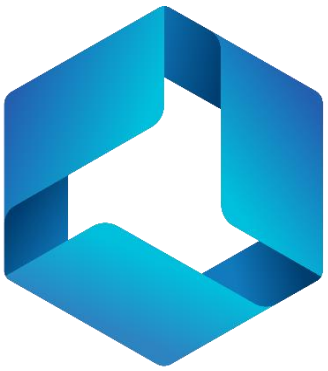


ALTIUM 365

Altium Designer

Essentials Course - Altium 365

Module 18: PCB Layerstack Manager

Table of Contents

Module 18: PCB Layerstack Manager

1.1 Purpose

The Layer Stack Manager in Altium Designer allows designers to define and manage the layer stackup of a printed circuit board (PCB). The layer stackup is a critical aspect of PCB design, as it determines the arrangement and properties of the various layers that make up the PCB. The Layer Stack Manager provides a centralised interface for configuring the layer stackup to meet specific design requirements

In this exercise, we will examine the layers associated with our design and learn how to add plane layers to simplify power and ground connections used later.

1.2 Shortcuts

Shortcuts when working with Module 18: PCB Layerstack Manager

D » K:	Layer Stack Manager
Ctrl+S:	Save Document
L:	View Configuration panel

1.3 Preparation

1. **Close all existing projects and documents.**
2. Next, create a Copy / Clone of the Training Project `Module 18 Layerstack Manager`.
3. Select **File » Open Project…** to open the *Open Project* dialog.
4. Navigate to the predefined Training Project `Module 18 Layerstack Manager (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: `Module 18 Layerstack Manager - [Your Name]`.
   b) Add a description: `Altium Essential Training - Module 18 - [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.
7. Wait until Altium Designer created the copy of the project and opened the project for you at the *Projects* panel, this may 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 Layer Stack Manager

The additional Layers and Vias added during the following steps are only for the training, to show additional options. The original board is a 2 Layer Board.

1.4.1 Load Default Layer Stack

8. From the *Projects* Panel, open the `Module 18 Layerstack Manager.PcbDoc` document.
9. With the `Module 18 Layerstack Manager.PcbDoc` as the active document, go to the **Design** menu and select the **Layer Stack Manager…** . You can also use the shortcut keys **D » K**.
   a) A new document will appear as shown in Figure 1 below.
   b) The *Properties* panel will also appear, and you can dock it along the left or right side of the workspace.

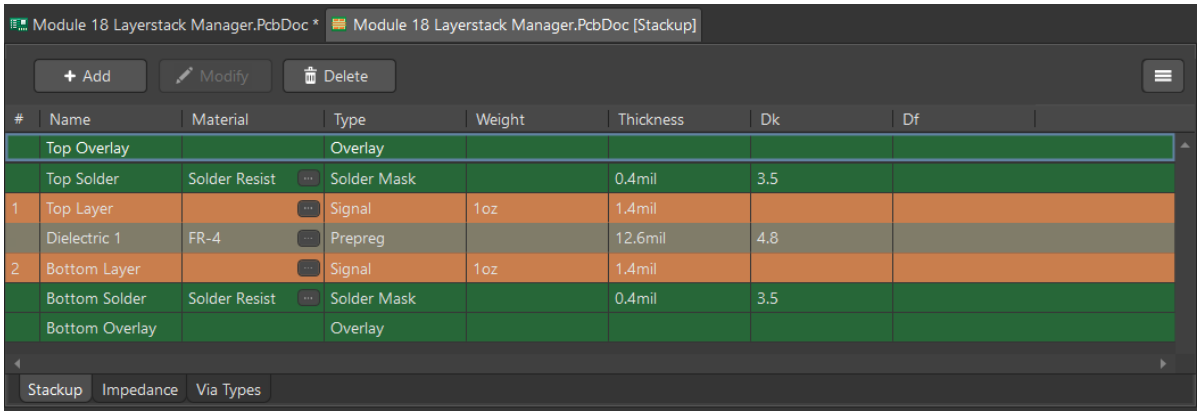


Figure 1. Layer Stack Manager

1.4.2 Load Stackup from Altium 365

10. Select **File » Load Stackup from Server…** .
    a) At the *Choose Item Revision* dialog navigate to the Section `Managed Content - Templates - Layer Stacks`.
    b) Select the Stack ALS-0000 , a 2 Layer + 0 Plane Stack.
11. Notice, after selection the layer stack is updated, as shown in Figure 2.

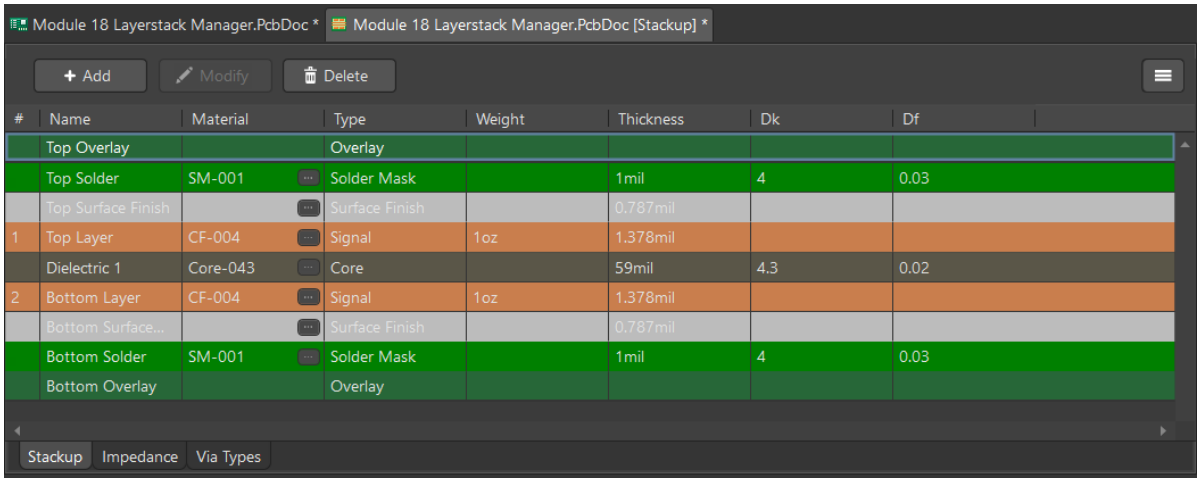


Figure 2. 2 Layer Layerstack loaded from Altium 365

1.4.3 **Modify the Layer Stack**

If you need a Layer stack that is not available in the Altium 365 Workspace, chose the one that is as close as possible to the stack you need and modify it.

12. Select the layer *Top Layer*.
13. Right click on *Top Layer*, select **Insert layer below » Plane** as shown in Figure 3 below.

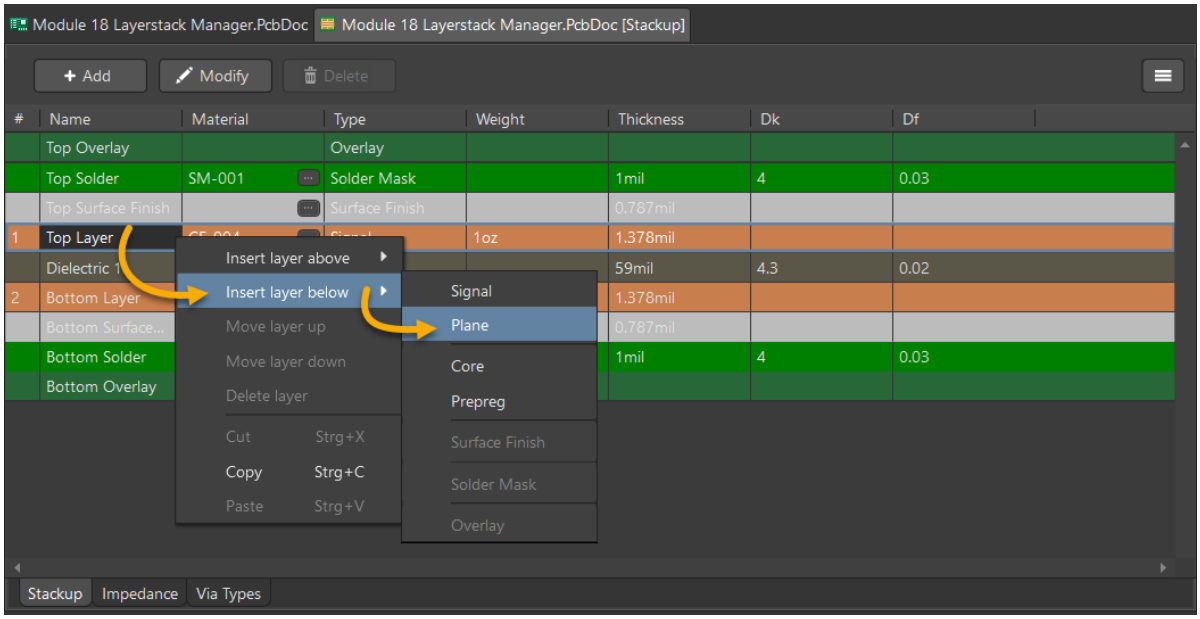


Figure 3. Adding Plane Layers

14. As a default configuration, two Plane Layers are added to create a symmetric stack as shown in Figure 4 below. The **Stack Symmetry** option can be enabled or disabled from the *Properties* panel.

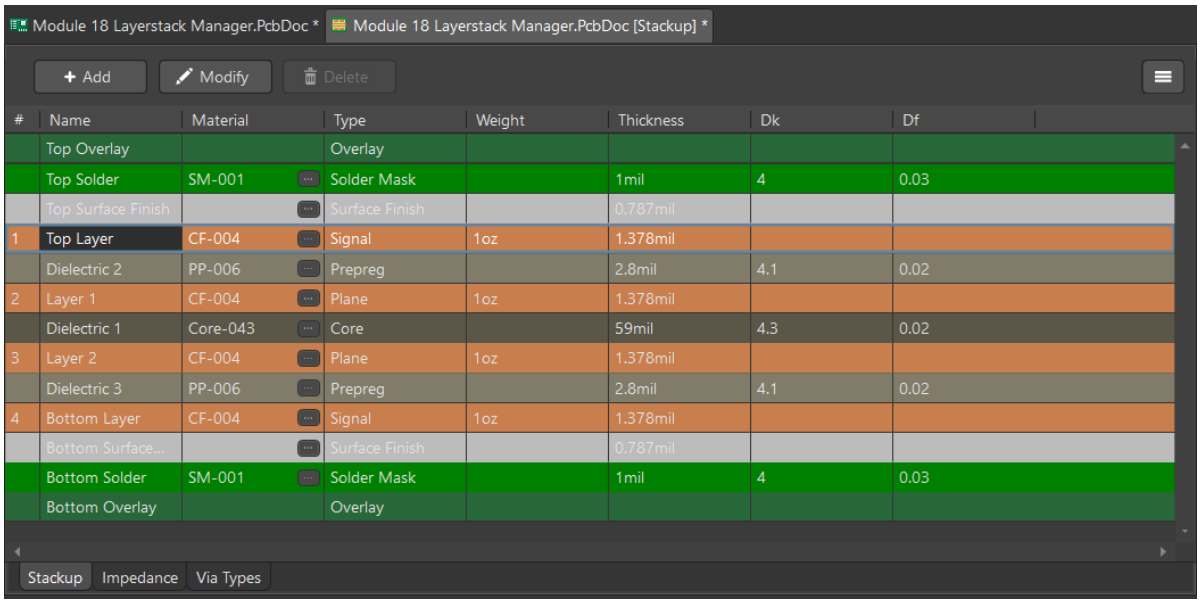


Figure 4. New Plane layers added

15. Click on the new *Layer 1* cell and rename it `GND` and hit **Enter**.

Edit text by selecting the cell, then click the cell again with left mouse button, or just press the F2 key.

16. Repeat the previous step to name the second Plane layer as `3V3`. The layer stack should look like Figure 5 below.

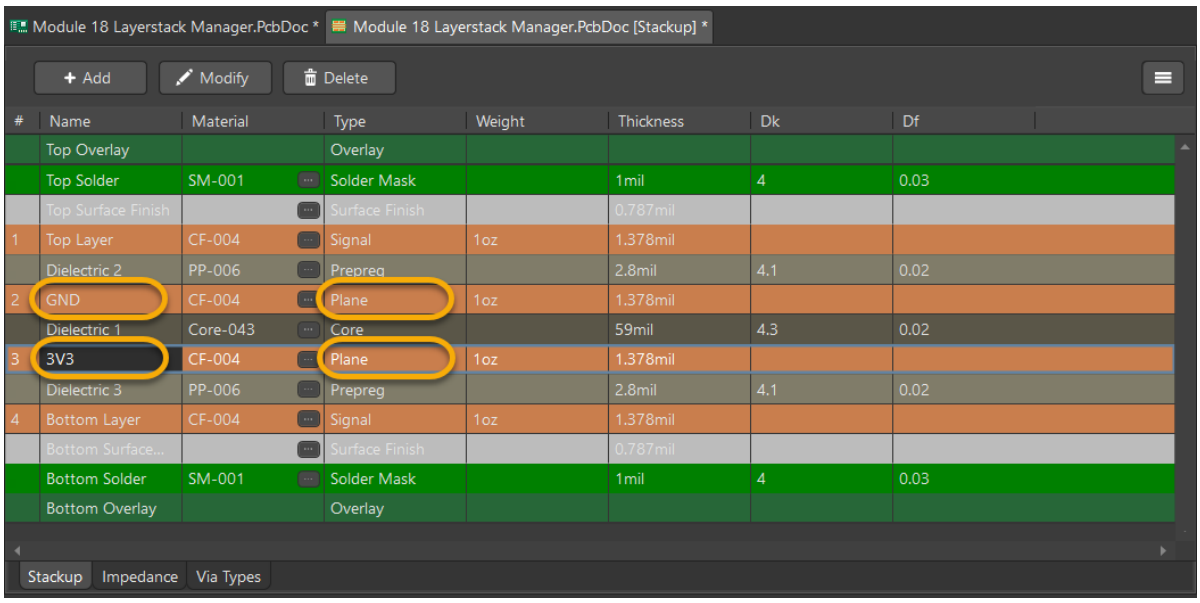
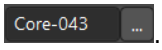


Figure 5. Layer Stack with renamed Plane Layers

17. Open the *Properties* panel. The *Properties* panel shows the layer information, e.g., the option for a symmetrical stack.
18. Select *Top Layer* and then *Dielectric 1* to observe the change in the *Properties* panel. You'll notice that the material and other parameters will be different between the layers.
19. Select the cell *Material* for *Dielectric 1* 

.
20. Click on Ellipsis button 

to open the *Select Material* dialog. Select a new core material, e.g., Core-29 as shown in Figure 6 below.
21. Click **OK**.

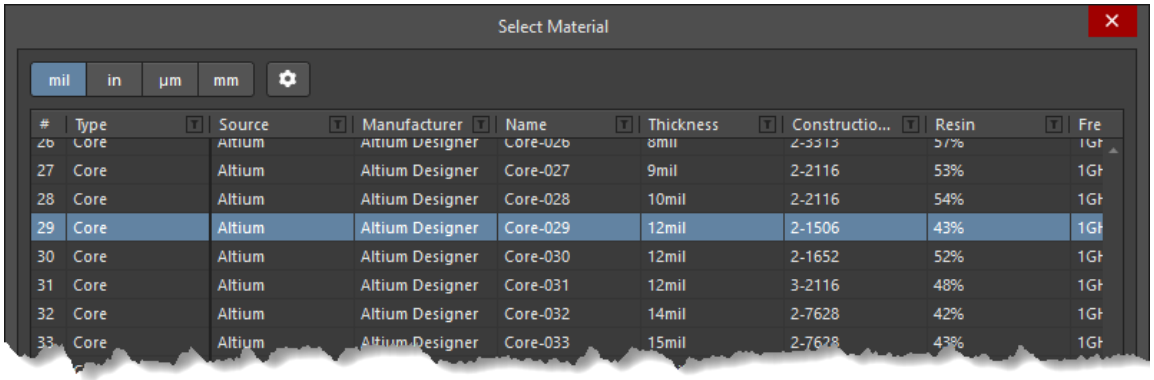


Figure 6. PCB Material database

1.4.4 **Adding additional Vias**

22. At the bottom of the *Layer Stack Manager* select the tab *Via Types*. By default, a through-hole Via from the Top Layer to Bottom Layer is available, as shown in Figure 7 below.

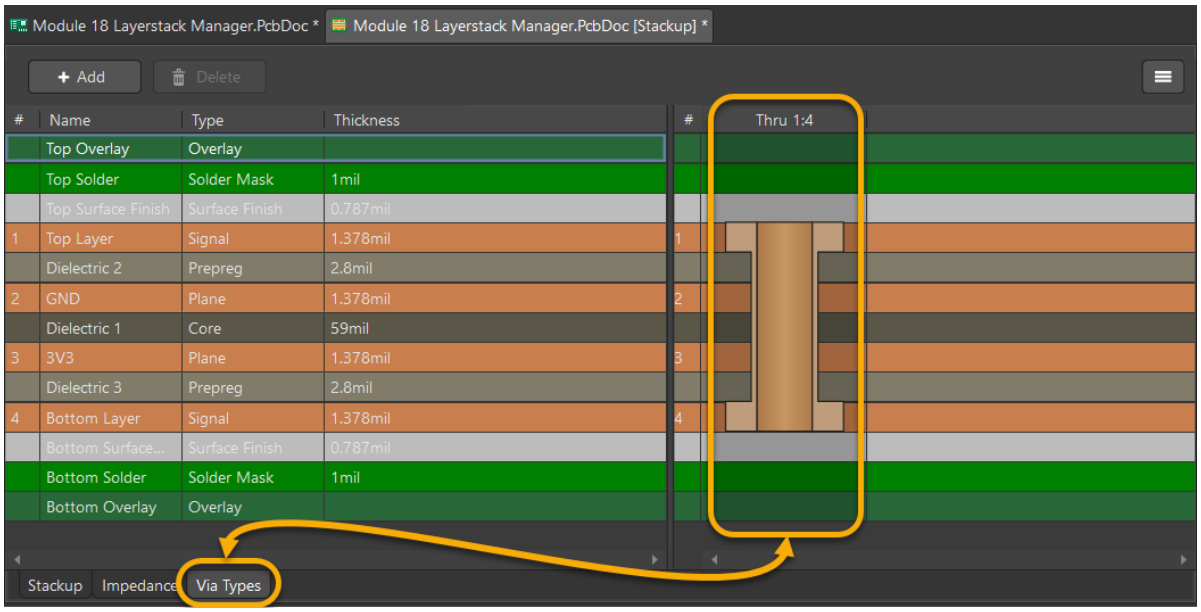


Figure 7. Layer Stack Manager with default via pairs

23. For Multi-Layer designs additional Vias like Blind, or Buried vias, or uVias for HDI Designs, can be added as well.
24. Close the *Layer Stack Manager* and save the changes.

1.5 Layer Stack Table

25. Select the Layer Set **Default** to activate all Layers, LS or **Design » Manage Layer Sets » All Layers**.
26. From the **Place** menu, select **Layer Stack Table** to add a table of the Layerstack in the workspace, on the right side of PCB, inside the PCB Title Block.
27. Press the **Tab** key before placing the Layer Stack Table.
28. Modify the following parameters in the *Properties* panel as shown in Figure 8.
    a) Change **Layer** to `19_FABRICATION_NOTES`
    b) Uncheck the **Show Board Map**
    c) Set **Line Width** to `8mil`
    d) Set **Text Height** to `30mil`
    e) Set **Stroke Width** to `5mil`

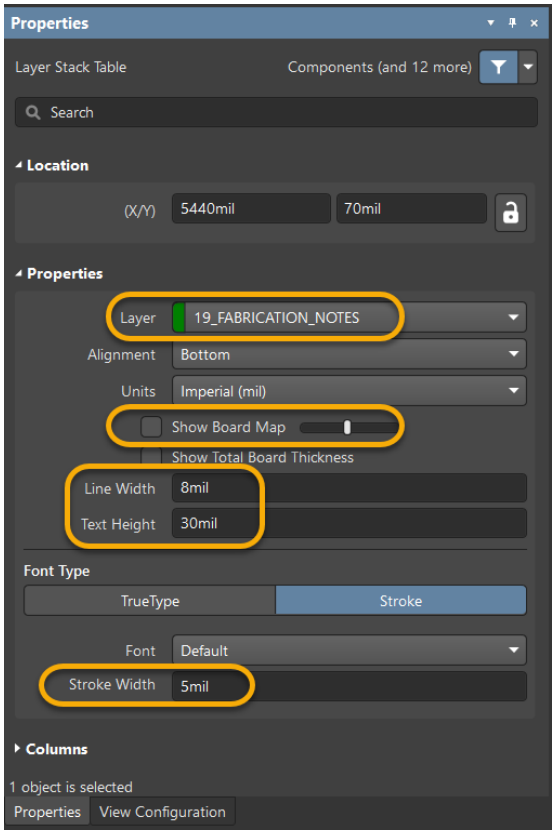
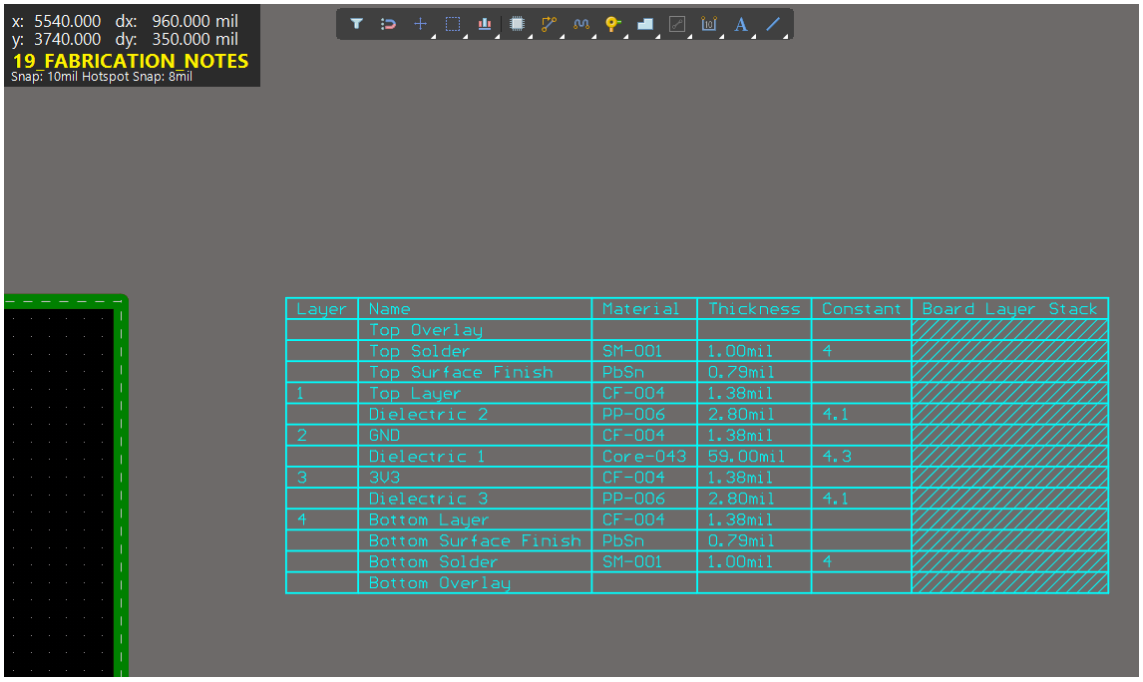


Figure 8. Layer Stack Table

29. Hit **Enter** or press the **Pause** icon in the middle of the workspace and position the Layer Stack Table, e.g., as shown in Figure 9 below. **Right Click** to exit Layer Stack Table placement command once you've placed it.



Layer	Name	Material	Thickness	Constant	Board Layer Stack
	Top Overlay				
	Top Solder	SM-001	1.00mil	4	
	Top Surface Finish	PbSn	0.79mil		
1	Top Layer	CF-004	1.38mil		
	Dielectric 2	PP-006	2.80mil	4.1	
2	GND	CF-004	1.38mil		
	Dielectric 1	Core-043	59.00mil	4.3	
3	3V3	CF-004	1.38mil		
	Dielectric 3	PP-006	2.80mil	4.1	
4	Bottom Layer	CF-004	1.38mil		
	Bottom Surface Finish	PbSn	0.79mil		
	Bottom Solder	SM-001	1.00mil	4	
	Bottom Overlay				

Figure 9. Place Layer Stack Table

30. Save all documents using **File » Save All.**
31. 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 18: PCB Layerstack Manager - [Add Your Name] - Finished.`
        iii) Select **OK.**
32. **When ready, close the project and any open documents.**

Congratulations on completing the Module!

Module 18: PCB Layerstack Manager

from the

Altium Designer Essential Course
with Altium 365

Thank you for choosing Altium Designer