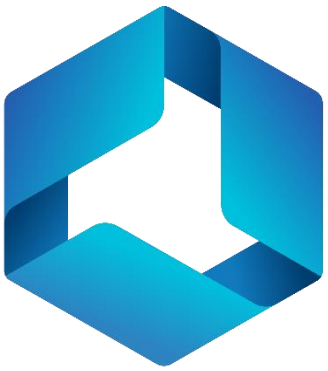


ALTIUM 365

Altium Designer

Essentials Course - Altium 365

Module 26: Draftsman

Table of Contents

Module 26: Draftsman

1.1 Purpose

In this exercise, you will learn to use and update Draftsman documents which can then be used for board production drawings for fabrication and assembly. You will see board views (assembly/fabrication/drill drawing), tables (drill table, layer stack legend, BOM) and other graphical notes or objects to improve clarity.

During the exercise you will open and modify and existing Draftsman document, and then using an existing template to create a new Draftsman document.

1.2 Shortcuts

Shortcuts when working with Module 26: Draftsman

F » N » D:	New Draftsman Document
T » A:	Add New Draftsman Sheet
P » F:	Board Fabrication View
P » B:	Bill Of Material
P » Y:	Layer Stack Legend
P » V:	Place additional Views
P » V » D » C:	Place additional Views **- Circle area detail view**
P » N » O:	Place Callout
Ctrl+S:	Save Document

1.3 Preparation

1. **Close all existing projects and documents.**
2. Next, we create a Copy / Clone of the Training Project `Module 26 Draftsman`.
3. Select **File » Open Project…** to open the *Open Project* dialog.
4. Navigate to the predefined Training Project `Module 26 Draftsman (Top\Projects\Altium Designer Essentials Training Course\...)`.
5. Select **Open Project as Copy…** 

.
6. At the new dialog *Create Project Copy:*
   a) Add your name to the project name: `Module 26 Draftsman – [Your Name]`.
   b) Add a description: `Altium Essential Training - Module 26 – [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.

   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

For details how to Copy / Clone the predefined training project see Module 8 Making the Connection, Step 1.3 Preparation.

1.4 Updating an Existing Draftsman Document

In this part of the exercise, we will open an existing Draftsman document, and additional information, such as: Detail Views, Dimensions and Notes for further information.

7. Open the `Module 26 Draftsman.PcbDoc` document.
8. Open the `Module 26 Draftsman - Assembly.PcbDWF` document.

1.4.1 Page Configuration

9. First, we will check the Sheet Template for the Draftsman document.
10. Open the *Properties* panel and select the *Page Options* tab. Under the *Formatting and Size* pane, you see the used *Template* for this document, as shown in Figure 1.

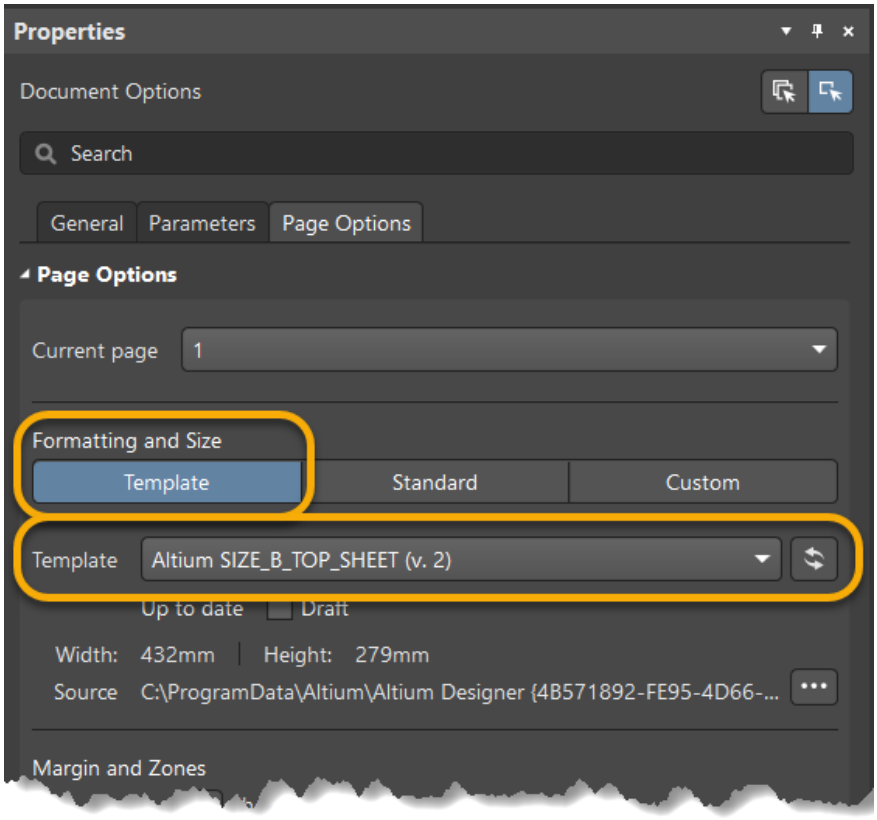


Figure 1. Changing the document template

11. Select the drop-down list to see the different templates that are available. If later needed it is possible to change the template e.g., change the page size from Size B to Size C.

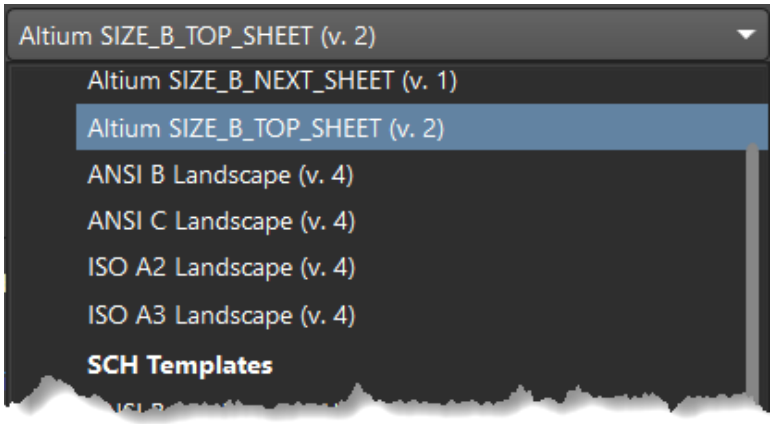


Figure 2. Drop-Down list of Example Template.

1.4.2 **Add Additional Objects to the Draftsman Document**

Next, we will place some additional objects to the Draftsman document. Use Figure 3 as reference for the next steps.

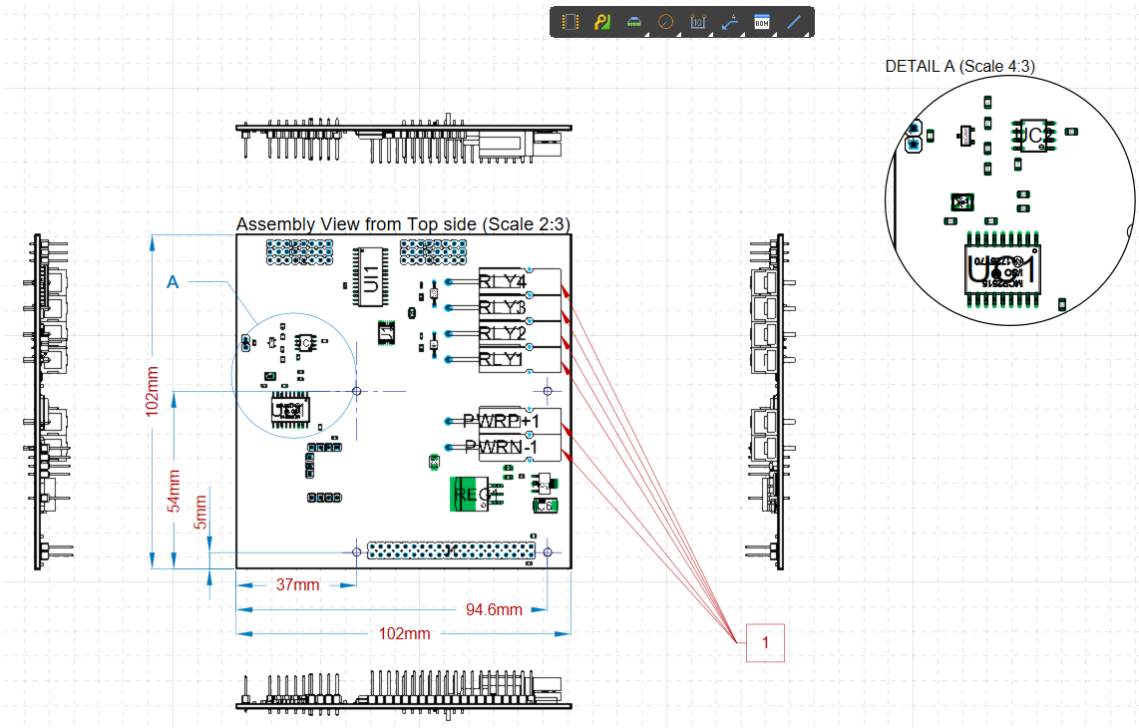


Figure 3. Assembly View with Detail and Dimensions

1.4.2.1 Detail View

12. Right-click anywhere in the document and select **Place » Additional View » Board detail View » Circle area detail view.** This view, similar to other views, can also be placed from the **Place** menu or the **Active Bar.**
    a) Left Click do define the center of the **Circle area detail view**
    b) Move your mouse to define the size of the **Circle area detail view.** If you have the size you like, left click to define he size.
    c) Now the **Circle area detail view** is attached to your mouse. Position the view right of the *Right PCB View* with a left click.
    d) Feel free to reposition the existing BOM table and / or change the scaling for the **Circle area detail view** by opening the *Properties* panel.

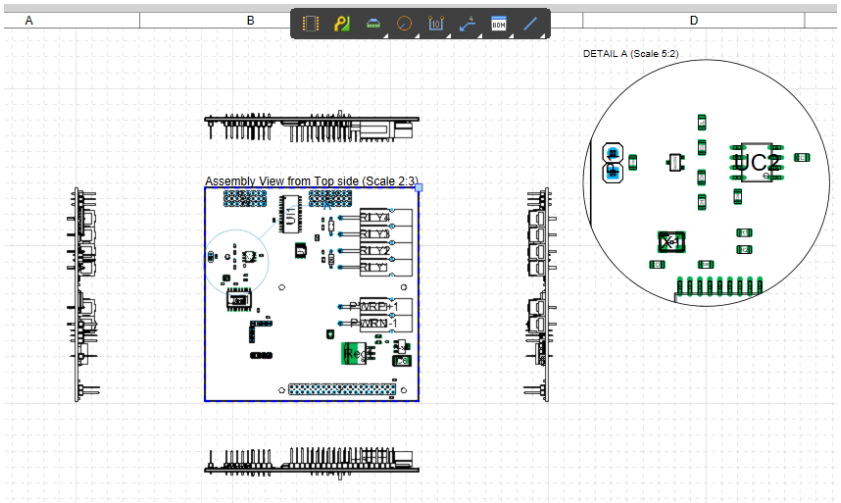
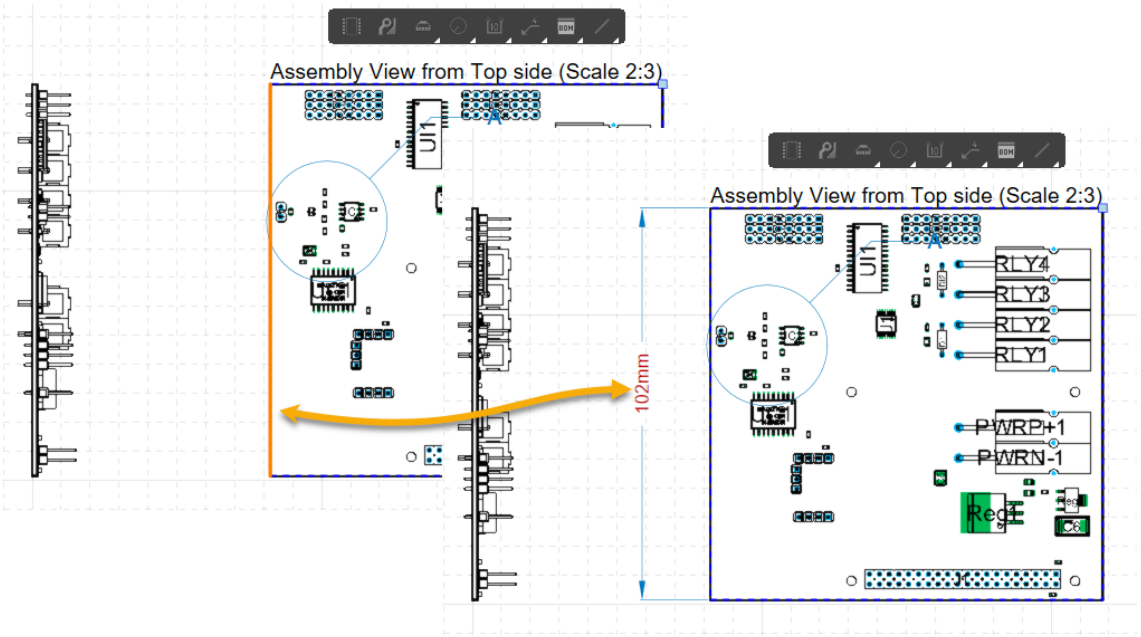


Figure 4. Detail View

1.4.2.2 Dimension

13. We will now place linear dimensions on the *Top Side* assembly view by going to **Place » Linear Dimension**.
    a) Left-click on the left vertical edge of the *Top Side* view. An orange line will appear, indicating that you will place the dimension on the vertical edge.
    b) With the dimension on your cursor, move the cursor away from the edge and left-click to place the dimension. The units of the dimension can be changed in the *Properties* panel when the dimension is selected.
14. Repeat Step 13 to place the horizontal dimension.
15. For the Mounting Holes:
    a) Start the **Linear Dimension** command.
    b) Select the lower left corner of the PC.
    c) Select the Mounting hole to define the dimension.
    d) Position the Dimension.



16. The Template has a default Precision of 1, to change the Precision:
    a) With nothing selected, open the *Properties* panel.
    b) The pane *Units* allows you to change the **Unit** and the **Precision**, as seen at Figure 5.

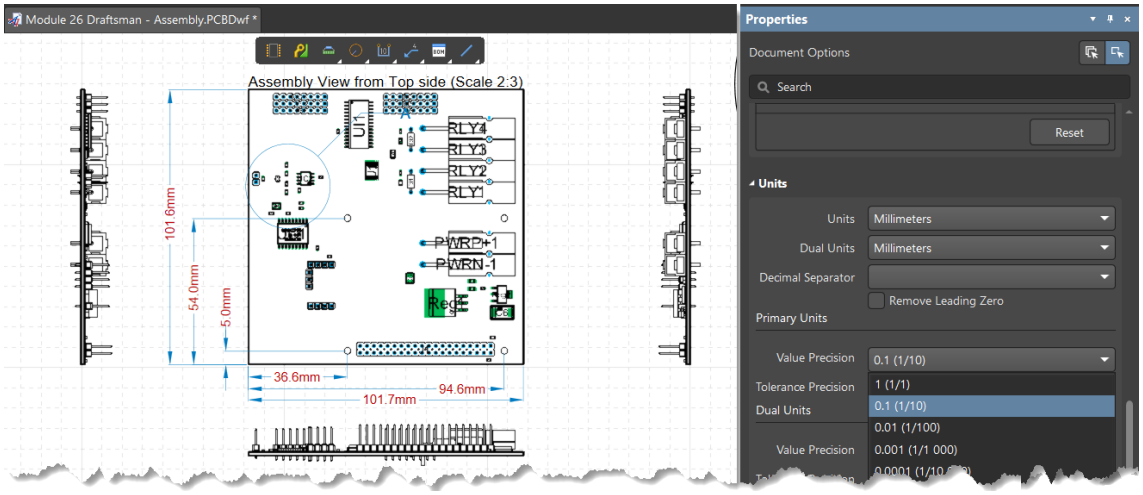


Figure 5. Units for Draftsman Document

1.4.2.3 Callout

17. Right-click anywhere in the document and select **Place » Annotations » Callouts.**
    a) Select RLY4 at the center of the right edge, as seen at Figure 6.

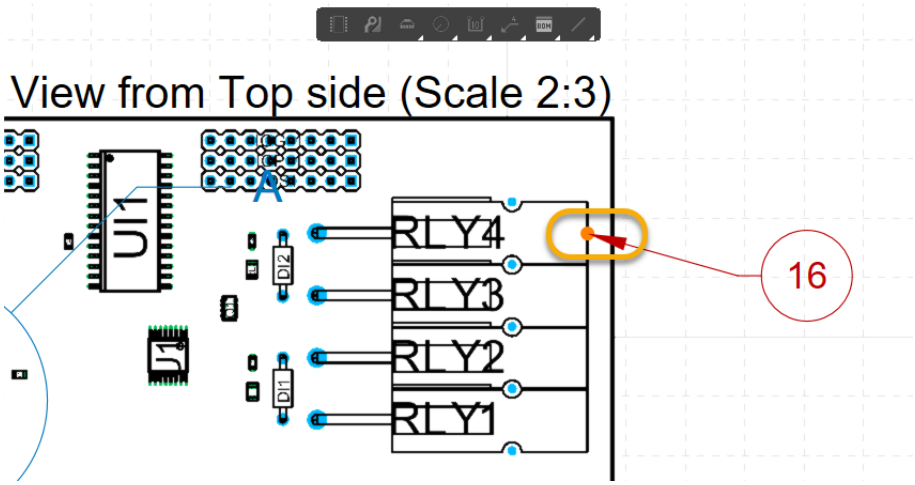


Figure 6. Select RLY4 for Callout

    b) Place the text field at the right side of the PCB.
    c) Position the next arrow, that automatically appear, at `RLY3`.
    d) Repeat Step c) for `RLY2, RLY1, PWRP+1, PWRP-1.` Right click twice to stop the placement for additional Callout arrows and to stop the Callout placement.

18. Select the Callout and open the *Properties* panel to configure the parameters for the callout, as seen at Figure 7.
    a) *Source Text*: **Note Item**
    b) *Note Item*: **1**
    c) *Border*: **Square**
    d) *Elbow Style*: **Elbow Before Tag**
    e) *Elbow Length*: `2mm`
    f) *Elbow Position:* **Right**

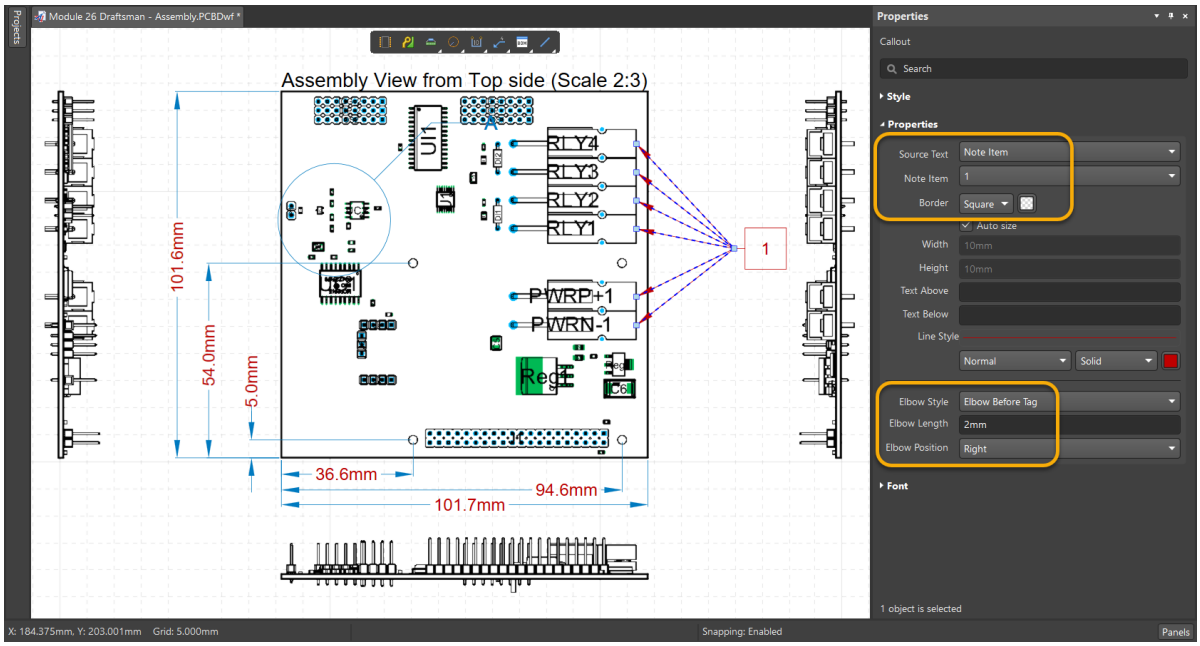


Figure 7. Callout configuration

19. Select **Note1** from the existing Note element.
20. Configure **Note 1**, as seen at Figure 8.
    a) *Note Item Number*: **1**
    b) *Border*: **Square**
    c) *Note Description*: `Note: Use 114555P2 -staple to fasten Anderson Connectors to PCB`
    d) Select **Add**

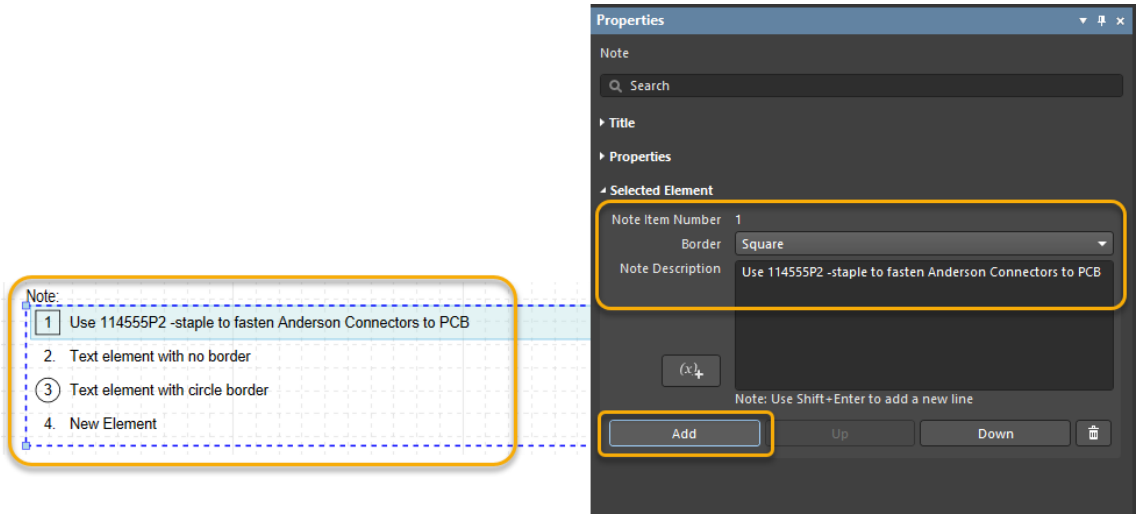


Figure 8 Note

21. Save your modifications for `Module 26 Draftsman - Assembly.PcbDwf`, **File » Save**.

1.5 Creating a Draftsman Document from a Template

Next, we will create a new Draftsman document based on an existing Template.
There are many benefits of using an existing Draftsman template:

- Reduce the time for creating the PCB Documentation
- Ensures that the documents have a uniform appearance.
- Ensure that all required basic information is part of the documentation

The example template represents a start point for the Fabrication documentation.

- The template could be updated to have e.g., more predefined Fabrication Views.
- Similar to the first Draftsman document, we could place additional Objects if needed.

1.5.1 Create Draftsman Document

22. Create new Draftsman document with the command **File » New » Draftsman Document**.
23. At the dialog *New Document* that opens,
    a) Select `Altium SIZE_B_Fabrication` as the template.
    b) At the bottom of the dialog, you see the link to the Project and the PCB that is part of the project.
24. Click **OK** to generate the Draftsman document.

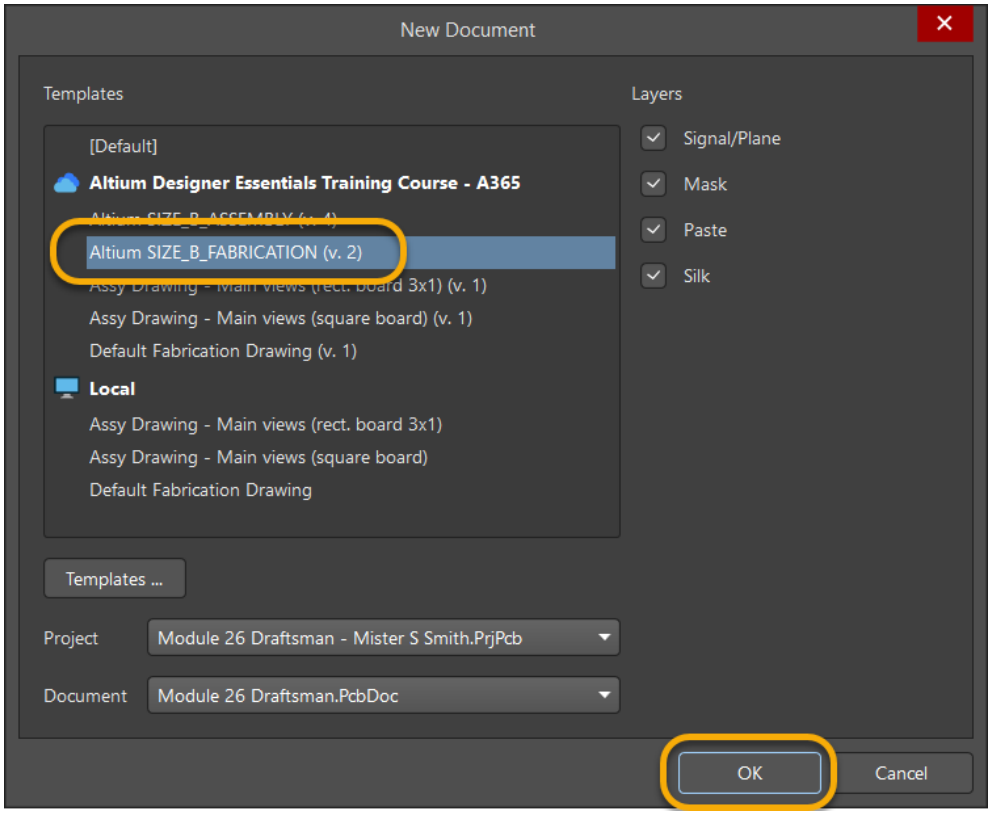


Figure 9.Create New Draftsman document

25. Next you will see a dialog that represents the sync process between the PCB and the new draftsman document, wait until the sync process is finished.

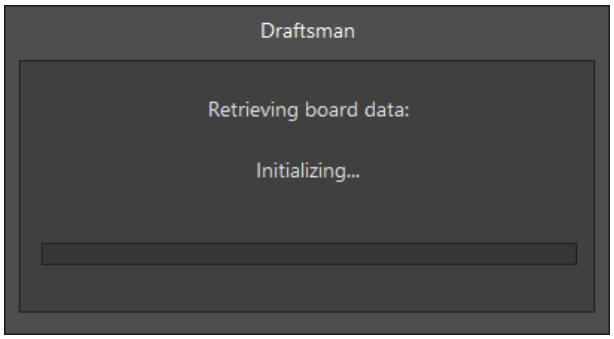


Figure 10. PCB-Draftsman sync process.

26. Go to **File » Save As** and save the Draftsman document as `Module 26 Draftsman - Fabrication.PCBDwf`

The template options may vary if your company has other templates stored in this location, or if this location is pointing to a company folder. These options would also vary if you're using a different Altium 365 Workspace.

27. Open the *Bookmark* panel.
28. The Draftsman document `Module 26 Draftsman - Fabrication.PCBDwf` has 3 Sheets:
    - Sheet 1: A Drill Drawing and a Drill Table
    - Sheet 2: A Layer Stack Table
    - Sheet 3: One Fabrication View (Top Layer)

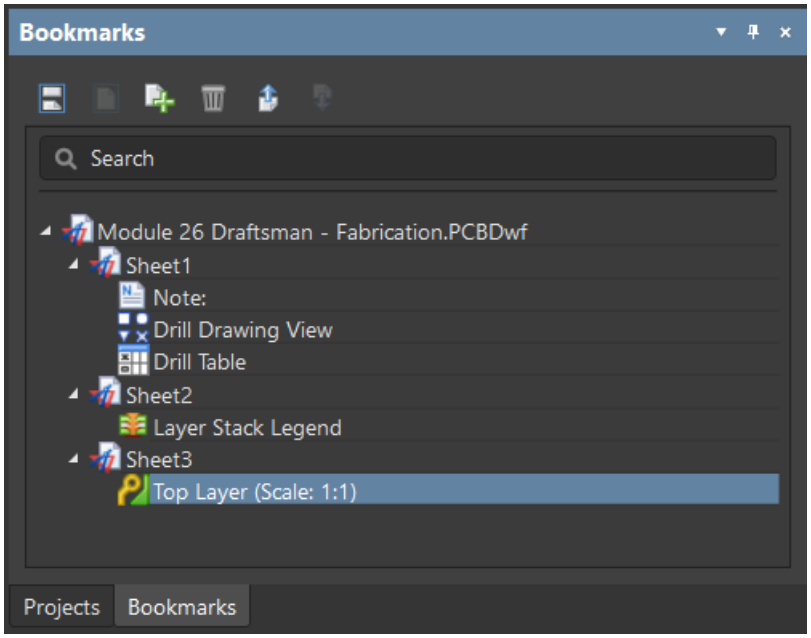


Figure 11 Bookmark panel for Draftsman Documents

29. Double-click a sheet or an object to jump to that sheet / object.
30. Select Sheet 3 to see the Top Layer view.

1.5.2 **Fabrication View – Layer View**

31. Right-click anywhere in the third sheet and select **Place » Board Fabrication View.**
    By default, the top-side view of the board will be provided. This view can also be placed from the **Place** menu or the **Active Bar.**
32. Place this view next to the existing view, more to the upper right of the sheet.
33. As alternative you could Copy - Paste the first Fabrication View.
    a) **Right-click** on the placed fabrication view and click **Copy** or use **CTRL+C**. A crosshair will be present, where you will need to select a reference point to **Copy** it.
    b) The Fabrication View can then be pasted into the document using **CTRL+V**.
34. Double-click on the second Fabrication View to open the *Properties* panel.
    a) Navigate to the section *Properties*, to see the option *Layer* with the Layer drop down list.
    b) Select from the drop-down List `Bottom Layer`.
    c) The second Fabrication View is automatically updated to show the Bottom Layer.
    d) Your view should now look like the example seen at Figure 12.

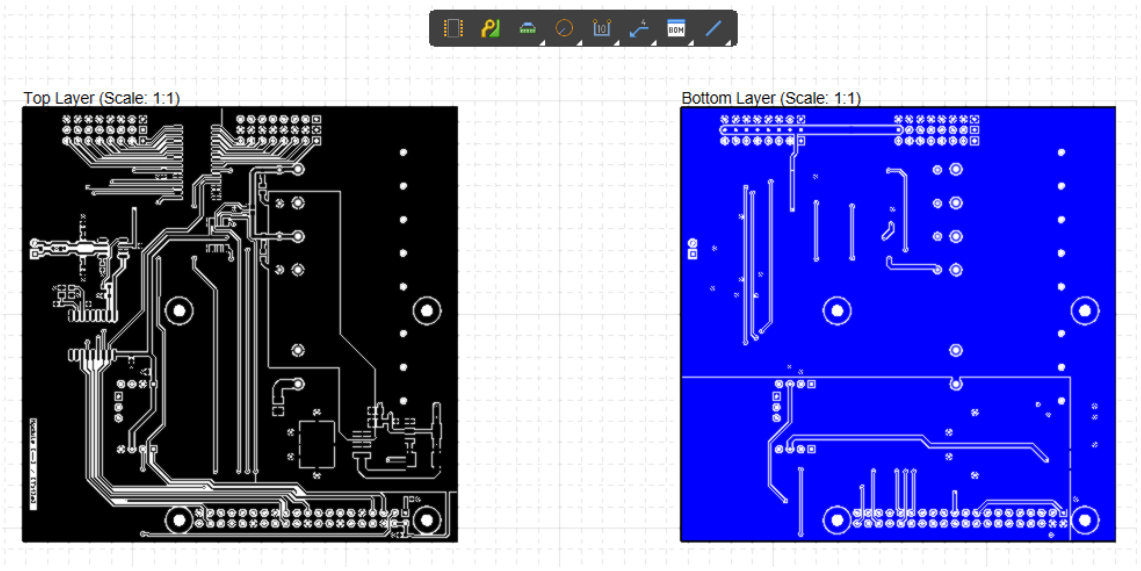


Figure 12. Fabrication View - TOP and Bottom layer

To Update your documentation, after the PCB is modified, execute **Tools » Import Changes From Board**. This command started the sync process for the Draftsman document.

35. Save all documents using **File » Save All**.
36. Save the modifications to the server:
    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) Activate the checkboxes for the files that are not under version control.
        ii) Add the comment `Module 26: Draftsman - [Add Your Name]` - Finished
        iii) Select **OK.**

Congratulations on completing the Module!

Module 26: Draftsman

from the

Altium Designer Essential Course
with Altium 365

Thank you for choosing Altium Designer