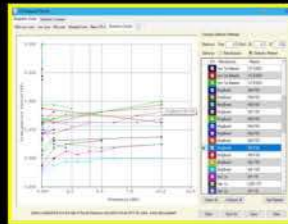
Simulation & Analysis Avoid Costly Re-spins

Now just $5,000*

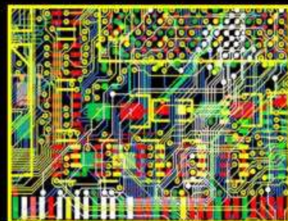
Analyzed by Design007 Columnist Barry Olney

A Comprehensive Report Includes:

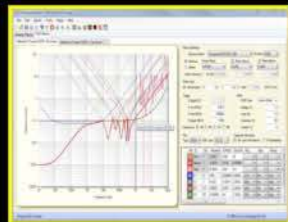


Material Selection for Cost/Performance to Required Frequency and Bandwidth, Design Constraint Review

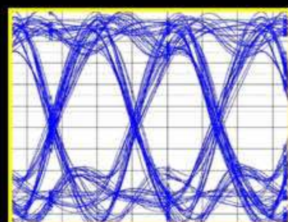
Stackup Impedance Analysis, Single-ended, Differential Pairs and CPW Blind and Buried Via Definition, Reference Plane Assignment Validation



Critical Placement and Routing, Plane Pour Definitions, Return Current Paths, Plane Cross-overs and Broadside Coupling Review



PDN Analysis - Minimizes AC Impedance, Decap Selection, Mounting Inductance Analysis, Plane Resonance Dampening



Critical Net Simulation, EMC Analysis to FCC Class B, Timing Measurement, Termination Review, Crosstalk Analysis

Ensure your next design works!

Discover More

iCD

In-Circuit Design Pty Ltd

** conditions apply*

www.icd.com.au

2D Field Solver—An Essential Tool for High-speed PCB Design

Beyond Design

Feature Column by Barry Olney, IN-CIRCUIT DESIGN PTY LTD / AUSTRALIA

A field solver is the engine behind signal and power integrity analysis. You never see it, but it performs all the magic of simulation. In its elementary form, the field solver can employ Maxwell's equations to calculate the parasitic elements of a solution space. This method is referred to as 2D extraction and is used to analyze and synthesize a stackup to achieve a target single-ended or differential impedance. The velocity of propagation can also be extracted to perform signal integrity analysis. A field solver can be used as a stand-alone tool or as part of a simulation environment. In this month's column, I will look at 2D field solvers.

An electromagnetic (EM) field is produced when a logic driver delivers a high-speed, fast rise-time pulse into a transmission line. The EM wave propagates along the length of the trace radiating into the surrounding dielectric material and coupling energy to nearby trace segments. These EM fields are not restricted to the multilayer substrate and if proper care is not taken may emit radiation causing electromagnetic interference. If the impedance of the driver and load do not match the transmission line, then reflections occur, and radiation is created. Hence, the need for impedance control.

iCD Design Integrity | Field Solver Technology

File Edit Export Import Tools Setup Help

Stackup Planner | PDN Planner | CPW Planner

2 Layer | 4 Layer | 6 Layer | 8 Layer | 10 Layer | 12 Layer | 14 Layer | 16 Layer | 18 Layer | 12 Layer - Isola 370HR

UNITS: mil — 6/1/2022 — Total Board Thickness: 62.74 mil

Differential Pairs > 50/100 Digital | 40/80 DDR3 | 90 USB

Layer No.	Via Span & Hole Diameter	Description	Layer Name	Material Type	Dielectric Constant	Dielectric Thickness	Copper Thickness	Trace Clearance	Trace Width	Current (Amps)	Characteristic Impedance (Zo)	Edge Coupled Differential (Zdiff)	Broadside Coupled Differential (Zdbs)
		Soldermask		PSR-4000BV / CA-40BB LPI (1GHz)	3.3	0.5							
1	8 4 8	Signal	Top	Conductive			2.17	13.2	4	0.43	51.96	99.99	
		Prepreg		370HR; 1080; Rc= 64% (10GHz)	3.76	2.8							
2		Plane	GND	Conductive			1.4						
		Core		370HR; 1-106BC; Rc=71% (10GHz)	3.6	2							
3		Plane	VDD	Conductive			1.4						
		Prepreg		370HR; 1080; Rc= 71% (10GHz)	3.63	3.5							
		Prepreg		370HR; 1080; Rc= 66% (10GHz)	3.72	2.9							
4		Signal	Signal	Conductive			0.7	13.2	5.12	0.23	50.38	99.31	
		Core		370HR; 1080/2113; Rc=56% (10GHz)	3.9	6							
5		Plane	GND	Conductive			1.4						
		Prepreg		370HR; 1652; Rc= 48% (10GHz)	4.09	5.3							
6		Signal	Inner 6	Conductive			1.4	17.72	4	0.31	49.51	98.09	72.12
		Core		370HR; 2-1080; Rc=58% (10GHz)	3.9	5							
7		Signal	Inner 7	Conductive			1.4	17.72	4	0.31	49.51	98.09	72.12
		Prepreg		370HR; 1652; Rc= 48% (10GHz)	4.09	5.3							
8		Plane	GND	Conductive			1.4						
		Core		370HR; 1080/2113; Rc=56% (10GHz)	3.9	6							
9		Signal	Signal	Conductive			0.7	13.2	5.12	0.23	50.38	99.31	
		Prepreg		370HR; 1080; Rc= 66% (10GHz)	3.72	2.9							
		Prepreg		370HR; 1080; Rc= 71% (10GHz)	3.63	3.5							
10		Plane	VCC	Conductive			1.4						
		Core		370HR; 1-106BC; Rc=71% (10GHz)	3.6	2							
11		Plane	GND	Conductive			1.4						
		Prepreg		370HR; 1080; Rc= 64% (10GHz)	3.76	2.8							
12		Signal	Bottom	Conductive			2.17	13.2	4	0.43	51.96	99.99	
		Soldermask		PSR-4000BV / CA-40BB LPI (1GHz)	3.3	0.5							

Design Rule Checker: — © 2022 In-Circuit Design Pty Ltd

Figure 1: A 2D boundary element method field solver. (Source: iCD Stackup Planner)

In a uniform cross-section transmission line, a propagating signal sees the same instantaneous impedance all the way down the line. There is one value of the instantaneous impedance (characteristic impedance) that distinguishes the transmission line. It depends only on the cross-section of the transmission line and the surrounding material properties. 3D field solvers are not more accurate than 2D field solvers in stackup design. When interconnects are uniform transmission lines, a 2D field solver can be more accurate, faster, and much easier to use.

When interconnects are uniform transmission lines, a 2D field solver can be more accurate, faster, and much easier to use.

Electromagnetic behavior is governed by Maxwell's equations, and all parasitic extraction requires solving some form of these equations. There are numerous types of field solvers but for 2D field solvers, the appropriate form of Maxwell's equations is typically solved by one of two classes of methods.

The first uses a differential form of the governing equations and requires the meshing of the entire domain in which the electromagnetic fields reside. Two of the most common approaches in this first class are the finite difference (FD) and finite element (FEM) methods. The resultant linear matrix that must be solved is large but sparse.

The second class of methods is integral equation methods which instead require an evaluation of only the sources of the electromagnetic field. Those sources can be physical quantities, such as the surface charge density or mathematical abstractions resulting from the application of Green's theorem. When the sources exist only on two dimensional surfaces for three dimensional problems, the method is often called method of moments (MoM), or boundary element method (BEM), as illustrated in Figure 1. For open boundary conditions, the sources of the field exist in a much smaller domain than the fields themselves, and thus the size of the matrix generated by the integral equations methods is much smaller than FEM. Since the BEM results in a smaller solution space, it is considered faster than other 2D solvers.

The most effective planning tool for optimizing the stackup of a PCB is a 2D field solver. This tool can be used to predict the characteristic impedance and differential impedance for all topologies, including microstrip, stripline, and dual stripline. The other advantage of a 2D field solver is its ability to include second-order effects such as trace thickness, the influence of solder mask, and mixed prepreg dielectrics. These solvers are limited to the numerical accuracy of solving Maxwell's equations, which generally are accurate to better than 1%. Therefore a 2D field solver should be used as your design tool of choice.

Accuracy is important because of yield. Any inaccuracy in the predicted impedance will shift the center position of the distribution of the PCB fabricators' target impedance. Customers usually request ±10% tolerance so, with an accuracy of 1%, the field solver is the least of the fabricators' worries. The absolute accuracy of the manufactured product can be evaluated by comparison to measurements on well-characterized test vehicles as well as on production boards. There are many steps to get the fabrication process right, and the field solver is one of the most reliable links in the chain. For instance, typical results of better than 3% tolerance (including fabrication variables) are achieved by iCD customers (Figure 2).

The field solver is very accurate but if one incorrectly models the structures or incor-

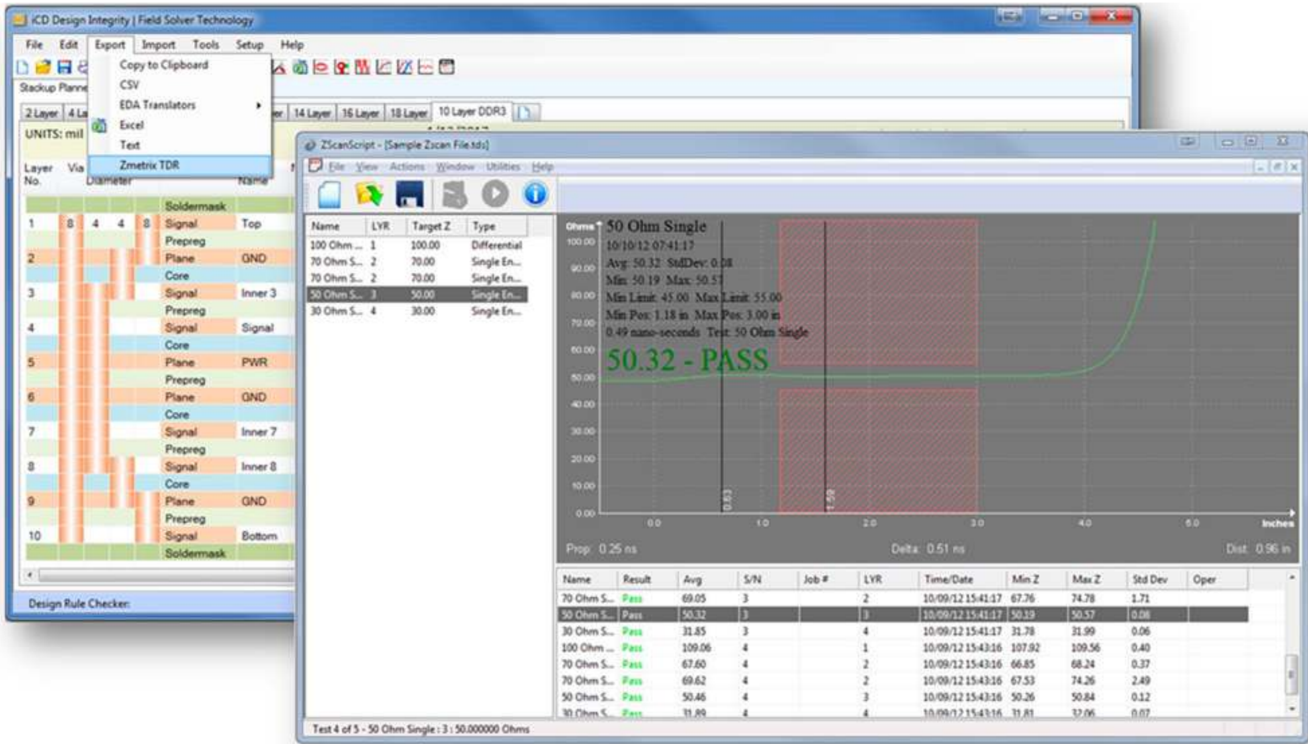


Figure 2: The iCD Stackup Planner coupled with the ZMetrix TDR and ZScan software.

rectly enters the dielectric constant or other properties, then the field solver is hardly to blame when you get erroneous results. Using the exact materials that are stocked by your preferred fabricator can increase accuracy by up to 5%. That is why the availability of a comprehensive dielectric materials library and dielectric material selection is so important.

Years ago, when clock frequencies were low and signal rise times were slow, selecting a dielectric material for your PCB was not difficult—we all just used FR-4. We didn't really care about the properties of the materials. However, with today's multi-gigabit designs, with extremely fast rise times and tight margins, precise material selection is crucial to the performance of the product. Materials used for the fabrication of the multilayer PCB absorb high frequencies and reduce edge rates. That loss in the transmission lines is a major cause of signal integrity issues. But we are not all designing cutting-edge boards and sometimes we tend to over-specify requirements that can lead to inflated production costs.

Over the years, a huge range of materials has been developed for multilayer PCB fabrication. In fact, to give you an idea, iCD now has a choice of over 700 series of dielectric materials from over 60 different manufacturers, in its dielectric materials library. When each material is used for the right target application, the resultant PCB will have the lowest possible cost while still satisfying the design and performance goals of the project. Selecting the best material for an application is often a daunting task. One needs to quickly sort through the vast array of choices to make an informed decision.

Figure 3 plots the loss properties of dielectric materials from a fabricator. One can easily see which materials are best for high-speed applications and can choose a few materials from this that are in stock. Cost-to-performance evaluations must still be done to ensure the lowest cost material that will do the job is

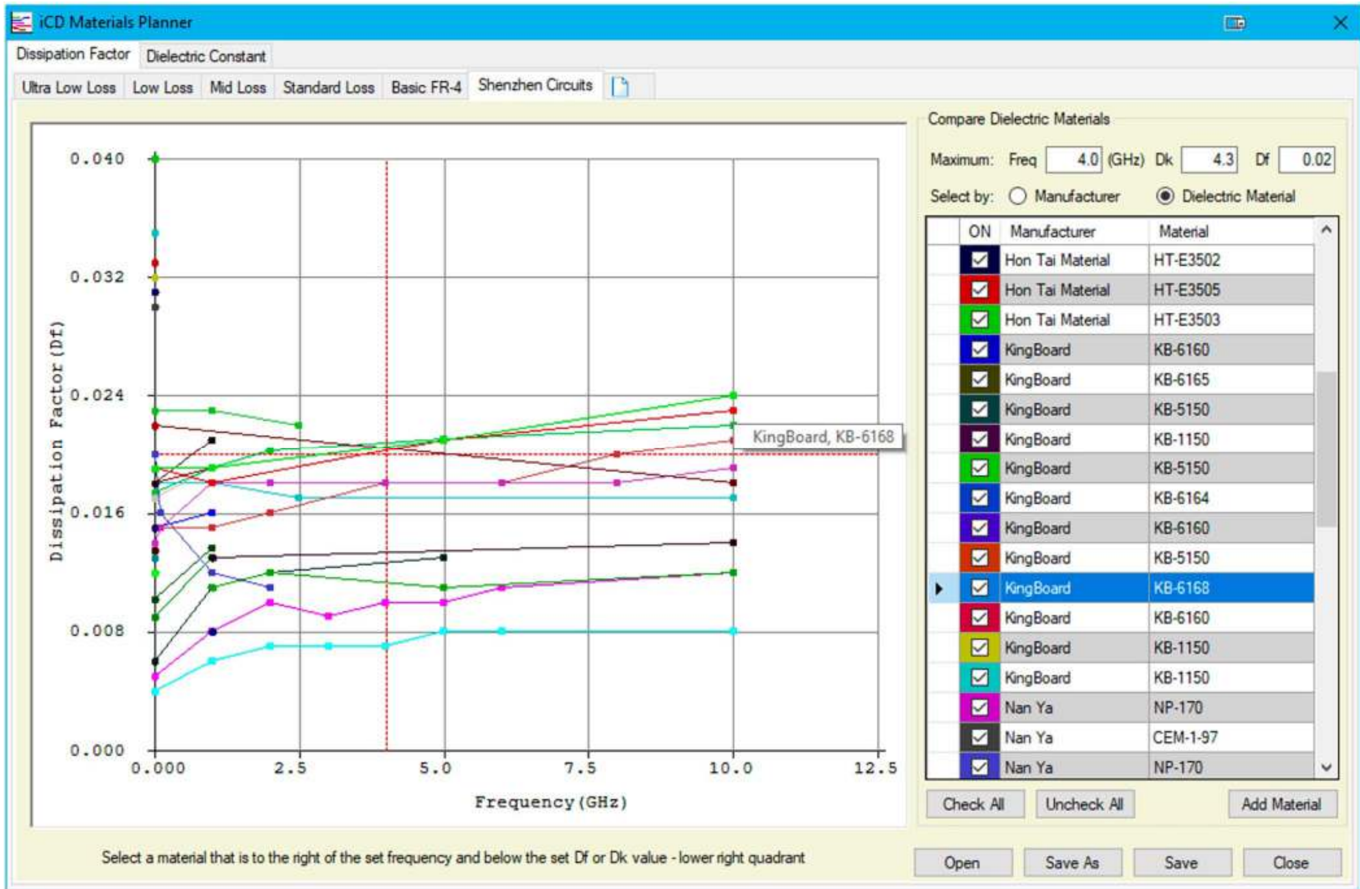


Figure 3: Example of a fabricator's dielectric materials loss profile. (Source: iCD Materials Planner)

selected. Keep in mind that material costs vary with quantity.

Matching material performance numbers of the dielectric constant (Dk) is also important. A small difference in this value between materials can impact impedance, line widths/clearances, and thus losses significantly. Also, the dielectric constant of a material determines the propagation speed of the signal in the medium. So, if Dk values vary on different layers of the substrate, then bus timing may also be an issue. One should consider construction options that allow a drop-in material that matches the impedance for each mirrored layer of the stackup.

As signal speeds get up into the gigahertz range, the 2D field solver is an essential tool that every PCB designer must employ to maintain design integrity and performance. Impedance is the foundation of the system performance and reliability, so using a 2D field solver to get it right is not an option.

Key Points

- Field solvers employ Maxwell's equations to calculate the parasitic elements of a solution space.
- EM fields are not restricted to the multilayer substrate and if proper care is not taken may emit radiation causing electromagnetic interference.
- In a uniform cross-section transmission line, a propagating signal sees the same instantaneous impedance all the way down the line.
- Since the 2D BEM solver results in a smaller solution space it is considered faster than other 2D solvers.
- Filed solvers are limited to the numerical accuracy of solving Maxwell's equations,

which generally are accurate to better than 1%.
- There are many steps to get the fabrication process right, and the field solver is one of the most reliable links in the chain.
- Using the exact materials that are stocked by your preferred fabricator can increase accuracy by up to 5%.
- Materials, used for the fabrication of the multilayer PCB, absorb high frequencies, and reduce edge rates; that loss in the transmission lines is a major cause of signal integrity issues.
- When each material is used for the right target application, the resultant PCB will have the lowest possible cost while still satisfying the design and performance goals of the project.
- If dielectric constant values vary on different layers of the substrate then bus timing may also be an issue. DESIGN007

References

1. Beyond Design: "Predicting and Measuring Impedance," "It's a Material World," "Stackup Planning Parts 1-6"
2. "Signal and Power Integrity—Simplified," by Eric Bogatin
3. Wikipedia: Electromagnetic field solvers.

Barry Olney is managing director of In-Circuit Design Pty Ltd (iCD), Australia, a PCB design service bureau that specializes in board-level simulation. The company developed the iCD Design Integrity software incorporating the iCD Stackup, PDN, and CPW Planner. The software can be downloaded at www.icd.com.au. To read past columns, click here.

IPC Issues Call for Participation in IPC E-Textiles 2023

IPC is now accepting abstracts for IPC E-Textiles 2023, the international conference for the e-textiles industry, to be held live and in-person on Monday, January 23, 2023, in San Diego, California, in conjunction with IPC APEX EXPO 2023, the industry's premier conference and exhibition for the electronics industry.

IPC E-Textiles 2023 provides an excellent opportunity for the e-textiles industry to learn, network, and collaborate on this vastly expanding technology space. By aligning IPC E-Textiles 2023 with IPC APEX EXPO, speakers and attendees will also have the opportunity to participate in IPC E-Textiles Committee standards meetings and take advantage of all events offered during IPC APEX EXPO.

IPC invites innovators and technologists from the e-textiles supply chain and academia to submit an abstract for a technical presentation during the conference. The IPC E-Textiles 2023 Program Committee seeks abstracts in the following topic areas:

- E-Textiles Success Stories
- Reliability and Standards
- Economic and Business Aspects
- Futuristic Thinking

A 250-word technical conference abstract should describe significant results from experiments and case studies, emphasize new techniques, discuss trends of interest, and contain appropriate technical test results.

Technical conference paper abstracts are due August 1, 2022. To submit an abstract or course proposal, visit IPC's website.

IPC also announces the IPC E-Textiles 2023 Program Committee, which will shape the technical deliverables and format for this exciting and educational event:

- Vladan Koncar, Ph.D., University of Lille, Ecole Nationale Supérieure des Arts et Industries Textiles (ENSAIT)
- Pratyush Rai, Ph.D., Nanowear, Inc.
- Sahar Rostami, Myant
- Yolita Wildman Nugent, Betazip, LLC

(Source: IPC)