

Altium Designer

Advanced Course

Module: Draftsman 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

Draftsman Templates	3
1.1 Purpose	3
1.2 Shortcuts	3
1.3 Preparation	3
1.4 Draftsman Preferences	3
1.4.1 Draftsman Defaults	3
1.4.2 Draftsman Templates Location.....	4
1.4.3 Using a Custom Template in a design	8

Draftsman Templates

1.1 Purpose


Using templates provides a standardized means for creating documentation. Like Schematic and PCB templates, Draftsman templates offer the opportunity to create custom Draftsman documents to facilitate the generation of fabrication and assembly drawings. In this exercise we will modify an existing Draftsman template and use it on a project.

The flow in this Exercise shows the options used for a standalone Altium Designer. Working with a Workspace offers additional options for creating and managing Draftsman templates.

1.2 Shortcuts



Shortcuts when working with Draftsman Templates


F1:	Help
 :	Preferences
P-S:	Board Assembly View
T-A:	New Sheet
CTRL+S:	Save Document

1.3 Preparation

1. Close all existing projects and documents.

1.4 Draftsman Preferences

1.4.1 Draftsman Defaults

2. Click on the gear icon  to open up the *Preferences*.
3. Select the *Draftsman* section and expand it to access the *Defaults*.
4. In the *Default Primitives* section of this page, we have the choice to use either *Mils* or *MM*. We will keep *MMs* as the default.
5. The *Primitives* drop down menu provides a means of navigating to the various groups of primitives available to change as shown in Figure 1.

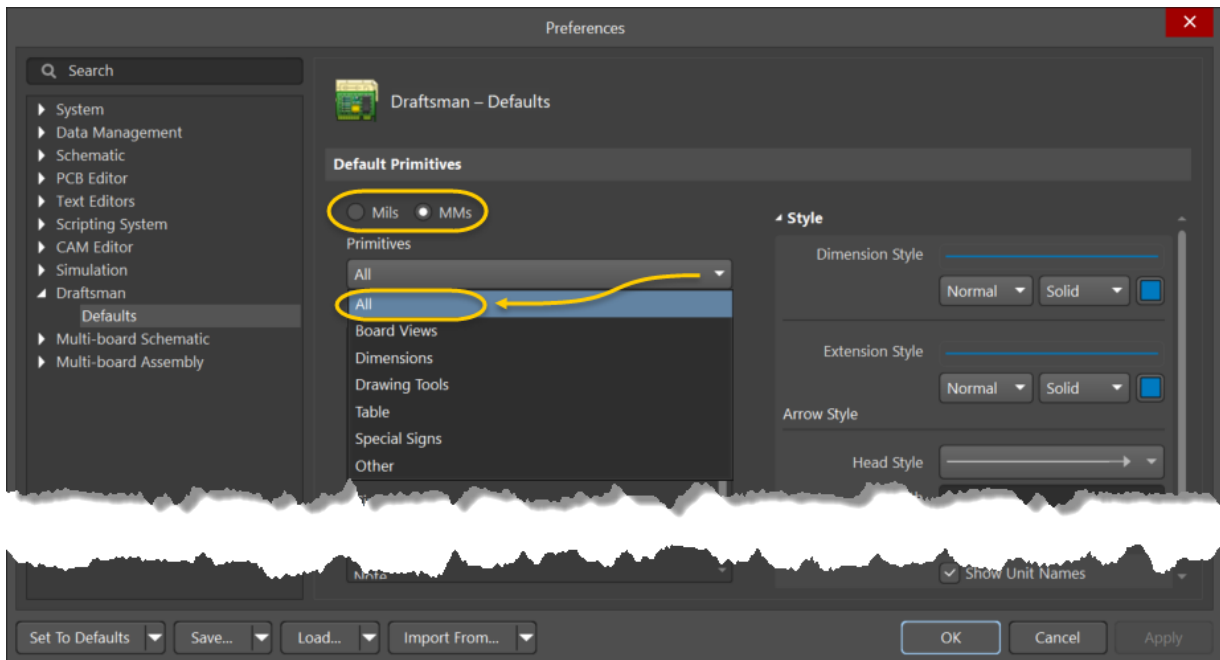


Figure 1. Draftsman Preferences - Default Primitive Drop Down menu

6. Ensure the drop-down menu is set to **All** to show all the possible primitives that can be configured. Although we will not be changing any, clicking on a primitive from the primitive list opens up its properties where the user could modify then as needed.

1.4.2 Draftsman Templates Location

7. From the *Data Management* section, open the *Templates* page as shown in Figure 2. This is the location in which Altium Designer stores all of the default templates included with the installation.

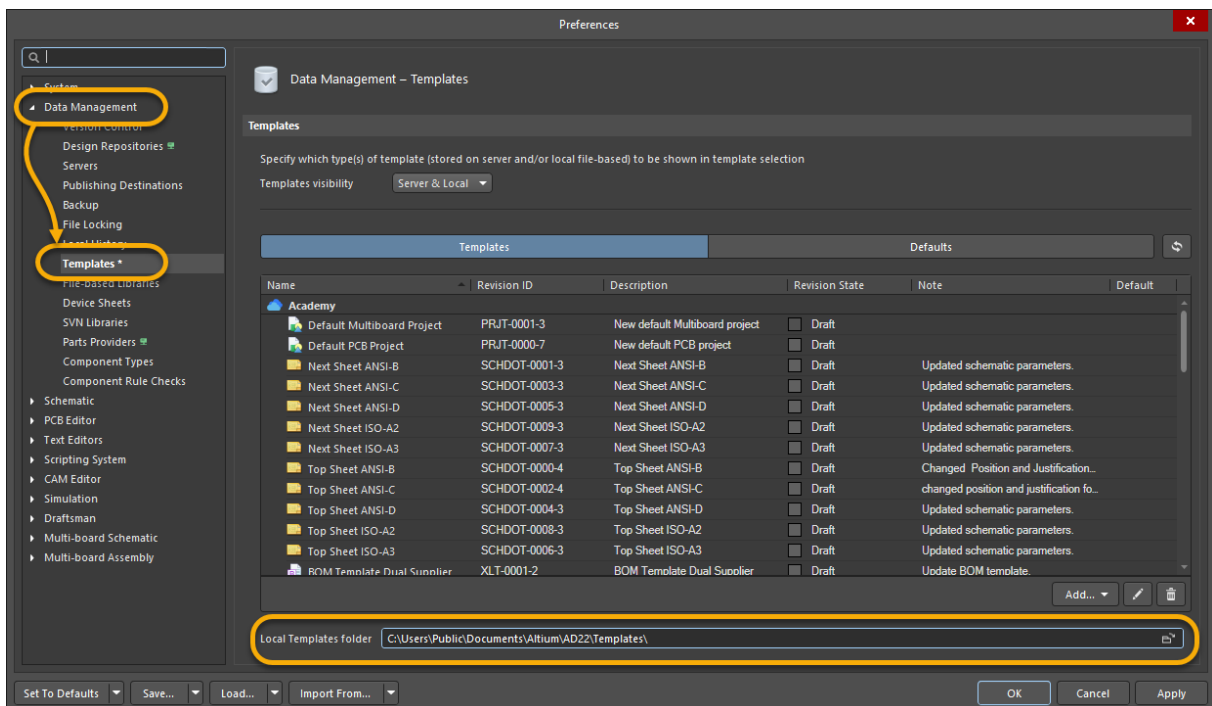


Figure 2. Template file location



The Templates shown here may vary depending on if you're connected to an Altium 365 workspace or not. The Local Templates Folder path may also be pointing to your company's shared folder where your local templates are stored.

8. At the bottom of this window, you'll find the *Local Templates folder* location, as highlighted in Figure 2.
9. Copy this path using **CTRL+C** and click **OK** to close the *Preferences*.
10. Open a new *File Explorer* tab and paste the copied path using **CTRL+V** and hit **Enter**. You should see something similar to what is shown in Figure 3.

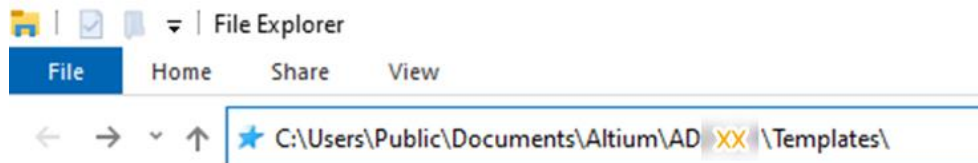
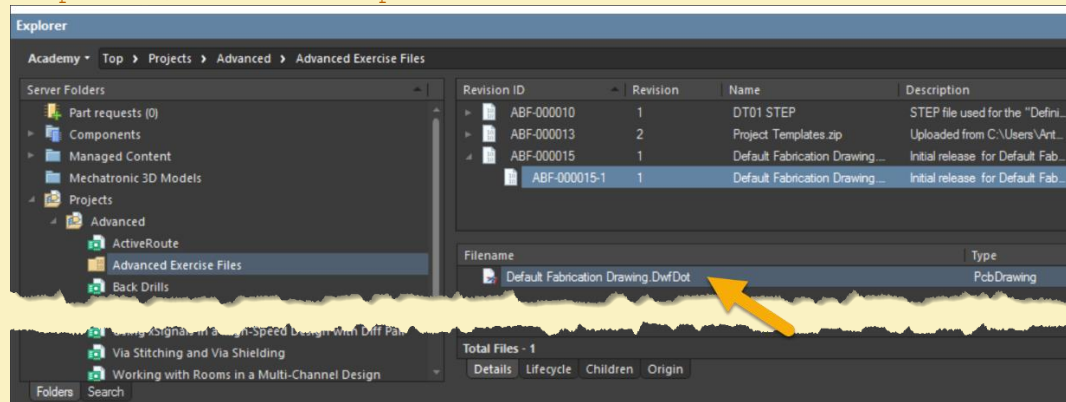


Figure 3. Paste template path into the file explorer

11. Looking at the existing templates, we can see both Draftsman Sheet *.DwsDot and Draftsman Document templates *.DwfDot, along with plenty of other types of templates. We will be copying a Draftsman Document Template for the PCB (with a *.DwfDot extension) and modify it for this exercise. Normally, it is easier to take an existing Draftsman template and modify it to meet your company needs than to create it from scratch.
12. From the list of templates, find the `Default Fabrication Drawing.DwfDot` template. It may be easier to sort the files based on Type in the file explorer.



If you could not find the `Default Fabrication Drawing.DwfDot` on your local template folder you can use the provided `Default Fabrication Drawing.DwfDot` in the `Altium Designer Advanced Exercises.zip` (`..\Draftsman Templates\Draftsman Templates`).



13. Copy the `Default Fabrication Drawing.DwfDot` file and paste it within this same template location.
14. Rename the copy to `Custom Fabrication Drawing` as shown in Figure 4.

Assy Drawing - Main views (rect. board 3x1).DwfDot	10/11/2016 2:20 PM	Altium Draftsman Document Template
Assy Drawing - Main views (square board).DwfDot	10/21/2016 1:21 PM	Altium Draftsman Document Template
Custom Fabrication Drawing.DwfDot	3/23/2018 3:58 PM	Altium Draftsman Document Template
Default Fabrication Drawing.DwfDot	10/21/2016 1:21 PM	Altium Draftsman Document Template

Figure 4. Renamed copy of Default Fabrication Drawing

15. Now open this file by double-clicking on it. Your view should return to Altium Designer.
16. Looking at the first page, we see a *Drill Drawing View*, a *Drill Table* and a *Notes* section. We will be making some slight changes to this template to make it our own.
17. With nothing selected, open the *Properties* panel.
18. Under the *Parameters* tab, we'll change the value for the *CompanyName* parameter to *Altium Inc.* as shown in Figure 5.

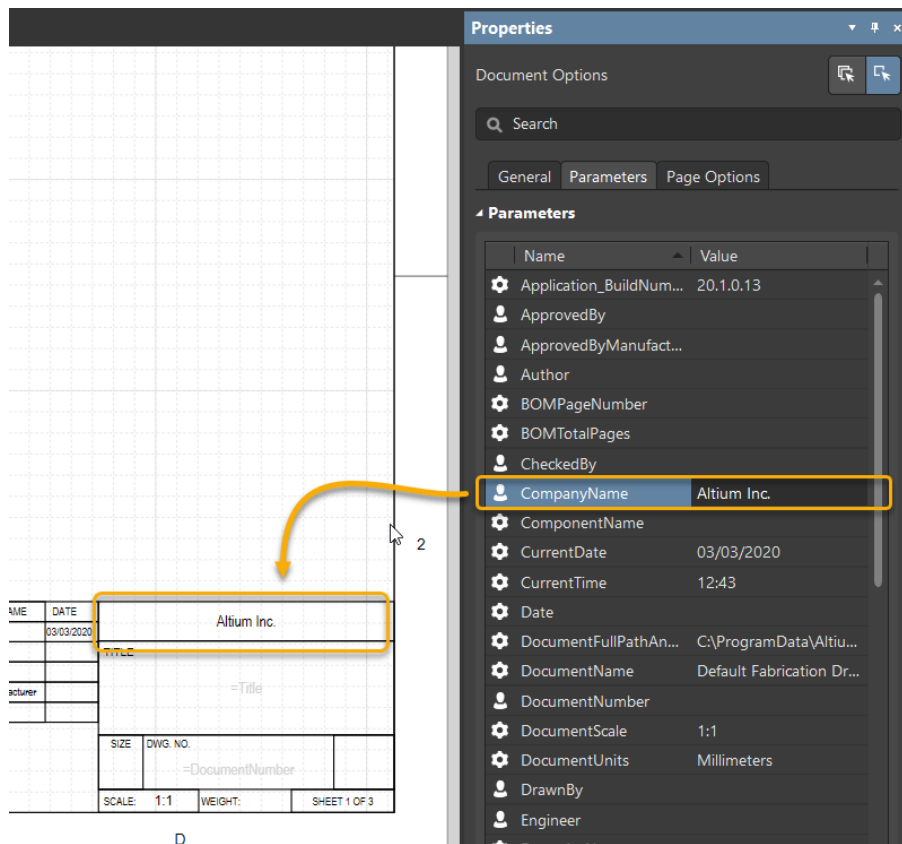


Figure 5. Editing Document Parameters

19. We will make some edits to the *Notes* and the *Drill Table* to provide some additional information.
20. Start by left-clicking on the *Drill Table*.
21. When selected, drag it from its original location, and place it just above the *Notes* section. Once you get close enough, the auto alignment guidelines will appear to help you align the top of the *Drill Table* with the *Drill Drawing* view as shown in Figure 6. This isn't mandatory, but it will make for a clean and professional drawing.

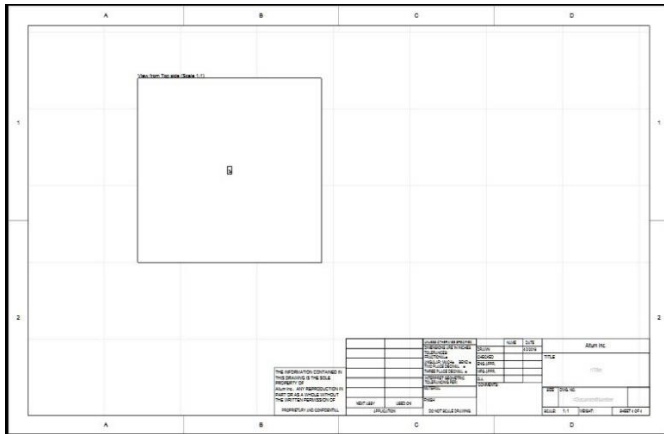


Figure 8. Added Board Assembly View

28. Save the Custom Fabrication Drawing.DwfDot file and close the file.

1.4.3 Using a Custom Template in a design

29. Open the Draftsman Templates.PrjPCB project found in its respective folder of the Advanced Training.
30. From the *Projects* panel, right-click on the project and **Add New to Project » Draftsman Document**.
31. From the *New Document* window, select the Custom Fabrication Drawing template that we just created, as shown in Figure 9. Note, that if you're not connected to an Altium 365 workspace or your company workspace, the templates available to you will be different.
32. Click **OK** and the new Draftsman document will open.

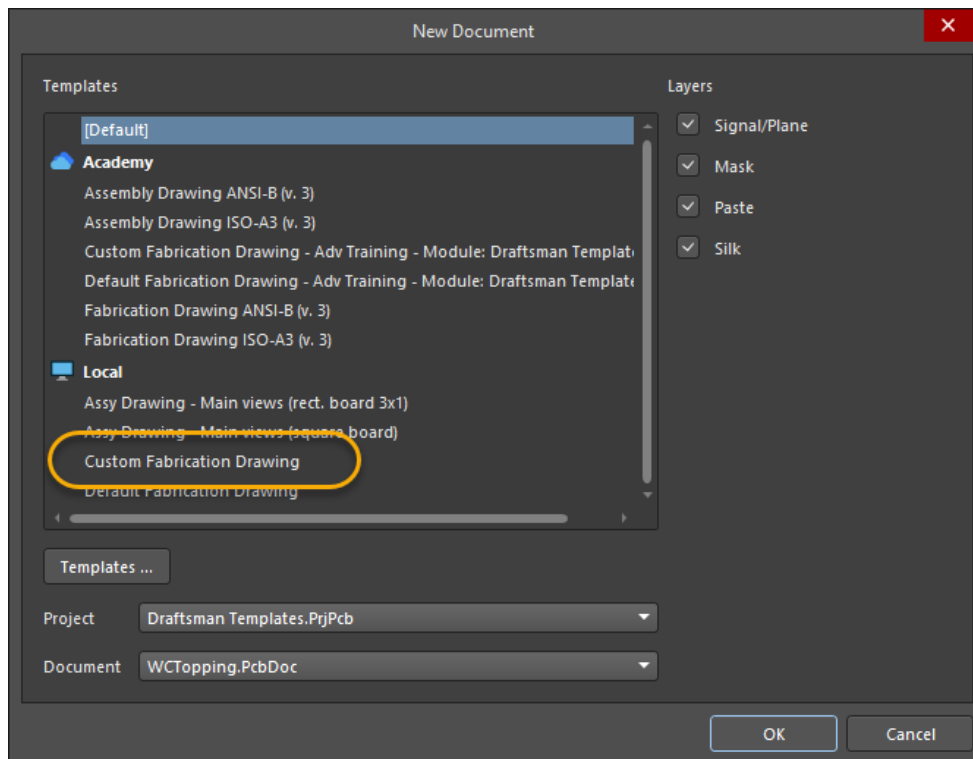


Figure 9. New Draftsman Document using the Custom Fabrication Template



If you do not see your template in this list, you may have saved it in the wrong location. This window will only show templates from the Default Template Folder path, along with templates from the Altium 365 Workspace.

33. When the Draftsman document opens, you'll notice that the information from the template has been populated in our Draftsman document, as it pulls information from the project's PCB document. It can now be reviewed and updated to your liking.
34. Scrolling down to Sheet 4, we see the *Top Side Board Assembly* view.
35. We can also add an isometric view by right-clicking on the sheet, and select **Place » Additional Views » Board Isometric View**.
36. Place the *Board Isometric View* to the right of the *Top Side Board Assembly* view as shown in Figure 10.

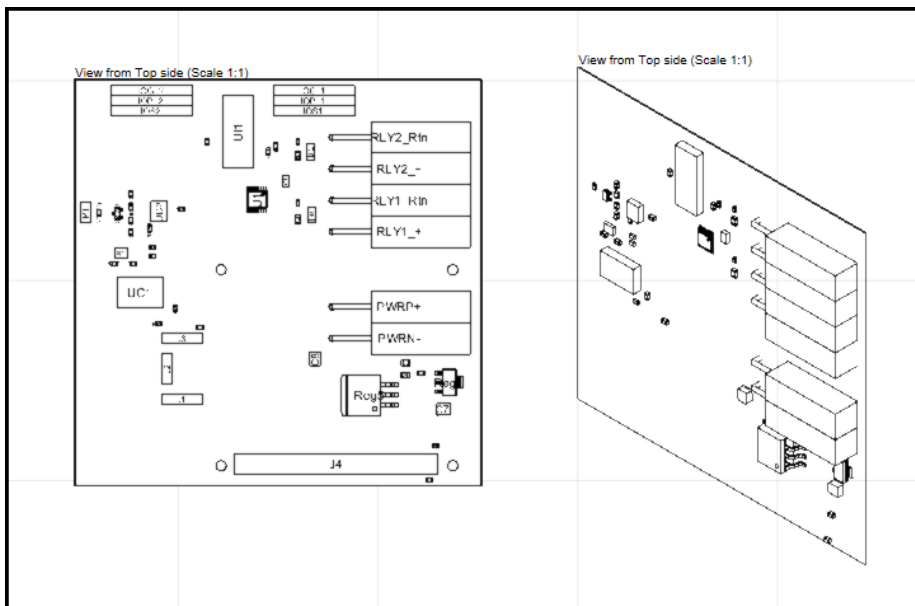


Figure 10. Added Isometric View to sheet 4

37. Feel free to save your modifications.
38. **Close the project and any open documents.**

Congratulations on completing module

Draftsman Templates

from the
Altium Designer Advanced Course

Thank you for choosing Altium Designer