



ALTium **365**

## **Altium Designer**

### **Essentials Course - Altium 365**

Module 18: PCB Layerstack Manager

Software, documentation and related materials:

Copyright © 2024 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®, A365™, Altium 365®, Altium Concord™, Altium Concord Pro™, Altium Designer®, AD™, Altium NEXUS®, Altium OnTrack™, Altium Vault®, Autotrax®, Camtastic®, Ciiva™, CIIVA SMARTPARTS®, CircuitMaker®, CircuitStudio®, Common Parts Library™, Concord™, Concord Pro®, Draftsman®, Dream, Design, Deliver®, DXP™, Easytrax®, EE Concierge®, Fearless HDI™, Geppetto®, Gumstix®, Learn, Connect, Get Inspired™, NanoBoard®, NATIVE 3D™, OCTOMYZE®, Octopart®, OnTrack™, Overo®, 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 affiliated companies. 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

<b>Module 18: PCB Layerstack Manager</b>	<b>3</b>
<b>1.1 Purpose</b>	<b>3</b>
<b>1.2 Shortcuts</b>	<b>3</b>
<b>1.3 Preparation</b>	<b>3</b>
<b>1.4 Layer Stack Manager</b>	<b>4</b>
1.4.1 Load Default Layer Stack.....	4
1.4.2 Load Stackup from Altium 365 .....	4
1.4.3 Modify the Layer Stack.....	5
1.4.4 Adding additional Vias .....	7
<b>1.5 Layer Stack Table</b>	<b>8</b>

# 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.

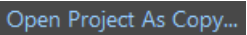

## 1.2 Shortcuts



Shortcuts when working with Module 18: PCB Layerstack Manager

<b>D » K:</b>	Layer Stack Manager
<b>Ctrl+S:</b>	Save Document
<b>L:</b>	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.

#	Name	Material	Type	Weight	Thickness	Dk	Df
	Top Overlay		Overlay				
	Top Solder	Solder Resist	Solder Mask		0.4mil	3.5	
1	Top Layer		Signal	1oz	1.4mil		
	Dielectric 1	FR-4	Prepreg		12.6mil	4.8	
2	Bottom Layer		Signal	1oz	1.4mil		
	Bottom Solder	Solder Resist	Solder Mask		0.4mil	3.5	
	Bottom Overlay		Overlay				

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.

#	Name	Material	Type	Weight	Thickness	Dk	Df
	Top Overlay		Overlay				
	Top Solder	SM-001	Solder Mask		1mil	4	0.03
	Top Surface Finish		Surface Finish		0.787mil		
1	Top Layer	CF-004	Signal	1oz	1.378mil		
	Dielectric 1	Core-043	Core		59mil	4.3	0.02
2	Bottom Layer	CF-004	Signal	1oz	1.378mil		
	Bottom Surface...		Surface Finish		0.787mil		
	Bottom Solder	SM-001	Solder Mask		1mil	4	0.03
	Bottom Overlay		Overlay				

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, choose 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.

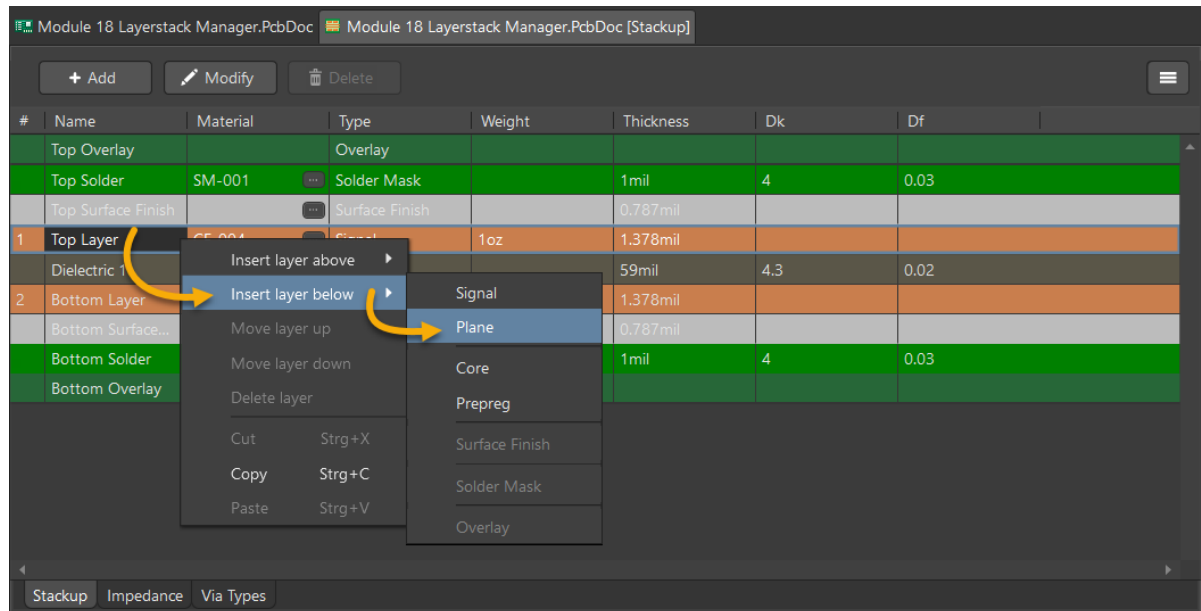


Figure 3. Adding Plane Layers

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.

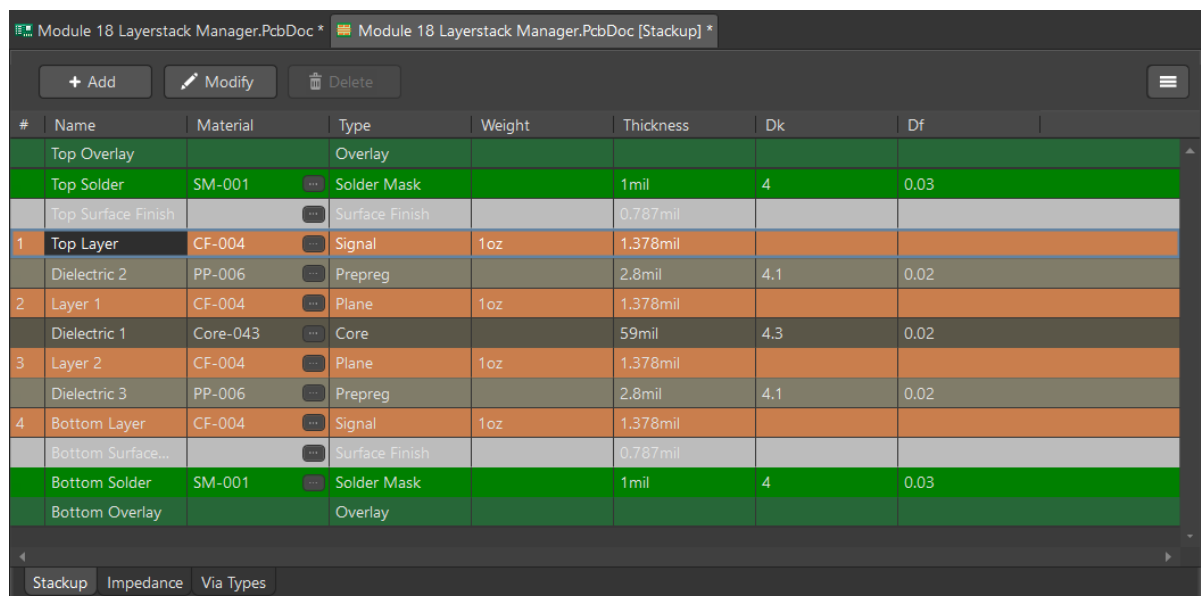


Figure 4. New Plane layers added

- 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.

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

#	Name	Material	Type	Weight	Thickness	Dk	Df
	Top Overlay		Overlay				
	Top Solder	SM-001	Solder Mask		1mil	4	0.03
	Top Surface Finish		Surface Finish		0.787mil		
1	Top Layer	CF-004	Signal	1oz	1.378mil		
	Dielectric 2	PP-006	Prepreg		2.8mil	4.1	0.02
2	<b>GND</b>	CF-004	<b>Plane</b>	1oz	1.378mil		
	Dielectric 1	Core-043	Core		59mil	4.3	0.02
3	<b>3V3</b>	CF-004	<b>Plane</b>	1oz	1.378mil		
	Dielectric 3	PP-006	Prepreg		2.8mil	4.1	0.02
4	Bottom Layer	CF-004	Signal	1oz	1.378mil		
	Bottom Surface...		Surface Finish		0.787mil		
	Bottom Solder	SM-001	Solder Mask		1mil	4	0.03
	Bottom Overlay		Overlay				

Figure 5. Layer Stack with renamed Plane Layers

- Open the *Properties* panel. The *Properties* panel shows the layer information, e.g., the option for a symmetrical stack.
- 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.
- Select the cell *Material* for *Dielectric 1* Core-043 ....
- 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.
- Click **OK**.

#	Type	Source	Manufacturer	Name	Thickness	Constructio...	Resin	Fre
26	Core	Altium	Altium Designer	Core-026	8mil	2-3313	57%	1Gt
27	Core	Altium	Altium Designer	Core-027	9mil	2-2116	53%	1Gt
28	Core	Altium	Altium Designer	Core-028	10mil	2-2116	54%	1Gt
29	Core	Altium	Altium Designer	Core-029	12mil	2-1506	43%	1Gt
30	Core	Altium	Altium Designer	Core-030	12mil	2-1652	52%	1Gt
31	Core	Altium	Altium Designer	Core-031	12mil	3-2116	48%	1Gt
32	Core	Altium	Altium Designer	Core-032	14mil	2-7628	42%	1Gt
33	Core	Altium	Altium Designer	Core-033	15mil	2-7628	43%	1Gt

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.

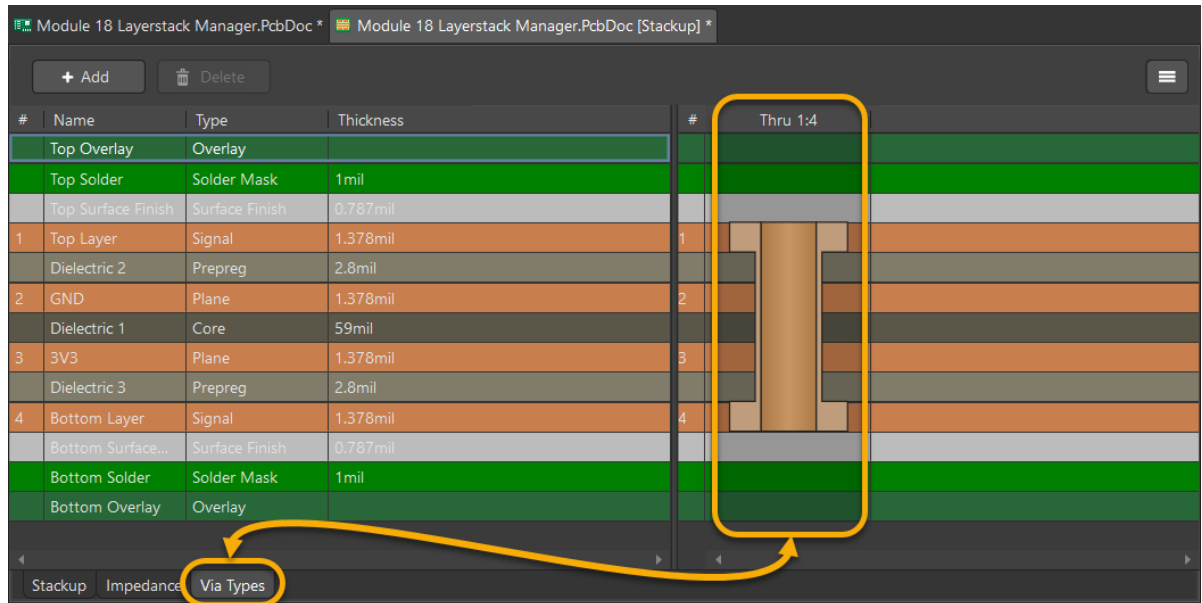



Figure 7. Layer Stack Manager with default via pairs

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,  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

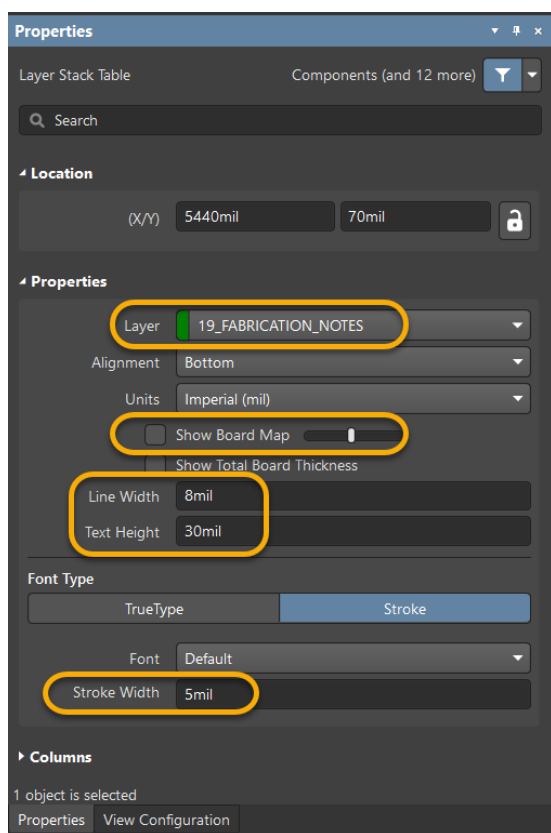


Figure 8. Layer Stack Table

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.

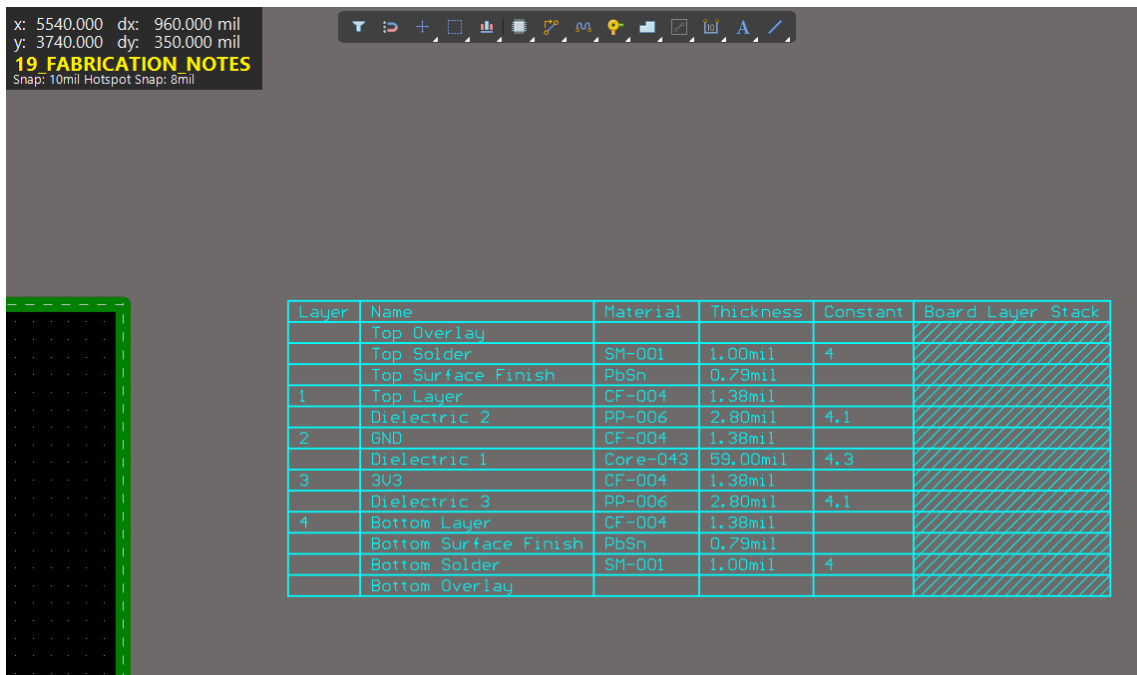



Figure 9. Place Layer Stack Table

30. Save all documents using **File » Save All**.
31. Save the modifications to the server:
- At the *Project* panel, next to the Project name you find the command **Save to Server** .
  - Select **Save to Server**.
  - At the dialog *Save [Project Name]*:
    - Activate the checkboxes for the files that are not under version control.
    - Add the comment *Module 18: PCB Layerstack Manager - [Add Your Name] - Finished*.
    - 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**