Altium Designer

Advanced Training with Altium 365

Template 2 - Project Templates

Software, documentation and related materials:

Table of Contents

Template 2 - Project Templates

1 Purpose

Project templates help automate the creation of documents and files for new designs. The project template serves as the starting point for new designs, with preformatted settings to streamline documentation and file generation. You can predefine assembly and fabrication drawing packages to quickly generate them with just a few clicks in the OutJob file, ensuring consistent and accurate results every time. A preconfigured, organized file structure makes it easy to find and understand the generated outputs. In this module, you will create a size B document project template.

2 Shortcuts

Shortcuts used when working with Template 2 - Project Templates

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

3 Preparation

1. Close all existing projects and documents.
2. Open the *Explorer* panel from the **Panels** button, or with the combination **K»R**.
3. Navigate to the folder `Training Download`, Figure 1.

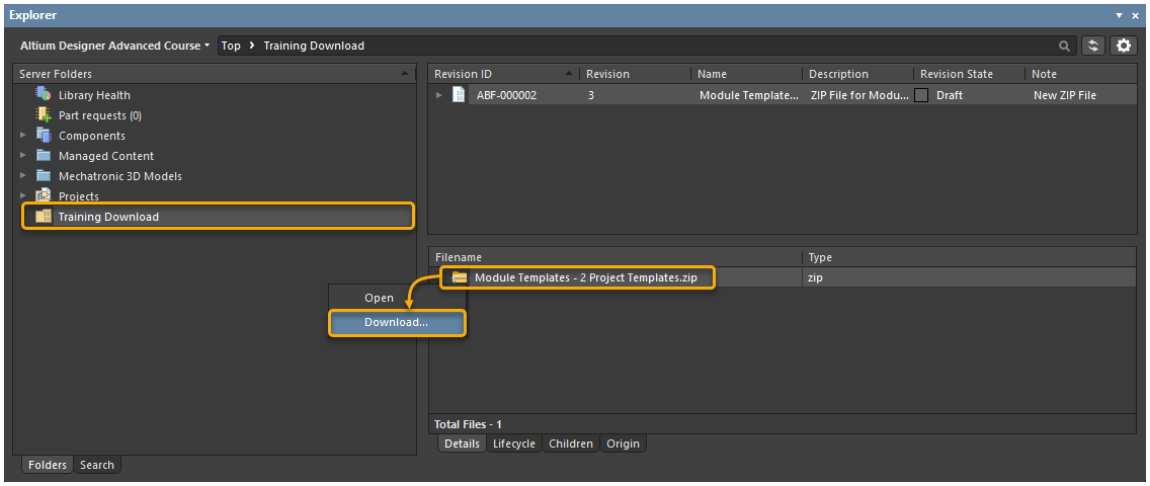


Figure 1. Training Download folder with a predefined file

4. Right-click `Module Templates - 2 Project Templates.zip` file from the *Filename* branch.
5. Select `Download...` to download the ZIP file. Download the file and save it on the Desktop, or any other folder of choice.
6. From the dialog *Download from Vault,* select **Explore** to open a Windows Explorer, Figure 2.

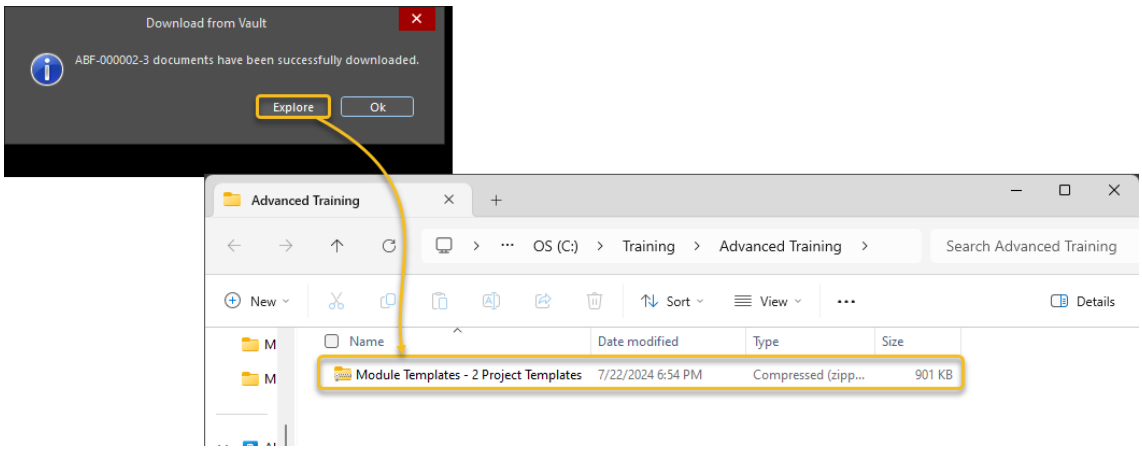


Figure 2. Compressed file in your Windows Explorer

7. Extract the files from the compressed folder. Keep the Windows Explorer open. You will come back to the downloaded files later in the exercise.

4 Creating New Project Template

Next, you will create two Project Templates. You can create a project template by starting a new project and manually adding the necessary information. Another option is to modify an existing project to make it ready as a template.
First, you'll start a project from scratch to understand the overall process. Later, you'll speed up the process by using a predefined project as the starting point for the template.

8. Open the *Explorer* panel from the **Panels** button, or with the combination **K»R**.
9. Navigate to the Branch *Managed Content - Templates – Project Templates*, Figure 3.

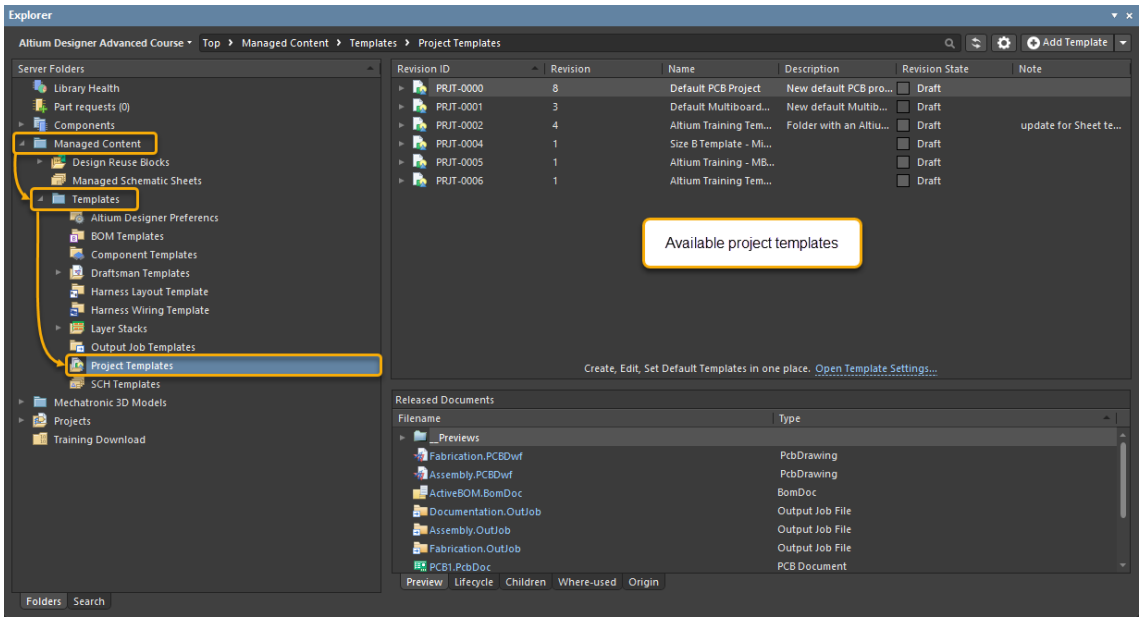


Figure 3. Available project templates

10. You will see the current available project templates that are offered in the Create Project dialog. Alternatively, to display the available project templates, you can execute the **File » New » Project...** command.
11. Close the *Explorer* panel.

4.1 Creating New Project Template from Scratch

4.1.1 Creating the Project

12. Go to **File » New » Project...** to launch the *Create Project* dialog. The dialog shown in Figure 4 will appear.

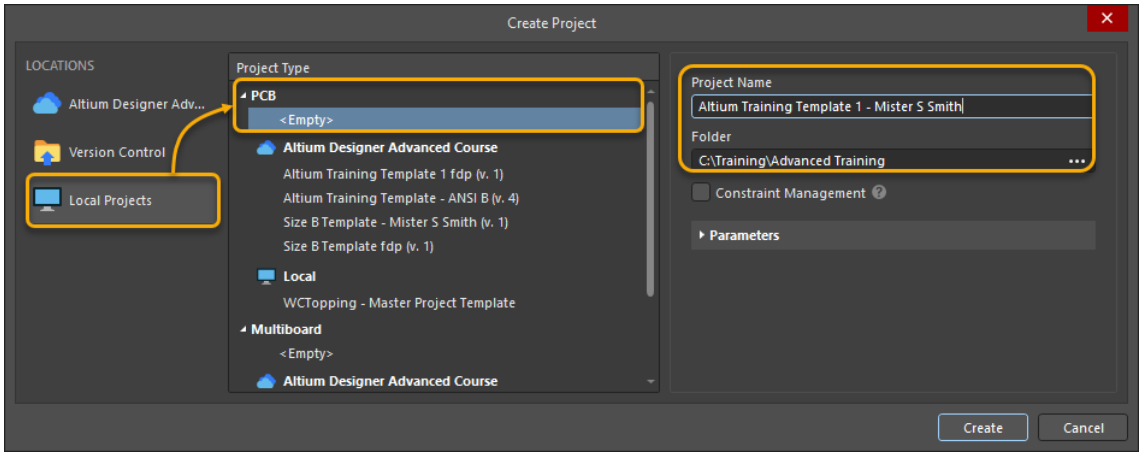


Figure 4. Create Project dialog

13. You may notice the **Constraint Management** option, that won't be used for this project. Use Figure 4 above as a reference for the following steps:
    a) Select the Local Projects from the *Locations* area.
    b) Select **<Empty>** in the ***Project Type*** pane.
    c) Enter the project name: `Altium Training Template 1 - [Your Name].`
    d) Select the location for your project: the Desktop or any other folder you prefer.
    e) Select the **Create** button to create the new project. It may take a few seconds for Altium Designer to create the new project.
14. In the *Projects* panel, you will see the project you just created.
15. Go to **File » New » PCB** to add a PCB to the Project.
16. Go to **File » New » Schematic** to add a Schematic to the project

> Note: Depending on your configuration, you can see various types of schematic documents. This project creates a generic schematic with a standard title block.

17. Save the project and the documents with **File » Save All**. Save the Schematic and PCB with the default names `Sheet1` and `PCB1`.

4.1.2 Uploading the Project to the Workspace as a New Template

With a minimal configuration for a Project Template, you can upload this project as a template to the workspace.

18. Go to **File » Save Project As Template to Server... .**
    a) A *Confirm* dialog with information about the name and location for the template pops up, Figure 5.
    b) If the information you see is correct, select **OK** to confirm the creation of the Project template.

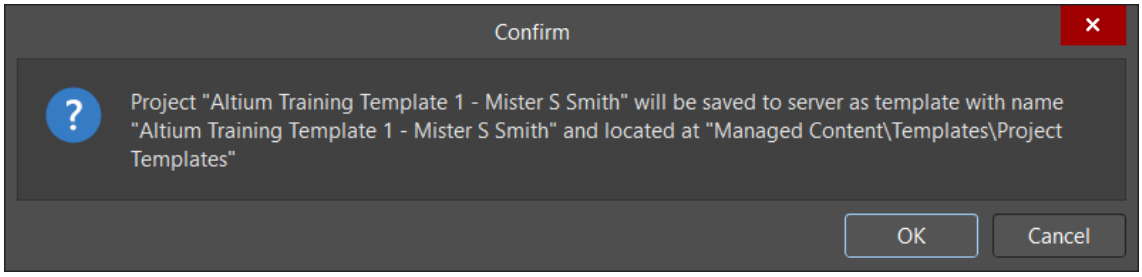


Figure 5. Confirm to create the Project template

19. After the project was uploaded to the workspace, you can see a *Save Project as template* dialog, Figure 6. Select **OK** to continue.

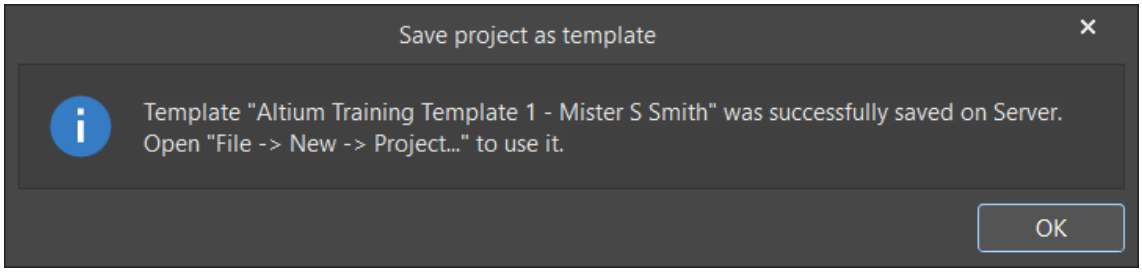


Figure 6. Save Project as template dialog

20. Close the Project `Altium Training Template 1 - [Your name]`.
21. The Project `Altium Training Template 1 - [Your Name]` saved locally on your PC is no longer needed.

4.2 Creating New Project Template from Existing Project

To speed up the Project template creation, you will use an existing project. Let's modify and update the project before you can upload it as a template to the workspace.

22. Change the focus back to the Windows File Explorer with the project you downloaded from the workspace as ZIP. To recap, see section **3 Preparation**.
23. Rename the project file from `Size B Template.PrjPCB` to `Size B Template – [Your Name].PrjPCB`.
24. Open the Project `Size B Template – [Your Name].PrjPCB` in Altium Designer.
25. Open the Project Options **C»O.**
26. In the *Project Options* dialog, select the *Parameters* tab at the top-right of the dialog, Figure 7.

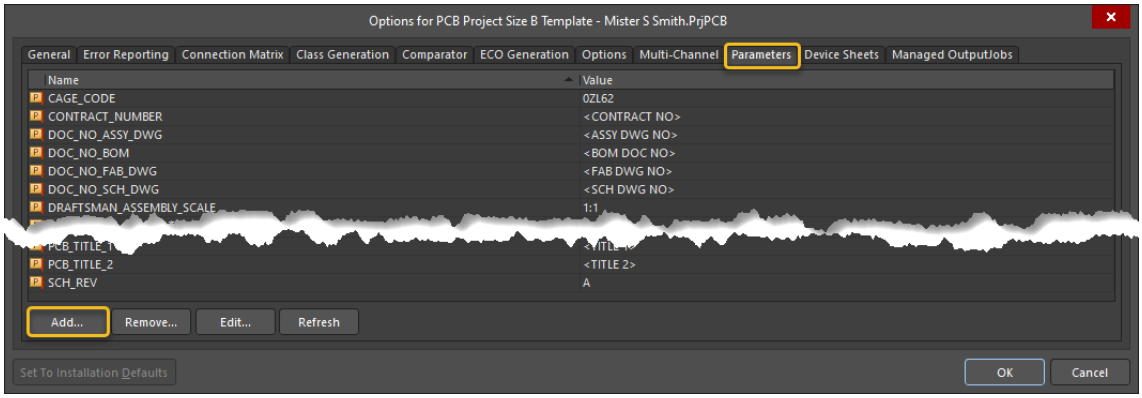


Figure 7. Project Options - Parameters

27. Note the various preconfigured Project level parameters.
28. Select the **Add** button to add a new parameter.
29. In the *Parameter Properties* dialog set the *Name* field to `PCB_REV` and *Value* field to `A`, as in Figure 8.

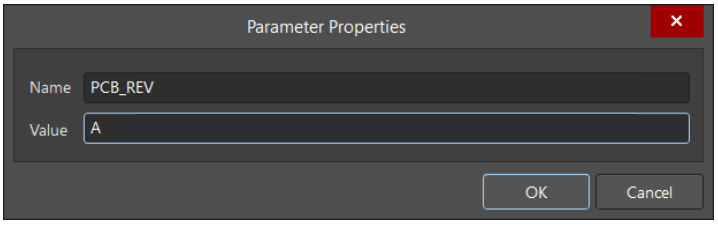


Figure 8. Adding a new Project Parameter

30. Select **OK** to close the dialog.
31. Repeat the Previous Step to add a Second Parameter
    a) Name: `Altium Training`
    b) Value: `Altium Advanced Training with Altium 365`
32. Select **OK** to close the dialog.
33. Select **OK** again to close and accept the changes for the *Options for Project* dialog.
34. Select **File » Save All**.

4.3 Checking Schematic Templates

35. Double click on the SchDoc files to open the two schematics from the project:
    a) `[ProjectName]_TopSheet_Template_B`
    b) `NextSheet_Template_B`

> Note: The [ProjectName] from the file Name `[ProjectName]_TopSheet_Template_B.SchDoc` is a placeholder. Later, when this Project is used as a Template, [ProjectName] wil be replaced with the Project name you choose for your new project.

36. Inspect each of the document title block.
    a) The `[ProjectName]_TopSheet_Template_B` has a full title block.
    b) The `NextSheet_Template_B` has a subset of the title block commonly referred to as a continuation block.

The title blocks are populated with text referred to as special strings (also known as parameters). These special strings enable title blocks to reference Project or Document parameter names and display their associated values. If a value is assigned to the parameter, it will be displayed. If no parameter value is provided, the special string name will be displayed instead, for example, =DOC_NO_BOM. The equal sign designates it's a special string.

> Hint: 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*.

37. The two schematics are linked to Sheet templates. Let's check the link for the top sheet.
38. Make the `[ProjectName]_TopSheet_Template_B` the active document.
39. Open the *Properties* panel and select *General* tab.
40. Scroll down to see the Page *Options – Formatting and Size,* Figure 9.
    a) With the Option *Template* active, you see the Sheet Template used for this document.
    b) You can select a different template from the drop-down menu.

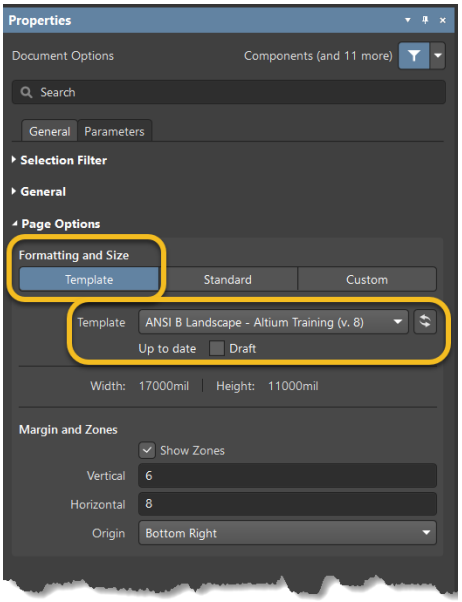


Figure 9. Properties Panel with Template information

41. Add a new Schematic to the Project with **File » New » Schematic**.
42. Save the new schematic as `NextSheet2_Template_B.SchDoc`.
43. Use Figure 10 as a reference for the following steps:
    a) With the third schematic, `NextSheet2_Template_B.SchDoc`, as the focused document, add the `Next Sheet` template to the sheet by using **Design » Sheet Templates » Server » Next Sheet – Size B**.
    b) In the *Update Template* dialog, focus on *Choose Parameter Actions* section and select the **Replace all matching parameters** option.
    c) Select **OK** to attach the template.
    d) Select **OK** in the *Information* pop up box.

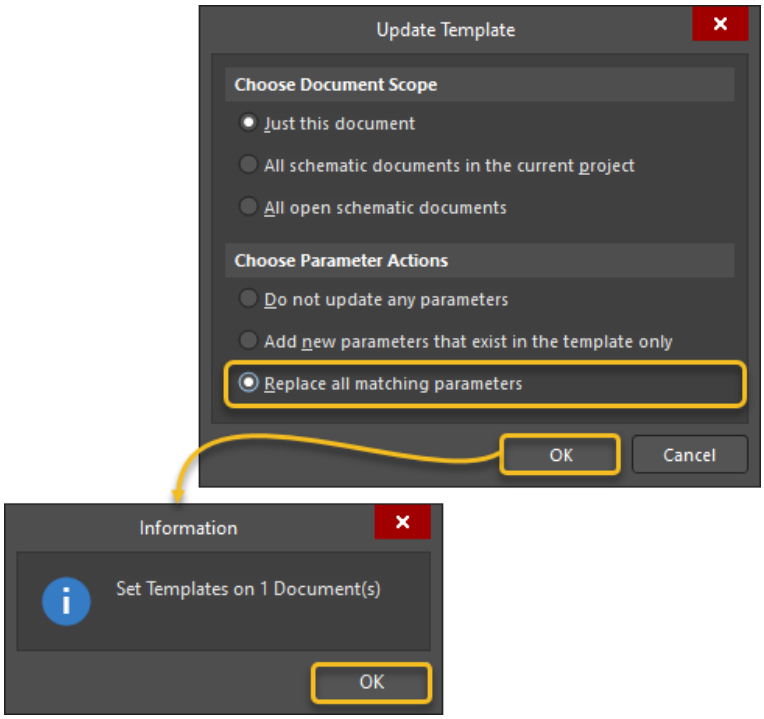


Figure 10. Updating Template settings

44. Try selecting the parameters in the title block for either of the project schematics. Note that the title block in each of these files cannot be selected. The values will be automatically filled in using the Document and Project parameters. To modify the Title Block, Parameter, or Logo, you need to open and update the template.
45. Feel free to change the order of the documents in the *Projects* panel using drag and drop.
46. Save the Project and all files using **File » Save All**.
47. Close all Schematics.

4.4 PCB Template

To streamline template creation, we included a PCB in the example that already has basic configurations, for example, a Predefined Board Shape, a Title Block (drawn by hand or imported from DXF), Place documentation tables, and so on.

48. Open the `[ProjectName]_B Template` PCB document.
49. Open the *View Configurations* to view the mechanical layer allocations by pressing **L**, or with **View » Panels » View Configuration**.
50. Scroll down to the Mechanical Layers (M) section, see Figure 11.

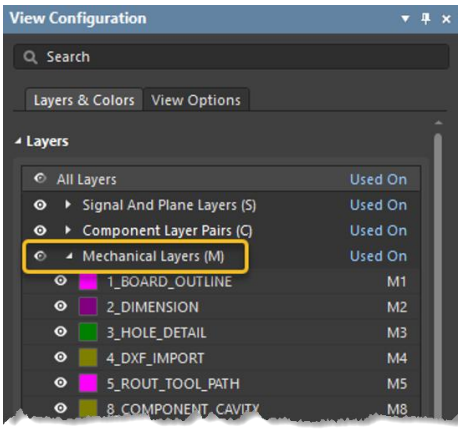


Figure 11. Mechanical layers

51. The layers visible in the PCB should align with the layers from your footprints and any additional PCBs you include in other project templates:
    a) Scroll to the *Component Layer Pairs* section.
    b) If not already expanded, select the arrow to expand the section.
    c) Paired mechanical layers won't appear in the Mechanical Layers section but will be shown in the Component Layer Pairs section. Mechanical layers from 9 to 16 have been paired, as shown in Figure 12.

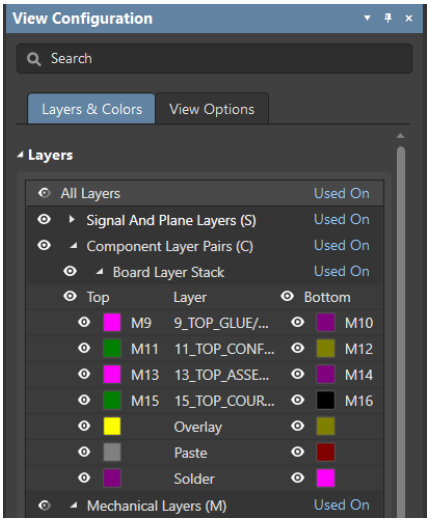


Figure 12. Component Layer Pairs. Paired Mechanical Layers showing Top (left column) and Bottom (right column) associated pairs

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 work similarly to top and bottom solder mask layers, overlay layers, and paste mask layers.

Next, you will export the PCB Mechanical Layer setup to upload that file to the workspace.

52. To export the PCB Mechanical Layer setup, select menu **Tools » Export Mechanical Layers...**
53. When prompted, name the stackup file `Advanced Training Mechanical Stackup – [Your name].stackup`, and save it to the `..\Project` folder.
54. Open the *Explorer* panel from the **Panels** button, or with the combination **K»R**.
55. Navigate to the branch *Managed Content - Templates – Layer Stacks - Mechanical Layer Stackup*. Use Figure 13 as reference.
    a) Right click in the area of the Item list and execute **Create Item » Binary File**.

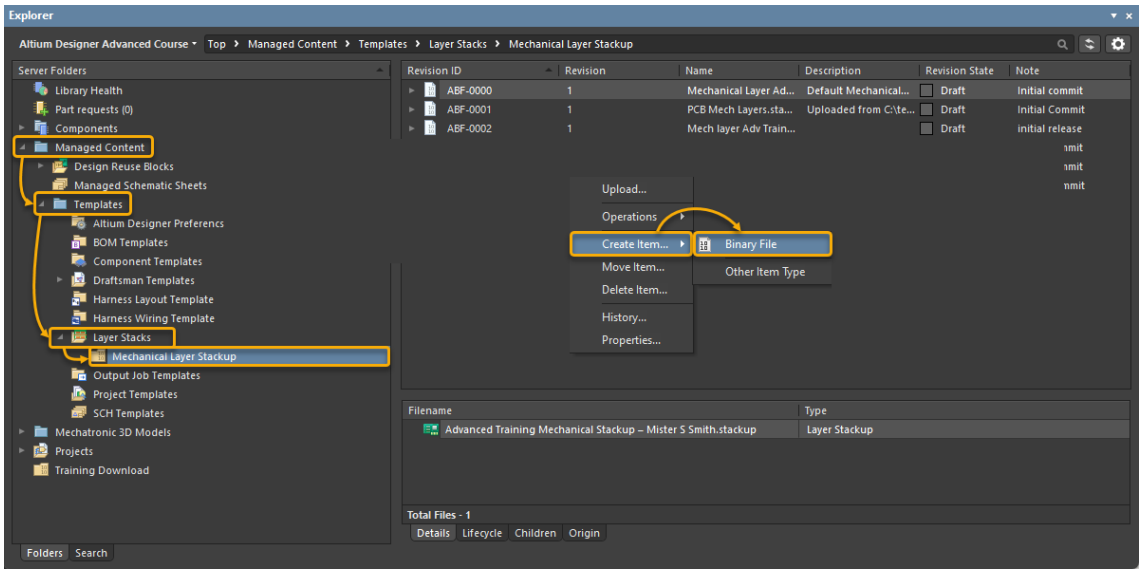


Figure 13. Create new Item for a Mechanical Layer stack

56. In the dialog *Create New Item* (Figure 14):
    a) Add the Name: `Mechanical Layer Advanced Training - [Your Name].`
    b) Add a Description: `Default Mechanical Layer stack for all new PCBs and Footprint - Advanced Training - [Your Name].`
    c) Select **OK** to create the Item. A new empty container is available in the workspace. Next, you will upload the Mechanical Layer Set into that container.

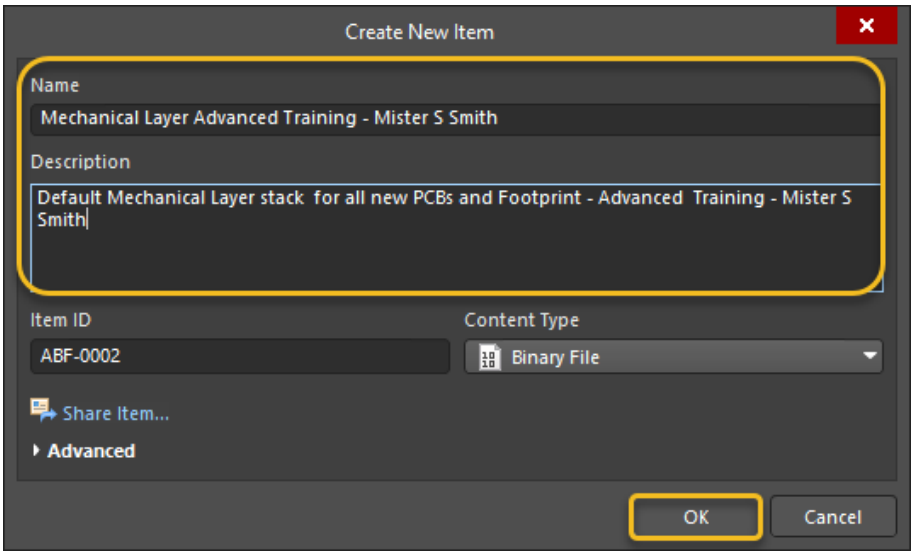


Figure 14. New mechanical Layer for the Workspace

57. Select the item you created: `Mechanical Layer Advanced Training - [Your Name].`
58. Start the *Create new Revision* dialog with **Right click » Upload**.
    a) In the new dialog add a release note: `Initial commit`.
    b) Using the **Add** button or with **drag and drop**, add the `Advanced Training Mechanical Stackup - [Your name ].stackup` from your Windows Explorer folder.
    c) Select **OK** to upload the file to the workspace. The layer set is now saved and can be accessed and downloaded by any workspace user with the appropriate permissions.
    d) Switch back to the PCB.

Hint: To add mechanical Layer set to a new (blank) PCB without preconfigured Mechanical Layers, open **Tools » Import Mechanical Layers...** and load the [Mechanical Layes].stackup file.

59. Open the Layer Stack Manager with **D»K** or use **Design » Layer Stack Manager...** from the main menu.
60. The PCB for this example is an 8 Layer PCB. The training workspace doesn't include a default 8-layer stackup (you might be seeing the stackup from your default user, Mister S Smith), so let's save this stackup to the workspace.
61. Select **File » Save to Server...**

62. In the dialog box *Choose planned Item Revision*, navigate to the branch *Managed Content - Templates – Layer Stacks*.
    a) Create the 8 Layer Stack Item through the *Choose Planned Item Revision* dialog (Figure 15):
        i) Right-click in the revision list region to activate the pop up menu. Select **Create Item**, and then select **Layerstack**.

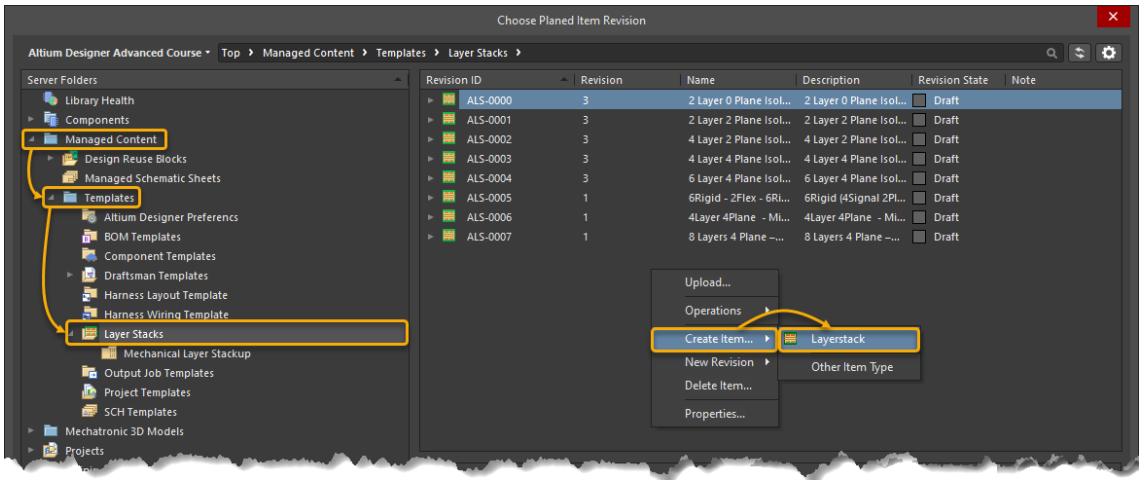


Figure 15. Adding a Layerstack in Choose Planned Item revision dialog

    b) Use Figure 16 as a reference for the next steps. In the *Create New Item* dialog that opens, disable the **Open for editing after creation** option to avoid entering direct editing mode.
    c) In the Create New Item dialog, add the **Name** and **Description**:
        i) Name: *4 Layer 4 Plane – [Your Name]*
        ii) Description: 4 Layer *4 Plane – [Your Name]*

Hint: When you choose an existing Layerstack, you update it with new information and generate a new revision.

    d) Select **OK** to close the *Create New Item dialog*.

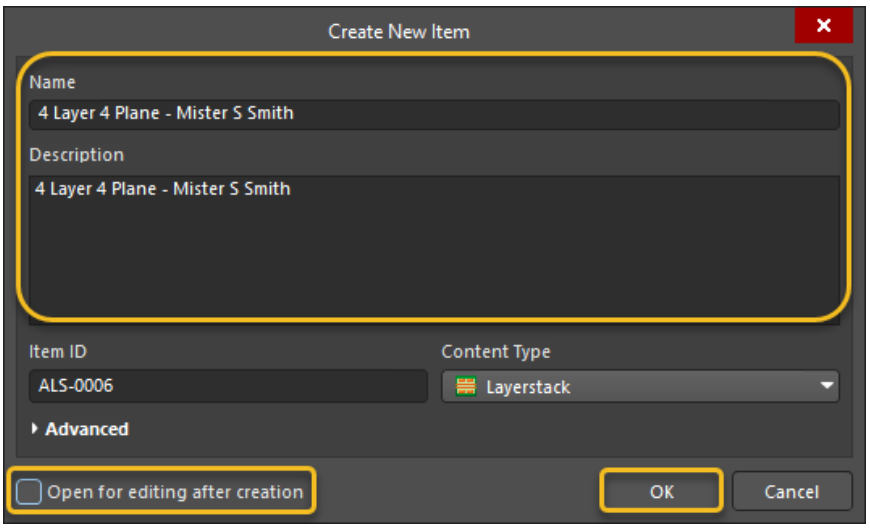


Figure 16. Create New Item dialog

63. With the new item selected, select **OK** to close *the Choose Planned Item Revision dialog*.
64. In the dialog *Edit revision 1 for Item ALS-[Number]* that pops up, add a release note `Initial release`, Figure 17.

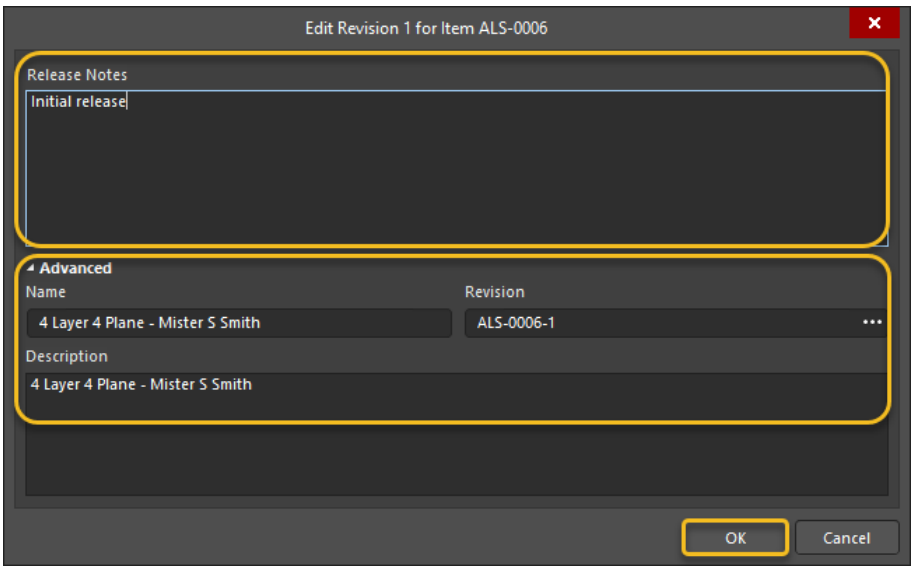


Figure 17. Edit revision 1 for Item dialog

65. Feel free to open the **Advanced** section and check the Name and Description.
66. Wait until the tool uploaded the Stackup to the workspace.

> Hint: You can open the Explorer panel, navigate to the branch `Managed Content - Template - Layer Stacks` to see the new Stackup uploaded to the workspace.
> To load an existing Layerstack onto a new, blank PCB without preconfigured layers, use the command **File » Load Stackup From Server...** .

67. Close the Layer Stack without saving modifications you may have done by accident.

68. Back in the PCB, zoom into the lower left corner of the title block and double-click on the text, *Printout_Name is not interpreted until output*.
    a) 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, Figure 18. Note the special string *.Printout_Name* is a special string which exists in the PCB document by default.
    b) Close the *Properties* panel.

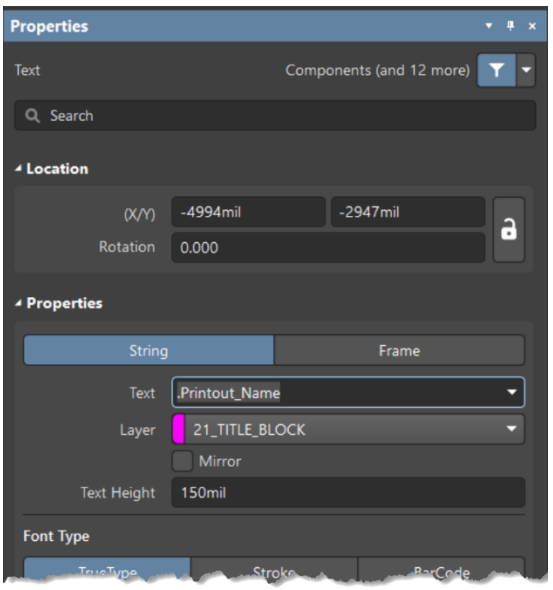


Figure 18. Printout_Name special string

> Note: The special string *.Printout_Name* is filled in when documents are created using the associated layer where the string is located. It's populated with the sheet name defined in the Output Job file for fabrication and assembly drawings. The use of the special string *.Printout_Name* will be explored in the following section.

69. Zoom in on the title block located at the lower right of the PCB, Figure 19. Take note of the Project-level parameters found in the Reference Documents section. These parameters allow document names to be set once in the Project Options Parameters tab, automatically updating the title block for reference and naming the files during output generation.

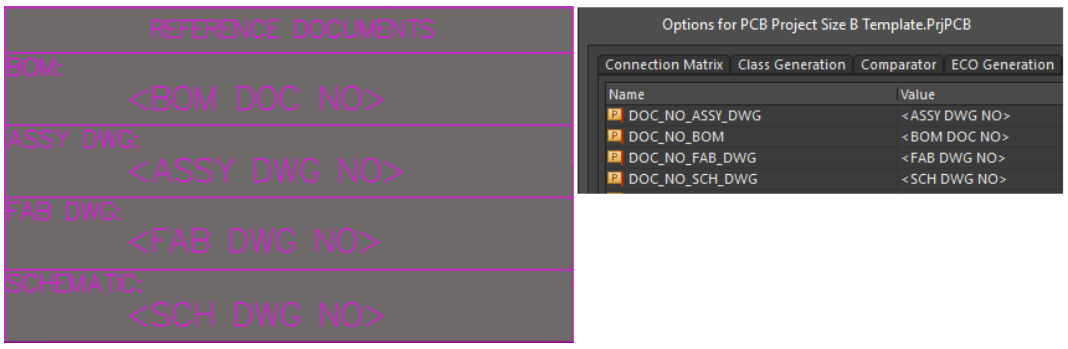


Figure 19. Reference Documents section of title block

70. Save the PCB document.

4.5 Adding Draftsman Templates

Just like in Module `Template 1 - Draftsman Template`, in the section `Using your Custom Template in a Design`, you will now include Draftsman documents in the project template.

Add a new Draftsman document to the Project Template, **File » New » Draftsman Document**. Refer to Figure 20, which may display different templates than those available in the workspace you are connected to.

a) In the *New Document* dialog, choose the `Fabrication Drawing - [Your name]` template. This is the drawing you created in the Module `Template 1 - Draftsman Template`. If you have skipped Module `Template 1 - Draftsman Template`, choose the default template `Default Fabrication Drawing`.

b) Select **OK** to close the dialog.

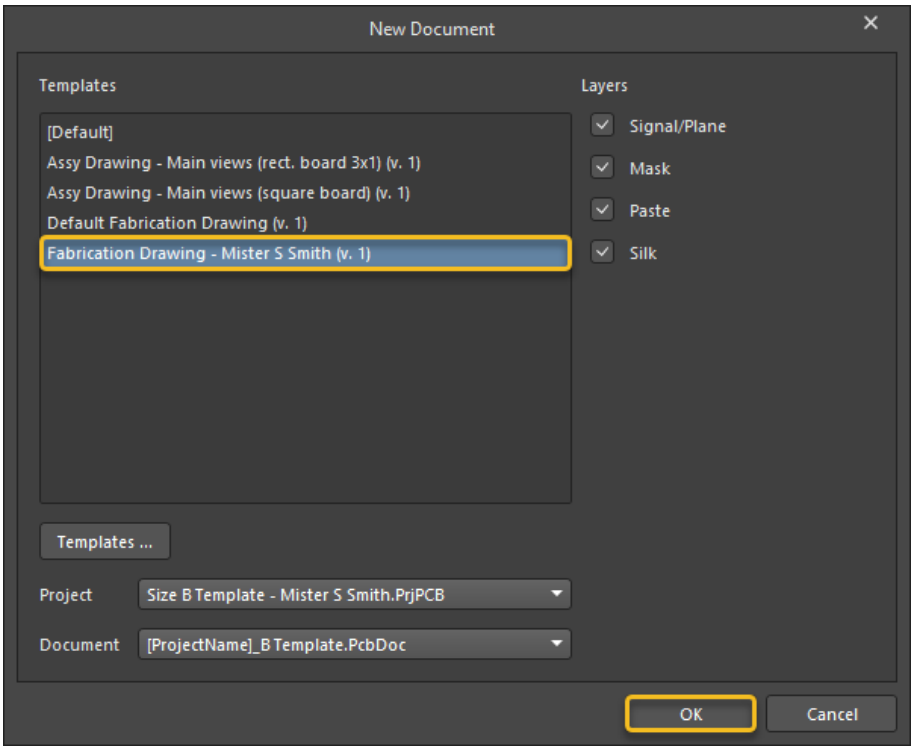


Figure 20. Choose the Draftsman template

71. Wait for Altium Designer to retrieve all PCB data and update the Draftsman document with the necessary information for creating and updating the documentation elements.
72. Open the `[Project Name] B_Template PCBDfw` Draftsman document, if not already opened.
73. As you did in the Module `Template 1 - Draftsman Template`, continue with the following steps:
    a) Execute **Tools » Import Changes From [Board name]** to update all documentation elements with the current PCB information.
    b) Optimize the position for the documentation elements.
74. Select **File » Save** to save the new Draftsman document (PCBDwf). Save the file with the name `[ProjectName]_My Fabrication Drawing`.
75. Save the Project, **File » Save All**.

4.6 OutJob Templates

76. Open the *Explorer* panel from the **Panels** button, or with the combination **K»R**.
77. Navigate to the branch *Managed Content - Template – Output Job Templates*. You can view several predefined Output Job templates for different fabrication options, as well as assembly and validation.
78. You can copy the existing templates or create a new Output Job Template from Scratch. Let's copy the Gerber Output Job template to demonstrate the basic setup of an Output Job File. Use Figure 21 as reference.
    a) Select the `Fabrication Gerber` template.
    b) **Right click » Operations » Make a copy...** to copy the template.
    c) The copy is created and opened for editing in the Altium Designer workspace area.
    d) Close the *Explorer* panel.

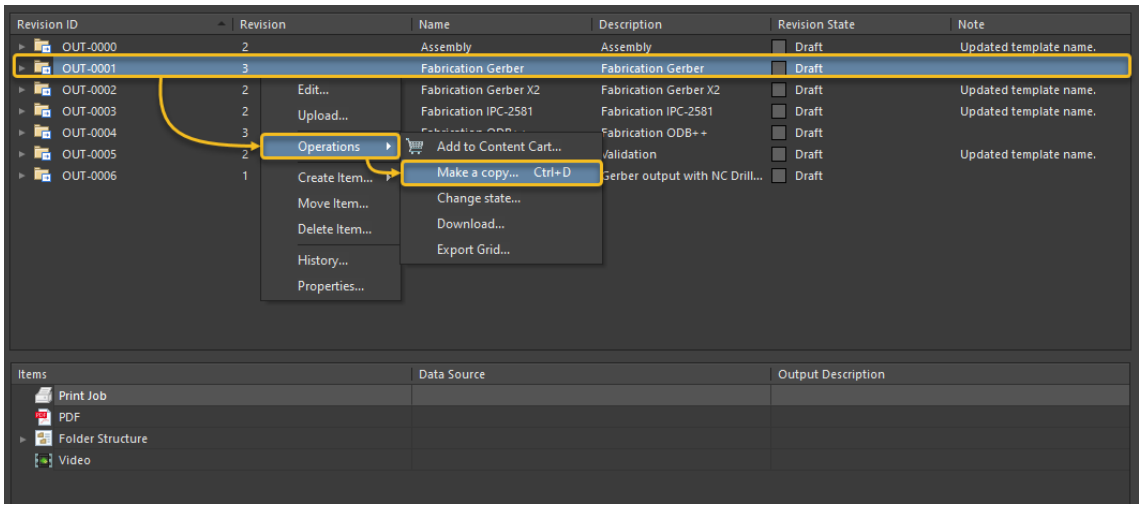


Figure 21. Copy an Output Job Template

79. On the right side of the OutJob editor, you'll find, among other elements, two Output Containers: a PDF (which is currently not in use) and a file generator named `Folder Structure` for Gerber and NC Drill, Figure 22.

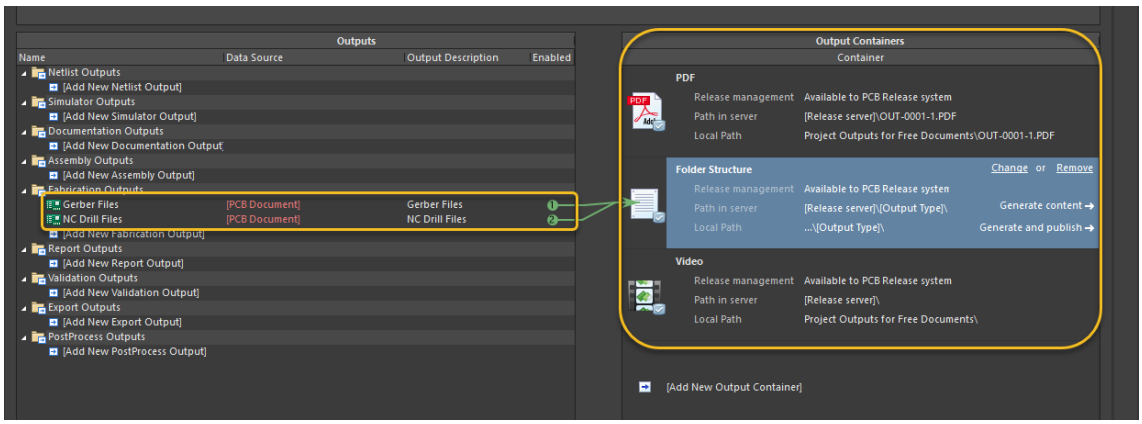


Figure 22. Output Containers

80. Select `Folder Structure`, press F2 functional key (or **fn+F2**) on your keyboard to change its name to `Gerber and NC Drill`. Alternatively, left-click the `Folder Structure` title twice with a short pause in between to enter edit mode for the title.

81. On the left side of the OutJob editor, within the Fabrication Outputs section, you'll find, among other elements, two outputs: one configuration for Gerber and one for NC Drill.
82. Between the two Outputs and the Output Containers for Gerber and NC Drill, you'll notice two green connections indicating that the Gerber and NC Drill files are being saved to the Hard Drive.
83. Select the Output (left side) Gerber Files. Double click or **ALT+Enter** to open the *Gerber Setup* dialog.
    a) You can adjust settings such as Units, Decimal, file name extensions, and more.
    b) At the moment, the Template lacks a PCB source, which is why only the Board Profile Layer is visible and no additional layers are shown.
    c) Close the Gerber Setup using **Cancel**, without saving any modifications.

Hint: Just like with the Gerber configuration, you can open the settings for any Output available in an Output Job File. The specific configuration options depend on the selected Output.

84. Next, you'll add a new Output.
    a) In the Section *Documentation Outputs*, click on *[Add New Documentation Output]* and select **Draftsman » All Draftsman Documents**, Figure 23.

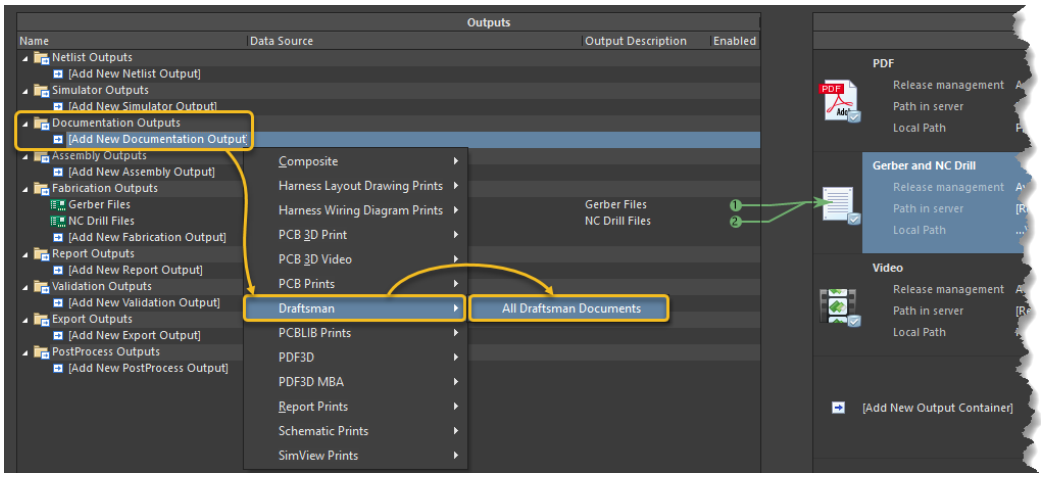


Figure 23. Adding new documentation output

    b) A new Output is added to you Output Job File.
    c) Since Draftsman document settings are handled within Draftsman, there's no separate configuration for it in the Output Job file.
85. Next, you'll create a connection between the Output Draftsman and the Output Container PDF. Follow the steps below:
    a) Select the Output Container PDF.
    b) Click on the title `PDF` and press F2 (or **fn+F2**) to rename it to `Draftsman`.
    c) Select **Change** or use **right click » Properties** within the container to open the configuration for the PDF Output Container.
    d) In the main configuration, you can adjust the save location on your hard disk. By default, PDFs are saved in the same location as the Altium Designer Project that uses the Output Job file.
    e) With the Advanced option, you can configure some PDF specific options.

f) An example configuration with modified path setting is shown in Figure 24.

g) Close the Configuration.

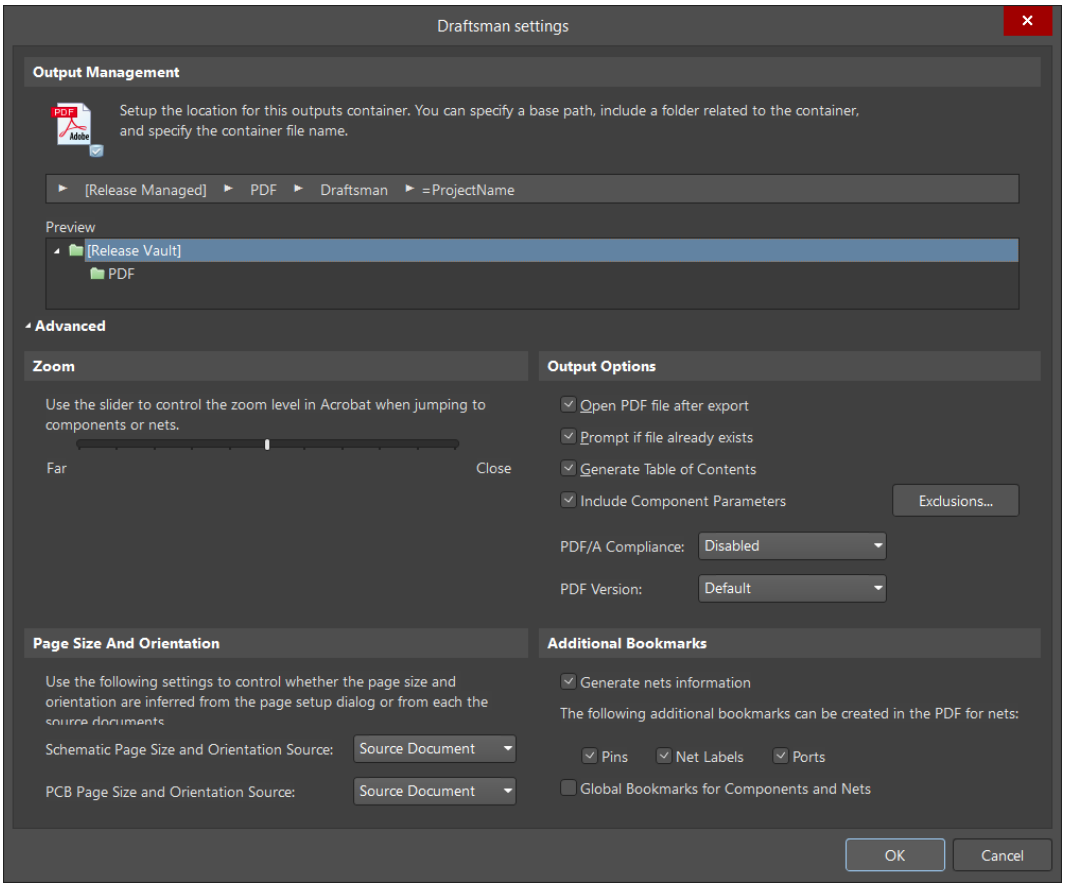
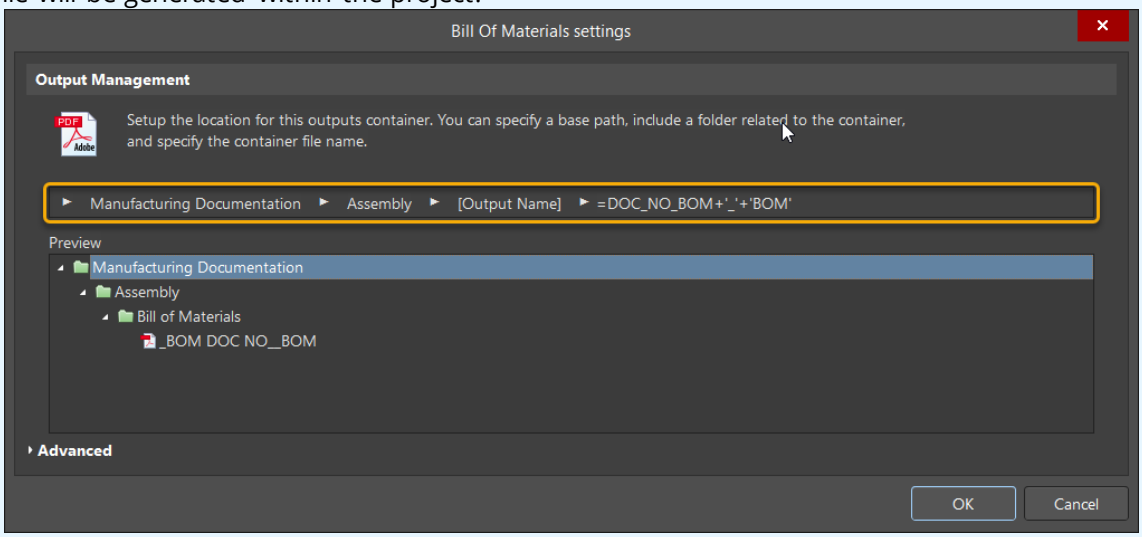
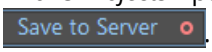


Figure 24. Out put Job Configuration for a Draftsman Output

Hint: The image below shows a configuration for a Bill of Materials document. The document is 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.
In the *Preview* area, you can see the file name, directory structure, and location of where the file will be generated within the project.



You can change document names or other parameters in the **Project Options**, **Parameters** tab. The names will carry through the entire project and can also be used to name output files.

86. Save the Output Job File.
87. Save the Template to the server.
    a) In the *Projects* panel, next to the Project name, find the command **Save to Server** 

    b) Select **Save to Server**.
    c) In the dialog *Create New Item* add the name `Gerber – [Your name]`.
    d) Add the description `Gerber output with NC Drill and Draftsman`.
    e) Open the Advanced section and check that the default path is correct `…\Templates\Output Job Templates`.
    f) Select **OK** to create the document. The document is automatically removed from your project window.
88. Open *Explorer* panel **K»R** and check the branch *Managed Content - Templates – Output Job Templates*. You'll find your own template, and templates from the other attendees.

4.7 Adding Out Job Files

89. Next, you'll link Output Job Files from the workspace to the Project template. The Output Job files include the default Output Job Files and the Gerber-[Your Name] file that was uploaded to the workspace.
90. Open the Project Options **C»O.**
91. In the *Project Options* dialog, select the *Managed OutputJobs* tab at the top-right of the dialog.
92. Selecting **Add** to add the first Output Job File, Figure 25.
    a) In the dialog *Select Configuration Item* choose `Gerber-[Your Name]` as first file.
    b) Select **OK** to close the dialog.
93. Repeat the previous step to add the configuration `Fabrication ODB++,Assembly` and `Validation`.

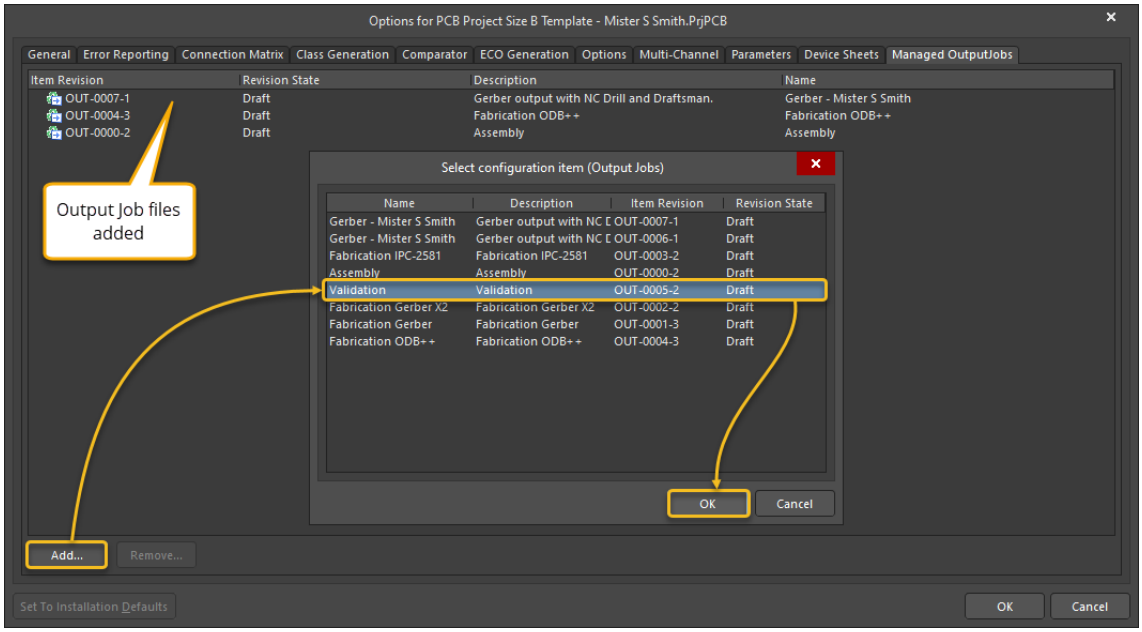


Figure 25. Adding Output Job Files

94. Close the *Project Options* with **OK** and save the Project.

5 Uploading Project to Workspace as New Template

Now that a Project Template is configured, you can upload the project as a template to the workspace.

95. Check that you have a unique Project Template names. If needed, rename the Project to Size B Template – [Your Name].
96. Go to **File » Save Project As Template to Server... .**
97. A *Confirm* dialog pops up. Check the information and select **OK** to confirm the creation for the Project template.
98. After the project is uploaded to the workspace, you can see a *Save Project as template* dialog,
    a) Select **OK** to continue.
    b) Close the Project and all open documents, **Window » Close all**.

6 Creating New Project from Template

6.1 Creating New Project

99. Go to **File » New » Project...** to launch the *Create Project* dialog, Figure 26.

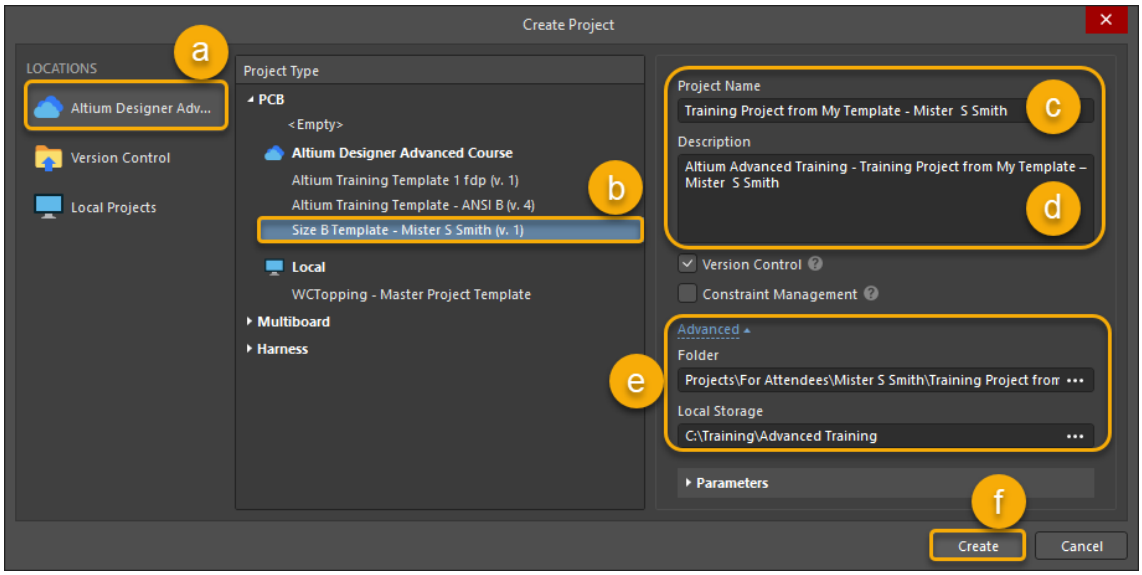


Figure 26. Creating a New Project

100. Use Figure 26 above as a reference for the following steps. You might see the **Constraint Management** option that won't be used for this project.

a) Select the workspace `[XYZ] - Altium Advanced Course A365 [ab]` in the *Locations* area.

b) Select **Size B Template – [Your Name]** in the *Project Type* pane to create the new project with the preconfigured settings.

c) Enter the project name: `Training Project from My Template – [Your name].`

d) Add the following Description: `Altium Advanced Training - Training Project from My Template – [Your name].`

e) Open the **Advanced Settings**. This is the configuration for where your project will be saved:

i) *Folder*: Select the ellipsis ••• and select the *Folder* `Projects » For Attendees » [Your Name]` that you created at the beginning of this training.

ii) *Local Storage*: Check if the predefined *Local Storage* Path is okay for you. If not, feel free to change the path.

f) Select the **Create** button to create the new project. It may take a few seconds for Altium Designer to create the new project.

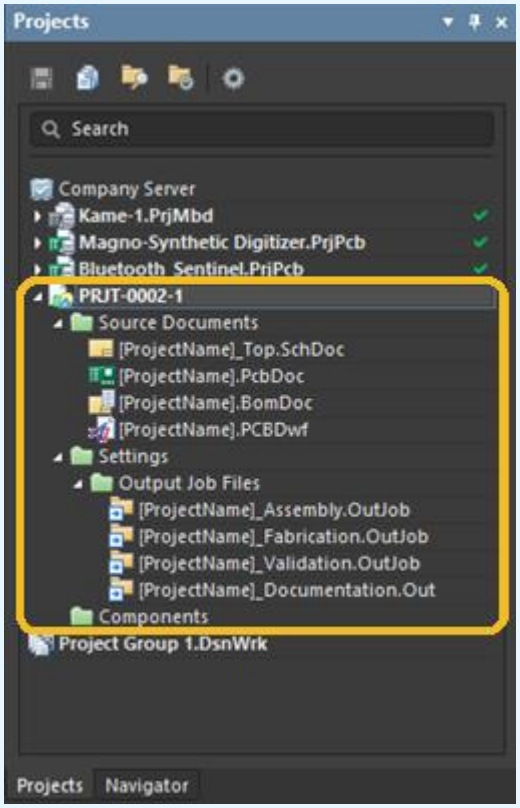
101. Return to Altium Designer and look at the Projects panel. You'll see a project named `Training Project from My Template - [Your name]`, which includes three schematic documents, a PCB document, Draftsman documents, and Output Job files.

102. Next, you can begin with the schematic entry and PCB design.

Hint: Automated Document Naming
Automated naming of documents when applying a Project Template is a useful feature in Altium Designer. This is done using a special [ProjectName] syntax in the template names, which gets replaced with the actual project name for new projects.

For instance, if a schematic document in a Project Template is named [ProjectName].SchDoc and you use this template for a new project called Flux_Triangulator, the resulting schematic file will be named Flux_Triangulator.SchDoc. The naming string can be combined with other characters, so a template like [ProjectName]_Top-Level-Structure.SchDoc will become Flux_Triangulator_Top-Level-Structure.SchDoc in the new Flux_Triangulator project.

To use preprocessing names in the Project Template, edit the document names with the [ProjectName] string before saving and releasing the project template. If updating an existing template, open its revision, modify the document names, and then release the updated version back to the server.



An example project set up as a template uses the [ProjectName] syntax for automatic document naming when the template is reused.

103. Close the project and any open documents.

Congratulations on completing the Module!

Template 2 - Project Templates

from

Altium Designer Advanced Training with Altium 365

Thank you for choosing **Altium Designer**