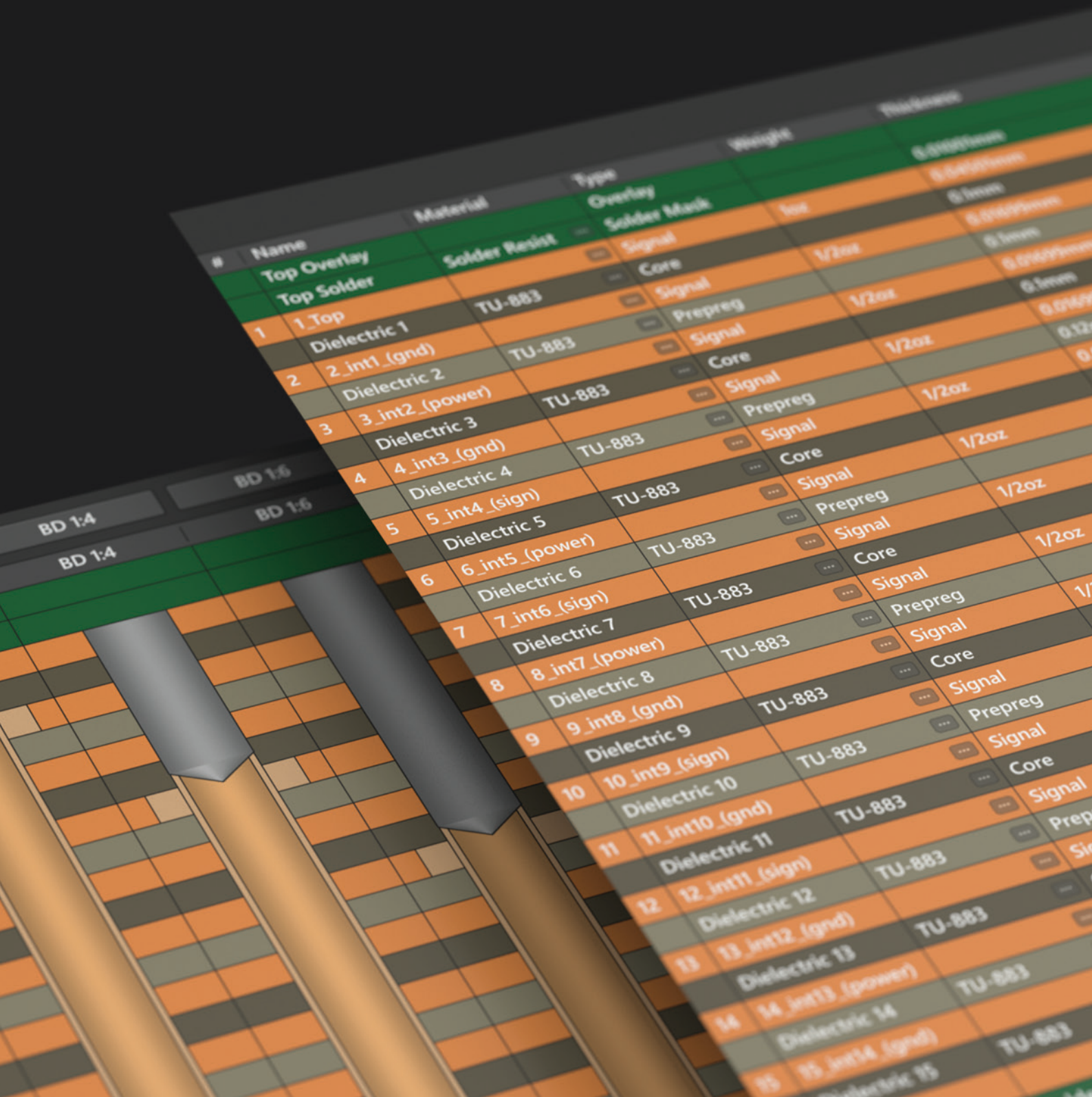
Altium

Impedance Calculation

Guide for Altium Designer



IMPEDANCE CALCULATION

INTRODUCTION

In modern devices signals are operating at higher frequencies, rising and falling edges are reduced (1 ns or less), and the frequency is increased (to tens of GHz). This requires the use of printed circuit boards (**PCB**) with controlled impedance in order to prevent distortion of the signal when it is transmitted over the conductors.

A conductor on a PCB is no longer just a track linking the contact pads and vias, but a transmission line that has to transmit a signal with low-loss of shape, amplitude and velocity.

When designing a PCB, the developer must determine which layers contain impedance-controlled conductors (or differential pairs) and which layers contain ground and power reference layers. The developer's task is to perform preliminary calculations of the PCB structure and design it taking into account the calculated values of the conductor width in the specified layers.

Modern CAD for PCB design should support different transmission line structures and consider as many parameters that can affect the calculation of impedance as possible. The impedance calculator must use the most accurate formulas for calculation.

Controlled impedance of PCBs raises the process of design, material selection, structure and production of PCBs to a new level.

BENEFITS OF THE ALTIUM DESIGNER IMPEDANCE CALCULATOR

- Supports different line structures:
  - Microstrip - the outer layers
  - Stripline - is constructed with a flat conductor suspended between two ground planes, where the conductor and ground planes are separated by a dielectric.
  - Embedded Microstrip - a flat conductor suspended over a ground plane with a dielectric between it and another dielectric material above the conductor
  - Asymmetric Stripline - is most commonly found in a pcb where the distance from trace to planes is not the same distance above and below
  - Coplanar - lines that lie on the same plane
- Uses a material library
- Provides transmission line calculation for several impedance values (including single layer)
- Automatic impedance calculation
- Visual presentation of complete information about transmission line, including layer structure with parameters and line constructions Accounts for side etching of conductor
- Considers mask thickness above the conductor and above the board
- Enables Selection of model and roughness parameters
- Performs Delay Calculation
- Provides Inductance calculation
- Provides Capacitance calculation
- Available for FREE as part of Altium Designer 20 and beyond

IMPEDANCE CALCULATION

EXAMPLE OF IMPEDANCE CALCULATION

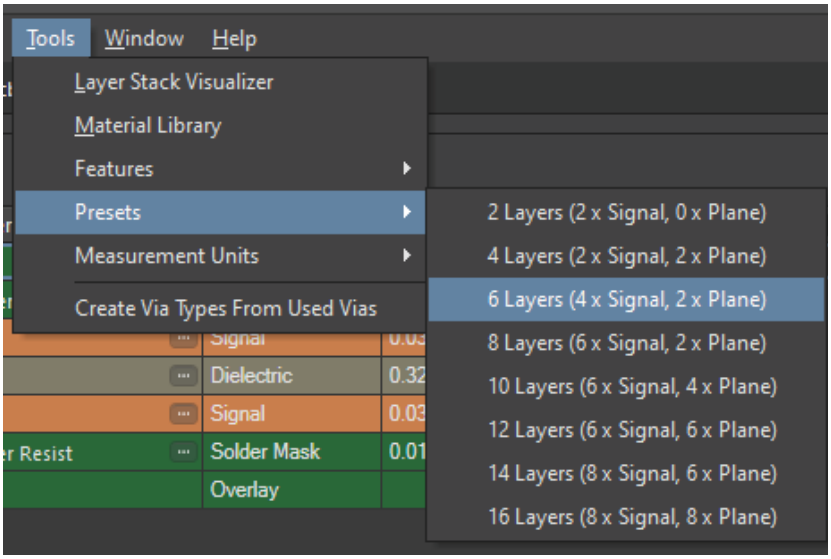
When designing a board with controlled impedance, engineers need to minimize manufacturing costs. Therefore, at the initial stage, engineers tend to focus on certain parameters including conductor thickness and gap. Then, using the selected parameters, the developer attempts to select the appropriate materials and PCB stack. There is an example below showing how this can be done using the Altium Designer Layer Stack Manager and its impedance calculation capability. It is worth noting that the opposite situation is possible in projects when the PCB stack is a constant and the user must determine the width of the conductor and the gap.

Example:

It is necessary to design Edge-Coupled Microstrip on the outer layers of the 6-layer board.

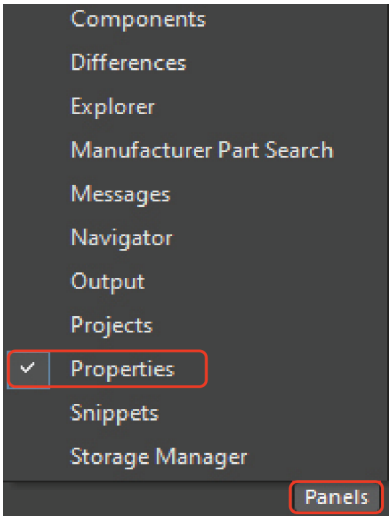
Input parameters:
Line width 0.2 mm
Gap between the lines 0.2 mm
100 Ohm impedance, tolerance not more than 5%.
It is necessary to select the stack materials.

1. Load the PCB stack of 6 layers into the Layer Stack Manager.



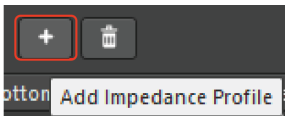
IMPEDANCE CALCULATION

2. Display the **Properties** panel. Using the **Panels** at the bottom-left of the workspace, select the **Properties** panel. This panel should always be open when working in the Layer Stack Manager.

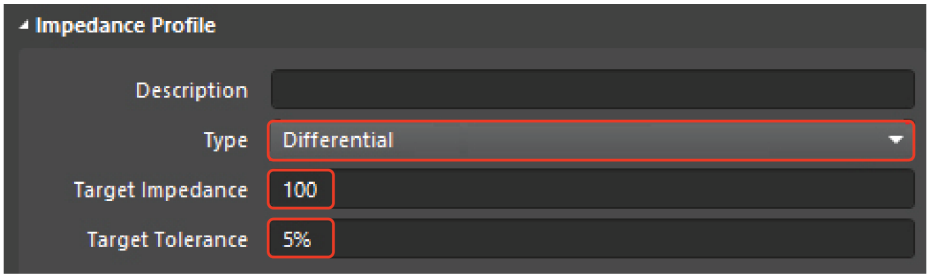


3. Select the **Impedance** tab in the Layer Stack Manager.

4. Add the Impedance Profile by clicking the + button.



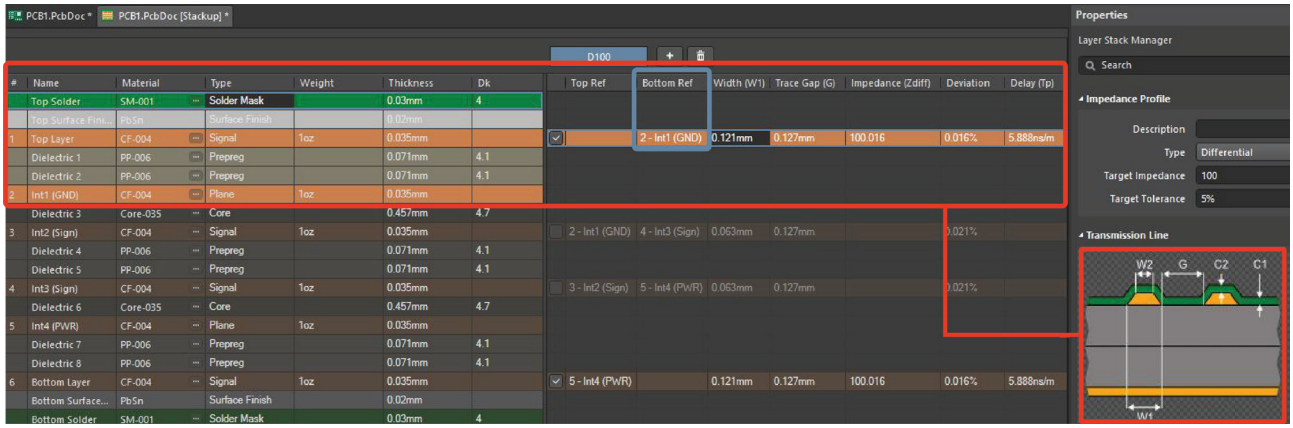
5. In the **Properties** panel in the **Impedance Profile** section, select the **Differential** type, Set the **Target Impedance** value to **100** and **Target Tolerance** field to **5%**.



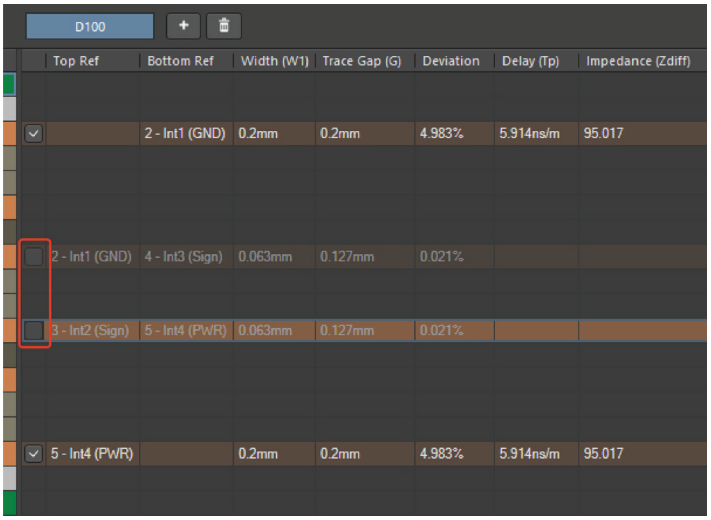
IMPEDANCE CALCULATION

6. Select the layer where the transmission line will be located. Specify **1 layer (Top Layer)** in the impedance profile, to highlight all layers in the stack-up that participate in the calculation of this transmission line (display the structure and parameters of this line). In the impedance profile in the **Bottom Ref** column - select the nearest reference layer **2 - Int1 (GND)**.

   In the **Properties** panel in the **Transmission Line** section, the selected line construction is displayed.



7. Disable Layers **3-Int2 (Sign)** and **4-Int3 (Sign)** in the profile, as the task is to count only microstrip transmission line.



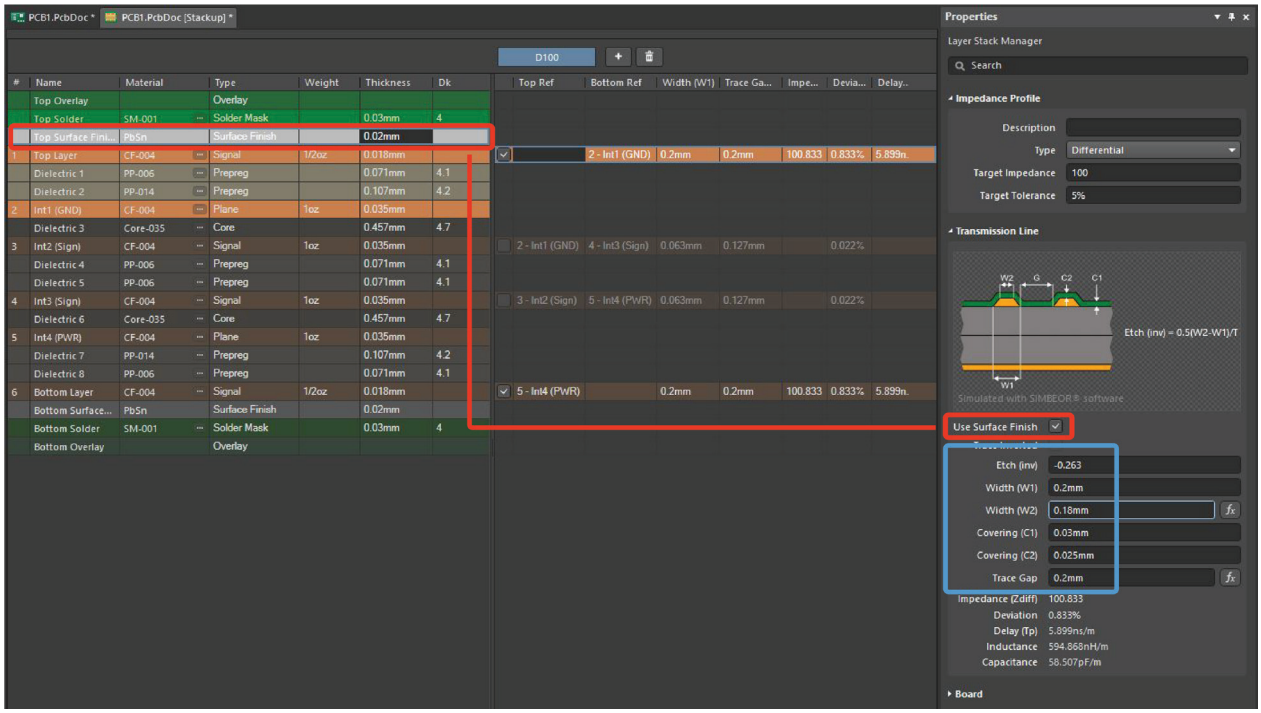
8. Set the checkbox for the use of the **Surface finish** in the **Properties** panel, which requires a **Surface finish** layer in the stack-up.

9. In the **Properties** panel - change the line width **W1 = 0.2 mm**, **W2 = 0.18 mm**. Etching (W2) - this setting depends on the technological specifics of the PCB fabricator.

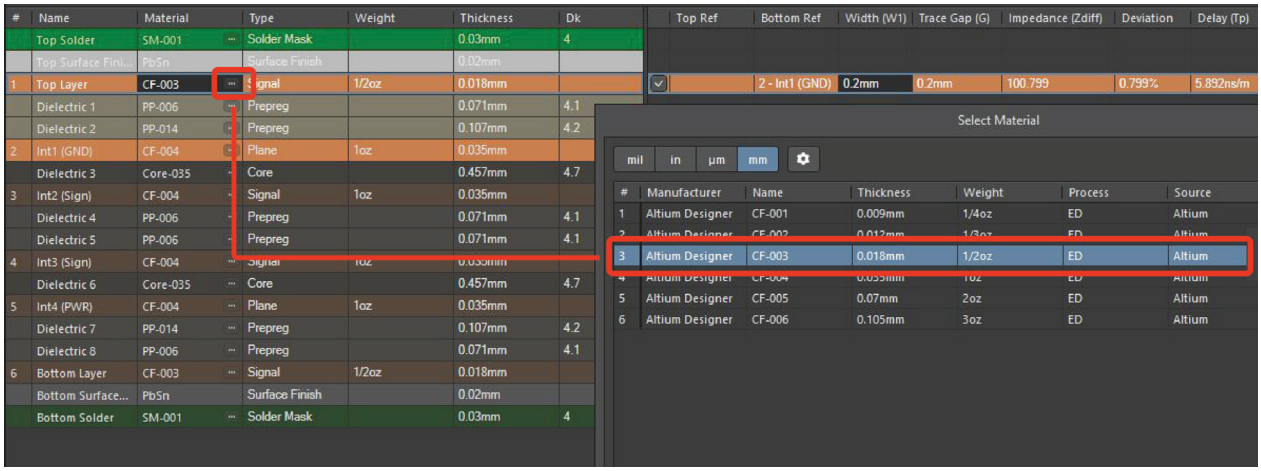
10. Change the thickness of the mask over the board **C1 = 0.03 mm**, over the conductor **C2 = 0.025 mm**. This parameter depends on the material and technological specifics of the PCB fabricator. As a rule, the mask thickness above the dielectric and above the conductor are different.

11. Set the gap between the differential pairs **G = 0.2 mm**.

IMPEDANCE CALCULATION

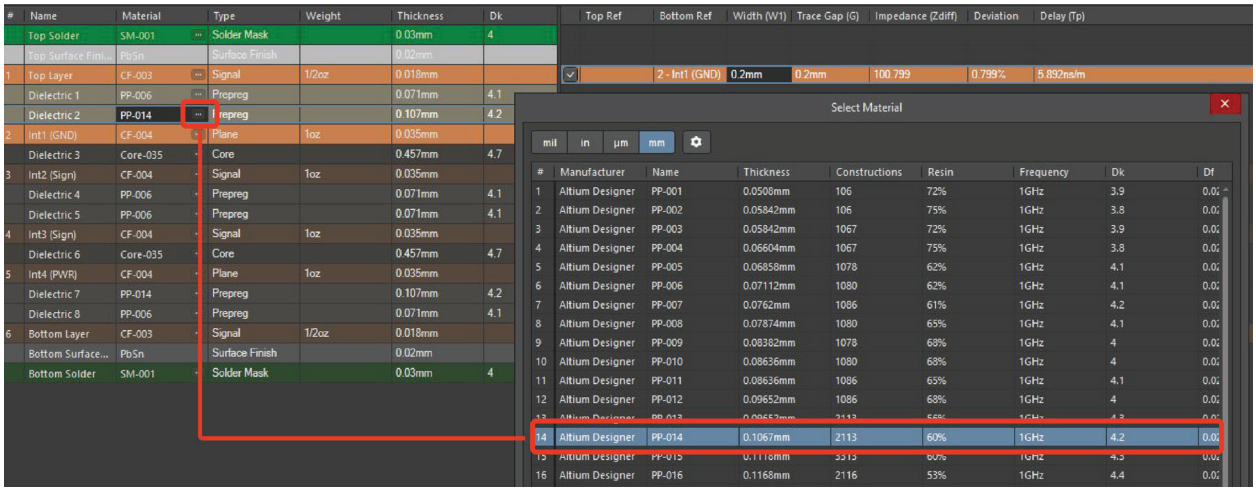


12. Select another material for the outer layer copper library by **½ oz** (**Thickness = 0.018mm**). Since the copper thickness affects the impedance, and total thickness of the outer layers of the metal layer will be added to the copper plus the surface finish, it is possible to change the copper thickness to adapt the calculation to a given impedance. If such a material is available from the fabricator.

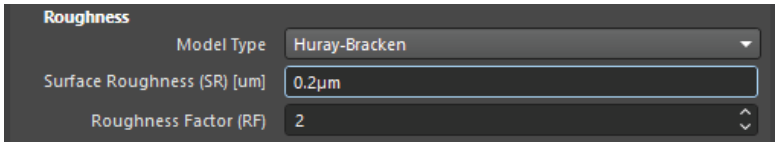


13. Change the prepreg for the Dielectric 2 layer to **PP-014** (Thickness=0.107 and Dk=4.2). Also note that the impedance calculation is more influenced by the dielectric thickness, the width of the conductor and, to a lesser extent, the Dk parameter. In this example, it is necessary to raise the impedance value, but we cannot change the width of the conductor. The simplest solution is to use a thicker dielectric. Of course, the engineer has to be in regular contact with the fabricator to assure that the material is available and that this solution is technologically feasible.

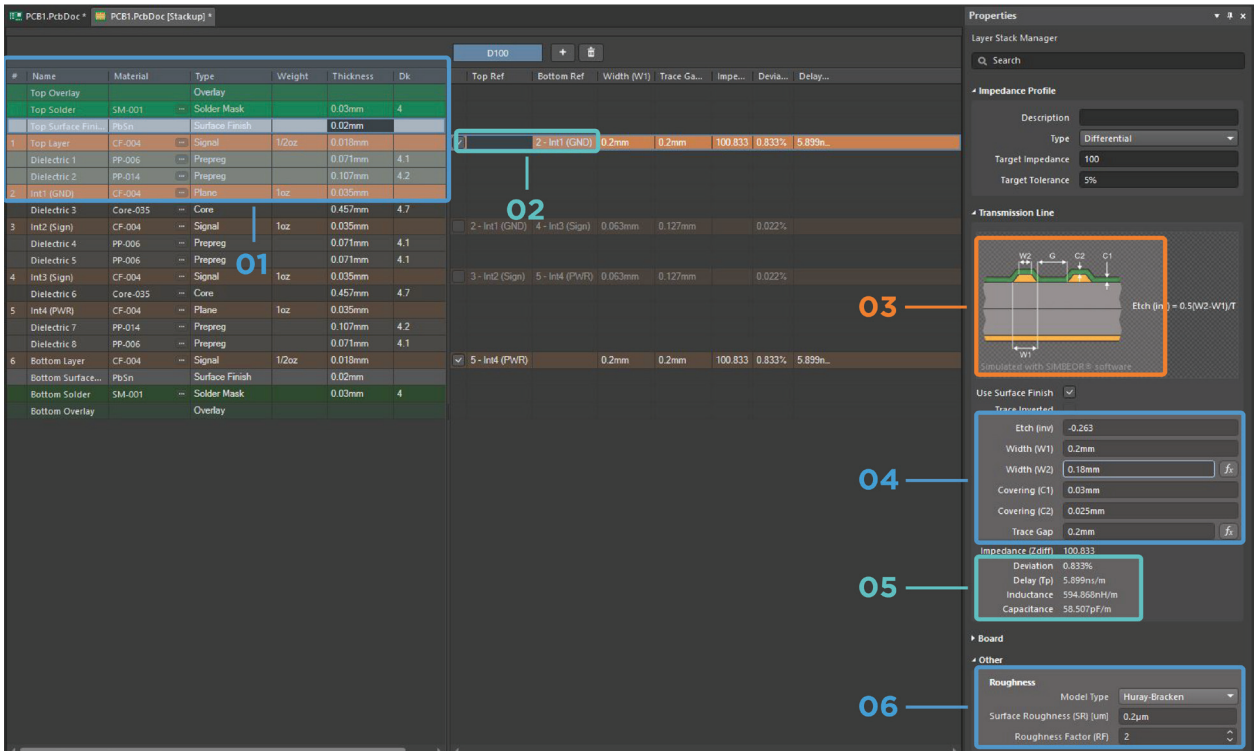
IMPEDANCE CALCULATION



14. In the **Properties** panel, in the **Other** section - specify the model and roughness parameters. Use the following as an example of the copper roughness setting Model Type - **Huray-Bracken**, **SR=0.2**, **RF=2**. This parameter is set for high-speed transmission lines.



15. View of the Impedance tab after all of the calculations have been changed

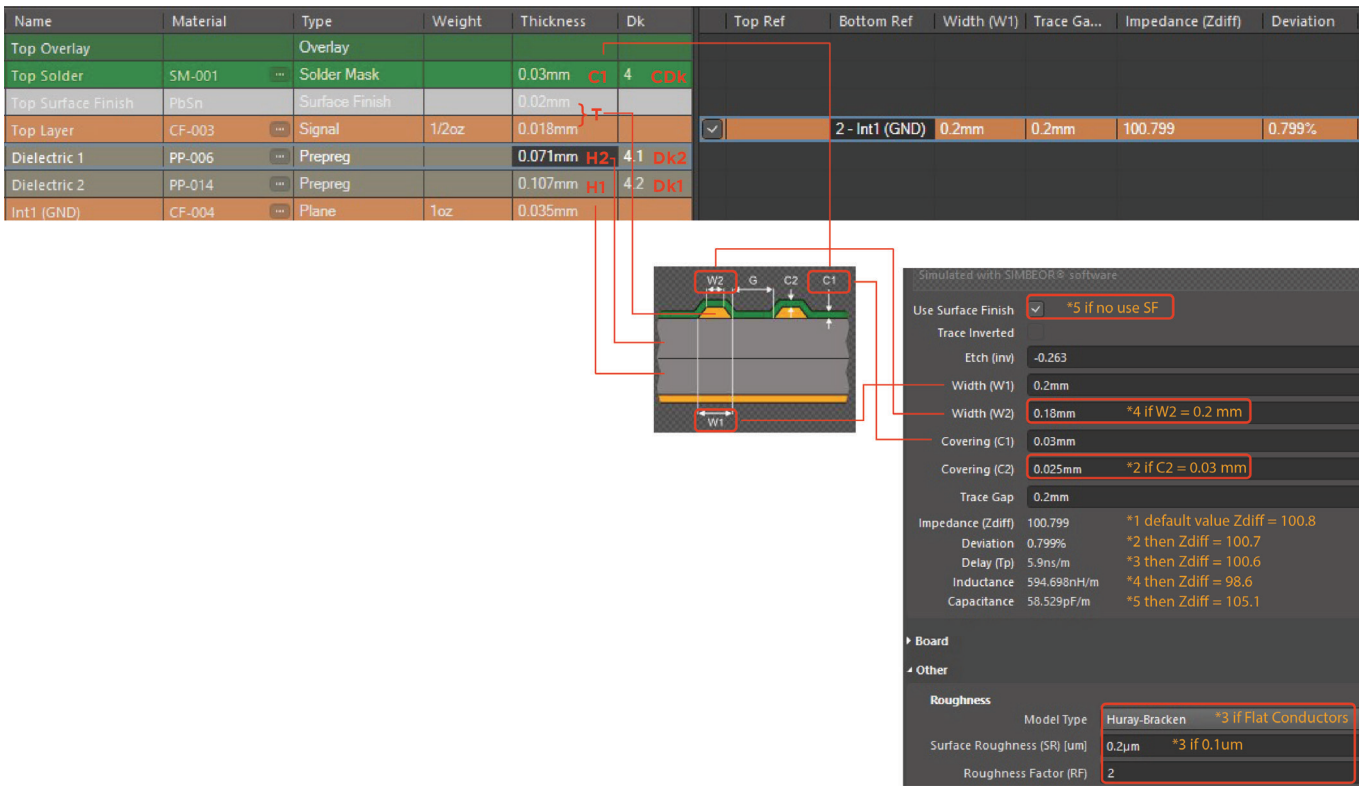


01 - Contents Transmission Line
02 - Reference layers
03 - Structure Transmission Line
04 - Advanced parameters for input
05 - Calculated parameters
06 - Roughness

IMPEDANCE CALCULATION

16. Final impedance calculation:

Use case	Description	Zdiff (calculated)
Basic use case	**Target Impedance value:** Zdiff = 100Ohm **Stackup parameters:** H1 = 0.107 mm, Dk1 = 4.2, H2 = 0.071 mm, Dk2 = 4.1, T=0.38 mm, W1 = 0.2 mm. W2 = 0.18 mm, Trace Gap (G) = 0.2 mm, Surface Finish = 0.02mm **Solder Mask :** C1 = 0.03 mm, C2 = 0.025 mm, CDk = 4 **Roughness:** Model Type-Huray-Bracken, SR=0.2µm, RF=2	Zdiff = 100.8 Ohm
Without solder mask differentiation	C1=C2=0.03 mm, other parameters without changes from basic case	Zdiff = 100.7 Ohm
Without the roughness of copper	If Model Type = Flat Conductors and SR=0.1µm, other parameters without changes from basic case	Zdiff = 100.6 Ohm
Without etching	If W2 = 0.2 mm, other parameters without changes from basic case	Zdiff = 98.6 Ohm
Without surface finish	Surface Finish checkbox disabled, other parameters without changes from basic case	Zdiff = 105.1 Ohm



IMPEDANCE CALCULATION

COMPARISON OF BASIC CALCULATORS BY FUNCTIONALITY

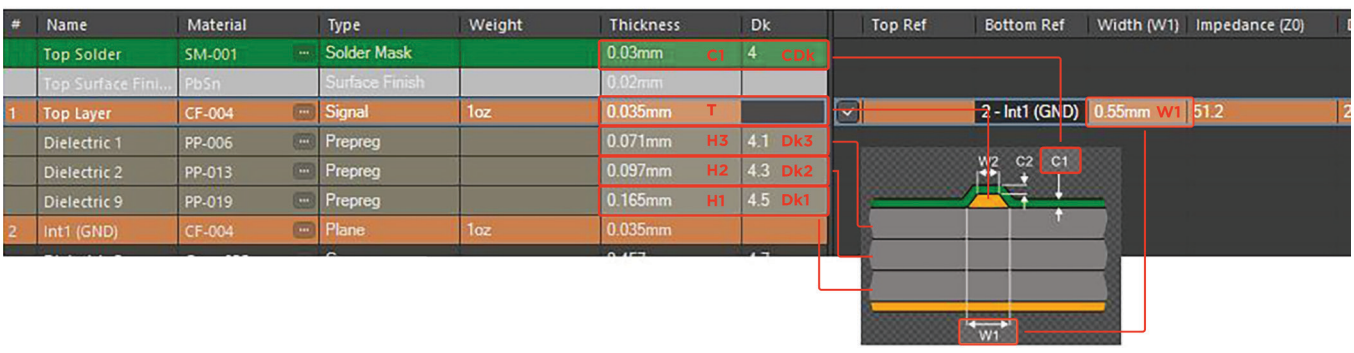
№	Feature name	Altium Designer	High End Calculator	Embedded in CAD PCB Calculator	Online Calculator
1	Basic transmission lines (Microstrip and Stripline)	✓	✓	✓	✓
2	Materials Library	✓	-	-	✓
3	Interrelation of stack-up and transmission line	✓	-	✓	-
4	Coplanar transmission line	✓	✓	-	-
5	Dielectrics with different thicknesses and different Dk	✓	✓	✓	-
6	Accounting for side etching of conductor	✓	✓	✓	-
7	Accounting of the mask thickness above the conductor and above the board	✓	✓	-	-
8	Selection of model and roughness parameters	✓	✓ (no model selection)	✓ (no model selection)	-
9	Cost	Free	$$$	$	Free
10	Delivery	Part of PCB Design tool	Separate software	Extra option	Web page

COMPARISON OF BASIC CALCULATORS IN CALCULATION ACCURACY

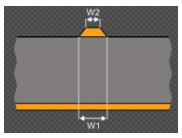
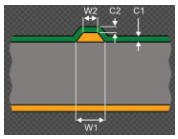
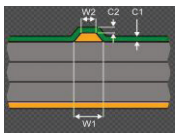
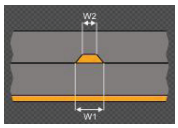
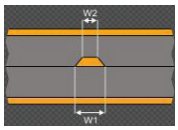
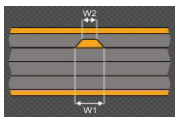
Input data:

- Model and roughness values for copper are not defined. The default model used in AD is Flat Conductors (SR=0.1µm, RF=2).
- T - Trace Thickness with Surface Finish (mm)
- H - Substrate Height (mm), 1,2,3... dielectric numeration from bottom to top in the transmission line
- Dk - dielectric constant, 1,2,3... numbering of corresponding dielectrics
- W1 - base trace width (mm)
- W2 - etched trace width (mm)
- C1 - mask thickness above the board (mm)
- C2 - mask thickness above the conductor (mm)
- G - Gap between lines in diff. pair (mm)
- S - clearance to the reference layer for coplanar lines (mm)

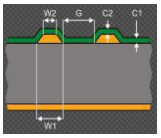
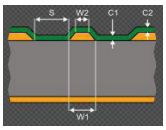
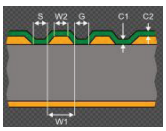
Example



IMPEDANCE CALCULATION

№	Transmission line type and source data	Altium Designer	High End Calc	Embedded in CAD PCB Calc	Online Calc
		Impedance value			
	Microstrip Single (Target Impedance = 50 Ohm)				
1	 T=0.055, H1=0.1, Dk1 = 4.6, W1=0.16, W2=0.14	50.34	49.35	49.7	49.52 without W2
	Microstrip with Soldermask (Target Impedance = 50 Ohm)				
2	 T=0.055, H1=0.1, Dk1=4.5, W1=0.14, W2=0.12, C1=0.03, C2=0.02, CDk=4	50.05	49.52	49 C1=C2= 0.03	48.46 without W2,C1, CDk
	Microstrip with Soldermask and few dielectrics (Target Impedance = 50 Ohm)				
3	 T = 0.055, H1 = 0.08, Dk1 = 4.2, H2 = 0.12, Dk2 = 4.4, H3 = 0.1, Dk3 = 4.6, W1 = 0.5, W2 = 0.48, C1 = 0.03, C2 = 0.02, CDk = 4	50.07	not supported	49.9 C1=C2= 0.03	not supported
	Embedded Microstrip (Target Impedance = 50 Ohm)				
4	 T = 0.035, H1 = 0.2, Dk1 = 4.2, H2 = 0.2, Dk2 = 4.2, W1 = 0.3, W2 = 0.28	49.85	49.74	49.4	48.68 without W2
	Symmetric Stripline (Target Impedance = 50 Ohm)				
5	 T = 0.035, H1 = 0.2, Dk1 = 4.4, H2 = 0.2, Dk2 = 4.4, W1 = 0.15, W2 = 0.13	50.11	50.80	49	47.89 without W2
	Asymmetric Stripline (Target Impedance = 50 Ohm)				
6	 T = 0.035, H1 = 0.18, Dk1 = 4, H2 = 0.16, Dk2 = 4.2, H3 = 0.14, Dk3 = 4.4, H4 = 0.12, Dk3 = 4.6, W1 = 0.13, W2 = 0.11	50.40	not supported	49.4	not supported

IMPEDANCE CALCULATION

№	Transmission line type and source data	Altium Designer	High End Calc	Embedded in CAD PCB Calc	Online Calc
		Impedance value			
	Edge-Coupled Microstrip with soldermask (Target Impedance = 100 Ohm)				
7	 T = 0.055, H1 = 0.2, Dk1 = 4, W1 = 0.14, W2 = 0.12, C1 = 0.03, C2 = 0.02, CDk = 4, G = 0.13	99.99	99.58	99.5 C1=C2= 0.03	105.74 without W2,C1, CDk
	Microstrip Coplanar (Target Impedance = 50 Ohm)				
8	 T = 0.055, H1 = 0.17, Dk1 = 4.5, W1 = 0.14, W2 = 0.12, C1 = 0.03, C2 = 0.02, CDk = 4, S = 0.1	50.07	50.24	not supported	not supported
	Edge-Coupled Microstrip Coplanar (Target Impedance = 100 Om)				
9	 T = 0.055, H1 = 0.5, Dk1= 4.3, W1 = 0.16, W2 = 0.14, C1 = 0.03, C2 = 0.02, CDk = 4, S = 0.1, G = 0.2	99.90	100.2	not supported	not supported

CONCLUSIONS

- To design high-speed and high-frequency devices modern CAD must support different transmission line structures. Unlike other programs, Altium Designer supports most of them.
- Parameters such as mask height, side etching of the conductor and roughness must be taken into account to calculate the impedance accurately. This is especially true for high-speed devices. Altium Designer allows you to do this to the fullest extent.
- The transmission line impedance is calculated as a part of the entire stack-up, which allows the engineer to see the whole picture.
- The wave impedance depends largely on the material. Parameters such as resin content, mesh density influence dielectric permeability and therefore impedance. The use of the material library allows you to select the necessary material quickly thereby reducing the time to design the structure of the transmission line structure design.
- When designing equipment with controlled impedance, calculations are used to predict the capacitance and inductance on which the impedance depends. The formulas used in Altium Designer are based on empirical dependencies and are unique for different configurations.

Simberian Electromagnetic Solutions

Impedances in Altium Designer are simulated with Simbeor® software.