Refresh* as shown in Figure 19 to bring up the *Preview* as shown in Figure 19.

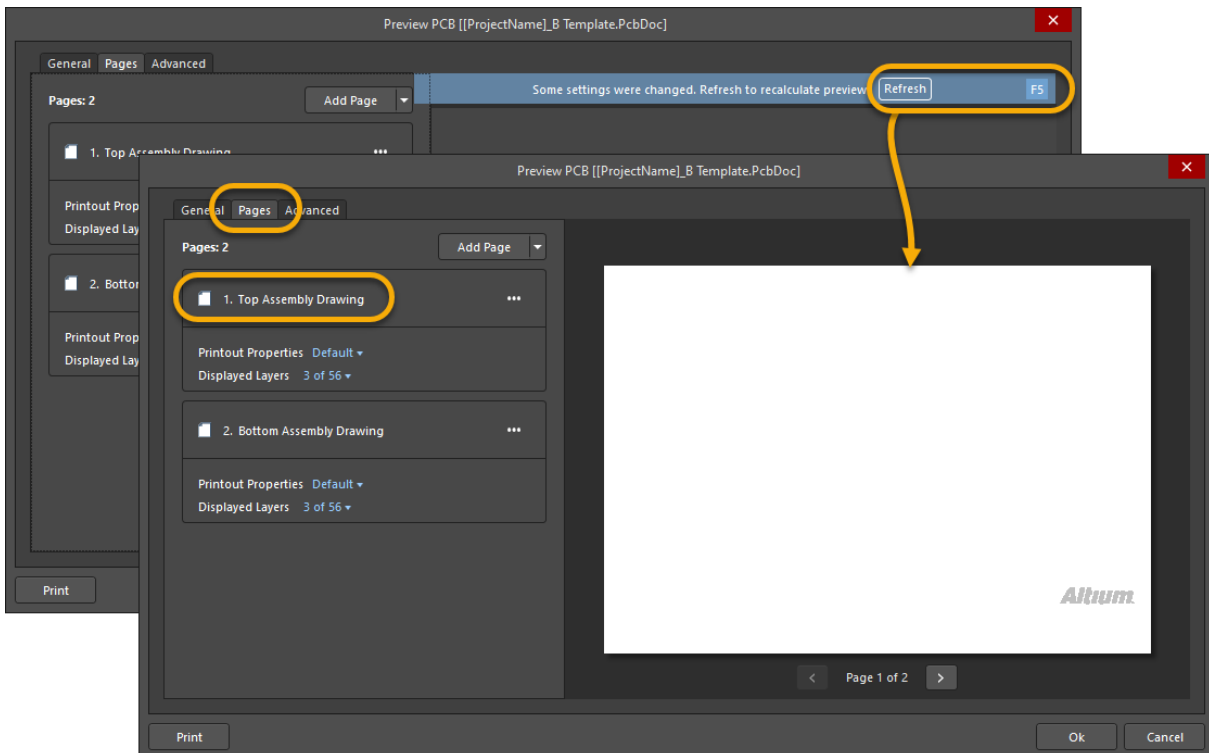


Figure 19. Preview properties dialog

67. Select the *Display Layers* to bring up the *Preview PCB* dialog, Figure 20.



The Printout Name parametrically populates the special string, *.Printout_Name*, in the PCB Template. By including the mechanical layer on which the *.Printout_Name* has been placed in the PCB Template, each Printout (sheet) of the drawing package will have this title applied.

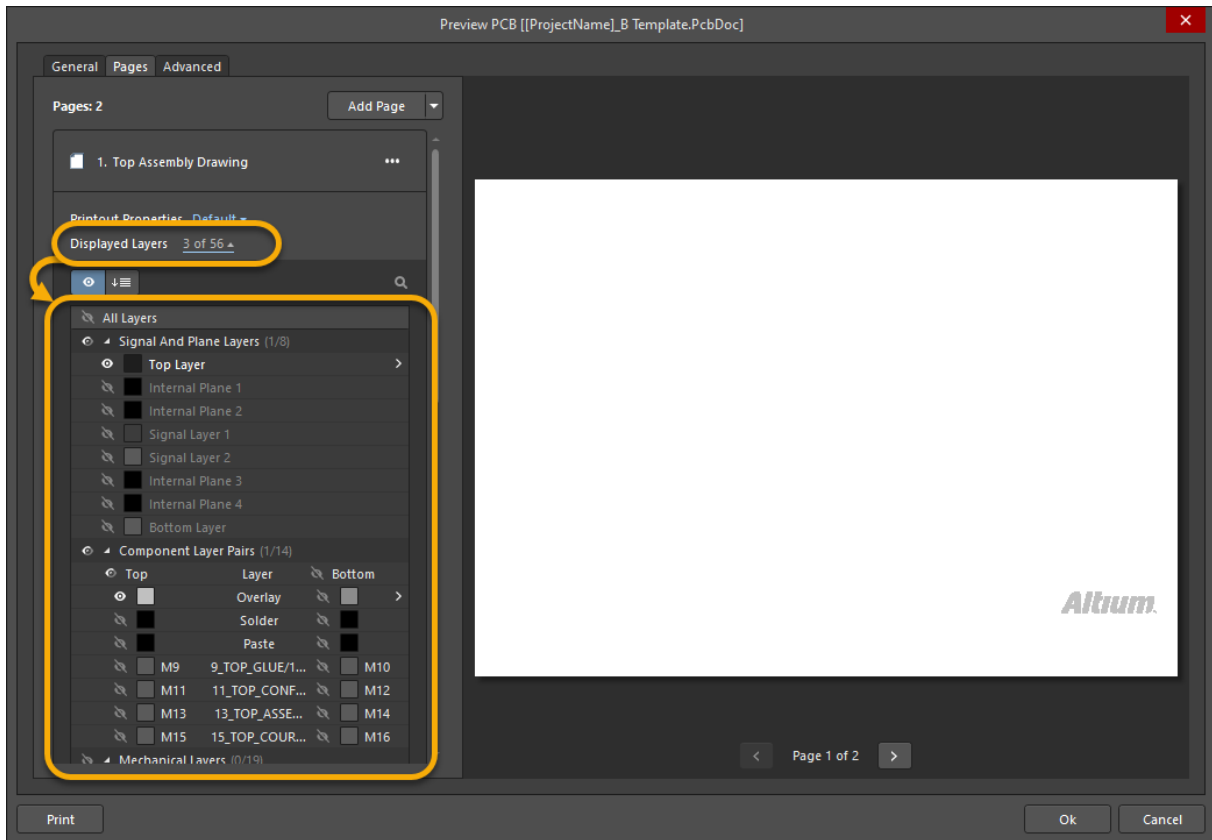


Figure 20. Preview with Layer Option

68. Using Figure 21 as a reference, add and remove layers for this printout by selecting from the list of available layers and then selecting refresh, lets select layers 21 and 24, and then select **Refresh** as shown in Figure 21, or F5 from your keyboard.

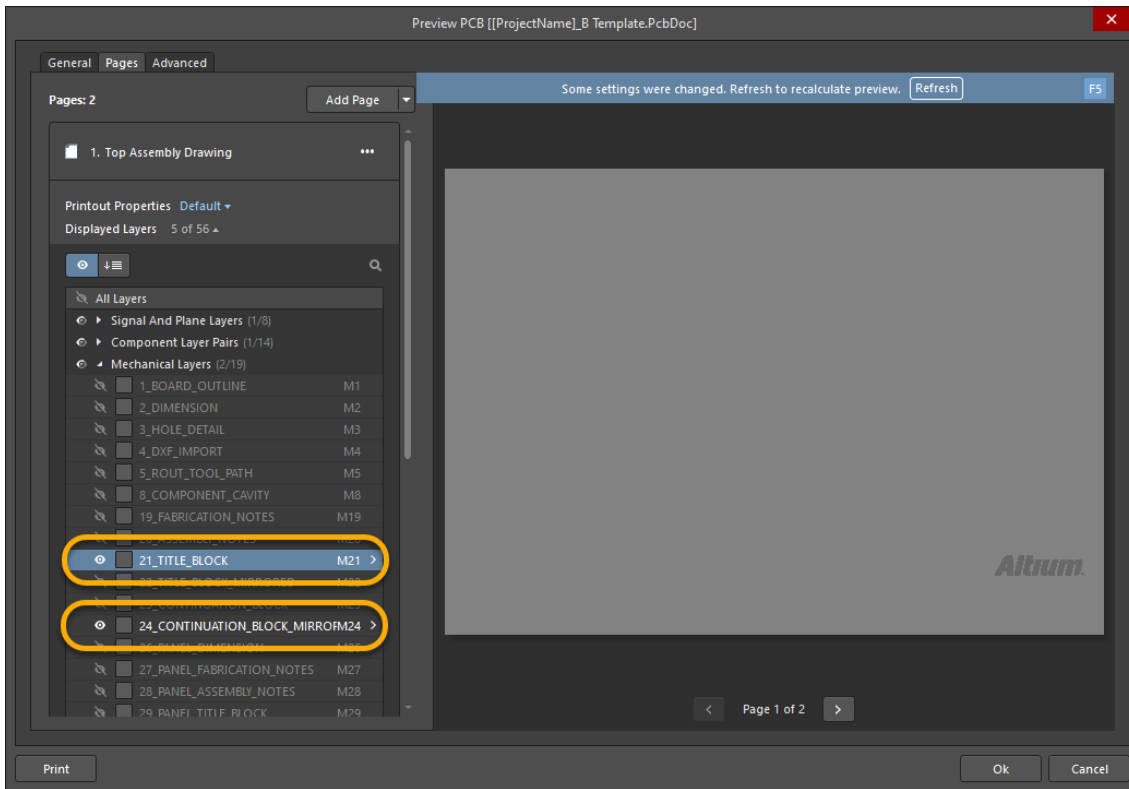


Figure 21. Final PCB Printout Properties

69. The result should be as seen in Figure 22.

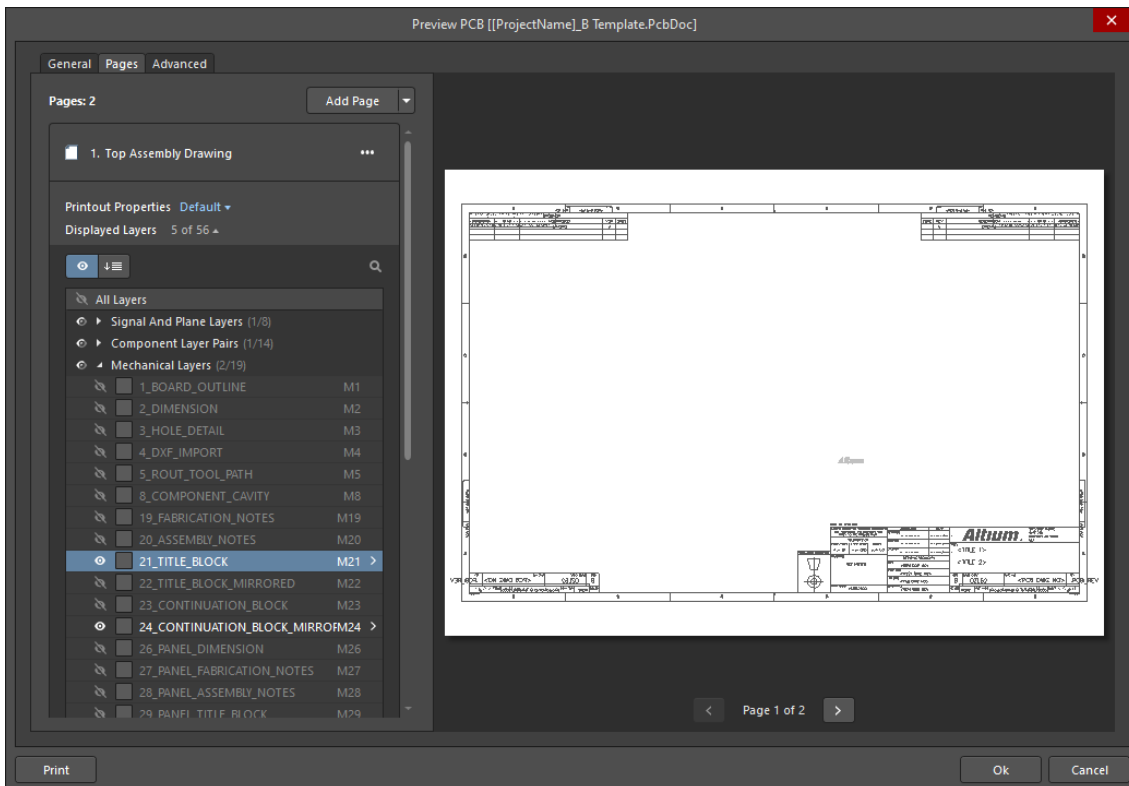


Figure 22. Final PCB Preview

70. Click **OK** to save changes and close the Preview.
71. Add a new Draftsman output to the OutJob by clicking **Add New Documentation Output** under the *Documentation Outputs* section of the OutJob. Select **B Template.PCBDwf**, Figure 23.

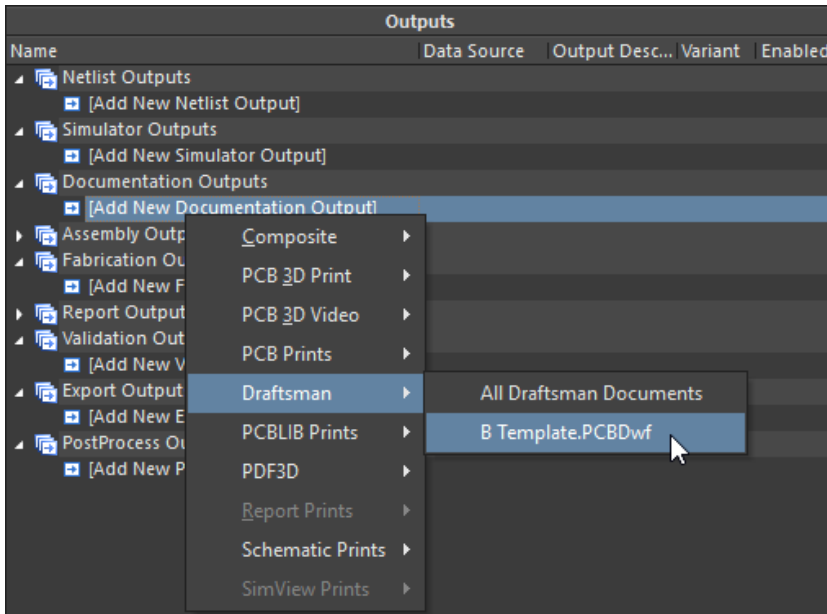


Figure 23. Adding the Draftsman document to the OutJob

72. With the Assembly Drawing Output Container selected, click the radio button for both the Draftsman and Assembly Drawings Outputs we just created, as shown in Figure 24.

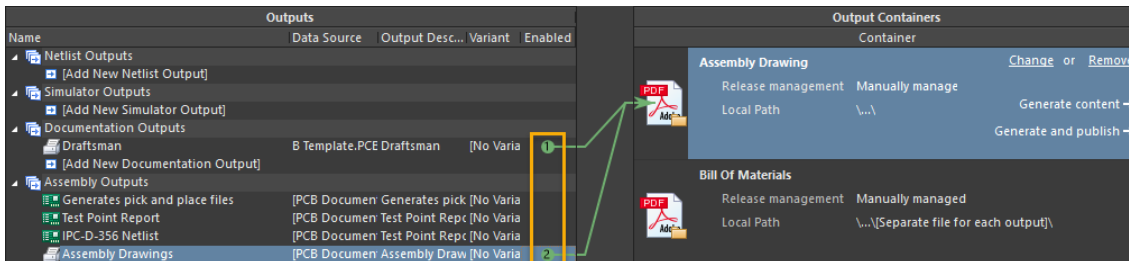


Figure 24. Targeting Assembly Drawing output to the Output Container

73. Select **File » Save All** to save the project and all other files in the *Projects* panel.
74. The final project should appear similar to what is shown in Figure 25. Note, if you validate the project the Components folder will appear. Close the project.

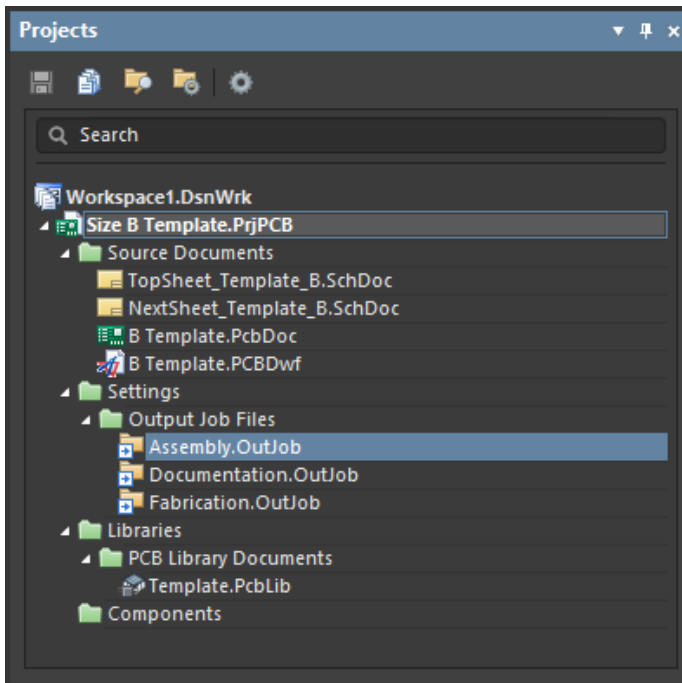



Figure 25. Final Template project

1.4.5 Using the Project Template

75. We will save this new project template to the default template directory to make this available as a local project template.
76. Access the Preferences by clicking the *System Preferences* gear icon .
77. Navigate to **Data Management – Templates**.
78. Copy the Template location path and paste it into Windows File Explorer to open the directory.
79. Copy the entire Project Template folder containing our newly made project and files into the Altium Designer Templates folder location.



The Release specific subfolder of Altium in the Public Documents section is subject for deletion on a Complete Uninstall. Another option is to copy the Release Specific Template folder to the Altium base folder, e.g., C:\Users\Public\Documents\Altium. Then, modify the Data Management » Templates path to point to this location.

1.4.5.1 Using New Templates with New Document Defaults.

80. In the next steps we will replace the default files produced during new file creation with our own template files.
81. Modify the New Document Defaults to reference the template files
 - a) While still in **Data Management – Templates** in the *System Preferences*, click the *Defaults* tab to the right of the *Templates* tab.
 - b) Click **Add... » From File » Schematic**
 - c) Browse to the templates path we copied earlier, select `NextSheet_Template_B.SchDoc` from the Project Template folder.
 - d) Add templates for the various file types, Schematic, PCB, etc. as shown in the Figure 26 below.

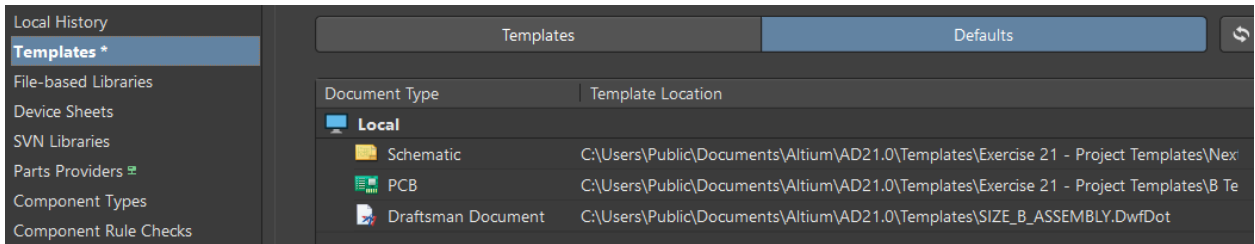


Figure 26: New Document Defaults targeting the newly created templates

82. Close the *Preferences*.

1.4.5.2 Create a New Project from the Template



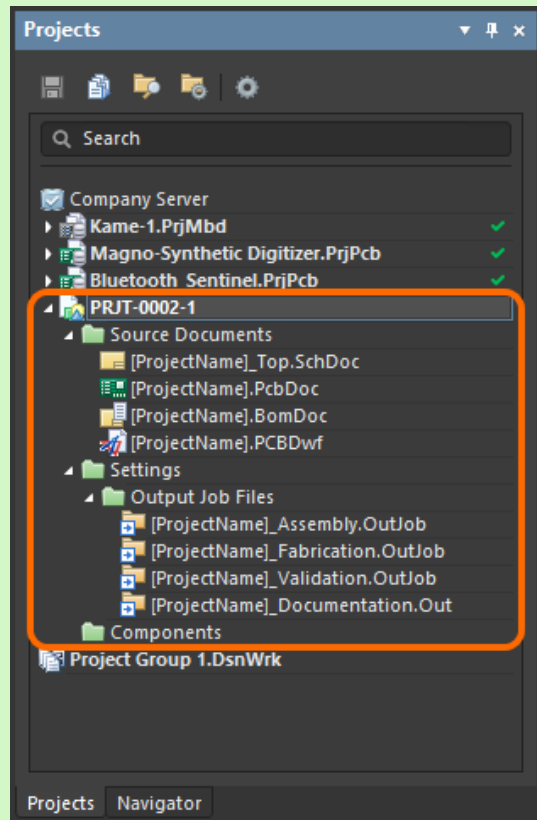
Automated Document Naming

A valuable feature that becomes available when Altium Designer applies a Project Template to a new design, is the automated naming of the template's constituent documents. This capability is enabled by including a special [ProjectName] syntax in the template document names, which will be replaced by the nominated project Name when the template is used for a new project.

So for example, if a schematic document in a Project Template is named [ProjectName].SchDoc, and that template is then used for a new Altium Designer project called Flux_Triangulator, then the schematic file created in the project will be named Flux_Triangulator.SchDoc. Note that the naming string can coexist with other characters, so a template document such as [ProjectName]_Top-Level-Structure.SchDoc will become a project document named Flux_Triangulator_Top-Level-Structure.SchDoc in a newly created project that has been named Flux_Triangulator.



To include the preprocessing name capability when creating a Project Template, simply edit the document names accordingly (Save As) before releasing the project to the server as a Project Template (or if just saving it locally). Alternatively, open the revision of an existing Project Template Item and edit the document names to include the [ProjectName] string, then release back to the server into the next revision of that Item.



Example project readied as a template, and using the [ProjectName] syntax to facilitate automated document naming when that template is reused.

83. Let's create a new Project by going to **File » New » Project...**
84. In the **Create Project** dialog, select our new local template, Figure 27. Choose any name and your choice of folder.

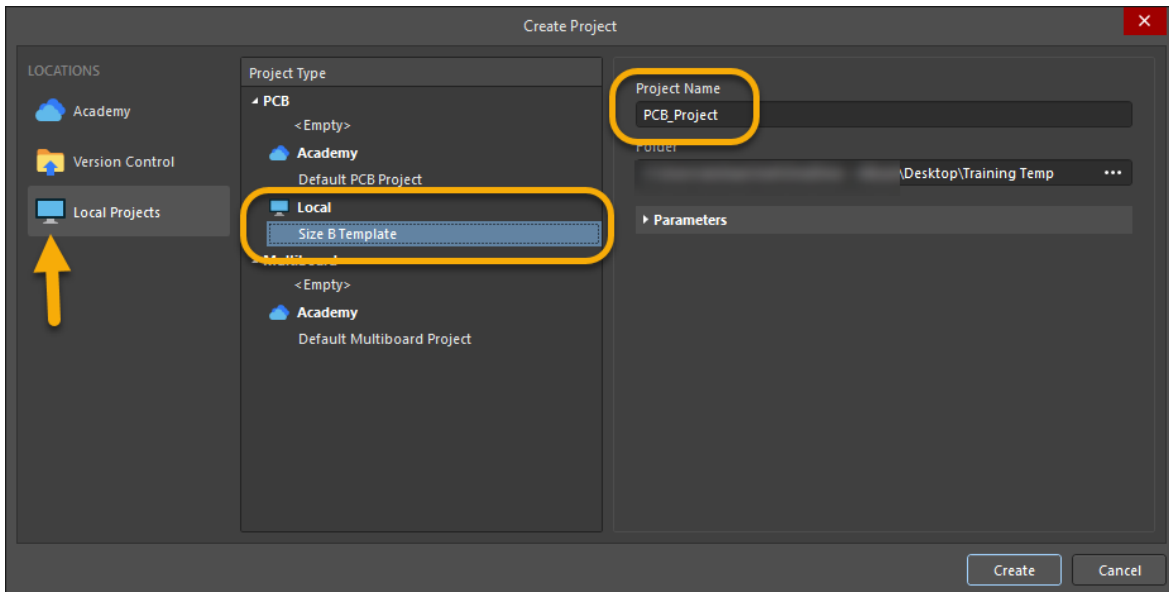


Figure 27: Create Project Dialog

85. Expand out the folders in the Projects panel, compare to Figure 25. They should be identical, as shown in Figure 28 below.

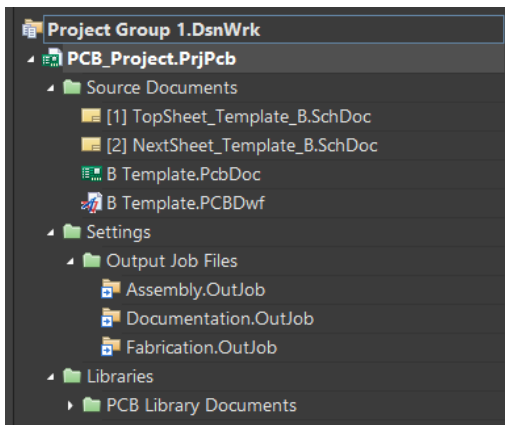


Figure 28: New Project from our Template

86. Change the Value of the Project Parameter `DOC_NO_ASSY_DWG` from `<ASSY DWG NO>` to `ABC123` and click **OK**.
87. Open the schematic `TopSheet_Template_B.SchDoc` and note the **ASSY DWG:** field in the **REFERENCE DOCUMENTS** region of the title block has changed to `ABC123`, Figure 29.

REFERENCE DOCUMENTS	
BOM:	<BOM DOC NO>
ASSY DWG:	ABC123
FAB DWG:	<FAB DWG NO>
PCB DWG:	<PCB DWG NO>

Figure 29. REFERENCE DOCUMENTS region of the schematic title block

88. Open the PCB document `B_Template.PcbDoc` and note the **ASSY DWG:** field in the **REFERENCE DOCUMENTS** region of the title block has changed to `ABC123`, Figure 30.

BOM:	<BOM DOC NO>
ASSY DWG:	ABC123
FAB DWG:	<FAB DWG NO>
SCHEMATIC:	<SCH DWG NO>

Figure 30. REFERENCE DOCUMENTS region of the PCB title block

89. Open the `Assembly.OutJob` and click **Change** in the Assembly Drawing Output Container. Note the file naming convention for the assembly drawing has changed to `ABC123_ASSY`, Figure 31.

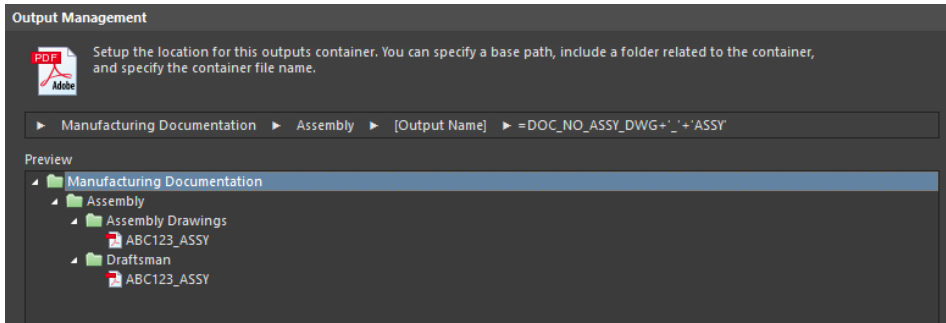


Figure 31. Assembly Drawing settings dialog

90. Close the new project, feel free to save the files.
91. Close the project and any open documents.

Congratulations on completing module

Project Templates

from the

Altium Designer Advanced Course

Thank you for choosing Altium Designer