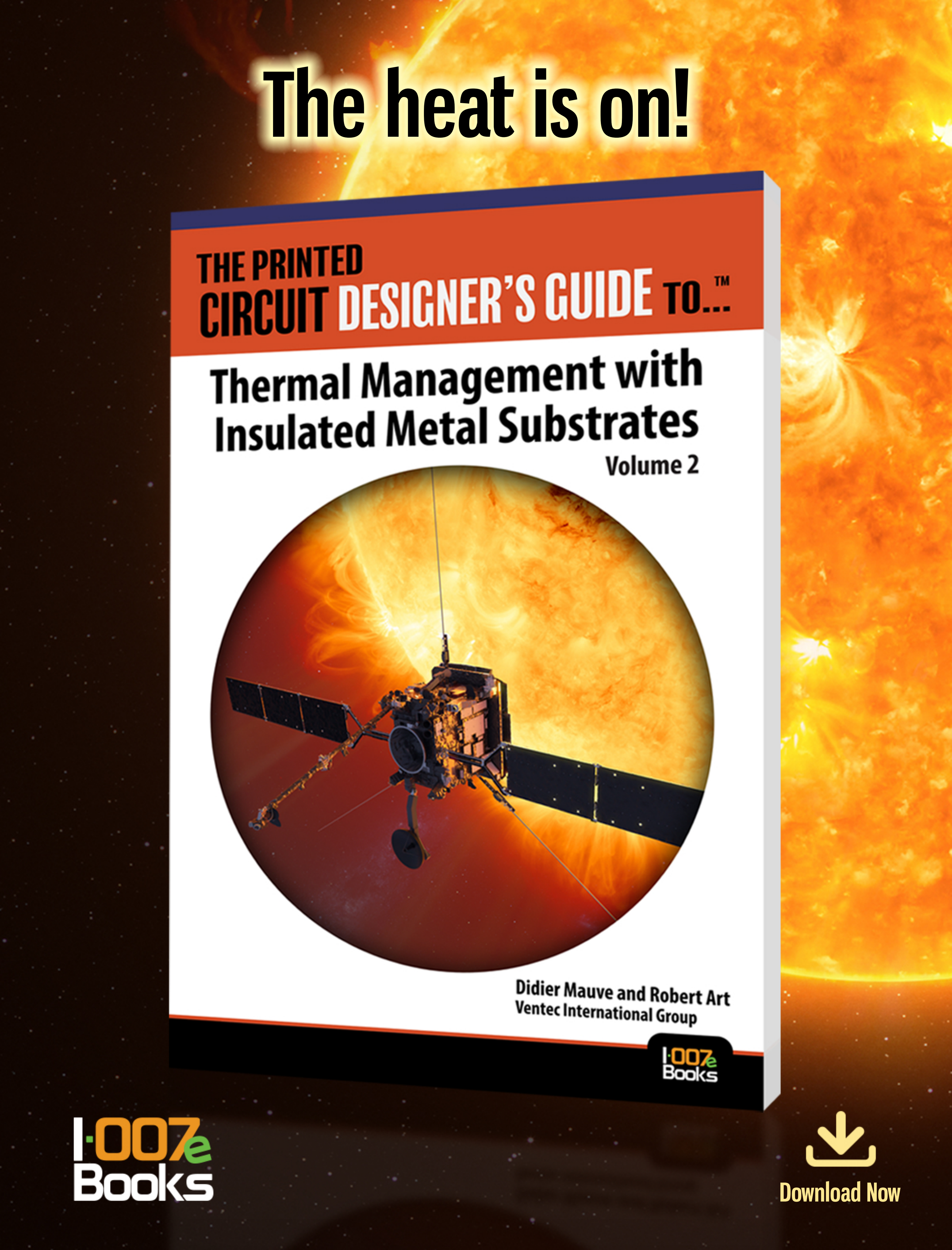
I-Connect007

JULY 2022

DESIGN007 MAGAZINE



The heat is on!
THE PRINTED CIRCUIT DESIGNER'S GUIDE TO...™
Thermal Management with Insulated Metal Substrates
Volume 2
Didier Mauve and Robert Art
Ventec International Group
I-007e Books
I-007e Books
Download Now

DESIGN007 MAGAZINE

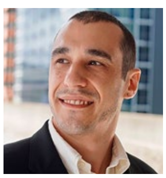
Field Solvers

Thanks to the “left shift” of signal integrity functionality too early in the design cycle, more designers are using field solvers than ever before. But even experienced SI engineers risk “garbage in, garbage out” if they input the wrong data. This month, we focus on the proper use of field solvers and how to avoid getting hurt by GIGO.

FEATURE INTERVIEWS & ARTICLES

10 **Surveying the Land of Field Solvers**
Interview with Todd Westerhoff and Bill Hargin

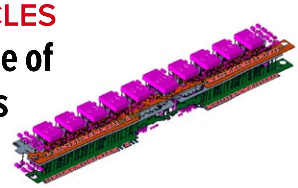
20 **The Great Divide in PCB Simulation Software**
by Zach Peterson



34 **Cadence Provides ‘Clarity’ in Design Tool**
Interview with Brad Griffin

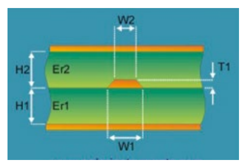
FEATURE ARTICLES

52 **The Practical Side of Using EM Solvers**
by Heidi Barnes

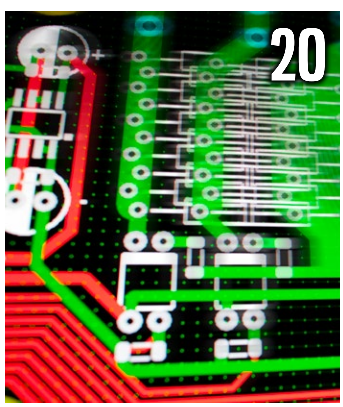


FEATURE COLUMNS

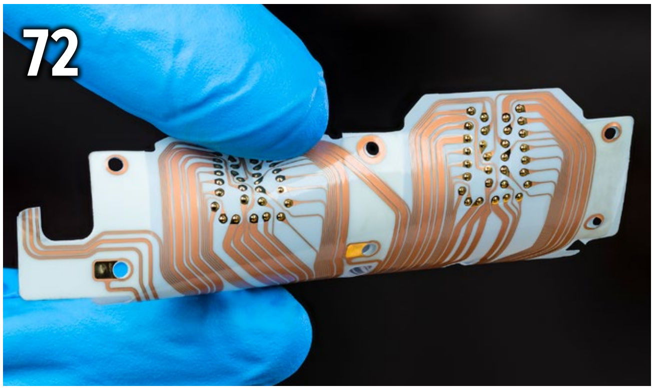
26 **Field Solver Finesse for Modelling Transmission Lines**
by Martyn Gaudion



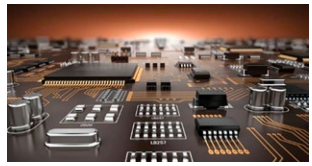
40 **2D Field Solver—An Essential Tool for High-speed PCB Design**
by Barry Olney



DESIGN007 MAGAZINE



ARTICLES



58 **Final Finishes—ENIG and ENIPIG**
by Anaya Vardya

72 **The Importance of Rigid-Flex PCB Design Guidelines**
by Cody Stetzel

SHORTS

24 **BOOK EXCERPT: *The Printed Circuit Designer's Guide to... Thermal Management with Insulated Metal Substrates, Volume 2***

39 **Infographic: EV Battery Critical Materials**

45 **IPC Issues Call for Participation in IPC E-Textiles 2023**

61 **University of Illinois Researchers Create Low-cost, High-yield Plastic Microprocessors**

65 **All Systems Go: Accelerate Your PCB Designs with Machine Learning**

COLUMNS

8 **With Field Solvers, GIGO Hurts**
by Andy Shaughnessy

46 **Supply Chain Resilience, Part Two: The Solution**
by Stephen V. Chavez

62 **Can Solvent-free UV-Cure Coatings Increase Stability and Throughput?**
by Saskia Hogan

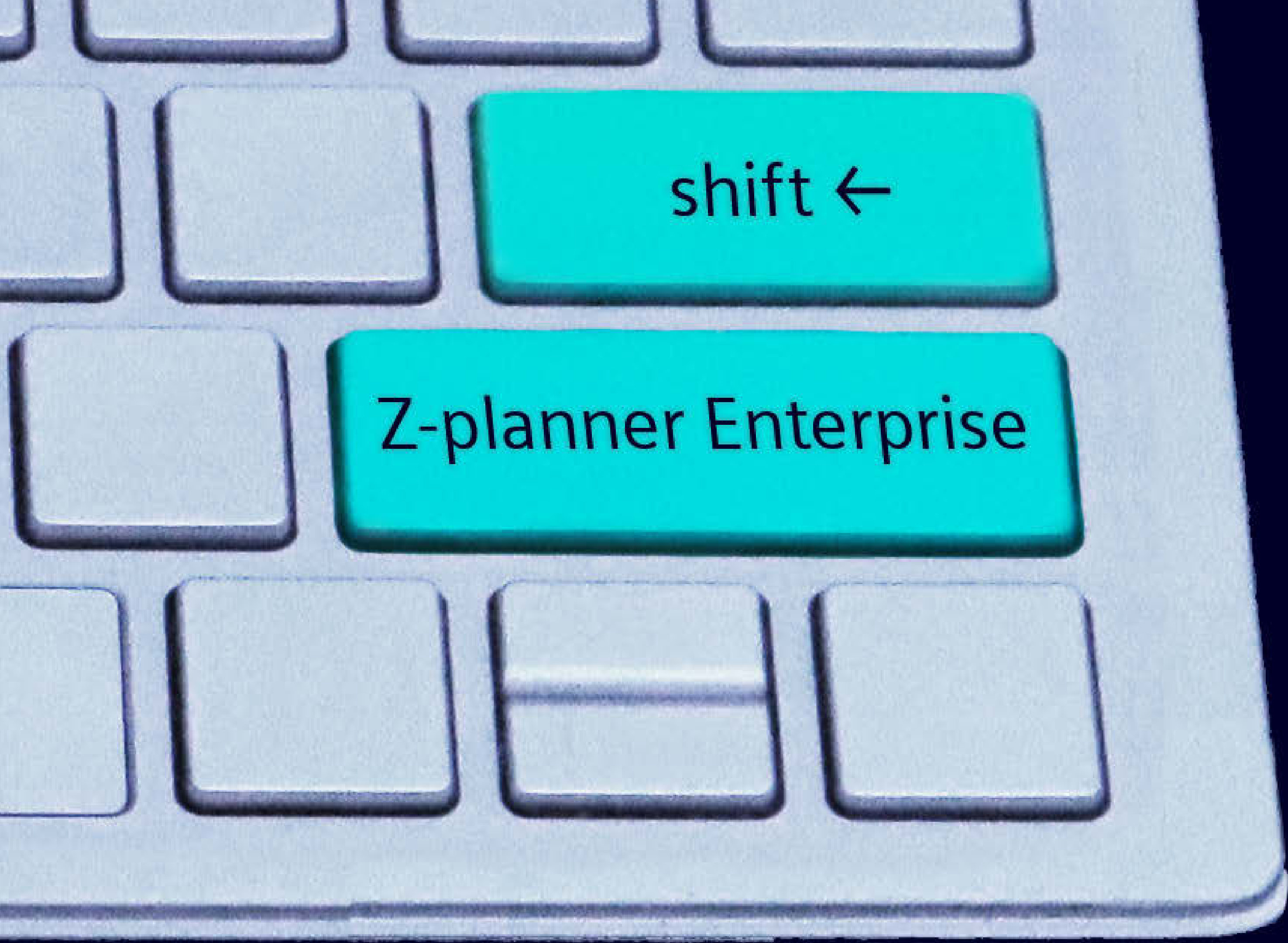
66 **Manufacturing Documentation: Keep the Builder in Mind**
by Tim Haag

DEPARTMENTS

81 **Career Opportunities**
94 **Educational Resource Center**
95 **Advertiser Index & Masthead**

HIGHLIGHTS

32 **PCB007**
56 **MilAero007**
70 **Flex007**
78 **Top Ten Editor's Picks**


shift ←
Z-planner Enterprise

With Field Solvers, GIGO Hurts

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

The "left shift" concept has been underway for at least five years, as EDA tool providers offer more powerful functionality earlier in the stages of PCB design and layout. This month, we focus on one tool that's been shifting leftward for some time now: the field solver.

Once the domain of the experienced SI engineer, these computational tools—or at least some of their functions—are now available in PCB design software tools. With improved user interfaces, field solvers are being used by PCB designers and design engineers, many of whom have not mastered electromagnetics theory.

Electromagnetic theory all starts with James Clark Maxwell, whose four differential equations form the framework for understanding e-mag theory. Field solvers are complicated programs that solve a subset of Maxwell's equations. Field solvers come in 2D, 2.5D, and 3D formats, and use several methods, such as the finite element method (FEM), finite difference (FD), and boundary element method (BEM) to extract parasitic circuit models.

When we first started planning this issue on field solvers, one thing became clear right away: Field solvers can be fairly difficult tools to use, even for veteran SI specialists, and it's

even tougher to make sure that you're using the correct data so that you get the results that you need. Signal integrity experts we spoke with said that even experienced engineers could wind up with "garbage in, garbage out" (GIGO) if they aren't careful.

There are hundreds of ways to go wrong with a field solver. As we see in this issue, some cases of GIGO derive from plugging in the wrong data, such as using IPC spec sheet info about board thickness instead of the "as pressed" number from your fabricator. Other designs have gone awry when field solver users didn't account for Dk changing with temperature. We heard a chorus of engineers saying, "Don't trust datasheets."

This particular left-shift is happening for a reason: There just aren't enough full-time signal integrity experts with advanced degrees and decades of experience in the industry. And if you can find them, they (rightly) charge a lot of money for their services. There are far more PCB designers than there are SI experts, so left-shifting computational horsepower into the hands of the PCB designer is a great way to mitigate this shortage of SI gurus.

But for a non-degreed PCB designer, using a field solver might feel like drinking from a firehose.

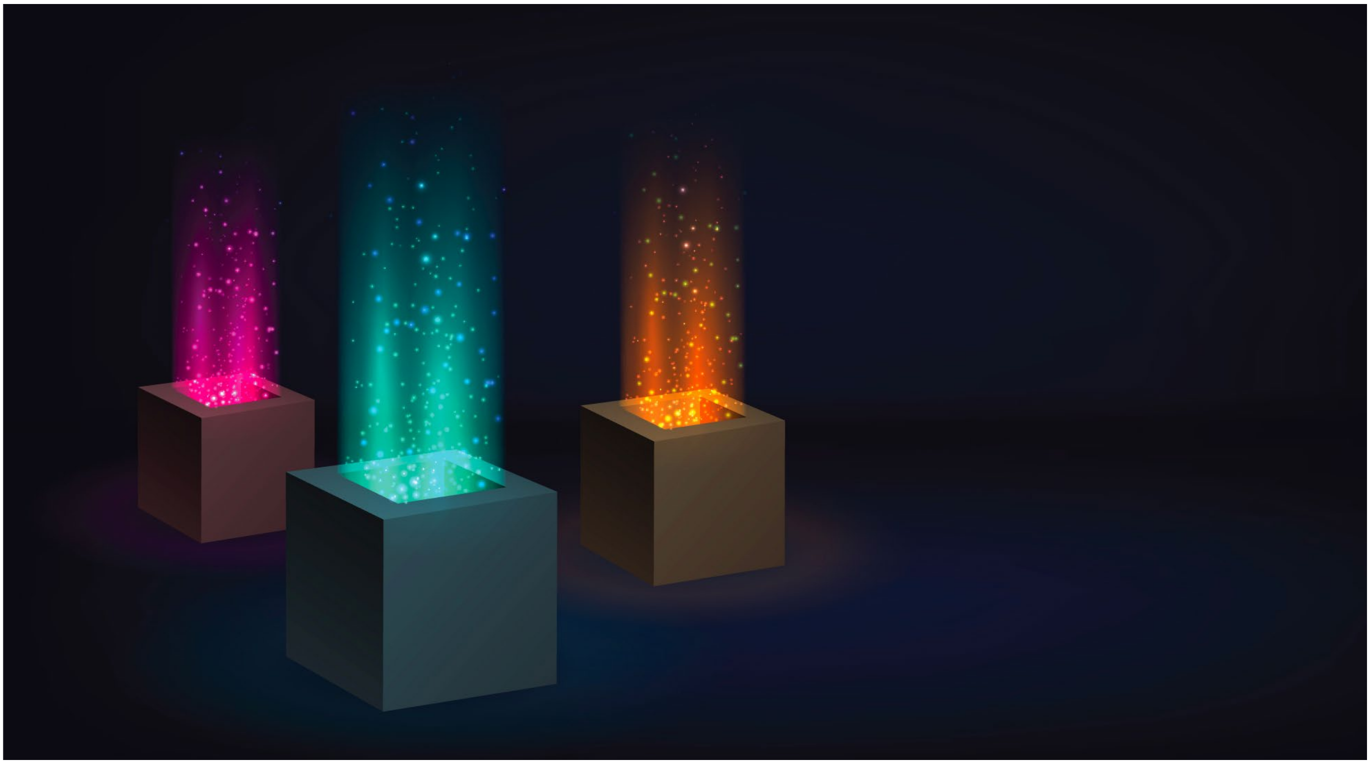
Design tool companies would love to develop a field solver with "PhD" (Push here, dummy) functionality. But we're not there yet. You still must know something about Maxwell's equations, or you're going to have problems.

So, for this issue, we asked our expert contributors to share their thoughts on field solvers, and what new users need to know to get up to speed. Todd Westerhoff and Bill Hargin discuss the evolution of field solvers and offer some handy tips and techniques for new users. Zach Peterson focuses on the divide between typical field solvers and layout software, and the need for the two to converge into one environment. Columnist Martyn Gaudion details a bit of "field solver finesse" for the modeling of transmission lines. Brad Griffin and Nolan Johnson explore Cadence's Clarity field solver functions, and columnist Barry Olney digs deep into cutting-edge techniques for the 2D field solver. Heidi Barnes takes a look at "the practical side" of field solvers and offers tips for avoiding GIGO.

We also bring you columns from Stephen Chavez, Tim Haag, and brand-new Electrolube columnist Saskia Hogan. We have an article by Cody Stetzel on flex guidelines, and another installment in Anaya Vardya's continuing DFM101 series.

It's summertime, but it doesn't look like this industry is taking much of a vacation! See you next month. **DESIGN007**

Andy Shaughnessy is managing editor of *Design007 Magazine.* He has been covering PCB design for 20 years. To read past columns, click here.



Surveying the Land of Field Solvers

Feature Interview by the I-Connect007 Editorial Team

Electromagnetic field solvers have traditionally been used by a small slice of engineers—the full-time signal integrity experts—and their use has been limited to the most demanding designs. But as design speeds increase, problems requiring field solvers are becoming more mainstream and field solvers are popping up all over the place, including free or inexpensive solvers available through an internet search. Many companies are claiming that their field solvers can be used by hardware design engineers without the SI background that solvers once required.

We asked Todd Westerhoff, product marketing manager for Siemens EDA, and Bill Hargin, CEO of Z-zero, to cut through the fog of field solvers and explain how engineers can ensure they avoid a "garbage in, garbage out" (GIGO) scenario when using field solvers to analyze their designs.

Andy Shaughnessy: Let's start by defining a field solver. What exactly does a field solver do?

Bill Hargin: I think the simplest answer, Andy, is that a field solver is a black box that converts mechanical design parameters and material properties into a simulatable electrical model using Maxwell's equations. You give a field solver a precise dimensional description of a structure you plan to build, along with the materials it will be built from, and it gives you back a model that you can either examine directly or use as part of a system simulation.

There are two broad classes of field solvers used in PCB design: 2D solvers, which are used to predict transmission line (signal trace) behavior under specific conditions, and 3D solvers, which are more complicated to use, but provide accurate answers under gen-

Bill Hargin

eral-case conditions. 2D solvers have become mature enough that designers performing simulations may be using them without realizing it; they're embedded parts of simulation processes in some tools.

Todd Westerhoff: Let's take a step back. Why do we simulate our designs? Why do we simulate anything?

Hargin: So that we can predict the electrical behavior of our designs—the performance of our designs in advance.

Westerhoff: Precisely. We use simulation to predict whether something will work before we build it, so that we can debug and optimize our designs before we commit them to prototype fab. There's really only one question, or two, that we ever care about: Will it pass or fail against requirements, and by how much? The "by how much" part is operating margin and it's critically important. No simulation is ever 100% accurate; no matter how careful we are, we can't model and predict everything. Every simulation and every simulation result has a margin of error associated with it. When we simulate, we're looking to prove that the operating margin at the system level is substantially more than the margin of error in the simulation we performed. That's when we can say the design will work with room to spare.

When we talk about field solvers, we need to remember it's the margin at the system level that we care about. We need to model the structure in question (a transmission line, a package breakout, a differential via, etc.) as accurately as required to ensure adequate margin at the system level. But we don't want to model things more accurately than required, because that takes additional time and effort that isn't really moving the design forward; it's just analytical overkill.

There's a huge temptation to model everything as accurately as possible because we're engineers and we want to be precise—the hard part is knowing when that extra accuracy is required and spending our time judiciously. That's a hard call when we're dealing with field solvers. Eric Bogatin says the level of accuracy you need in your interconnect model goes up as your system operating margin goes down. I think that's a great way to frame it.

Hargin: At faster speeds, system margins decrease and the number things you need to worry about increases. You need to take a lot more care and consider the details in your design, because you need to precisely model the design as it will be built, not as you wish it could be built. That's the focus of my company, Z-zero, and our product, Z-planner Enterprise. We allow designers to model their PCB stackups more precisely, instead of just designing to an approximate stackup and hoping the new product introduction (NPI) engineer can work with a fab house to create a stackup that matches whatever we used for design simulations. There's really not much point in running detailed design simulations if you don't have an accurate description of the board stackup and material properties.

Nolan Johnson: Bill, when you said margins are dropping, are you talking design margins, constraint margins, or something else?

Hargin: Actually, both. Let's take PCI Express as an example because it's widely known. The data unit interval (UI) and its associated sampling region gets smaller from one generation to the next, because the data rate typically doubles. That means the signaling margin you must work with goes down, which ultimately means that the physical constraints on the layout have to become tighter.

Consider this analogy: You're driving through a course marked with traffic cones in a parking lot. If you're driving at 10 miles an hour, it's easy to stay on the path and avoid the cones. If you double your speed, it gets harder; if you double your speed again, it starts to get tough. That analogy assumes the cones stay in the same place; in the case of PCI Express, we'd not only be driving faster, but moving the cones closer together as well.

Happy Holden: Are all field solvers created equal? Is it true that the faster they operate or the more accurate they are, the more expensive they are?

Johnson: And how do you determine which field solver is the right one for you?

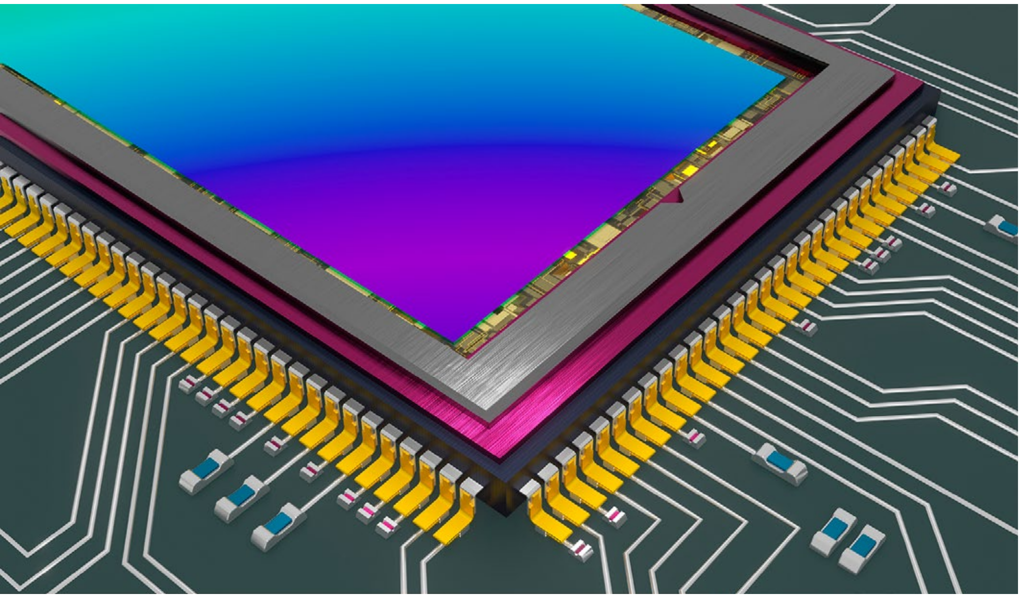
Westerhoff: Great questions. It depends on what you're trying to model, and the level of accuracy that you need. There are three different fundamental kinds of field solvers: 2D, hybrid, and 3D.

2D solvers are used when you have interconnect that has a constant cross-section, usually signal traces and cables. A 2D field solver provides the electrical characteristics per unit length, which are used as inputs for corresponding trace or cable elements in a circuit simulator. 2D solvers are well established, and they're built into many different simulation tools. If you've ever run any kind of post-layout simulation from a layout database, chances are you've used a 2D solver whether you knew it or not.

Todd Westerhoff

3D solvers are used when you want to solve an arbitrary three-dimensional structure with high accuracy, usually vias, package breakouts, or high-speed connectors. 3D solvers discretize (mesh) the structure's entire volume and solve Maxwell's equations, which requires prodigious amounts of compute power and memory. 3D solvers present the most demanding technical challenges and generate the most discussion, so when someone says "field solver," that's usually what they're talking about. There are multiple types of 3D solvers, but for our current purposes we can just say that 3D solvers are used when the structure to be modeled can vary arbitrarily in the X, Y, and Z dimensions.

Finally, hybrid solvers are in-between 2D and 3D solvers. They're best suited to large structures that are essentially planar, like printed circuit boards. They use a variety of different techniques to identify and solve the different regions and behaviors in a structure, then integrate those results to provide the interconnect's overall behavior. They're well suited



simulations they perform are more accurate, because they reflect a real stackup, instead of a hypothetical one. Designers can also use it to share stackups back and forth with their fab house and collaborate on potential changes. Z-planner is integrated with HyperLynx for SI and PI simulation, so designers can use accurate data for their design and verification work.

People commonly compare results from the HyperLynx 2D field solver to the corresponding ANSYS and Polar tools. These three provide very similar results and I think that provides a high degree of confidence in their accuracy. If you're using a different 2D field solver, it would be wise to compare it to one of these tools. As tempting as it is to say simulated results should always be verified against measurement, in cases like this, it's better to compare results to known good field solver results, because bench measurements introduce other issues.

For instance, when I measure a board, am I really measuring what I thought I was measuring? Let's say I'm measuring impedance and I want to compare two field solvers to a bench measurement. How do I know what the dielectric constant (Dk) of the laminate used in my bench measurement really is? There's typically a ±10% variance on Dk values, so if I'm going to correlate a field solver to measurement, I must figure out the exact material properties for the particular board I'm measuring first. In this case, comparing to another known good field solver is a better way to validate how good a field solver is.

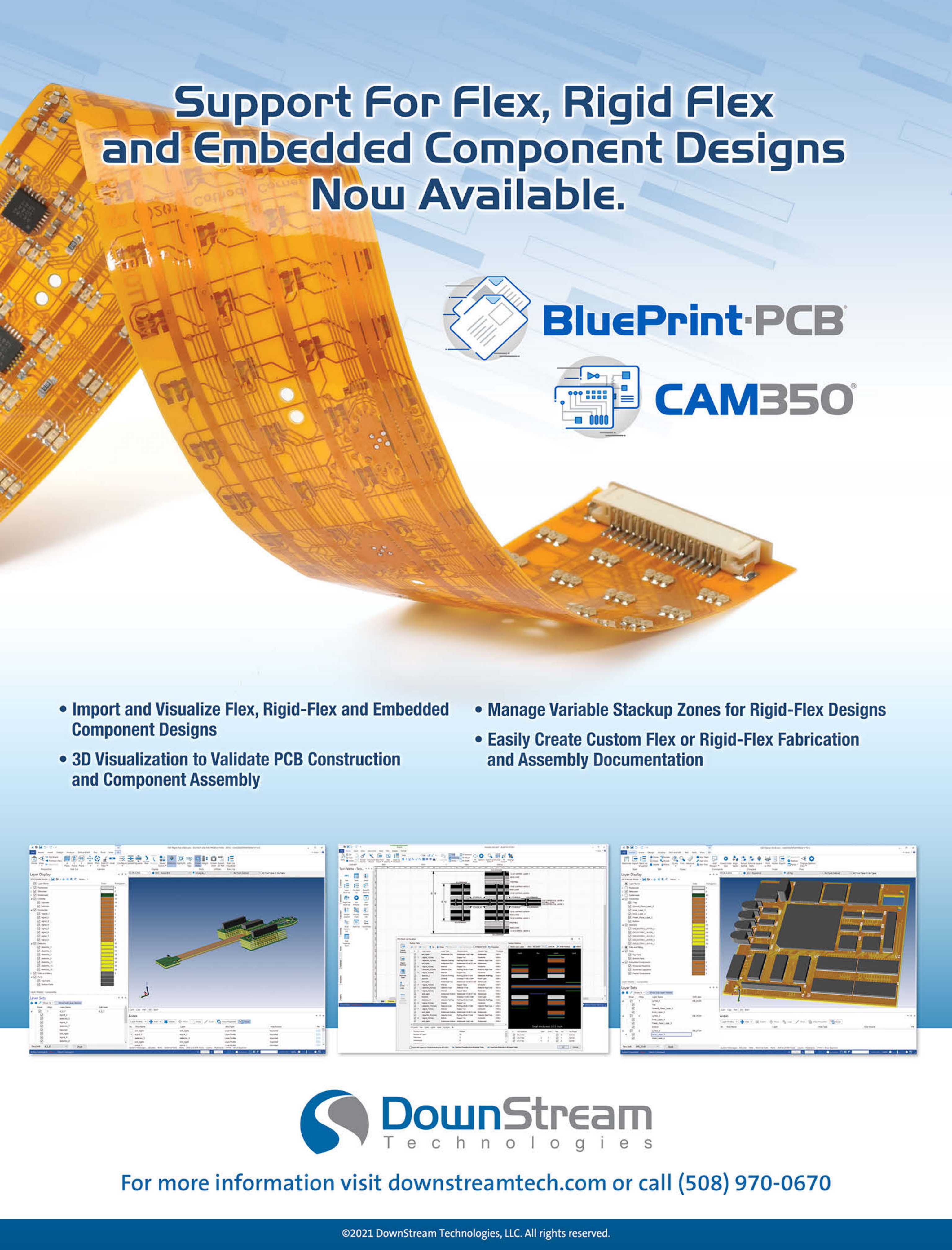
for modeling large sections of boards where good, but not ultimate, accuracy is required, so they're often used for power-aware simulations and PDN analysis. Hybrid solvers are sometimes called 2.5D solvers, but that's really a misnomer.

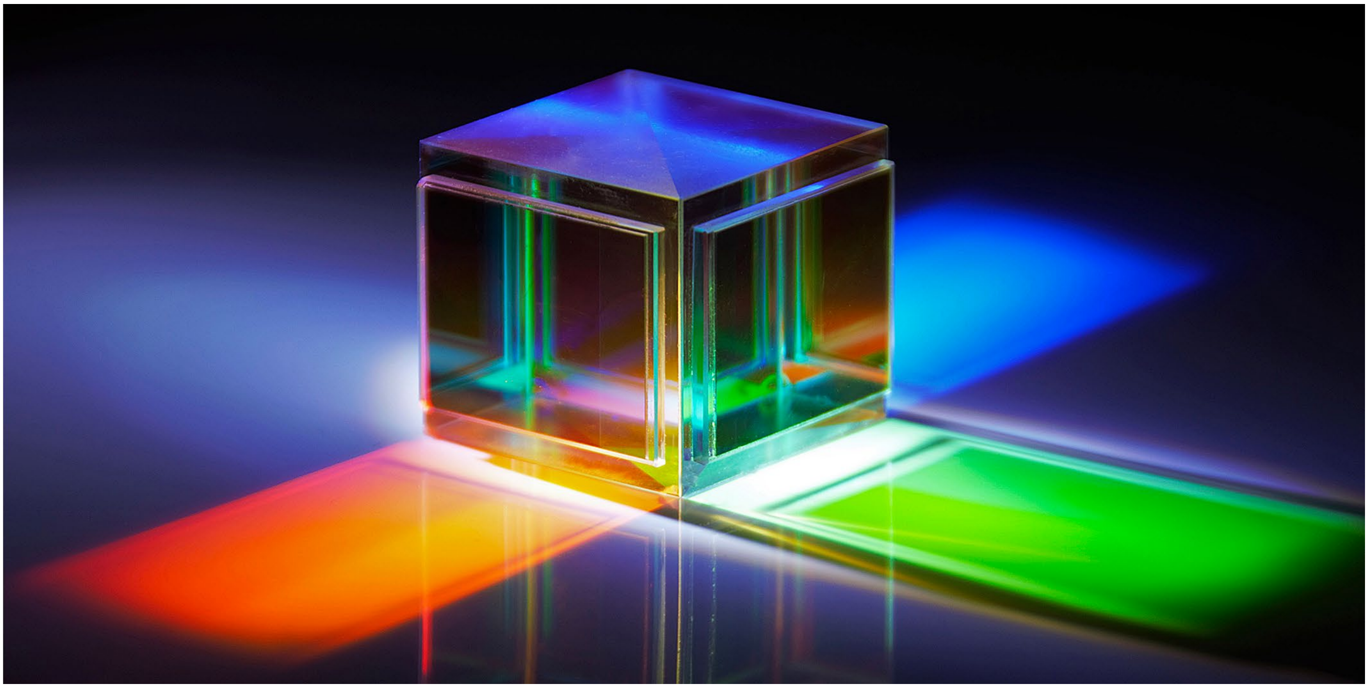
Hargin: No matter which field solver you're using, providing an accurate description of the structure to be modeled is essential. There's no point of doing a detailed 3D simulation if the structure doesn't match what you will build. We're well past the point where we can get away with guesstimating important parts of the design like the stackup, and leaving the details to the manufacturing folks to figure out later.

Traditionally, the NPI engineer took direction from SI experts on what trace impedances were required, then worked with the fab house to create a stackup that would meet those targets. Most fab houses used 2D field solvers like the Polar 2D solver to develop the stackup and document the resulting impedances. Key to this process was an accurate library of PCB materials and their electrical properties, and each fab house maintained their own.

With Z-planner Enterprise, we've gathered that material data and made it available to the system designer, so they can generate accurate stackups themselves. That means the design

Westerhoff: Field solvers and simulators assume structures can be manufactured with precise material properties and physical dimensions, which they can't. If you want to predict how a


Support For Flex, Rigid Flex
and Embedded Component Designs
Now Available.
BluePrint-PCB®
CAM350®
• Import and Visualize Flex, Rigid-Flex and Embedded Component Designs
• 3D Visualization to Validate PCB Construction and Component Assembly
• Manage Variable Stackup Zones for Rigid-Flex Designs
• Easily Create Custom Flex or Rigid-Flex Fabrication and Assembly Documentation
DownStream
Technologies
For more information visit downstreamtech.com or call (508) 970-0670
©2021 DownStream Technologies, LLC. All rights reserved.



structure will behave in production, you need to add another process around the solver itself, so you can predict a real-world distribution of design behavior. That's tricky with a 2D solver, where the simulations run fast, and really tough when you start using 3D solvers, because the run times are much longer and the number of experiments you can run is limited.

We think an important (but under-discussed) issue with field solvers is accessibility to everyday users. Field solvers have traditionally been expert tools and there aren't enough SI experts to go around. The "expert crunch" has been a problem for years, and it isn't getting better. We've reached the point where even mainstream designs require the use of field solvers, and our SI experts only have time to spend on the most demanding designs. That means most designs won't get expert SI attention, so what are the rest of us supposed to do?

Shaughnessy: Signal integrity skills are sort of like a pyramid, aren't they? You have the experts at the top, a small number of people who are full-time SI engineers. Those are the people using field solvers today. You've got a bigger group in the middle who are skilled system designers, who'd like to be able to run analyses themselves but can't, because they don't have the right tools and processes. Finally, you've got the largest group at the bottom, who don't have access to simulation tools or enough understanding of SI principles to know how to apply them.

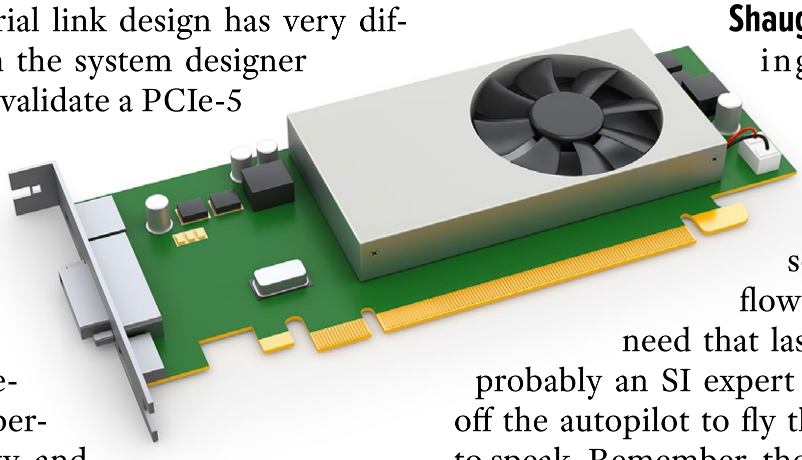
Westerhoff: That's especially true for 3D field solvers. There are lots of technical details associated with setting up and running a 3D simulation that produces meaningful results. The key word here is "meaningful." 3D solvers are complicated and it can be tough to get a simulation to just run in the first place. We can fall prey to the temptation to assume every result is valid simply because the solver itself is "known good." That's not true; correct simulation setup is critical to producing a good result and setting up 3D simulations is an expert user skill.

Here's a recent example of real-world "garbage in, garbage out." A customer was comparing our 3D solver to one they considered "known good." The two simulators weren't producing similar results, and the user assumed the problem had to be the HyperLynx 3D simulator. They contacted our AE and asked what was wrong with HyperLynx. Our AE realized the problem was actually with the simulation setup in the other tool; the signal traces were too close to the crop boundary, and that was throwing the results off. Once that was corrected, the two solutions correlated well. Interestingly, HyperLynx had set up the simulation correctly. The mistake occurred because the other solver just did what the user told it to. The lesson is that you must know how to set up a 3D field solver correctly, or it's "garbage in, garbage out."

The big challenge is this: How can we make these powerful simulation tools accessible to system designers? We can't expect system designers to take the time to learn all the ins and outs of setting up a 3D field solver.

Shaughnessy: It seems like some of the new field solvers coming out are aimed at this group.

Westerhoff: They claim to be. The challenge here is accessibility, not technology. The issue with 3D field solvers is that their development has always been driven by designs at the state of the art. The expert modeling the interconnect for 224G serial link design has very different needs than the system designer that just wants to validate a PCIe-5 board that was laid out using rules that supposedly guaranteed it would work. SI experts at the state-of-the-art care about performance, capacity, and accuracy first and foremost. It doesn't matter if the solver is hard to use; they can make it work no matter what. The system designer with PCIe-5 is literally the inverse: performance, capacity, and accuracy have to be good enough to produce a meaningful result, but the driving issues are integration, workflow, and usability. System designers need tools that they can deploy without having to become SI experts.



If you walk into the cockpit of a Boeing 737, your first impression is probably, "This is really complicated. I could never fly this!" The fact is, that plane has an autopilot, and with minimal training and common sense, you could fly that plane.

Most field solvers are still like flying that 737 without the autopilot, so they require a SI expert. We've integrated the HyperLynx solvers with our SI tools and put a layer of automation around them so that process of identifying the area to be modeled, setting the crop boundary, defining signal ports, and setting up solver options is done for you. It's like using a 3D field solver on autopilot.

The challenge has been taking this sophisticated technology and developing algorithms that will let system designers generate meaningful results, then ensuring that automation works for a variety of real-world designs. We want to bring 3D field solvers to mainstream design by changing 3D analysis from an art practiced by a few skilled craftsmen to something more like 3D EM for production use.

Shaughnessy: Are you trading off some level of accuracy for accessibility?

Westerhoff: With some of the automated flows, yes. If you really need that last few percent, you're probably an SI expert already and will turn off the autopilot to fly the plane manually, so to speak. Remember, the goal here is to enable a larger audience to run analysis themselves, reducing the demands on those overloaded experts. There are a lot of designs to get out the door, only a few of which require ultimate capacity and accuracy. That last few percent comes at an incredible cost in tool complexity and user expertise; we want to reserve SI experts for those cases where they are really needed.

Shaughnessy: So, you're looking to enable system designers who want to run analysis themselves. What advice would you give to a recent grad who has just started using field solvers? What's the takeaway for those guys?

Westerhoff: I think it's important to start with your ultimate goal. If you're trying to figure out whether your design works (and you should be), then everything else is a supporting detail. You need to know if your design passes or fails and by how much. You should therefore consider each element in the design in the context of system margin. Ask, "What process am I using to model the entire design? What process am I using to simulate it and assess success or failure?" When you need to use a field solver, ask, "What's the overall workflow? How will the field solver help me get my job done?" There needs to be an established workflow that someone who's not a SI expert can follow.

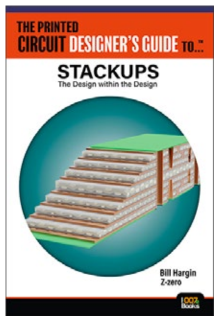
Shaughnessy: Bill, do you have any advice for anyone facing challenges using a field solver?

Hargin: You mentioned young engineers figuring out what's what. As I said earlier, you need caution and experience when comparing field solver results to bench measurements. Simulations assume everything is defined exactly, and the real world doesn't work that way. If you're comparing to bench measurements, you need to determine the correct material properties and dimensions for the as-manufactured device, and that can be difficult. If you're simply putting spec data into a field solver and getting different results from what you measure, that's probably why.

If you're new to the subject and asking about a good field solver, I cited three known good ones that correlate well enough that everything else should be compared to them. Finally, beware the free field solver, because we all know that you can go broke saving money.

Shaughnessy: Good advice. Thank you both, guys. This has been great.

Westerhoff: Sure. Thank you, Andy. DESIGN007



Bill Hargin is the author of *The Printed Circuit Designer's Guide to... Stackups: The Design Within the Design.*

The Great Divide in PCB Simulation Software

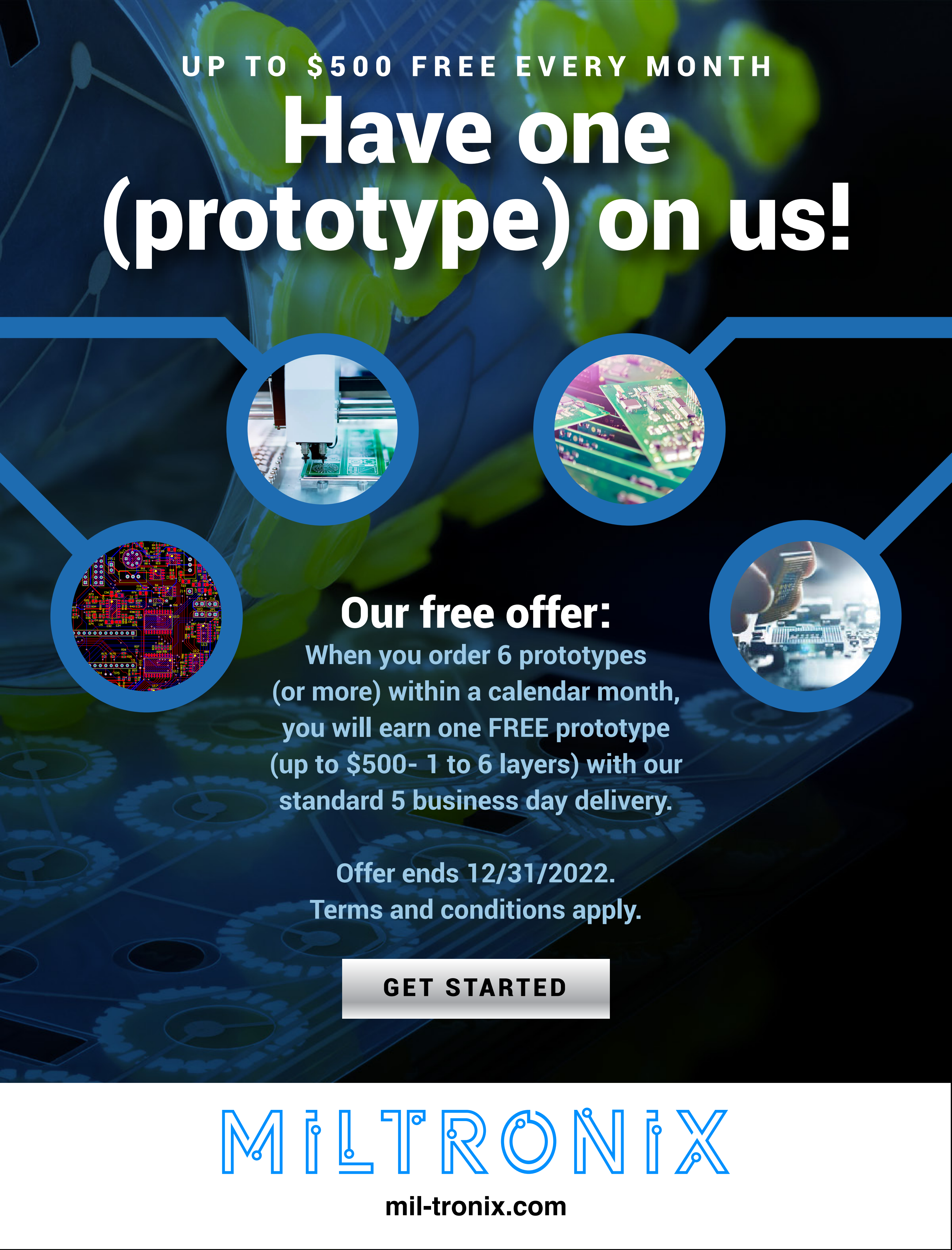
Feature Article by Zach Peterson
NORTHWEST ENGINEERING SERVICES

Today's PCB design engineers have more layout and analysis tools at their disposal than ever before. Over the years we've seen layout tools become more automated, rules-driven, and more integrated. Now we even have integration between design tools from different vendors and ranging across domains, starting with basic circuit design, and spanning up to PLM and ERP integration. It really is a great time to be a designer.

But there is one area that continues to be a bit disconnected from the rest of the design process: simulation and analysis. Those of you who know me will know that I love simulation, and I love analyzing interconnects to death. If you're like me, then you probably prefer to do all this by hand. If not, there are plenty of electronics and electromagnetics simulation software suites to help you along the way.

The problem with today's simulation software options is not their capabilities; if you shop around enough, you'll find a simulation for just about anything. The problem with the industry-standard simulation tools lies in where they fit into the standard PCB design workflow, as well as the user experience. Simulation software used in electronics and PCB design generally falls into one of the following categories.

- Circuit or schematic simulation tools (e.g., SPICE)
- 2D interconnect simulators implementing BEM or MoM
- Electromagnetic field solvers implementing FDTD, FEM/FEA, or a similar numerical method


UP TO $500 FREE EVERY MONTH
Have one (prototype) on us!
Our free offer:
When you order 6 prototypes (or more) within a calendar month, you will earn one FREE prototype (up to $500- 1 to 6 layers) with our standard 5 business day delivery.
Offer ends 12/31/2022.
Terms and conditions apply.
GET STARTED
MILTRONIX
mil-tronix.com

All engineers and designers are probably familiar with SPICE; if you're not, you should be. I consider the ability to run a SPICE simulation a mandatory skill for any PCB designer. Advanced applications will go far beyond SPICE and should use some level of simulation to verify signal integrity, channel compliance, and EMI.

2D Simulators

Not all PCB design applications include a 2D interconnect simulator that can return impedance, reflections, crosstalk, and return path tracking in a PCB layout. Applications that do include these simulators will not have it running in the online DRC engine. For everyone else, there are both free and paid calculator applications that will give you a rough estimate of crosstalk and reflections.

> Within the standard workflow, 2D simulators can work as a verification tool for interconnects once layout and routing are completed.

Within the standard workflow, 2D simulators can work as a verification tool for interconnects once layout and routing are completed. In most of these tools, the user interface is suboptimal. These are not point-and-click simulation tools that operate like DRCs. Some configuration is needed, and you need to know some inputs with high certainty to gain meaningful results. For this reason, these simulators are not often accessed while the design is being created; you'll find that users wait until the design is near completion to use these if they are used at all.

Electromagnetic Field Solvers

Field solvers are great for back-end verification once a design is completed, or for demonstrating feasibility of a circuit/interconnect on the front end. In between, while you're actually creating the design, they are practically absent. In any case, they are practically absent from the workflow for most designers. There is another challenge here; users of these applications need to be well-versed in methods for solving Maxwell's equations in all their various forms. That usually entails a graduate degree in engineering, physics, or mathematics.

The results from these programs are very accurate, but only as long as all the input data for simulations are also accurate. These programs suffer from "garbage in, garbage out"; if the input data used in a field solver do not match reality, then the field solver results won't match reality either. They also follow the same antiquated workflow you find in ECAD/MCAD collaboration; you have to export the design from your ECAD software, then import it into the field solver.

So, what should simulation tools of the future look like? How can the CAD vendors and simulation software providers do better?

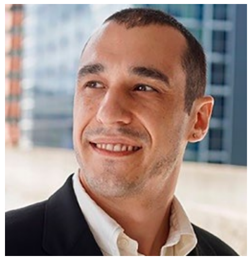
Where PCB Simulation Tools Can Improve

Clearly, there is a lot of room for improvement in terms of user experience and workflow integration. Because the user experience for these tools can be difficult to manage for less-experienced designers, they get pushed to the beginning or the end of the design process; there is little simulation happening in the middle.

It would be great to see a simulation toolset that is built into PCB design applications and that can be invoked like your DRC engine. Personally, I would love to see something that is invoked like DRCs, but that is qualifying the design against a broader range of SI/PI/EMI metrics than just impedance, reflection, and crosstalk.

The best possible feature for the high-speed PCB designer would be instant qualification against signaling standards at the click of a button. This is a simple pass/fail type of evaluation tool. I'd like to know whether my DDR4 routing is actually going to comply with DDR4 standards before I get too far along in a design, otherwise we risk ripping up tracks and adjusting placement.

To be fair, some simulation platforms are working toward automating common field solver-based analyses that are required to evaluate SI/PI/EMI, as well as channel or standards compliance. Still, these tools come with a huge price tag, so the individual designer is effectively priced out of the market. A stripped-down, highly automated PCB simulation suite inside ECAD software would help level the playing field for designers of all skill levels. **DESIGN007**



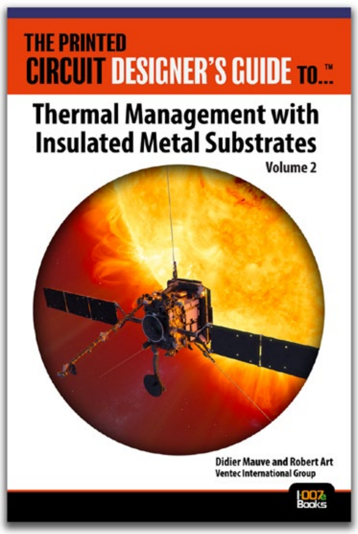
Zach Peterson is the founder of Northwest Engineering Solutions and an Altium consultant.

BOOK EXCERPT

The Printed Circuit Designer's Guide to... Thermal Management with Insulated Metal Substrates, Volume 2

As the second in a two-part series, this book from Ventec builds on the material presented in the first by describing up-to-the-minute products and design techniques for thermal management with IMS.

The first volume comprehensively covers principles of heat transfer and modeling of thermal systems to help engineers quantify the thermal-management challenges in the application at hand, the nature of IMS, typical material parameters, and the choices available for designers to reach the desired performance.



We recommend reading the first book in order to understand the issue from first principles, how to navigate the many material choices offered, and appreciate how adjusting the various combinations of material types and thicknesses can help you arrive at a solution that meets your requirements from all standpoints, particularly system reliability, size, cost, and manufacturability.

The Printed Circuit Designer's Guide to... Thermal Management with Insulated Metal Substrates, Volume 2, covers additional use cases and materials, including those used for high-temperature applications as well as high-emissivity surface treatment that can significantly enhance thermal performance. A web-based thermal calculator is demostrated, which helps arrive at the exact size of baseplate needed to achieve the desired thermal performance in the smallest possible volume—much smaller than would be possible using a suitably sized heatsink.

Volume 2 delves deeper into issues such as test methods and how these can affect assessments of IMS materials and analyzes the effects of tolerances in material properties and thickness on the overall thermal conductivity. Understanding this can critically influence measurement error, ultimately determining whether or not an accurate result for reliability can be calculated. The authors look at how problems such as solder cracks affect thermal performance, and share application examples showing design with multilayer IMS.

As application challenges have evolved, and new component types and thermal materials have entered the market, there is more to say about thermal management and the options available to designers striving for an effective, economical, and compact solution to meet system-reliability targets.

Download this title today!

BENDING THE POSSIBILITIES

FLEXFINER

BY TAIYO

AVAILABE IN TWO COLORS

AMBER

BLACK

- Excellent bendability
- High Resolution by photolithography
- High Reliability
- Simplified process
- UL94 VTM-0 certified
- Low spring back force
- Resolution Line/Space=50/50µm
- Solder mask defined opening (30µm diameter)

OUR BIGGEST FLEX YET!

Contact your local sales representative today!

Visit www.taiyo-america.com for more information.

TAIYO AMERICA

Follow us on social media

Field Solver Finesse for Modelling Transmission Lines

The Pulse

Feature Column by Martyn Gaudion, POLAR INSTRUMENTS

When I-Connect007 asked me to contribute to this issue on field solvers, I wondered what more could be added to this subject. But as a supplier and developer of field solvers, Polar is still asked some of the same questions—both by experienced customers who are perhaps exposed to a new scenario and, as is most welcome, by new entrants to the industry.

I will start by saying that all field solvers are accurate; they solve Maxwell's equations by one or another of the available mathematical methods. When all are fed with the same data, all should generate very similar results, and any differences observed will be orders of magnitude less than the variations in the PCB transmission lines caused by the composite nature of PCB substrate dielectrics and the variations of the plating and etching processes.

However, field solver is a very generic term for a range of tools in this application for predicting the behavior of PCB transmission lines. It is important to remember that some areas of field solvers are not actually field solving. Calculating the loss owing to surface roughness of copper is a prime example of this. The surface nature of plated copper is so complex that full field solving would be impractical, so most commercial "solvers" will overlay the core field-solving function with empirical techniques: Hammerstad, Groisse, Huray, and Cannonball-Huray, to name just a few. These empirical extensions extend the capability of the field solver into modelling parameters that are:

- Vital to model insertion loss
- Impractical to field solve given the complexity of the surface profile and the available compute power in the hands of even the best equipped SI engineers

Feeding the solver with the correct dimensions is vitally important, as no tool will give an accurate result if fed with incorrect start parameters. Customers frequently ask whether the transmission line structure height should include the trace thickness in the total height. This is easy to answer if you are working "backward from a microsection" but if you are imagining the finished PCB from a simulation, then it's less obvious.

My question in Figure 1 is set as a puzzle as it is one of the most frequently asked items

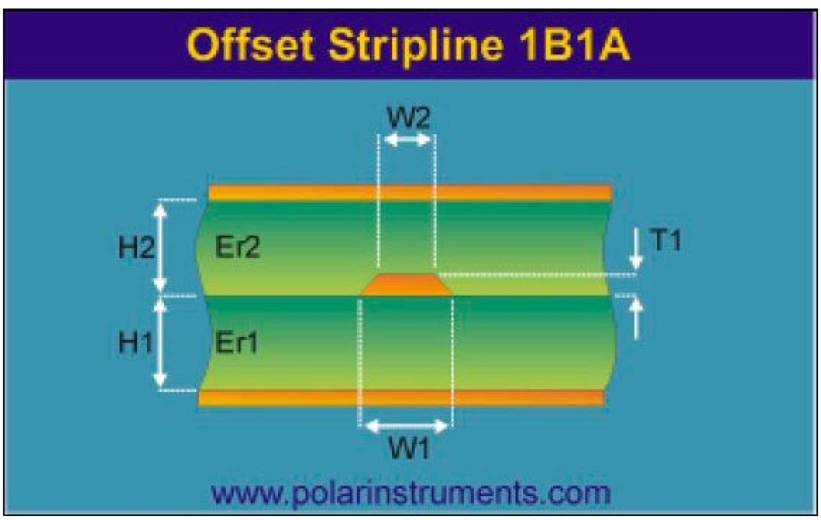


Figure 1: Does H2 include the copper or not?

regarding transmission line modelling. The H1 dimension represents the core thickness, and the H2 dimension is that from the top of the core to the foil. You can see H2 is denoting prepreg as the trapezoid is pressed into the Er2 region. By how much? Well, if you work from a microsection, the dimensions will be clear to see. But when modelling with a solver before the board is built, you must make a prediction as to how much the T1 will impact the H2 dimension.

Think about it: If there is a lot of copper on the signal layer, most of the T1 will need to be added to the pressed thickness of the prepreg; if the routing density is low, then far less. This is where a stackup tool comes in handy, as a good stackup tool will virtually press the prepregs and consider the Cu density on the signal layer to calculate the optimum value for H2 to feed into the solver. This is the point I was stressing earlier: Commercial PCB transmission line field solvers must possess a variety of tools and capabilities over and above the core solver engine to feed it with good mechanical data to solve.

As you look at Figure 2, consider the question posed by Figure 1. By using Speedstack to pre-process the material data, the pressed height of H2 and the impact of T1 have been calculated to feed into the solver engine. The left-hand side of the image shows the raw prepreg thicknesses. The signal trace (shown in blue) has a 5-mil core below and two sheets of 3-mil prepreg above. Another core is placed above the two prepregs at the top extent of the highlighted area. Now look at the structure on the right-hand side of the picture. This is an offset stripline located with the signal on the blue layer on the LHS.

As you would expect, H1 = 5 mil—the lower core thickness—but H2 does not equal 6 mil (3 mil plus 3 mil prepreg). H is calculated as 6.28 mils, the combined thickness of the structure when the two sheets of prepreg are pressed into the copper distributed on the signal layer. Taking care of this type of pre-processing is

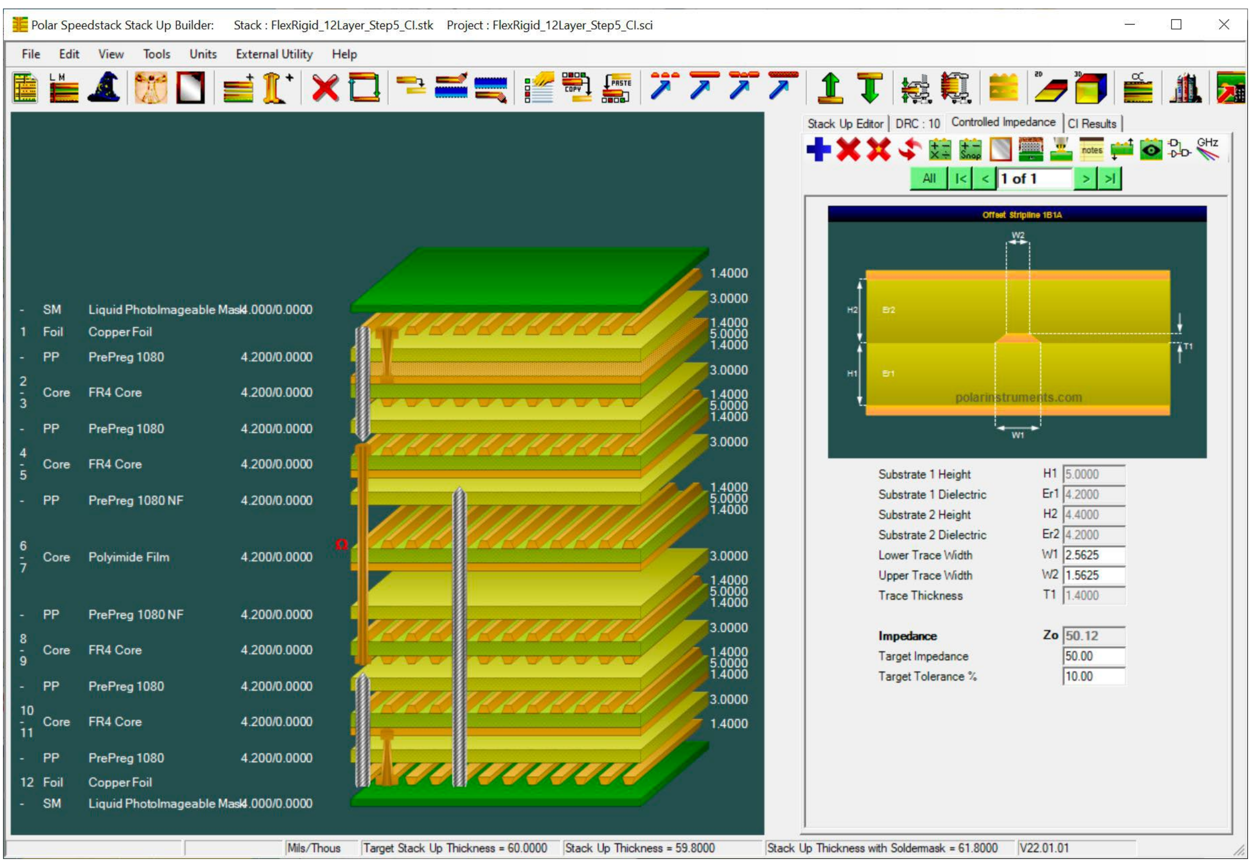


Figure 2: Auto virtual pressing and calculation of H2.

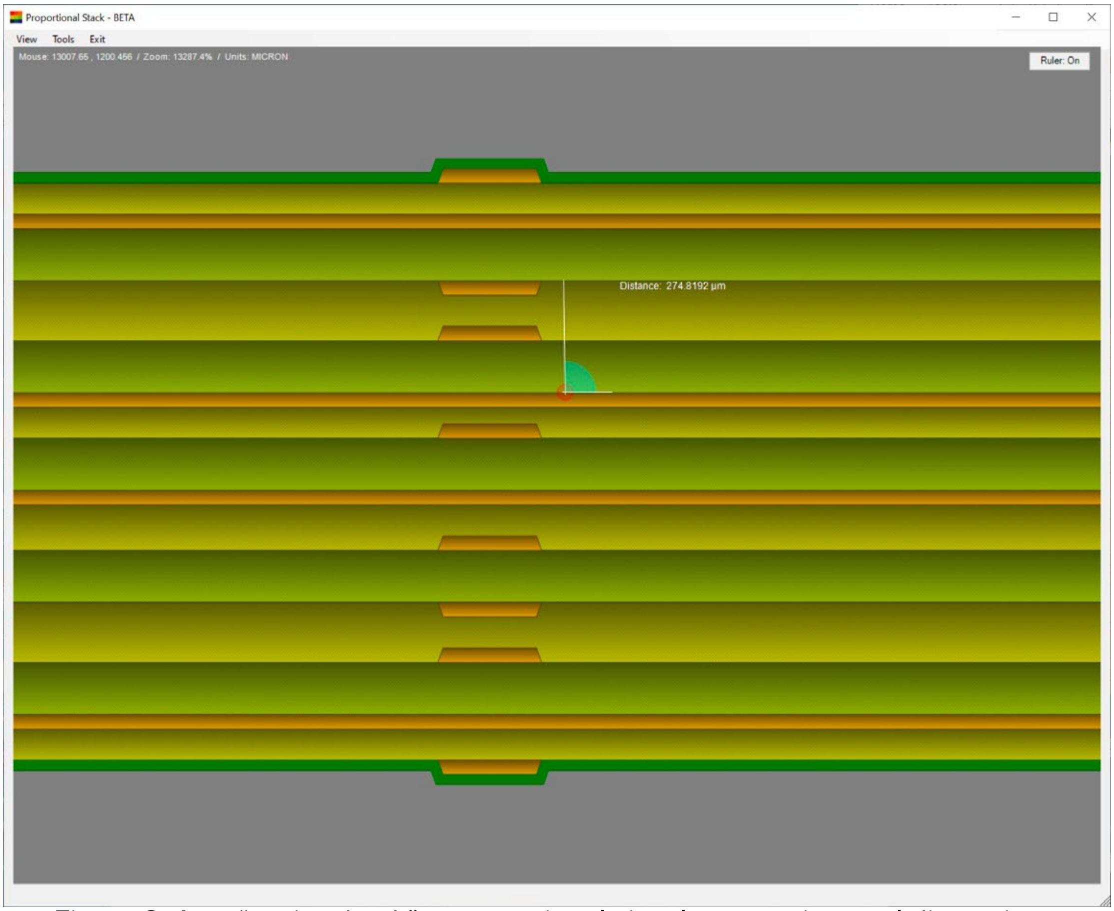


Figure 3: As a "sanity check" a proportional view lets you view real dimensions.

the key to obtaining accurate predictions from your solver.

Viewing the stack in Figure 2 as a schematic view with fixed size layers for materials is ideal from a planning perspective, but once the stack is complete and pressed, a proportional view makes it plain to see if you have added incorrect materials in error—in the blink of an eye.

Conclusion

I have used a small sample of the techniques you need to deploy to ensure that a field solver engine is fed with accurate data. Quality material data from the supplier is also key in this process, but the key takeaway is that, when looking at the appropriate field solver for your requirements, you must always remember that all solvers are accurate; it is the pre-processing of the data being fed to the solver which unlocks its full potential. This is true for lossless lines up to around 2GHz on through to ultra high-speed lines where insertion loss needs serious consideration too. You should also take care that measurement data is validated, but that is the topic of another article—already partially covered in my April 2022 column in *Design007 Magazine*, "Using Touchstone Files to Build Measurement Confidence." **DESIGN007**

Martyn Gaudion is managing director of Polar Instruments Ltd. To read past columns, click here.

Additional content from Polar:

- ***The Printed Circuit Designer's Guide to... Secrets of High-Speed PCBs, Part 1*** by Martyn Gaudion
- ***The Printed Circuit Designer's Guide to... Secrets of High-Speed PCBs, Part 2*** by Martyn Gaudion
- **View other titles in our full I-007e book library here**

PCB007 Highlights

Review: Institute of Circuit Technology 2022 Annual Symposium ▸

The British Motor Museum in Warwickshire, housing the world's largest collection of historic British cars, was venue for the 2022 Annual Symposium of the Institute of Circuit Technology, which attracted a substantial gathering of manufacturers and suppliers from the UK printed circuit industry.

Happy's Tech Talk #8: Copper Etchant Regeneration ▸

Copper has become a valuable metal, and with the growth of EV has come higher currents needed with increasing weight of copper in PCBs. This creates the need for increased copper etching and consumption of copper etchants.

FTG Circuits Fredericksburg Requalifies for IPC-1791 QML ▸

Firan Technology Group Corporation announced that IPC's Validation Services Program has awarded requalification for IPC-1791, Trusted Electronics Fabricator Requirements Qualified Manufacturers Listing (QML) to FTG Circuits Fredericksburg, Virginia.

The Chemical Connection: How Industry 4.0 Shapes PCB Wet Processes ▸

Earlier this year, I had the opportunity to attend IPC EXPO 2022 where I was able to network with many great people in our industry. Although that might have been my main takeaway, there was another trend that I couldn't ignore: the fourth generation of the industrial revolution is alive and well in the industry.

Atotech Sells 1,000th Horizontal Electrolytic Copper Plater ▸

Atotech, a market leader in advanced electroplating solutions, announced the sale of its 1,000th horizontal equipment for electrolytic copper plating.

IPC Establishes European Subsidiary ▸

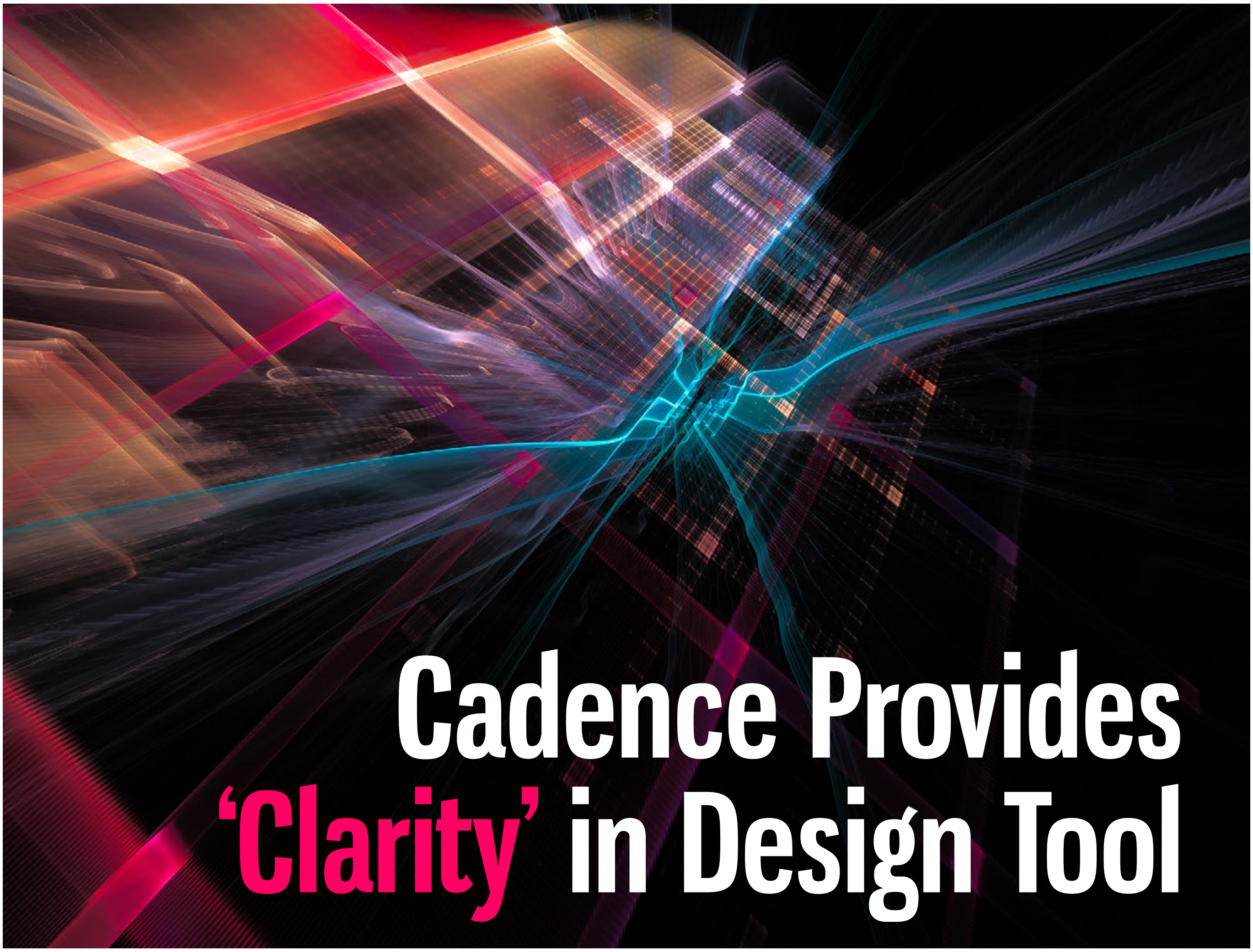
To better support our European members' interests, IPC announces the formation of IPC Electronics Europe GmbH. The subsidiary was officially registered with the German government on May 4, 2022. The new entity's main office is located in central Munich.

Mike Carano Joins the Averatek Team as Vice President of Quality ▸

Averatek is pleased to announce that Michael V. Carano has joined our corporate leadership team as vice president of quality. Mike brings 40 years of electronics industry experience, with special expertise in manufacturing, chemicals, metals, semiconductors, medical devices, and printing.

Calumet Students Take First Place in 'Michigan Girls Future Flight Challenge' ▸

Two Upper Peninsula girls recently took the Michigan aerospace industry by storm! CLK Washington Middle School students Jordan Hicks and Kristen Ylitalo outperformed more than 20 teams throughout the state in the "Michigan Girls Future Flight Challenge," hosted by the Women of Aerospace Industry Association of Michigan.



Cadence Provides 'Clarity' in Design Tool

Feature Interview by Nolan Johnson
I-CONNECT007

Nolan Johnson recently spoke with Brad Griffin, product marketing director for Cadence Design Systems, about Cadence's Matrix solver technology. They discuss its use as a multi-disciplinary field solver as well as Cadence's focus on thermal analysis and utilizing the power of the cloud.

Nolan Johnson: Brad, now more than ever you must make decisions in design that are going to impact manufacturing.

Brad Griffin: I agree with you. Cadence is very uniquely positioned because we've always had PCB and IC packaging design tools. So much of what we used to do on a printed circuit board is now done in silicon on a silicon interposer. The tools that are used at the board and package level have had to evolve to be able to support the new types of designs.

We're engaged through the whole process. That's where field solvers come in because if you're a printed circuit board designer and you're laying out a board, you may not even think about a 2D field solver running in the background, calculating impedance for you to tell you that you've met the 50 ohms of impedance that you were trying to get to. Those tools are embedded in the design tools, and the users don't even know they're running an EM solver. But as you get into more involved work, where you're sending 64 bits simultaneously at

Brad Griffin

don't want to put any vias in place because they know it's going to cause some discontinuity, but they need to find out if it will work, then this is where, in design, you can call on Clarity in the middle of design. Clarity can give you an S parameter. You can plug that S parameter into a circuit simulator, and you can look at the eye diagram and determine whether it will work. You can make that decision in design instead of waiting until the end, with your fingers crossed hoping that it's going to work.

That's the case where you probably get your answer back in minutes. There are other cases where you're looking at something bigger and you're saying, "I'm willing to let this run overnight." Again, I think this is where Clarity comes into play, where other field solvers might be running for weeks before they get the answer. The speed and the capacity of Clarity is really opening up what signal integrity and power integrity engineers can do in the design process.

five gigabits per second on a DDR5 bus, things get to a point where you need to look at the signal, the power, and the ground because now they all interact. That's the same, whether on a board, on the package, or whether you're on an interposer; you just may use different material properties to be able to model the interconnect.

A sophisticated, full wave 3D field solver can work in all those types of applications if it has the speed, the capacity, and the ability to crunch that large of a problem. That's where Clarity 3D Solver comes in. It's unique in the industry in its ability to apply Maxwell's equations to make accurate, full-wave 3D assessments of how the interconnect is going to behave, do so on a very large scale, and in a timeframe that allows an engineer to make a decision based on that simulation results.

Johnson: Not in post-processing, but during the design process?

Griffin: Again, it's always going to vary, but you're right. For example, if somebody is routing that differential pair and they really

Johnson: I've been hearing a lot about Sigrity X. Tell us about it.

Griffin: Sigrity X is what we call the new version that has the 10X faster simulation capabilities. That's where the matrix solver comes in.

Johnson: Are we still living in a world where you can have these processes running for days and weeks?

Griffin: Oh yes, absolutely. We see this all the time. If the compute power is there, people will use it. They must simulate this altogether as one structure, and they're willing to wait a week for that answer, if that's what it takes. Then we show them they can get it done in 10 hours now. That's a big difference.

It's amazing how much faster some of these solvers are, but once they've got that, they'll just make the simulation bigger, so our customers will still run the Clarity solver for days. We've connected Clarity to the cloud through AWS, so now you have almost unlimited computing resources, but there's always a cost associated with it. While the solver can run much faster, maybe they don't want to use all the licenses required, so they might say, "I can get it done in 10 hours, I'm okay with that, even though I could use 10 times the licenses to get it done in one hour."

Johnson: You do get to trade that off: how much resource you want to pay for vs. how fast you want to get your results. Is Clarity standalone up on the cloud now?

Griffin: It's a hybrid. You do all your setup locally from your machine because that's interactive. It's not really compute intensive; you're making decisions. Then, when you hit the simulate button, specify to simulate in the cloud. It just packages everything up, encrypted, and runs the simulation in a secure chamber where nobody has access to it. When complete, it then ships it all back to your machine. To you as the user, they don't really see a difference. I think that's the real power of what we've created with the cloud version of Clarity.

Johnson: How is this going to change the engineers' way of doing their job?

Griffin: Engineering is a game of "get me the best answer you can as fast as you can." What we've done largely before Clarity is we've seen many of our customers rely on what is called a hybrid solver. For some of these jobs I was talking about that are taking hours, the hybrid solver might take 10 or 15 minutes. You can get a basic idea of how things are going. Engineers will continue to use different tools at different times. The biggest breakthrough is that with all these compute resources now available—either on-premises or in the cloud—there's software written to take advantage of that. The main thing that Clarity brings to the table is that the other tools on the market were written when people still used single-core machines.

I think it's just so empowering for an engineer to have this capability when they want it. They're still going to make the quick changes, quick decisions in-process. But at some point, they're going to run their simulation overnight.

Johnson: I am hearing a series of iterations in the design process: quick checks while working with small sections, transmission lines, etc. You want to make sure that everything is maintaining spec, piece by piece, and then move up to a module within your entire design. Even though you were in-spec at the transmission line level, it still behaves and within spec when put all together.

We're finding most thermal tools are bent toward the mechanical engineer. But really, the electrical power is what's causing thermal issues in the first place.

Griffin: And no surprises when you get that prototype back; that's what you're aiming for. We're seeing the same thing in thermal because all these processors are so power intensive these days, especially as things get smaller.

We're finding most thermal tools are bent toward the mechanical engineer. But really, the electrical power is what's causing thermal issues in the first place. If you can take all the detail of the power delivery network and make that part of your simulation, that's thanks to what we call a massively parallelized matrix solver, that we use in Clarity. We also use it in our thermal tool called Celsius. And so the massive parallelization of being able to break that problem apart and solve it in parallel and then bring it all together and give accurate results. That's what really empowers us to do that.

We have this massive parallelization because they solved this on the IC side a long time ago for the Voltus power delivery network tool. That matrix solver was developed to solve a much bigger problem. And then as these system problems become the bigger problems, the technology is waiting for us, ready to be reused in the PCB space.

Johnson: Where else is it being used?

Griffin: PDN power integrity. The main areas are SI, PI, thermal, and then the chip level power as well. We had a customer call us, and he was using the version of our tool from two years ago. He said, "It's taking five days to solve this." This was with the version I told you normally happens in 15 minutes. This guy was running a really big problem and it was taking five days. We said, "Well, have you tried the new version?" He said, "Oh, I don't have a license for that." So, we got a demo license for everything. He ran the simulation, and results came back in eight hours.

Johnson: Five days down to eight hours. That's nice.

Griffin: It works. It basically works across all the different tools that we have. We have all this technology in the IC side, we've brought it into the system tools. It's no longer a feature thing. Engineers are empowered. It's just so frustrating when these full wave 3D field solvers force you to break it down to something so small that your computer will actually solve it. Then when you solve it, you don't know if the results are right. It's so frustrating. With us, you just don't break it down. You just put the whole thing in, you solve it. It uses potentially more computing resources, but it's also more efficient.

We used to talk about the three types of simulations for signal integrity matters. The first is while you wait; push the button, get your answer back. Then we have the coffee break because you talk to somebody for a few minutes, while your answer comes back. Then there was overnight. What happened is the problems got more and more complicated; overnight became over the weekend and over the weekend became over the week. Now what we've done in many cases is brought it back to overnight.

Johnson: Right. And six and a half days into a seven-day run and it crashes.

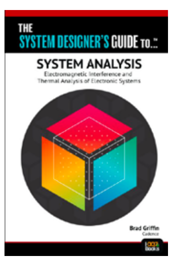
Griffin: That can certainly happen when you're consuming the amounts of memory that these tools are. I don't want to say we're perfect. But we do hear from customers that the other tools crashed six times, but the Clarity tool worked. With limited resources to simulate within his boundaries we were able to solve this problem.

Johnson: That's the magic. Anything else about this that we should talk about?

Griffin: I think we talked through the cloud, and we talked about Celsius. We talked about Sigrity X, which is what we call the new version that has the 10X faster simulation capabilities. And that's where the matrix solver comes in. So those are the three things that I would like to highlight.

Johnson: Excellent. Brad, thank you.

Griffin: Sure. My pleasure. DESIGN007



Download *The System Designer's Guide to... System Analysis* by Brad Griffin along with its companion book *The Cadence System Design Solutions Guide.* You can also view other titles in our full I-007e book library here.

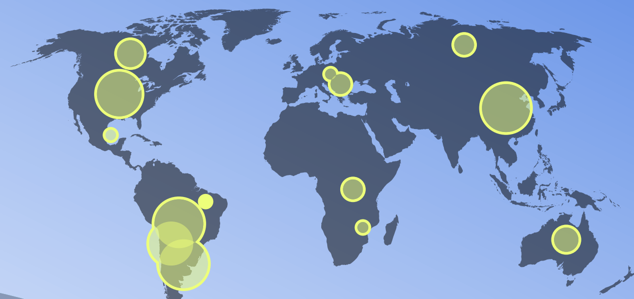
The EV Critical Battery Materials Boom

1000%

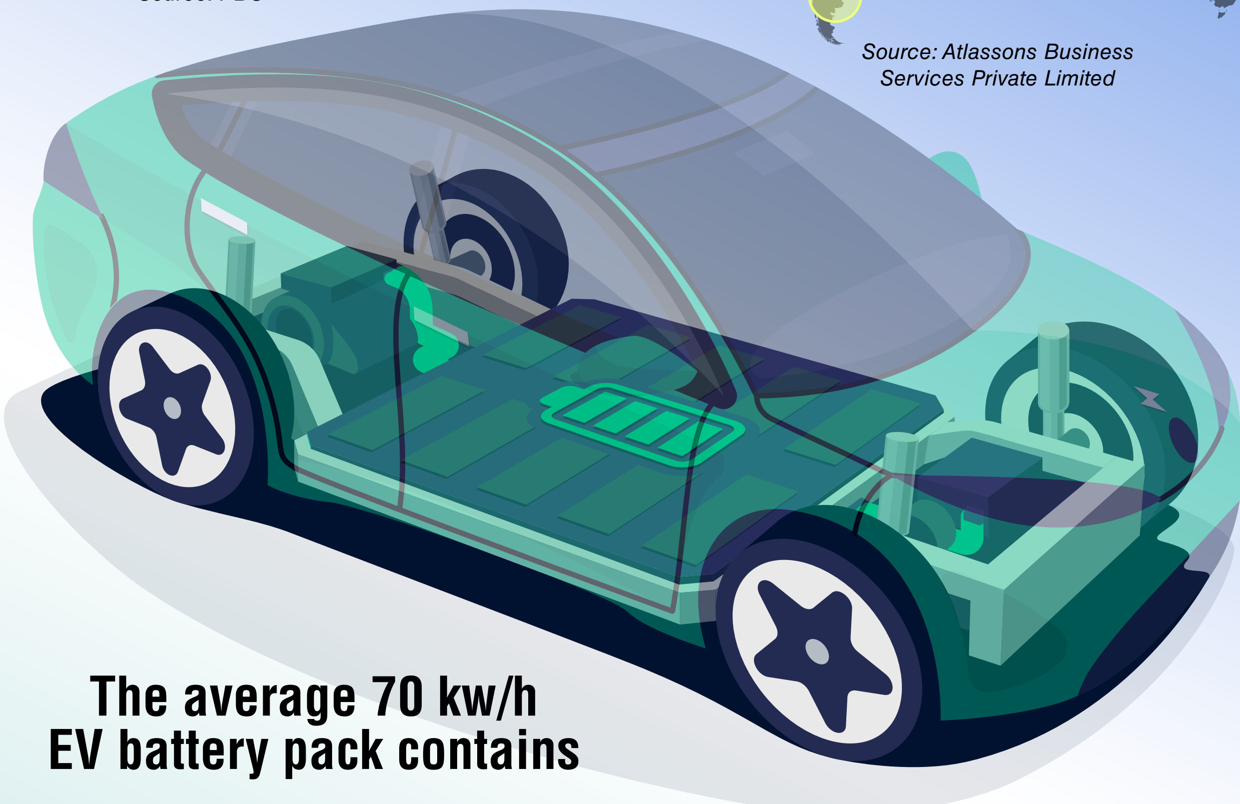
Expected growth in demand for minerals needed to make EV batteries by 2050

Source: PBS

WHERE DOES THE LITHIUM ORE COME FROM?



Source: Atlassons Business Services Private Limited

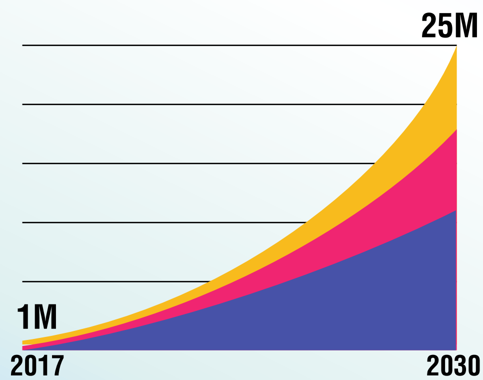


The average 70 kw/h EV battery pack contains

Ni Nickel	Co Cobalt	Li Lithium
77 lbs.	31 lbs.	18 lbs.

Source: Energy X

AS EV SALES INCREASE, MORE LITHIUM WILL BE NEEDED



United States China All other countries

Source: Bloomberg, NEF, Marklines

I-007e FAST FACTS

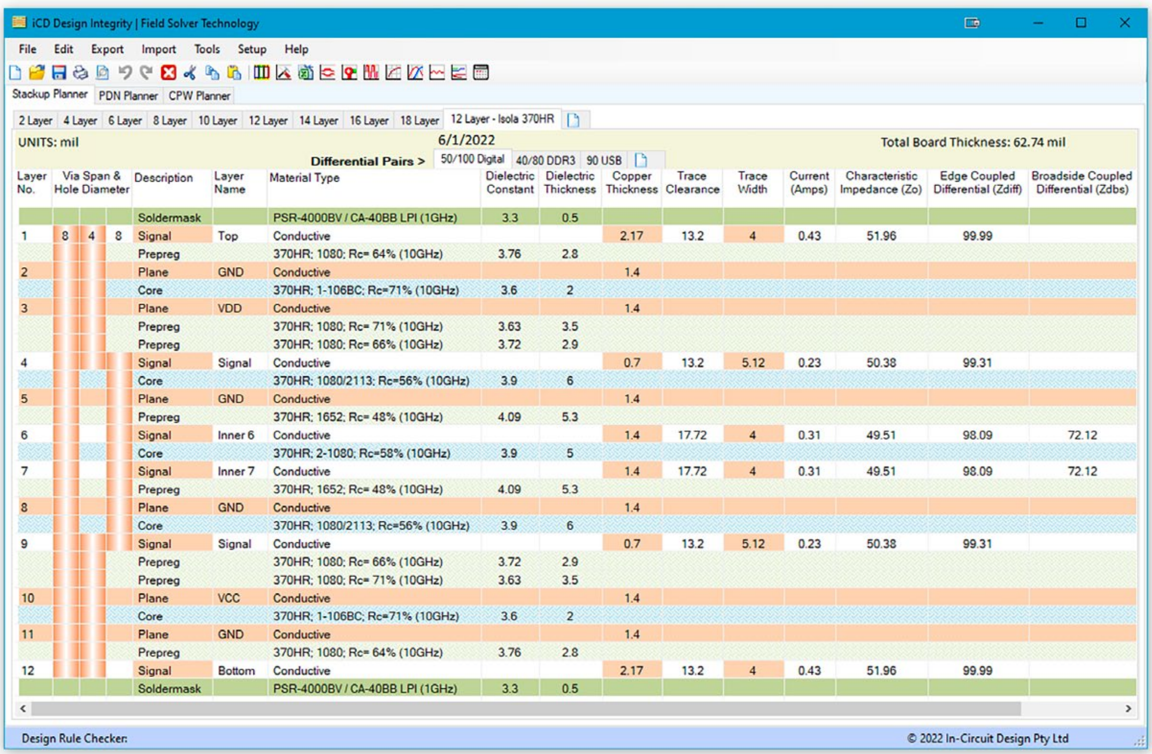
2D Field Solver—An Essential Tool for High-speed PCB Design

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.



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							

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-

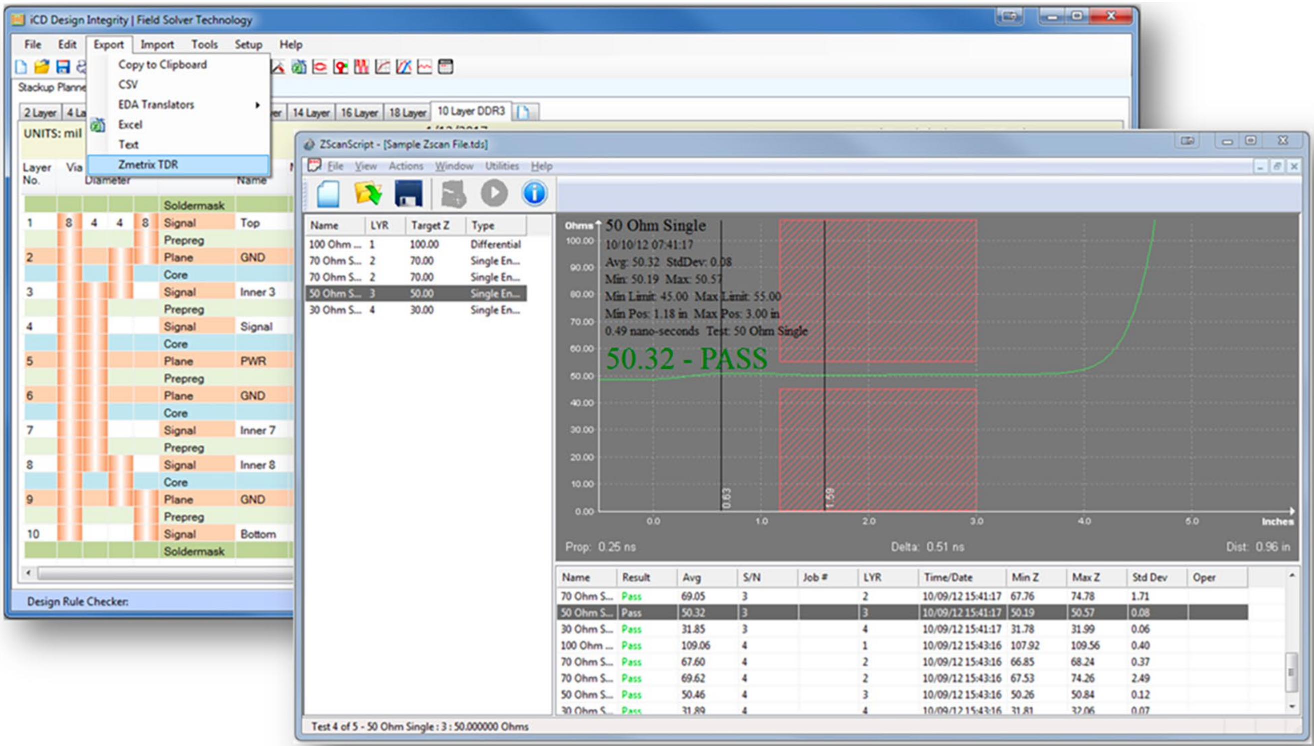


Figure 2: The iCD Stackup Planner coupled with the ZMetrix TDR and ZScan software.

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

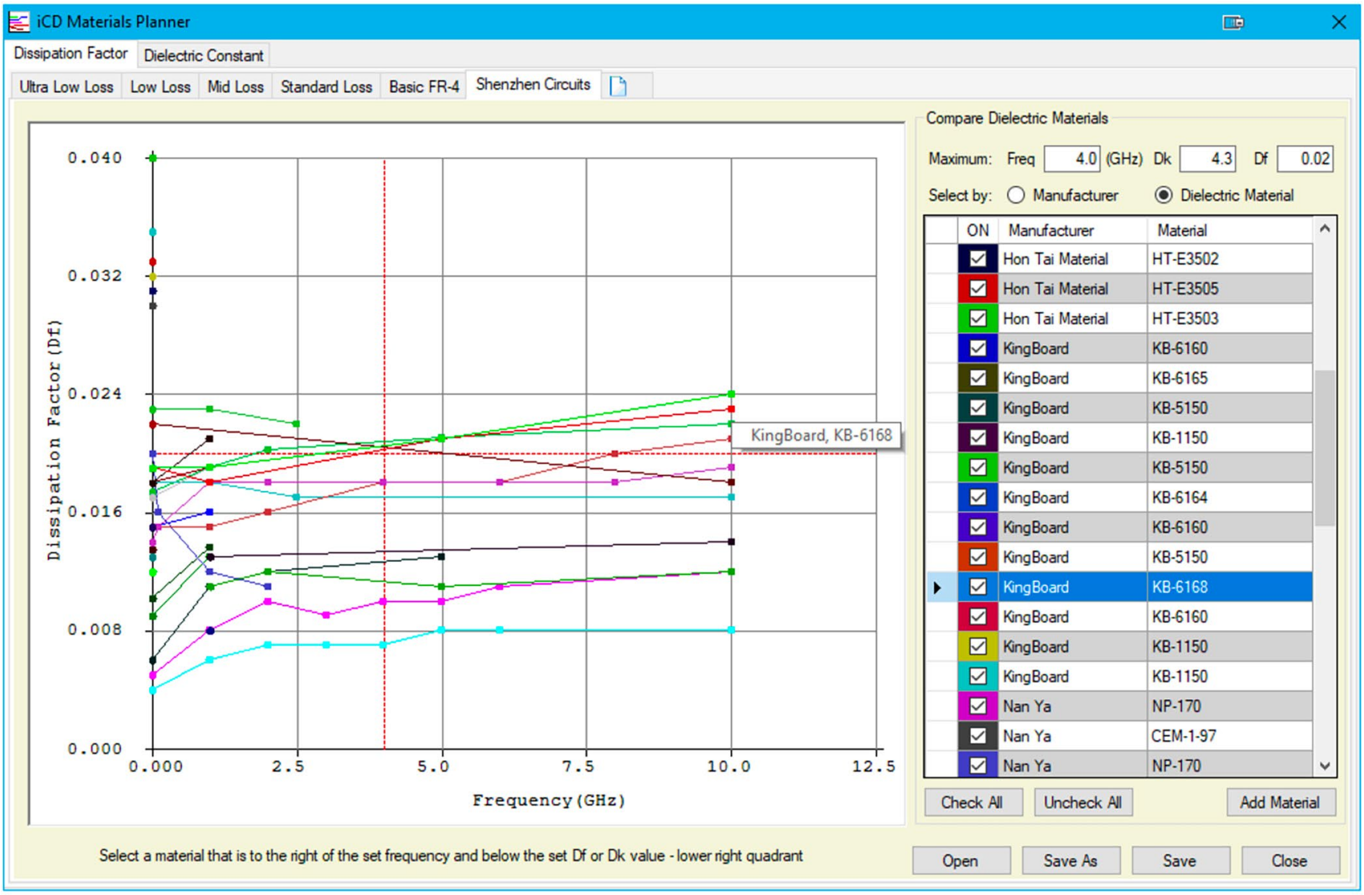


Figure 3: Example of a fabricator's dielectric materials loss profile. (Source: iCD Materials Planner)

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)

Supply Chain Resilience, Part Two: The Solution

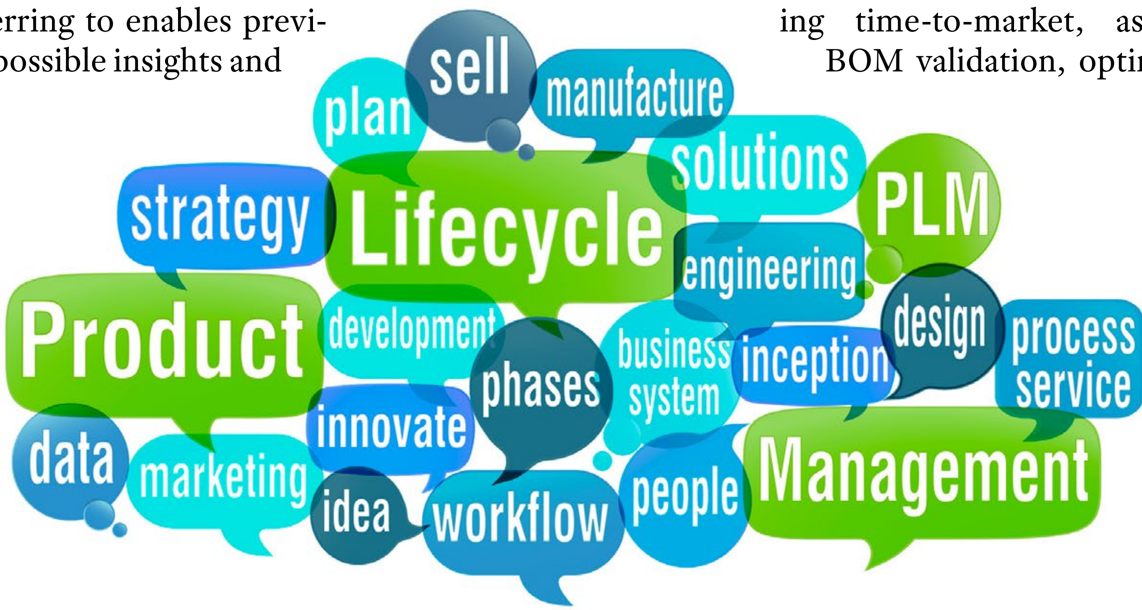
Digital Transformation

by Stephen V. Chavez, SIEMENS EDA

In part one of this two-part series on supply chain resilience, we addressed "the problem" being witnessed throughout the electronics industry regarding supply chain disruptions and their negative effects. These impacts spotlight a worldwide vulnerability that has been brewing beneath the surface, quietly growing for many years, and which was further magnified by the global pandemic. Today, supply chain issues are daily headlines and, in one way or another, professionally or personally, we're all experiencing the negative consequences of these disruptions. The geo-political turmoil arising from the war between Russia and Ukraine presents the latest twist to the supply chain plot.

Now that we have established "the problem," let's get into "the solution." The solution I am referring to enables previously impossible insights and decision support. These bring new levels of supply chain resilience to the electronics value chain. In practice, supply-chain resilience is the capacity of a supply chain to persist, adapt, or transform in the face of change. According to McKinsey & Company, December 2021, "Supply-chain disruptions cost the average organization 45% of one year's profits over the course of a decade." This leads me to ask: How can businesses manage risk and plan for a more resilient future?

There are several available options in the industry today that can assist in addressing supply chain issues. A great example is Supplyframe NPI, an SaaS application that builds supply chain resilience into the product lifecycle from the beginning of the new product introduction (NPI) process, accelerating time-to-market, assuring BOM validation, optimizing



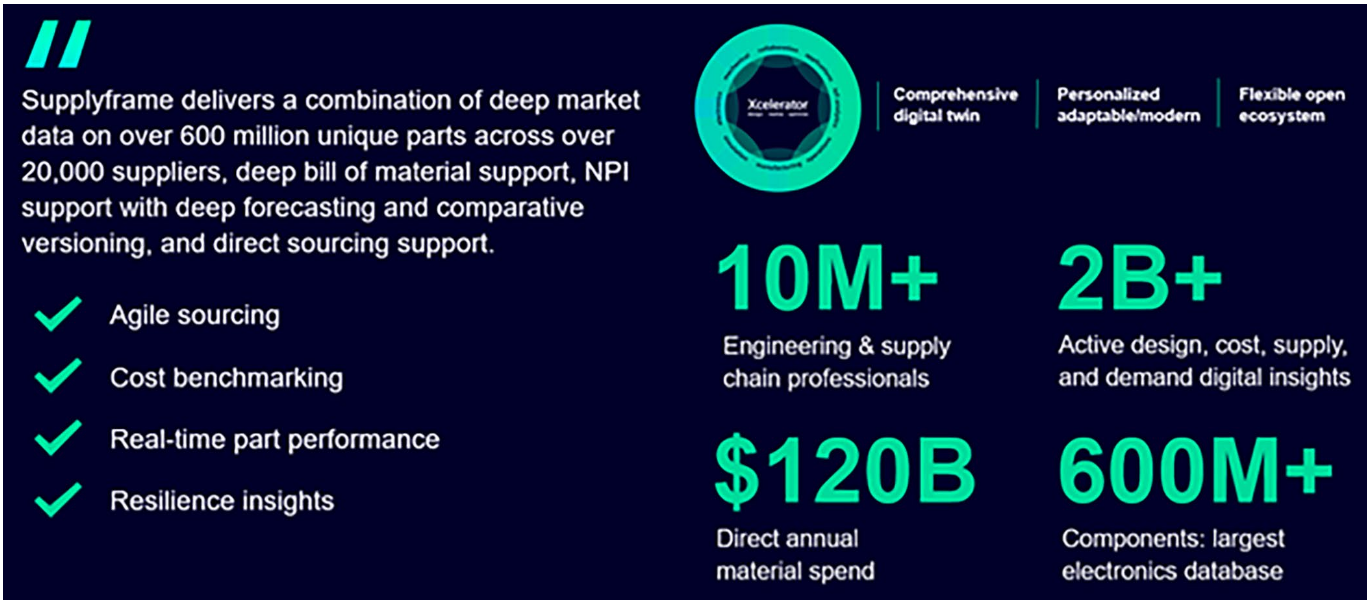


Figure 1: Supplyframe Design-to-Source Intelligence Network.

for target costs, and mitigating risk of downstream supply disruptions. At the core of Supplyframe NPI (Figure 1) is global real-time availability and lead times for over 600 million component parts, plus billions of data signals captured about part supply, demand, risk, and commercial intent. This solution is geared toward supply chain professionals at OEMs and other global manufacturers. However, it's important to note that this is not the solution to "the problem" on its own.

When we think about a complete supply-chain resilience solution (Figure 2), it needs to seamlessly connect the whole of a design organization with the electronics ecosystem (supply and manufacturing) they depend on to bring a successful product to market. At the heart of the solution is continuously updated, comprehensive component intelligence. For the design organization, this integrated source of AI-enabled data will be used to empower better decisions across the entire product lifecycle at the point of:

1. Component research and selection
2. Part creation
3. Design capture
4. The bill of materials (BOM) review and analysis
5. Product portfolio review
6. Sustaining engineering
7. All the way through to product end of life (EOL)

One solution is needed that can be fully integrated with the electronic systems design platform, and that breaks through the invisible silos in organizations so that better cross-functional decisions can be made, especially during NPI when upward of 80% of lifecycle risks and costs are locked in.

When deployed, this digitally integrated solution will reduce manual intervention, foster collaboration, improve transparency across disciplines, and reduce (or eliminate) respins between design and manufacturing. Fundamentally, this solution will "shift left" real-time component sourcing insights (e.g., cost, availability, lifecycle, compliance), in addition to performance and electrical parametrics, to the engineer's desktop (Figure 3). This allows for more informed decisions at the point of part selection and the point of design while enabling seamless collaboration between engineering and the procurement team. In addition to component sourcing from approximately 100 suppliers, engineers and designers will be presented with alternates

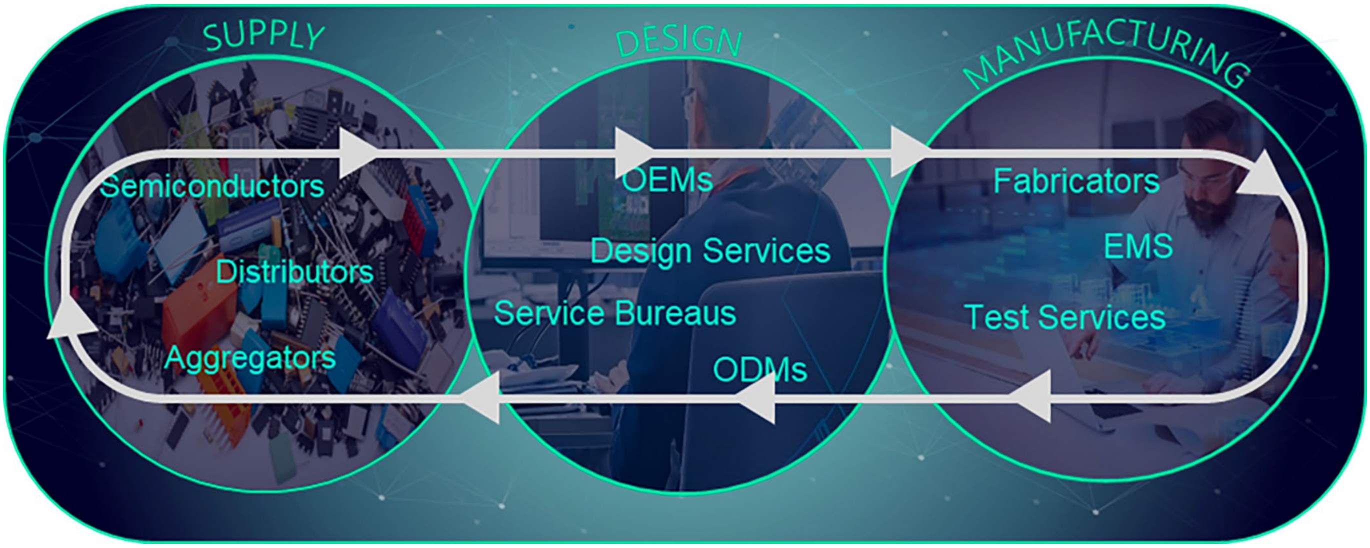


Figure 2: The foundation of supply-chain resilience.

and alerts for component risk graded at a single-part and BOM level.

With streamlined collaboration between procurement and product development established, the traditional serial handoff process is now concurrent, transparent, and integrated. Now fully aligned, design organizations can better manage, adapt, and thrive (competitive advantage) when faced with supply chain disruptions. Ultimately, designing for supply chain resilience can have a significant payoff by translating unexpected events into opportunities for growth.

Designing for supply chain resilience is based on empowering development organizations and can be implemented with these keys to resilient design, which will be deployed in three phases (Figure 4):

1. **Knowledge:** This phase is about arming engineers and teams with comprehensive

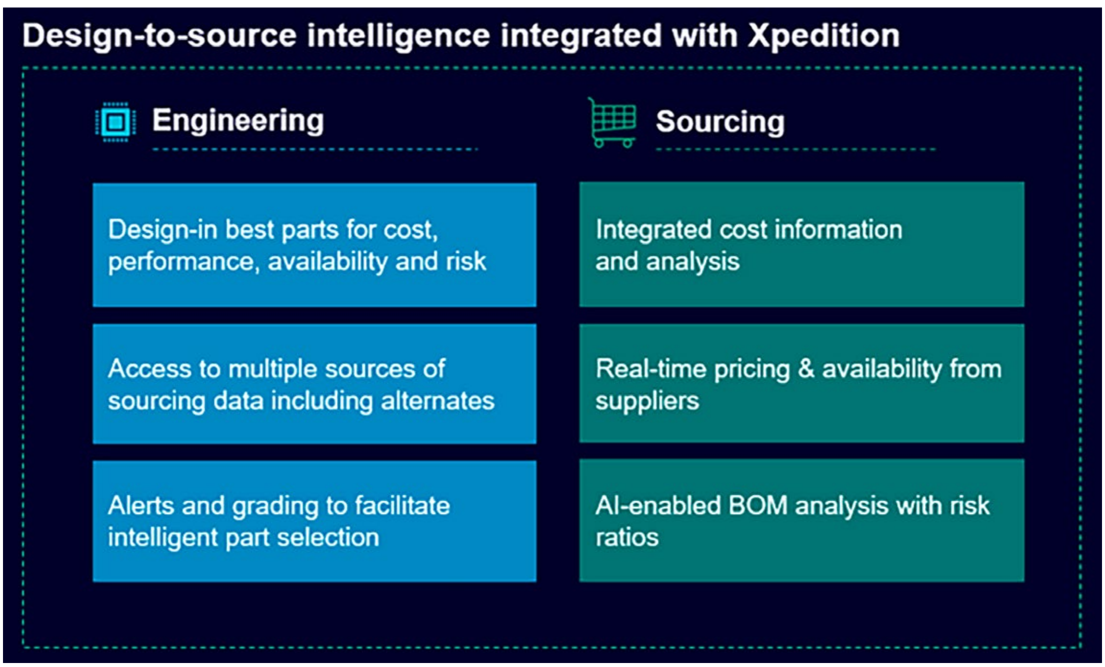


Figure 3: Shifting left design-to-source intelligence.

Figure 4: Keys to resilient design.

real-time component sourcing data so they can make more informed part decisions when the cost of change is lowest.

2. **Intelligence:** This phase will further apply the insights from component sourcing knowledge and couple it with part intelligence to empower more informed actions and workflows across the enterprise that solves for both cost and risk. This allows the enterprise to adapt quickly to supply disruptions.
3. **Optimization:** This phase will deliver a full closed-loop component management digital twin with built-in traceability, comprehended manufacturing experiences, and AI-driven analytics so that the most optimal decisions are made at every point of technical or business decision.

Today, Siemens EDA is currently empowering engineers and teams with the knowledge:

- Cloud connected access to real-time component sourcing knowledge from approximately 100 suppliers at the point of design entry:
  - Component pricing
  - Availability
  - Compliance
  - Lifecycle
- Links to component supplier datasheets
- Alerts to potential risks

While the knowledge phase is available today, work is already underway for the intelligence and optimization phases which are soon to follow. Stay tuned for future columns as well as LinkedIn activity regarding the evolution of supply chain resilience.

As stated in part one, the promising and effective solutions that are key to supply chain resilience are based on a three-phase approach that will transform organizations and shift supply chain resilience to the point of design, allowing companies to optimize not only their systems design process but also every link to the stakeholders in the global electronics value chain. By uniting the value chain with the engineer's desktop, system design companies will see higher levels of digital transformation and the greater profitability that will result as they are empowered to realize, with confidence, the challenges of tomorrow's designs today. DESIGN007

Stephen V. Chavez is senior product marketing manager, Siemens EDA, and chairman of PCEA. To read past columns, click here.

The Practical Side of Using EM Solvers

Feature Article by Heidi Barnes
KEYSIGHT TECHNOLOGIES

Electromagnetic (EM) solvers based on Maxwell's equations have proven invaluable in the advancement of digital electronics and wireline communications. Plain and simple, electrical engineers need to know what a circuit or electrical interconnect will do when excited by a dynamic or varying signal. In the signal integrity world, an interconnect that passes a DC connectivity check can completely fail at higher frequencies. In the power integrity world, a power rail that measures the correct DC voltage could easily go into oscillation when a dynamic load is applied. Learning the basic skills to fire up an EM simulator, obtain qualitative answers in minutes, and higher fidelity answers in a few days, can be the difference between sleepless nights of product failures vs. robust designs with wide design margins.

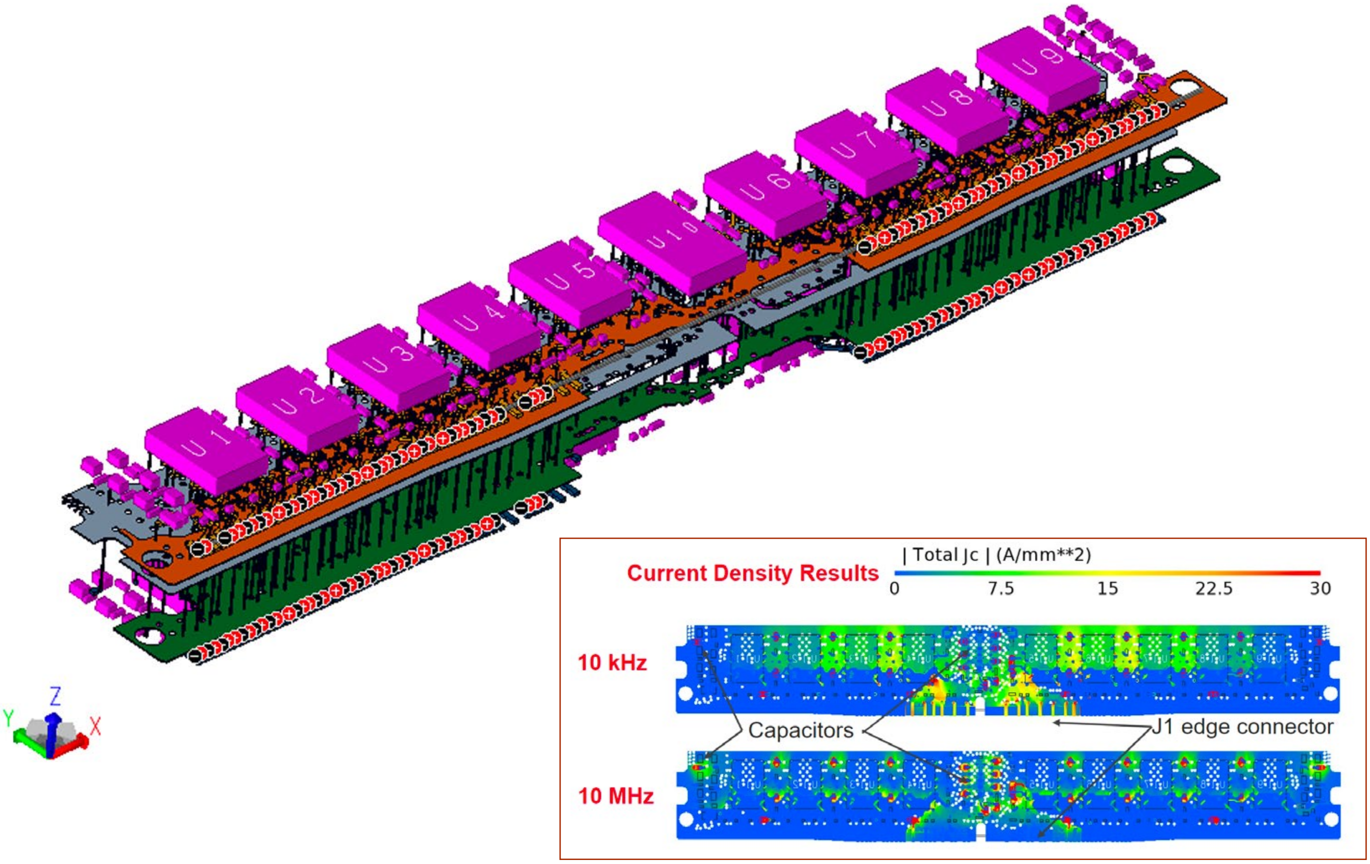


Figure 1: 3D EM simulators like PathWave ADS SIPro and PIPro make it easy to select nets and auto assign ports for fast EM setup and simulation. EM current density results on the right for a DDR4 DIMM board show how the power delivery at 10 kHz comes from the J1 edge connector, but at the higher 10 MHz frequency the power is being delivered by the on-board capacitors.

In the beginning and even today, EM simulators are constantly in search of the right balance between accuracy and speed to "mesh" a three-dimensional structure and acquire the right answer. In the search for the ultimate solution to Maxwell's equations, whether it is in the frequency domain for finite element methods (FEM) or in the time domain for finite difference time domain (FDTD), the user interface often suffers. Setting up ports, boundary conditions, and mesh topology might give the guru user additional flexibility. For most of us, we need automation to reduce the repetitive tasks.

The good news is the processing speed of modern computers and low-cost memory has made it much easier to maintain significant accuracy while increasing speeds. This has enabled increased investment in optimizing user interfaces for specific applications, like simulating multi-layer laminate PCB designs. The improved user interfaces like Keysight's PathWave ADS SIPro and PIPro rely on robust importing of EDA PCB CAD data, which include stackup, nets, and components for easy selection of nets and components for simulation. Ports can be automatically assigned, and default meshing and boundary conditions enable users to be up and running with an EM simulation in a matter of minutes.

However, simulators are notorious for "garbage in equals garbage out." Here are a few tricks of the trade to ensure a proper setup:

1. A tip from Eric Bogatin: Know what to expect by turning your complex EM model into a simple cascade of transmission lines and lumped components to get a quick idea of what the results might look like.

2. A tip from Lee Ritchey: Know what you paid for by creating a mechanical test structure where a trace is routed to the board edge on every layer, to visually check the layer-to-layer alignment, trace etch width, and mechanical height of the laminated layers.

3. My favorite tip from the measurement world: Measure something simple like a stepped impedance transition (the Beatty Standard) to verify dielectric material properties and check mechanical dimensions vs. algorithmic models and measurements.

4. A tip from the SI and PI world: Look at the data in both the time domain and the frequency domain.

5. Finally, put the model to work and check sensitivity to fabrication and component tolerances.

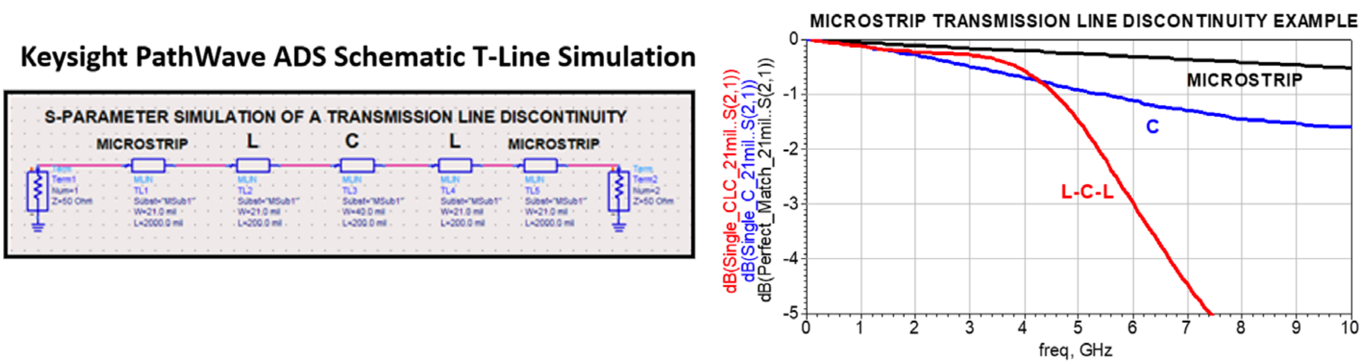


Figure 2: Simple cascade of algorithmic transmission line models in a schematic to understand what wider trace capacitive discontinuities or narrower trace inductive discontinuities will do to the transmission of high frequency signals.

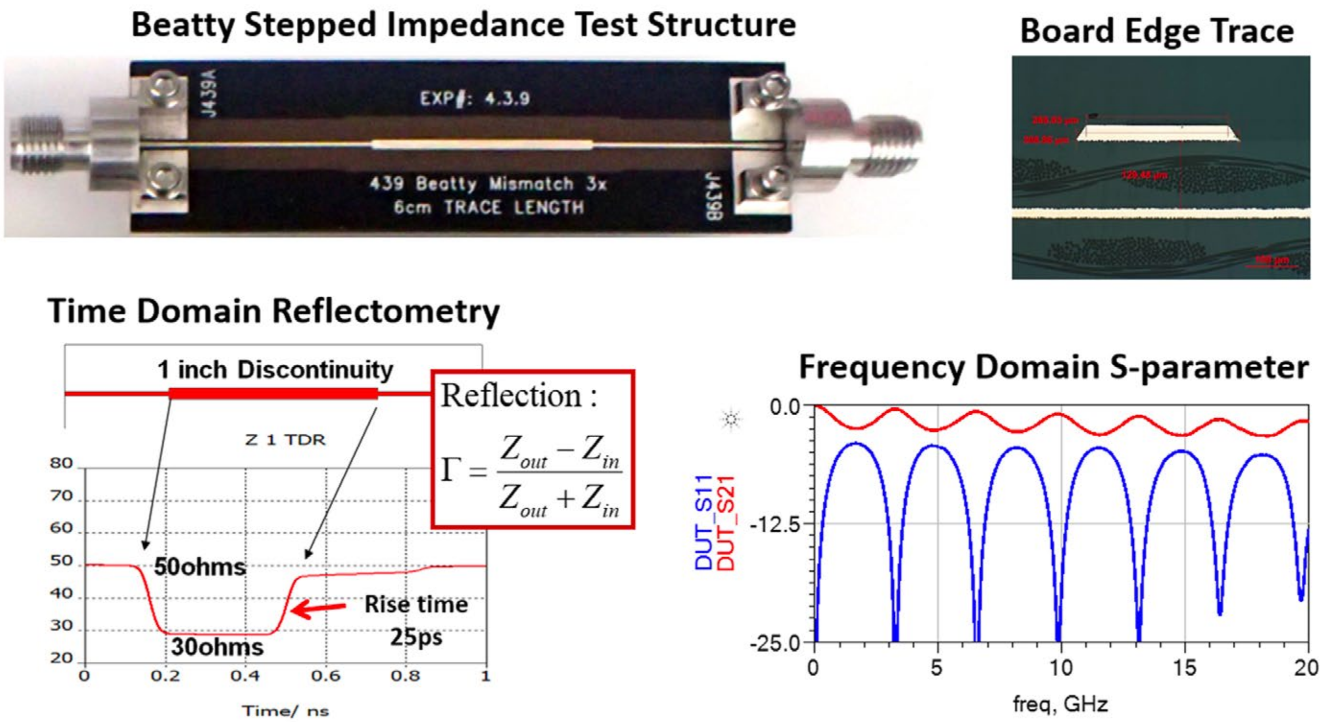


Figure 3: A simple stepped impedance Beatty test structure in the upper left is easy to understand with basic telegrapher equations for impedance reflections in the time domain shown in the graph on the bottom left. Traces to the board edge can help with verifying as-fabricated stack-up dimensions. In the bottom right is the frequency domain EM simulated S-parameter that is converted to the time domain for the TDR data.

The speed, accuracy, and ease of setup of EM simulators for PCB designs make it less acceptable to blindly leverage data sheets and past designs. Engineers need to add EM simulation to their checklist when verifying performance for dynamic operation. Simple DC checks and lumped SPICE modeling without PCB parasitics are no longer enough with the density and speed of multi-gigabit electronics powered by ever decreasing power rail voltages. **DESIGN007**

Heidi Barnes is a senior application engineer for high-speed digital applications at Keysight Technologies. She holds five patents and is a winner of NASA's Silver Snoopy award.

MilAero007 Highlights

The Government Circuit: U.S. Congress Gets Serious About Boosting U.S. PCB Sector ▶

Here in Washington, we are encouraged by new legislation indicating a bipartisan commitment to U.S. manufacturing that is long overdue. Keep reading for more details on this legislation, which would bolster the long-neglected PCB sector, and how you can express your support for it.

Standard of Excellence: From Partnership to Greatness ▶

"We choose to go to the moon in this decade. Not because it will be easy, but because it will be hard. And because that goal will serve to organize and measure the best of our energies and skills." That speech by JFK represents the onset of the greatest example of peacetime cooperative partnership this country has ever seen.

NASA Supports Small Business Research to Power Future Exploration ▶

NASA has selected hundreds of small businesses and dozens of research institutions to develop technology to help drive the future of space exploration, ranging from novel sensors and electronics to new types of software and cutting-edge materials. The project could help improve the efficiency of solar cells for space missions and use on Earth.

The Double-edged Sword of CMMC 2.0 ▶

For the past few years, those whose SMT provider organizations supply or contract with the U.S. Department of Defense (DoD) have been hearing about—or even gearing up for—implementation of the Cybersecurity Maturity Model Certification program, better known as CMMC. Today, we have CMMC 2.0, and there are several changes in the new version that impact both the standards for compliance and how you certify that compliance—especially if you run a small business.

Sierra Space, Spirit AeroSystems Partner to Bring Affordable Space Systems to Market ▶

Sierra Space and Spirit AeroSystems, Inc. announced the formation of a long-term strategic partnership intended to accelerate access to a vibrant commercial space economy of the future.

Sypris Wins Award for Electronic Warfare System Upgrade ▶

Sypris Electronics, LLC, a subsidiary of Sypris Solutions, Inc., announced that it has recently received a multi-year, follow-on award from a U.S. DoD contractor to produce and test multiple power supply modules for the upgrade of the electronic warfare suite of an important U.S. fighter aircraft program.

Deutsche Telekom, Inmarsat Collaborate with Tampnet to Strengthen European Aviation Network Connectivity ▶

Deutsche Telekom and Inmarsat have boosted the capacity of their award-winning European Aviation Network (EAN) inflight broadband solution by collaborating with Tampnet, a global leader in providing high capacity, low latency and reliable connectivity to offshore installations, mobile rigs, and vessels.

Final Finishes—ENIG and ENIPIG

Article by Anaya Vardya
AMERICAN STANDARD CIRCUITS

Introduction

Not understanding the cost drivers in the PCB manufacturing process is one of the biggest challenges facing PCB designers. This article is the latest in a series titled DFM 101 that discusses these cost drivers (from the PCB manufacturer's perspective) and the design decisions that impact product reliability.

Purposes of Final Finishes

Final finishes provide a surface for the component assembler to either solder, wire bond, or conductively attach a component pad or lead to a pad, hole, or area of a PCB. The other use for a final finish is to provide a known contact resistance and life cycle for connectors, keys, or switches. The primary purpose of a final finish is to create electrical and thermal continuity with a surface of the PCB.

There are several final finishes in use in the industry today, including:

- ENIG (electroless nickel, immersion gold)
- ENIPIG (electroless nickel, immersion palladium, immersion gold)
- ENEPIG (electroless nickel, electroless palladium, immersion gold)
- ImmAg (immersion silver)
- ImmSn (immersion tin)
- Sulfamate nickel/hard or soft gold (electrolytic nickel/gold)
- HASL (hot air solder leveling)
  - SnPb (63/37 tin/lead)
  - LF (lead free)
- OSP (organic solderability preservative)

Final finishes are primarily application driven, so there are several considerations that

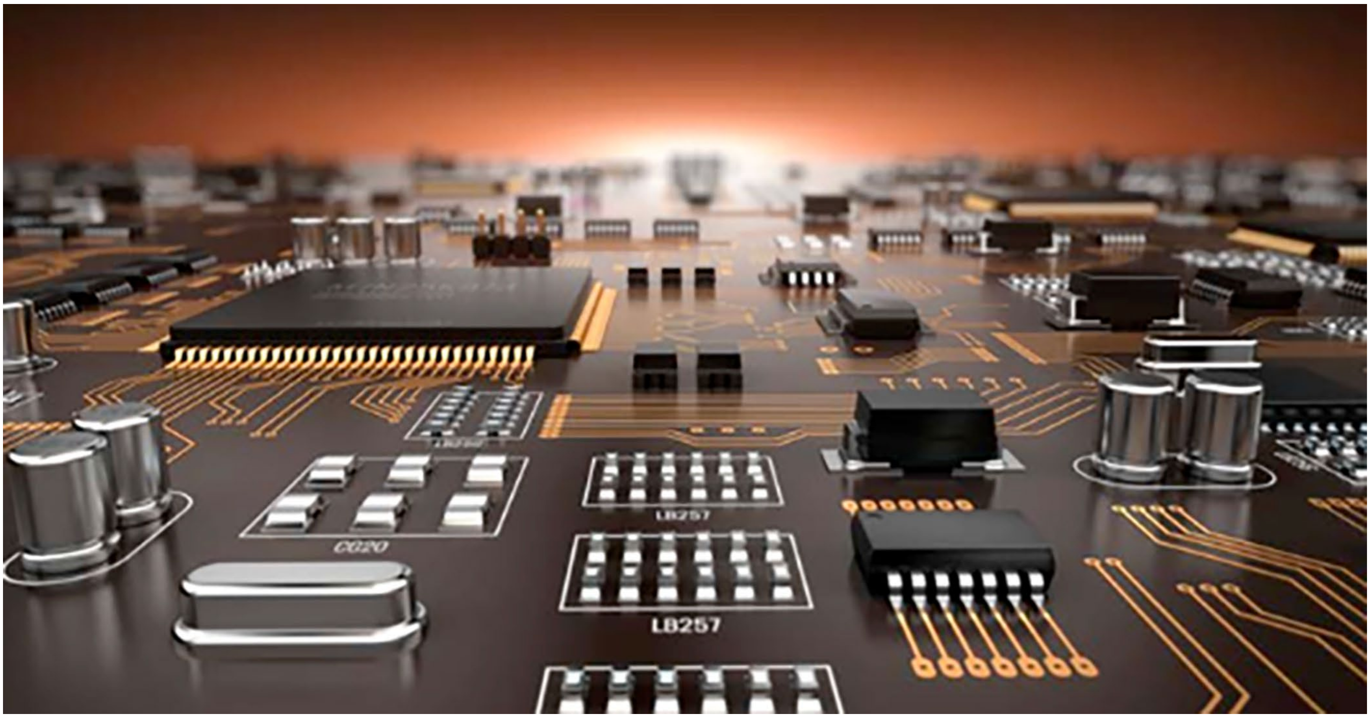


Figure 1: Lead-free finishes.

should be part of any decision to choose a final finish:

- Lead tolerant or lead-free (LF) process
- Shelf life
- Flatness
- Lead or ball pitch
- Wire bondability
- Lead insertion
- Solder joint integrity
- Corrosion resistance
- Potential problems
- Cost

Lead-free finishes are considered “RoHS” compliant (< 0.1% BW of finish, for Pb, Hg or Cd) with the single exception of tin/lead HASL. The RoHS compliant finishes include the following:

- ENIG
- ENIPIG
- ENEPIG
- ImmAg
- ImmSn
- Electrolytic nickel/gold
- LF HASL
- OSP

Lead-free PCBs require that the standard HASL surface finish cannot be used. There is still a significant amount of discussion on what the long-term surface finishes will be. Currently, the immersion silver and OSP surface finishes are the most prevalently specified surface finishes for solderable PCBs. Immersion tin is the prevalent surface finish for press-fit backplanes. Please contact your PCB fabricator for current information on where industry specifications are heading. I will discuss just two of them here.

Two Final Finishes

ENIG (Electroless Nickel, Immersion Gold)

ENIG is a two-layer metallic coating of 1.97 µin [0.05 µm] minimum, 2-5 µin [0.05 to 0.012 µm] typical, immersion deposited Au over 118.1-236.2 µin [3 to 6 µm] electroless deposited nickel, per IPC-4552 nominal pad size of 0.060” x 0.060” (1.5 mm x 1.5 mm).

ENIG is typically used for flat surface/fine pitch devices. The benefits of ENIG are a long shelf-life and environmental resistance, as well as low contact resistance.

This process plates a thin coat of nickel covered by a thin layer of gold. The gold provides a very good solderable surface. When components are soldered onto these pads, the gold diffuses into the solder joint. The gold layer is very thin, so it won't reduce the solder joint strength. This process is generally not utilized for high reliability, long lifetime, or high vibration applications.

ENIPIG (Electroless Nickel, Immersion Palladium, Immersion Gold)

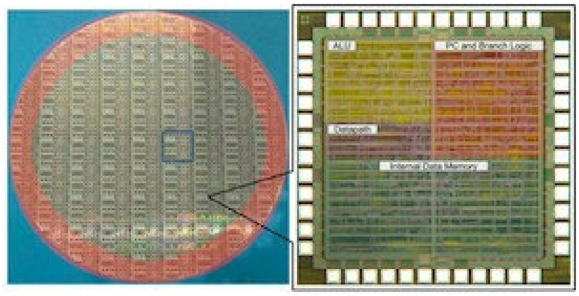
ENIPIG is a three-layer metallic coating of 1.97 µin [0.05 µm] minimum, 2-5 µin [0.05 to 0.012 µm] typical, immersion deposited Au over 2 to 12 µin [0.05 to 0.30 µm] immersion deposited Pd over 118.1-236.2 µin [3 to 6 µm] electroless deposited nickel, per IPC-4556, nominal pad size of 0.060" x 0.060" (1.5 mm x 1.5 mm).

ENIPIG is typically used for flat surface/fine pitch devices. The benefits of ENIPIG are a long shelf life and environmental resistance, as well as low contact resistance and excellent solder joint with SAC solders.

Understanding the cost drivers in PCB fabrication and early engagement between the designer and the fabricator are crucial elements that lead to cost-effective design success. Following your fabricator's DFM guidelines is the first place to start. **DESIGN007**

Anaya Vardya is president and CEO of American Standard Circuits; co-author of *The Printed Circuit Designer's Guide to... Fundamentals of RF/Microwave PCBs* and *Flex and Rigid-Flex Fundamentals;* and author of *Thermal Management: A Fabricator's Perspective*. Visit I-007eBooks.com to download these and other educational titles. He also co-authored "Fundamentals of Printed Circuit Board Technologies" and provides a discussion of flex and rigid flex PCBs at RealTime with... American Standard Circuits.

University of Illinois Researchers Create Low-cost, High-yield Plastic Microprocessors



Supercomputers can crunch massive data sets and give us insights into the wonders of the universe. But a sub-penny microprocessor that flexes to fit on a beer bottle and tells you whether your brew is still cold? Creating that technology has been surprisingly more difficult to achieve.

Recently, a team of researchers from the University of Illinois Urbana-Champaign said: hold my beer. Working with British flexible electronics manufacturer PragmatIC Semiconductor, the researchers developed the first commercially viable flexible plastic microprocessor chips, called FlexiCores, that can be manufactured at scale for less than a penny per unit.

The new processors could help even everyday objects become "smart."

"For example, you could put processors on bandages to detect whether a wound is healing, or add them to consumer goods packaging to track progress along the supply chain," said Rakesh Kumar, a professor of electrical and computer engineering and researcher in the Coordinated Science Lab at UIUC. "The challenge has been creating a processor that can be both cheaply produced and flexible enough to fit snugly even against uneven surfaces on our body, packages, or beer bottles."

The low gate-count in FlexiCores improves yield and reduces the bill of materials—the amount of material needed to produce each chip.

"These chips combine the flexibility and cost benefits of plastic technology with the high yield and low bill of materials enabled by our architecture," he said. "It will be interesting to see where we go from here."

(Source: University of Illinois)

Can Solvent-free UV Cure Coatings Increase Stability and Throughput?

Sensible Design

by Saskia Hogan, ELECTROLUBE

With environmental awareness at an all-time high, companies are looking for greener options in terms of the chemical products they use for the manufacture and protection of electronics, including conformal coatings that are applied to protect the metal surfaces from corrosion, condensation, dielectric breakdowns, and that mitigate against tin whisker formation and conductive metal particles.

Traditionally, these materials have been solvent-based, resulting in low viscosity, easy-to-use materials. However, the use of solvents has become more restricted and there is increasing demand for solvent-free technology. In this month's column, I'll be looking at a particular solvent-free coating group—UV cure coatings. I will examine the benefits that these particular coatings bring to the production line and unveil test data to reveal their performance values.

What Are the Key Benefits of Solvent-free Coatings to Manufacturers?

Aside from solvent-free materials being a smart choice, whereby solvent emissions from your facility will be much lower, there are cost savings due to compliance with legislative requirements, depending upon local, regional, and national legislation. UV-cure materials are also non-flammable and are a lower hazard to human health, resulting in potential cost-savings on insurance. Often, the energy demands required for curing these materials are much lower than for solvent-based materials, resulting in reduced energy bills and reduced CO_2 emissions.

But how do they perform? Solvent-free materials are generally applied slightly more thickly, resulting in increased coverage, and therefore greater protective capability. The formulations are also more modern and deliver

PREMIER GLOBAL SUPPLIER to the **MEDICAL DEVICE INDUSTRY**

FLEXIBLE CIRCUITS | EMS/ASSEMBLY | CONTRACT MANUFACTURING

Industry leading Flex Design Support

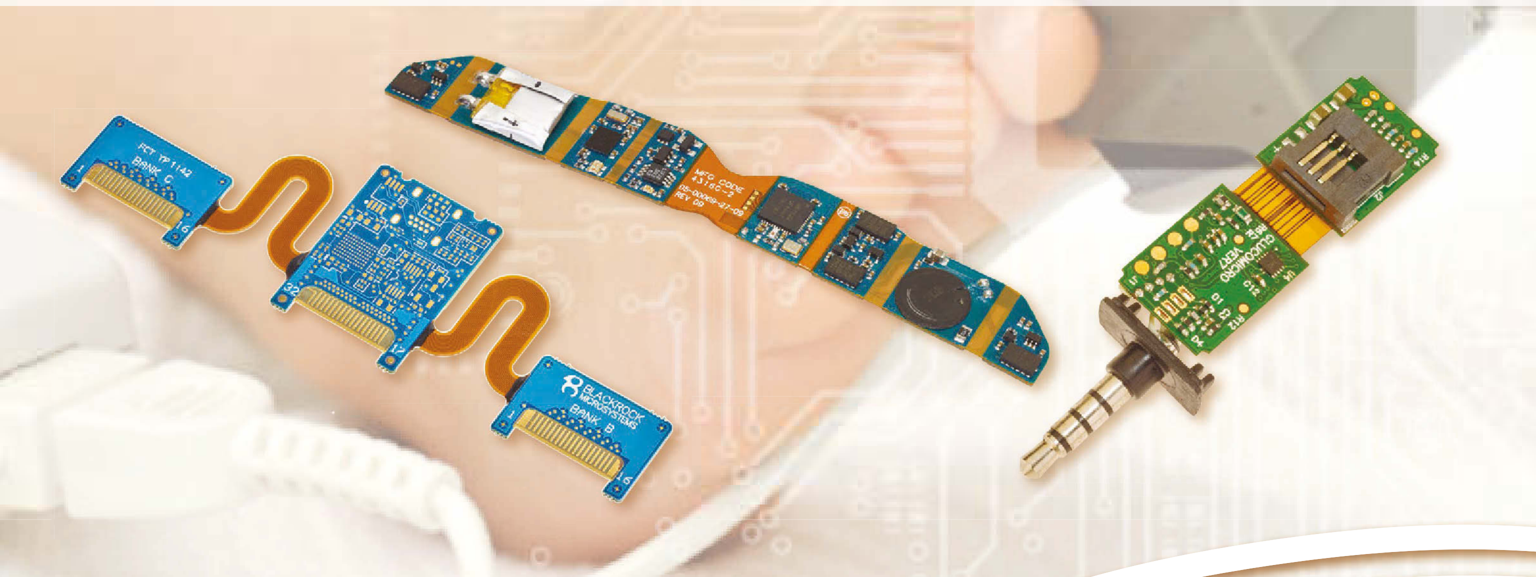
Flexible Circuits, Rigid Flex, Flexible Heat-ers Membrane Switches, Plastic Moldings

Specialized EMS/Assembly Services

Product Module to Complete Product Box Builds

ISO 13485 – FDA registered Box Builds

EXPLORE the POSSIBILITIES!



FLEXIBLE CIRCUIT Technologies®

Flexible Circuit Technologies
9850 51st Ave. N. | Plymouth, MN 55442
www.flexiblecircuit.com | +1-763-545-3333

a better capability to meet the increased protective demands of new applications, e.g., automotive industry thermal shock cycling or aerospace increased condensation resistance. UV-curable materials cure extremely rapidly (in seconds) when exposed to UV radiation of a suitable wavelength and intensity, making them an extremely interesting technology for increasing factory throughput and reducing production footprint. UV-curable materials also provide good protective properties and improved chemical resistance compared with other cure types.

Why Do We Need a Secondary Cure?

Unfortunately, light only travels in straight lines and curing is line of sight only. Due to the 3D nature of a PCB, it is likely that some areas of the board will not attain full exposure to the light, particularly material on the backside of component leads and underneath components. For this reason, UV-curable materials need to contain a secondary cure mechanism, which can be a moisture, chemical, or heat-based mechanism. Moisture-curing has traditionally been favored because it requires no additional processes. However, although the material is generally well cured in light exposed areas, the diffusion of moisture into the coating and the emission of the leaving groups can be difficult. The better the coating barrier, the longer the secondary-curing process; many days, weeks, months, and even years have been reported.

What Are the Issues With Heat Activated Curing?

Heat-activated secondary curing processes require additional processes and time, which eliminates the benefit of the rapid primary cure. Speaking of which, the initial rapid cure can generate significant levels of stress, and seldom leads to more than 70–80% of the theoretical conversion to polymer, meaning that the materials can contain reactive groups that remain dormant. Once exposed to high temperatures (100°C+) additional polymerisation can take place, resulting in the materials continuing to harden, change properties, and be more prone to cracking during thermal shock transitions.

Chemical Secondary Cure Time and Thermal Aging

Materials containing a chemical secondary process will cure completely within six to eight hours at room temperature after exposure to suitable long wavelength UV light. Due to the unique formulation of these materials, residual stress is minimised and the cure proceeds to a very high level of conversion, resulting in minimal changes in properties during thermal aging. After thermal age testing, the results showed that the conventional materials tended to be very stiff and inelastic at sub-ambient temperatures, whereas the secondary chemical cure system remained elastic until -20°C, but still retained a degree of elongation even at -40°C.

UV Chemical and UV Moisture Cure Test Results

While the lack of changes in physical properties during thermal aging are an important parameter in material selection, the key to performance in an end-user application is whether a material can survive the required thermal shock profile without cracking or impacting further stress on solder joints. To investigate this, we selectively coated 12 automotive engine control units at a normal thickness with five different coatings in a full thermal shock experiment test from -40°C to +130°C, 0°C to 130°C, and -40°C to +60°C.

After 250 cycles of each test cycle, the 12 boards were visually inspected for evidence of cracks in the coating, and the most striking observation from the tests was that the materials which had the most stable properties during the thermal aging process were the chemical cure and UV/chemical cure materials. The

UV/chemical cure remained virtually crack free after 1,000 thermal shock cycles (-40°C to +130°C) and also completed curing within six hours compared to the extremely long time required to complete the UV/moisture cure. The UV/chemical-cure material had far greater stability of the key physical properties during thermal aging when compared to UV/moisture-cure materials. This increased stability led to a considerably improved thermal shock performance when compared to the traditional UV/moisture-cure materials.

So, there you have it: Environmentally friendly materials are available now, including our unique bio-based UVCLX coating, which guarantees full cure within 24 hours and contains 75% bio-organic content from renewable sources. New developments are in the pipeline and, with time, these UV cure, solvent-free, and bio-based materials will open up new avenues for compliance across many disciplines. **DESIGN007**

Saskia Hogan is global product manager, conformal coatings, at Electrolube. To read past columns from Electrolube, click here. Download your free copy of Electrolube's book, *The Printed Circuit Assembler's Guide to... Conformal Coatings for Harsh Environments,* and watch the micro webinar series "Coatings Uncoated!"

All Systems Go

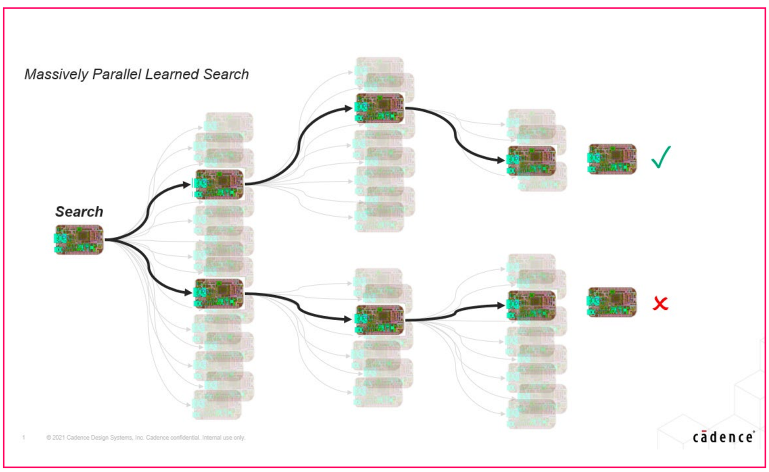
Accelerate Your PCB Designs with Machine Learning

by Jorge Gonzalez and Luke Roberto
CADENCE

Even though we hear the terms artificial intelligence (AI) and machine learning (ML) almost daily, there's still a lot of confusion about the actual meaning of these designations. In a nutshell, AI is an umbrella term embracing technologies that empower machines to simulate human behavior. ML is a subset of AI that allows machines to automatically learn from past data and events without explicitly being programmed to do so.

Another perception is that AI is a relatively new development, whereas, in reality, scientists and engineers have been working on AI-related technologies for decades. In fact, the founding event that led to AI as we know it today was a Dartmouth workshop—the Dartmouth Summer Research Project on Artificial Intelligence—in 1956.

We can divide the development of AI (and ML) into two eras. During the first era, until circa 2012, AI computational requirements doubled approximately every two years, roughly tracking Moore's Law. However, around 2012, developments in AI architectures and algorithms led to an inflection point that marked the beginning of the second (modern) era of AI, in which compute requirements for enterprise-level AI systems started to double approximately every three-and-a-half months (Figure 1). Fortunately, this demand for computational power can be satisfied using the tremendous XPU (CPU, GPU, FPGA, etc.) and memory resources made available by modern cloud computing environments.



Today, AI and ML technologies are becoming ubiquitous, from handwriting recognition applications on tablet computers to natural language speech recognition and generation in smart appliances, machine vision with object detection and recognition capabilities in robots, predictive maintenance applications in the automotive industry—the list goes on.

To read this entire column, click here.

Manufacturing Documentation: Keep the Builder in Mind

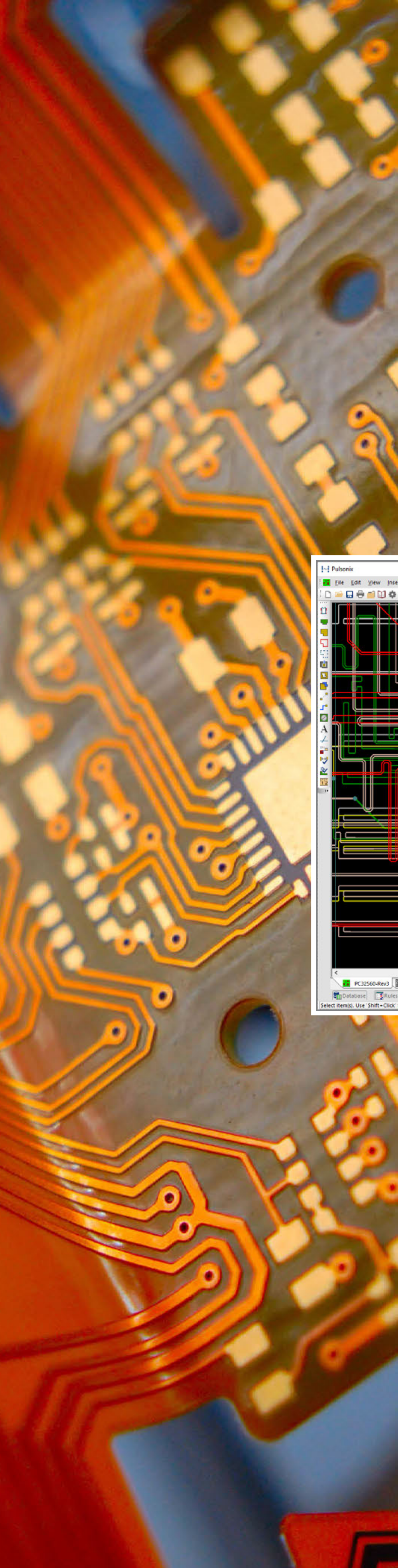
Tim's Takeaways

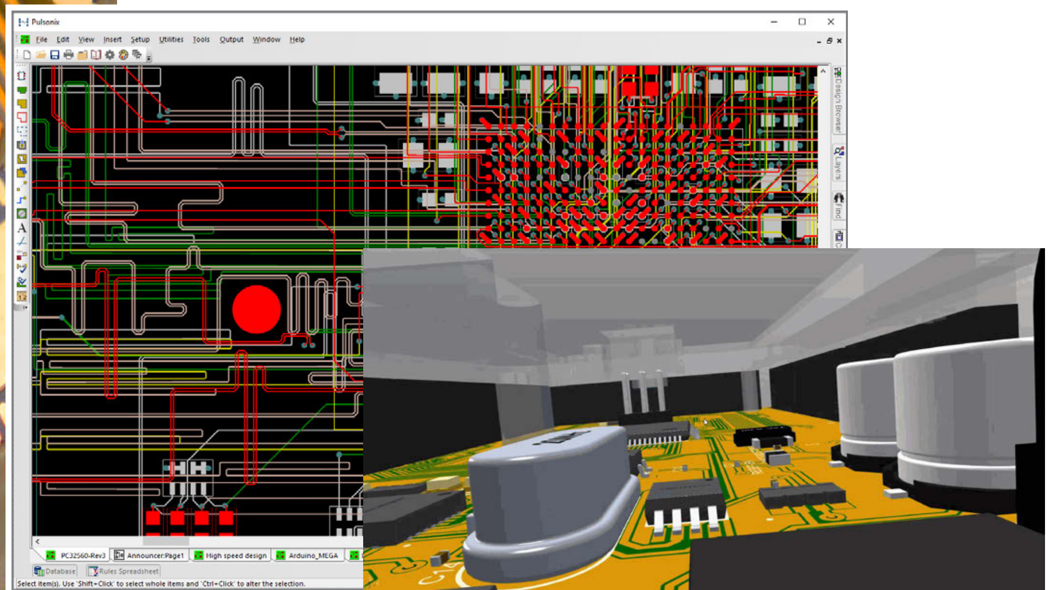
by Tim Haag, FIRST PAGE SAGE

It is the end of May, which among other things, meant that the Major League Baseball season was once again in full swing (pun intended). While my wife was happily settled into the couch with her Seattle Mariners cap, T-shirt, blanket, and coffee mug cheering on J.P Crawford and the rest of the team, I decided to turn on a movie instead. So, to keep with the spirit of the season, I re-watched "Field of Dreams, and was again mesmerized by the voice that speaks to Kevin Costner's character: "If you build it, they will come."

As circuit board designers, it's probably not all that unusual to hear similar voices speaking to us, especially after staring at a layout for hours, and hours, and hours. But in our case, the message is typically a little different, and sounds more like, "If you document it correctly, they will build it."

Yes, I'm talking about PCB documentation, and the importance of creating clear and understandable instructions for building the circuit board that you've just designed. To illustrate how essential this is, let me tell you about my latest project: the trike. The month of May isn't just about baseball in our house, it is also my wife's birthday. This year I gave her an adult-sized trike for cruising around the neighborhood and exploring campgrounds during our vacations. This trike promises to be a lot of fun and she is really looking forward to riding it, but first we have to put it together





This experience has given me a greater appreciation than ever before for the necessity of creating clear and concise manufacturing instructions when I design a printed circuit board. After all, if the rear wheels fall off the trike because I didn't mount them correctly to the "axis," then there is the very real potential for embarrassment or injury, neither of which is a desirable situation. Likewise, unclear fabrication and assembly documentation could also result in some negative consequences due to lost time and revenue from un-manufacturable circuit boards. To avoid problems like these, and many others as well, here are some ideas I try to keep in mind while documenting a PCB design:

and that is where the problem comes in. The assembly manual for the trike must be the absolute worst set of documentation that I've ever seen.

Not only is this document filled with translation errors from the language it was initially written in, but there are plenty of other spelling and grammatical errors as well. The manual appears to have been written for a different model of trike than what we have, as the pictures don't match and it isn't laid out sequentially from one step to the next as you would expect. For instance, step one shows some pre-mounted bolts in different locations than where they actually were installed at the factory. Step two describes how to mount the rear axis (and I'm guessing they really meant "axle" here instead) to the frame with a picture showing the wheels already mounted on the axle. Step three then instructs you how to mount those wheels to the axle even though the previous step already had them in place. Trying to make sense of it all requires a lot of detective work, and I'm constantly going back and forth between the manual and online videos to reverse engineer this mess and figure out what really needs to be done. However, I must confess, in spite of the extra effort taking up far more time than it should, I'm actually loving the technical challenge of it all.

1. Find out what your manufacturer needs and expects.

Many manufacturers end up spending unexpected time conferring with their customers over missing or unclear details in their build instructions. If the information isn't clear from the documentation, or is different than what was already agreed on, the manufacturer must resolve the differences before they can proceed with circuit board production. This lost time can increase the overall production cost or delay the build, but it can be easily avoided by talking first with the manufacturer and finding out exactly what they need. Some manufacturers have even told me that they end up correcting and regenerating their clients' prototype documentation to help them transition more efficiently into full production. If this documentation was created clearly to begin with, a lot of time and effort could have been saved.

2. Don't skimp on the details.

Manufacturing drawings and other build instructions usually end up with a lot of data and information in them. And although it is good to cut back on superfluous data for overall clarity in the documentation, designers need to avoid the temptation to cut back too much. Be sure to include adequate views, notes, and other details that specify how your design is to be fabricated and assembled. Also, be careful when using "standard" notes and other drawing elements stored in your PCB libraries. Stock drawing elements like these are great time savers, but they must be closely examined first to see if they need updates to match the specific requirements of the design. Minute details such as tolerances or surface finishes may be incorrect and can easily be overlooked.

3. Leverage the power of your design tools.

PCB design tools today have some incredible features within them to help with the creation of drawings and other manufacturing documentation, but these tools don't always get used. Often designers face some serious time challenges when getting a design out the door, and typically the drawings are one of the last pieces to the puzzle to be worked on. Therefore, many designers will avoid experimenting with new CAD features and simply go with what has always worked in the past. One of the biggest favors you can do for yourself is to invest some time (before crunch time) and find out what kind of unique features your design tools might have that you aren't aware of. View generators, table and chart wizards, automated dimensioning, and other advanced CAD features could end up saving you a lot of additional time and effort in the future.

4. During review, ask yourself, "Could I build what I just documented?"

I think that everyone is used to reviewing their work for completeness and accuracy, but have you ever looked at a drawing and asked yourself, "Could I build that?" I know that we are PCB designers and not manufacturers, but we are the ones creating the documentation for fabrication and assembly shops to build our designs. We should be able to tell if the manufacturers can do the job based on the documents we are sending them. If you are unfamiliar with the fab and assembly processes, spend some time with your vendors to better understand what they do and what they need from you to do it. As a designer, it is extremely interesting to tour the manufacturing facilities you are using, and see exactly how your design is being brought to life. After all, the more we know about what we are telling our vendors to do, the better the final product will be.

We should be able to tell if the manufacturers can do the job based on the documents we are sending them.

These are some of the ideas that run through my head when I'm preparing manufacturing documentation, and I hope that they will help you too. And with that, it's time for me to transition back out to the garage and pick up where I left off. I think that I'm ready to start step four of the great trike project, and I'm looking forward to it. Oh, and a quick note to self: first check and confirm that those rear wheels really are as tightly mounted as they should be. Until next time, then, everyone, keep on designing. DESIGN007

Tim Haag writes technical, thought-leadership content for First Page Sage on his longtime career as a PCB designer and EDA technologist. To read past columns, click here.

Flex007 Highlights

Flexible Patch Detects Real-time Changes in Water Temperature ▶

Researchers at Tokyo Tech invent a flexible patch containing carbon nanotubes and stretchable conductors that can fit inside a pipe to detect real-time changes in water temperature or the presence of contaminants, which may improve sanitation during industrial processes.

Cicor Expects 30% Growth in 1H22 Due to Strong Business Performance ▶

Cicor Group expects a sales growth of around 30% in the first half of 2022 compared to the same period in 2021 as a result of acquisitions made and the successful management of challenges in the current difficult business environment.

Smart, Dissolving Pacemaker Communicates with Body-area Sensor and Control Network ▶

Last summer, Northwestern University researchers introduced the first-ever transient pacemaker—a fully implantable, wireless device that harmlessly dissolves in the body after it's no longer needed. Now, they unveil a new, smart version that is integrated into a coordinated network of soft, flexible, wireless, wearable sensors and control units placed around the upper body.

Compeq Reports 27% Growth in May Revenues ▶

Taiwan-based Compeq Manufacturing Co. Ltd, a manufacturer of HDI, rigid-flex PCBs, and flex PCBs, has reported unaudited net sales of NT$5.1 billion ($173.9 million at $1:NT$29.53) for May, up by 26.7% year-on-year, but down by 1.7% from the previous month.

Ventec's Book on Thermal Management: The Summer Sequel You've Been Waiting For ▶

I-Connect007 is excited to announce the release of the second title in Ventec's series on thermal management, The Printed Circuit Designer's Guide to... Thermal Management with Insulated Metal Substrates, Volume 2. This second volume covers the latest developments in the field of thermal management, particularly in insulated metal substrates, using state-of-the-art products as examples, and focusing on specific solutions and enhanced properties of IMS.

Nan Ya PCB Posts 22% YoY Revenue Growth in May ▶

Nan Ya Printed Circuit Board Corp. (Nan Ya PCB), a Taiwan-based manufacturer of single-sided PCBs, HDI PCBs, and rigid-flex PCBs for motherboards, desktop and notebook PCs, home appliances, smartphones, and gaming consoles, has posted unaudited sales of NT$5.2 billion ($175.69 million at $1=NT$29.76) in May 2022, up by 18% from the previous month and by 21.6% year-on-year.

Taiflex to Expand in Thailand; Signs Land Purchase and Sale Agreement ▶

Taiflex Scientific Co., LTD. has signed the Amata City Chonburi Industrial Estate land purchase contract with AMATA Corporation in Bangkok. With the signing of the Regional Comprehensive Economic Partnership, Taiflex plans to set up overseas production bases to rise its global competitive advantage.

The Importance of Rigid-Flex PCB Design Guidelines

Article by Cody Stetzel
CADENCE DESIGN SYSTEMS

I have the tendency to try to replicate the delicacies I've ordered at restaurants in my own kitchen. One of my latest attempts at creating restaurant-worthy dishes was a Korean pancake that's crispy on the outside but soft on the inside. With my amateur cooking skills, it proved to be an impossible task—I could either make only a hard pancake or one that was total fluff.

While I'm still struggling to figure out the trick to bringing together the different textures of a Korean pancake, I've had more success in bringing together the hardboard elements and flexible PCB elements of a rigid-flex PCB. Compared to making Korean pancakes, striking the right balance of flexibility and rigidity on a rigid-flex PCB is easy if you abide by rigid-flex PCB design guidelines.

What Is a Rigid-Flex PCB?

For those who have spent their careers designing conventional PCBs on the FR-4 substrate, a rigid-flex PCB may be unfamiliar territory. As the name implies, a rigid-flex PCB is a PCB that combines both elements of a hardboard and a flexible PCB in a single piece.

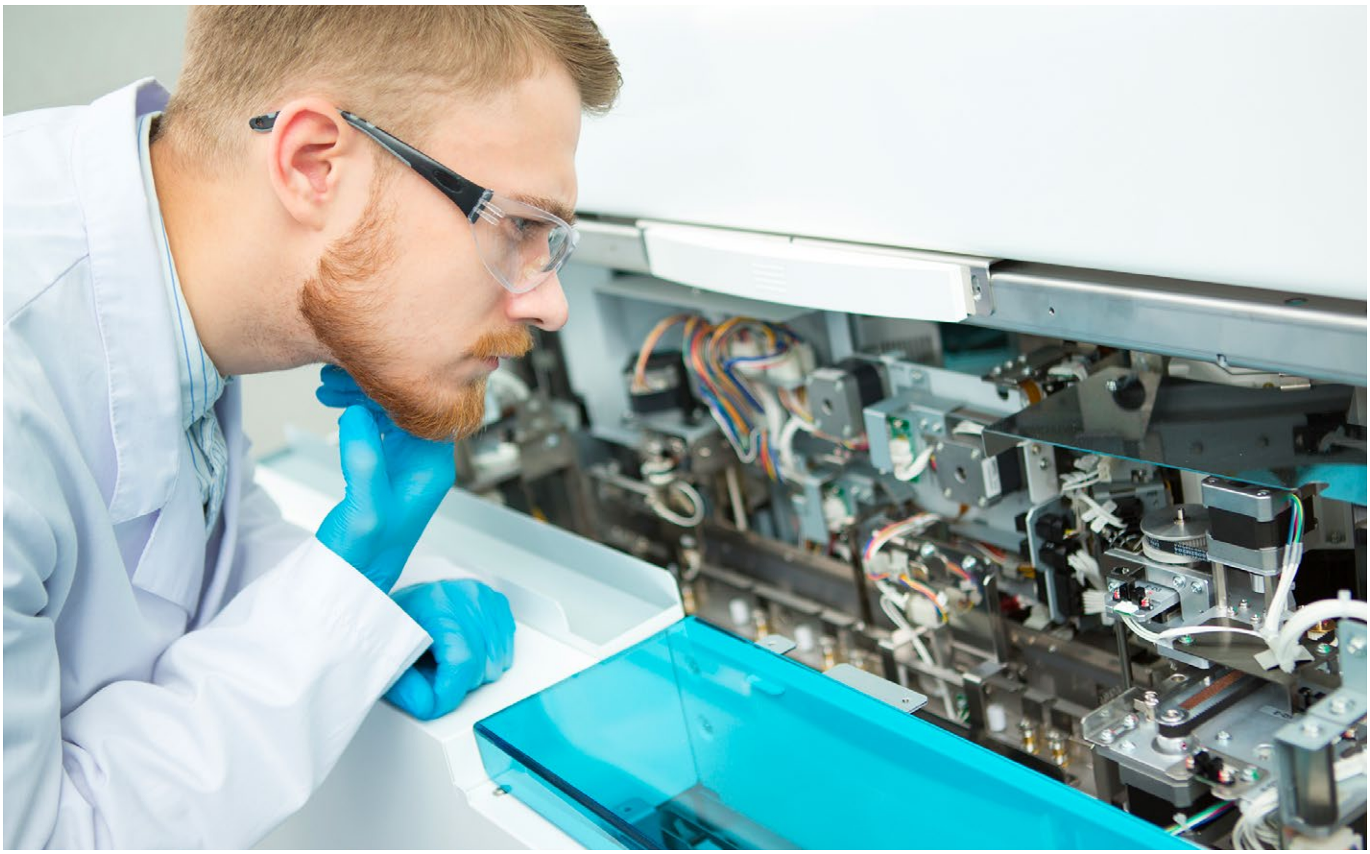


Figure 1: A rigid-flex PCB is space-saving and shock-resistant.

Figure 2: Talk to your manufacturer to ensure you select the right material for your rigid-flex design.

A rigid-flex design is typically made up of two or more rigid areas that are interconnected by a flexible strip. The synergistic property of the rigid and flex elements allow the PCB to be bent or folded in applications.

Rigid-flex PCBs are getting more attention in recent years, due to the demand for more compact, shock-resistant, and robust electronics. A rigid-flex design eliminates the need for connectors and interconnecting cables. Rigid-flex PCBS are also easier to install, as the entire design is manufactured on a single PCB.

Material Considerations for Rigid-Flex PCB Design

You'll want to consult your PCB manufacturer before starting a design. Depending on whether the PCB is meant for dynamic bend or stable bend, the choice of copper type, the number of layers, bend radius, and coverlays may differ.

A dynamic bend rigid-flex PCB is installed in an environment where it will constantly be subjected to bending. Therefore, it is recommended to use no more than two layers and ensure that the bending radius is at least 100 times the material thickness.

Meanwhile, it's possible to have up to 10 to 20 layers for a rigid-flex PCB that's meant for stable-bend installations. It is not subjected to repeated bending force and that means a smaller bending radius of about 10 times its material thickness is also possible.

Rigid-Flex PCB Design Guidelines

For all the advantages of a rigid-flex PCB, they also bring about challenges for PCB designers. Not only do designers need to deal with the electrical aspects of a design, but they also need to consider the mechanical dynamics of the PCB.

If you happen to be working on a rigid-flex PCB, the following guidelines will save you from committing costly mistakes.

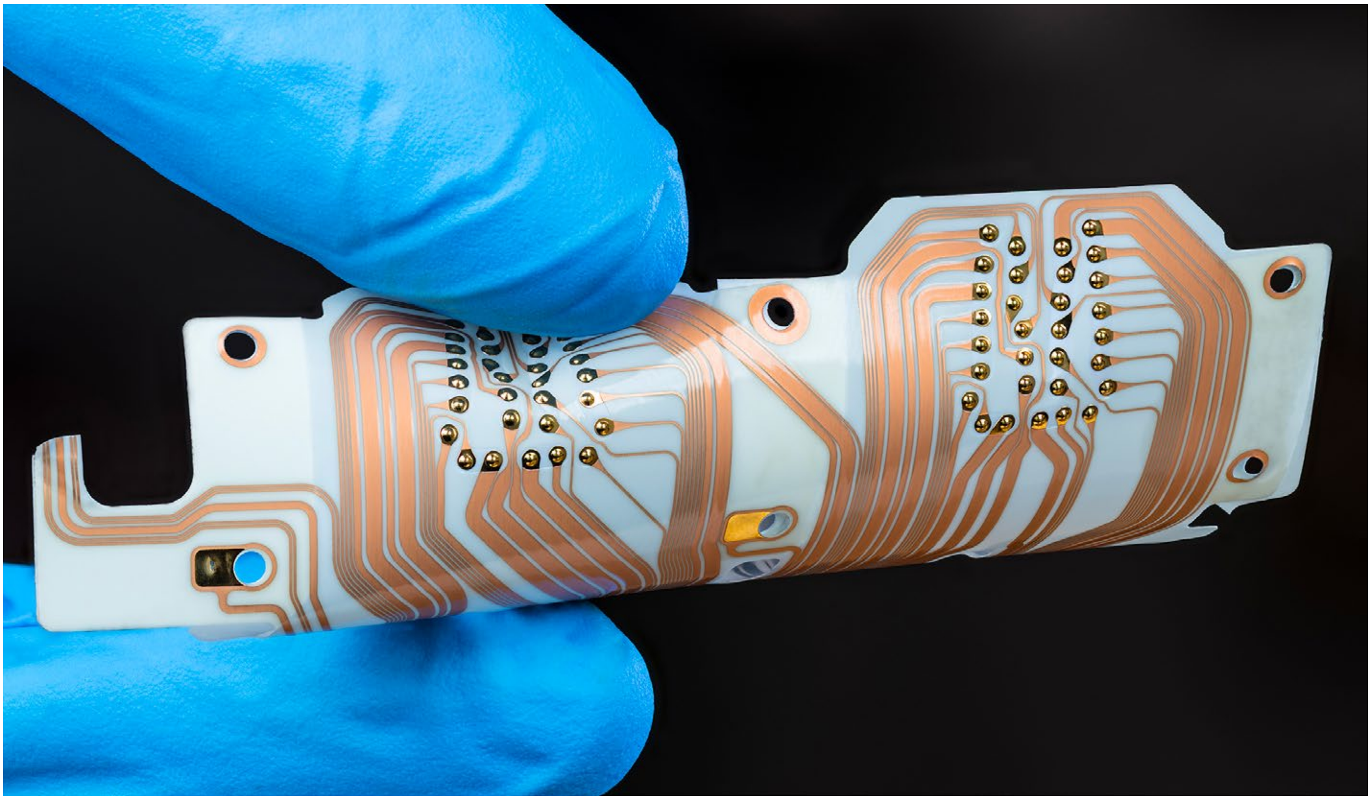


Figure 3: Use curved corners and teardrops for the flex area of a rigid-flex PCB.

Avoid Plated Through-holes on the Bending Areas

Avoid placing pads and vias on the bending area of the flex part of a PCB. Areas near the bending line will deliver mechanical stress that could jeopardize the structure of the plated holes.

Pads and vias can be placed on areas that are not subjected to bending, although it's not recommended. In such cases, use anchors to strengthen the plated holes. Additionally, use a teardrop to connect the trace to the plated hole for stronger joints. It's also a good practice to use larger pads and vias.

Pay Attention to Routing Across the Bending Area

Traces across the bending line should be kept as straight, perpendicular lines. It's better to use narrower traces that are spread evenly across the flexible area. Adding in dummy traces can help to increase mechanical sturdiness that protects the traces from breaking. For a double layer design, traces should be routed alternately on both the top and bottom layer.

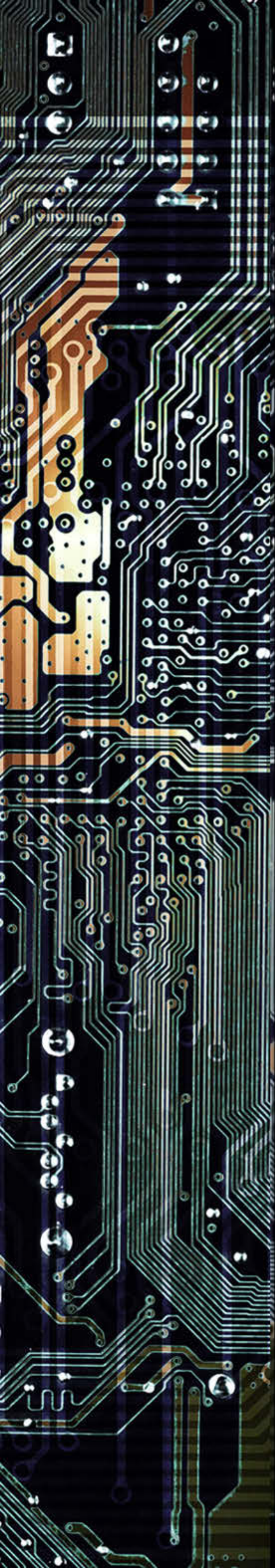
Avoid making any corners with the traces on the bending area. If the traces need to change direction on a flex PCB, use curves rather than sharp 45° or 90° corners.

Use a Cross-hatched Ground Plane

If you're pouring the ground plane as a solid area of copper, you'll risk putting a huge amount of stress on it and reducing its flexibility. Instead, use a cross-hatched ground plane on the flex area of the PCB.

It goes without saying that using a PCB design software that supports rigid-flex PCB design guidelines helps in creating an error-free prototype. DESIGN007

Cody Stetzel is a lead technical marketing engineer with Cadence Design Systems. This article originally appeared as a blog post on the Cadence Design Systems website.



TOP TEN EDITOR'S PICKS

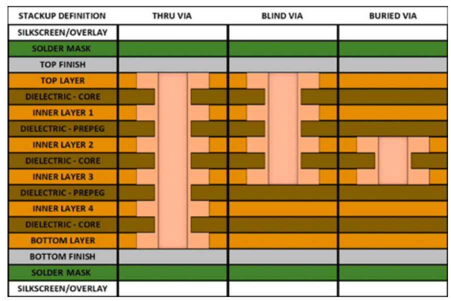
All Systems Go: Can You Design Without Electronic Data Management?

For any sizable design, PCBs are usually designed by a team of multiple design engineers (EEs) creating the schematic and multiple layout designers placing all the parts on the board and routing the traces. For a successful design, a multitude of interactions requires mandatory mechanisms to keep everyone on the same page during the design process.

Sensible Design: Why Some Conformal Coatings Fail in Service

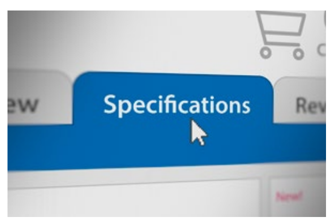
Why do coatings sometimes fail in service? What are the steps you can take to avoid failure in the first place? At Electrolube, we take great care in our research, product formulation and developing the appropriate processing and application techniques for our products, because we know that, on occasion, conformal coatings can fail.

Adventures in Engineering: A Penny Primer on PCB Design Fundamentals



The prevalent mindset of late seems to be "tools maketh the designer." I have noticed this in ECAD tool marketing, discussions with customers (past, present, prospective), and conversations with colleagues. While I am an advocate of having the right tool for the right job, this should be tempered with good old-fashioned know-how.

Dana on Data: DFM Issue Reduction—Company-specific PCB Acceptance Specifications

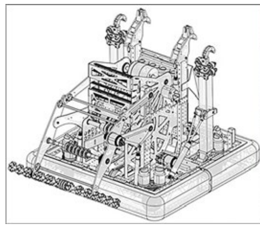
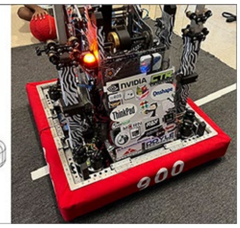


PCB data packages commonly generate fabricator DFM feedback questions that require resolution. Resolving these issues delays the manufacturing cycle time. There are many methods and techniques to reduce the DFM issues, such as working with the fabricator to review proposed stackup materials and impedance structures early in the design cycle. Another common method is to generate a company specific acceptance specification that provides requirements that are not covered in referenced IPC specifications and include negotiated DFM issue resolutions.

Connect the Dots: The Benefits of a Parts Library

To be effective at PCB design and layout, one needs to become proficient with the different tools of the trade. Parts libraries are among the most important.

Passionate People Can Do Fantastic Things

Electrical engineer Marshall Massengill is the first to admit that he has a pretty sweet gig. Marshall serves as a mentor for the Zebracorns, a robotics team based at a STEM-oriented high school in North Carolina. We recently spoke with Marshall about his work with the FIRST robotics team, and what it's like teaching PCB design to high school students.

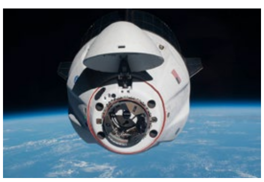
Tim's Takeaways: Today's Preparations for Tomorrow's PCB Designs

What skills actually prepare you for your future career? Tim Haag reflects on an eighth grade typing class that baffled those around him because "everyone knew that I had absolutely no aptitude for any sort of literary or language skills."

Beyond Design: Copper Pours in High-speed Design

The most common question I get asked by PCB designers is, "Do you need copper ground pours on digital multilayer PCBs?" The short answer is, "It depends." Copper pours are sometimes used incorrectly simply to fill in the unused space on a board. However, in some cases ground pours may be an advantage. In this month's column, I will look at where and where not to use ground pours.

IPC Training Course: PCB Design for Military & Aerospace Applications



This course addresses specific challenges encountered in military and aerospace applications, including the effects of vibration, shock, radiation, and altitude, extended operating temperature range, and other design considerations for high reliability applications.

Target Condition: Designing With the 5Ws and Other Acronyms

Who, what, when, where, and why with? I've slowly become disillusioned over the past decade with the whole "design-for" schtick. Design for manufacturing or "DFM" has become extremely subjective. I must say I am even becoming hesitant to splatter the acronym "DFM" when referencing job descriptions or when considering a layout tool's manufacturing analysis audit anymore because the term is so cliché.

For the latest news and information, visit **PCBDesign007.com**

Career Opportunities

Is your team growing?

Find industry-experienced candidates at I-Connect007.

For just $750, your 200-word, full-column ad will appear in the Career Opportunities section of all three of our monthly magazines, reaching circuit board designers, fabricators, assemblers, OEMs, suppliers and the academic community.

In addition, your ad will:

- be featured in at least one of our newsletters
- appear on our jobConnect007.com board, which is promoted in every newsletter
- appear in our monthly Careers Guide, emailed to 26,000 potential candidates

Potential candidates can click on your ad and submit a resume directly to the email address you provide, or be directed to the URL of your choice.

No contract required. Just send over your copy and company logo and we'll do the rest!

Contact barb@iconnect007.com to get your ad posted today!

+1 916.365.1727

I-Connect007
GOOD FOR THE INDUSTRY

smt007 pcb007 pcb design007

Career Opportunities

Career Opportunities

Prototron Circuits

Sales Representatives

Prototron Circuits, a market-leading, quick-turn PCB manufacturer located in Tucson, AZ, is looking for sales representatives for the New England and Northern California territories. With 35+ years of experience, our PCB manufacturing capabilities reach far beyond that of your typical fabricator.

Reasons you should work with Prototron:

- Solid reputation for on-time delivery (98+% on-time)
- Capacity for growth
- Excellent quality
- Production quality quick-turn services in as little as 24 hours
- 5-day standard lead time
- RF/microwave and special materials
- AS9100D
- MIL-PRF- 31032
- ITAR
- Global sourcing option (Taiwan)
- Engineering consultation, impedance modeling
- Completely customer focused team

Interested? Please contact Russ Adams at (206) 351-0281 or russa@prototron.com.

apply now

Koh Young America

Technical Support Applications Engineer

Full-Time — Duluth, GA

Koh Young Technology, founded in 2002 in Seoul, South Korea, is the world leader in 3D measurement-based inspection technology for electronics manufacturing. Located in Duluth, GA, Koh Young America has been serving its partners since 2010 and expanding team with an Applications Engineer to provide helpdesk support by delivering guidance on operation, maintenance, and programming remotely or on-site.

Responsibilities

- Provide timely, complete helpdesk support for Koh Young users
- Train users on proper operation, maintenance, programming, and best practices
- Recommend and oversee operational, process, or other performance improvements
- Effectively troubleshoot and resolve machine, system, and process issues

Skills and Qualifications

- Bachelor's in a technical discipline, relevant Associate's, or equivalent vocational or military training
- Knowledge of electronics manufacturing, robotics, PCB assembly, and/or AI; 2-4 years of experience
- SPI/AOI programming, operation, and maintenance experience, preferred
- Domestic and international travel (valid U.S. or Canadian Passport, required)
- Able to work effectively and independently with minimal supervision
- Ability to readily understand and interpret detailed documents, drawing, and specifications

Benefits

- Health/Dental/Vision/Life Insurance with no employee premium (including dependent coverage)
- 401K retirement plan
- Generous PTO and paid holidays

apply now

Career Opportunities

INSULECTRO

Are You Our Next Superstar?!

Insulectro, the largest national distributor of printed circuit board materials, is looking to add superstars to our dynamic technical and sales teams. We are always looking for good talent to enhance our service level to our customers and drive our purpose to enable our customers build better boards faster. Our nationwide network provides many opportunities for a rewarding career within our company.

We are looking for talent with solid background in the PCB or PE industry and proven sales experience with a drive and attitude that match our company culture. This is a great opportunity to join an industry leader in the PCB and PE world and work with a terrific team driven to be vital in the design and manufacture of future circuits.

apply now

CHEMCUT

Director of Operations
State College, PA

Chemcut Corp., a world leader in wet processing equipment for the manufacture of printed circuit boards and chemical etching of various metals, is seeking a Director of Operations.

Objectives of the Role:
- Collaborate with the CEO in setting and driving organizational vision, operational strategy, and hiring needs.
- Oversee manufacturing operations and employee productivity, building a highly inclusive culture ensuring team members thrive and organizational outcomes are met.
- Directly oversee manufacturing operations, production planning, purchasing, maintenance & customer service (product support) and partner with the CEO and controller on sales management to budget for sufficient investment capital to achieve growth targets.
- Aggressively manage capital investment and expenses to ensure the company achieves investor targets relative to growth and profitability.

Qualifications:
- Bachelor's degree in mechanical, electrical, or related fields
- 5+ years' experience in leadership positions
- Leadership skills, with steadfast resolve and personal integrity
- Understanding of advanced business planning and regulatory issues
- A solid grasp of data analysis and performance metrics
- Ability to diagnose problems quickly and have foresight into potential issues

Preferred Qualifications:
- Master's degree in business or related field
- International business experience

To apply, please submit a cover letter and resume to hr@chemcut.net

apply now

Career Opportunities

ventec
INTERNATIONAL GROUP
騰輝電子

European Product Manager
Taiyo Inks, Germany

We are looking for a European product manager to serve as the primary point of contact for product technical sales activities specifically for Taiyo Inks in Europe.

Duties include:
- Business development & sales growth in Europe
- Subject matter expert for Taiyo ink solutions
- Frequent travel to targeted strategic customers/ OEMs in Europe
- Technical support to customers to solve application issues
- Liaising with operational and supply chain teams to support customer service

Skills and abilities required:
- Extensive sales, product management, product application experience
- European citizenship (or authorization to work in Europe/Germany)
- Fluency in English language (spoken & written)
- Good written & verbal communications skills
- Printed circuit board industry experience an advantage
- Ability to work well both independently and as part of a team
- Good user knowledge of common Microsoft Office programs
- Full driving license essential

What's on offer:
- Salary & sales commission--competitive and commensurate with experience
- Pension and health insurance following satisfactory probation
- Company car or car allowance

This is a fantastic opportunity to become part of a successful brand and leading team with excellent benefits. Please forward your resume to **jobs@ventec-europe.com.**

apply now

MacDermid Alpha
ELECTRONICS SOLUTIONS

R&D Scientist III
Orange, CT

Job Description: The scientist will be a leader in technology for plating chemistry development, electrolytes, and additives. The position is hands-on, where the ideal candidate will enjoy creating and testing new aqueous plating processes and materials to meet the most demanding semiconductor applications related to Wafer-Level Packaging and Damascene. The qualified candidate will work as part of the R&D team while interacting with scientists, product management, and application engineers to commercialize new products for the advanced electronic solution business.

apply now

Technical Marketing Specialist
Waterbury, CT

This position provides information from the product team to the marketing communications team. It is a multifunctional role that requires some experience within electronics manufacturing supply chain or knowledge of how electronic devices are manufactured, specifically PCBs, semiconductors, and the chemical processes utilized therein. The primary function of this role is to help in the generation of product marketing collateral, but also includes assisting in tradeshow content development, advertising, and launches.

apply now

Career Opportunities

Regional Manager

Midwest Region

General Summary: Manages sales of the company's products and services, Electronics and Industrial, within the States of IL, IN & MI. Reports directly to Americas Manager. Collaborates with the Americas Manager to ensure consistent, profitable growth in sales revenues through positive planning, deployment and management of sales reps. Identifies objectives, strategies and action plans to improve short- and long-term sales and earnings for all product lines.

DETAILS OF FUNCTION:

- Develops and maintains strategic partner relationships
- Manages and develops sales reps:
  - Reviews progress of sales performance
  - Provides quarterly results assessments of sales reps' performance
  - Works with sales reps to identify and contact decision-makers
  - Setting growth targets for sales reps
  - Educates sales reps by conducting programs/ seminars in the needed areas of knowledge
- Collects customer feedback and market research (products and competitors)
- Coordinates with other company departments to provide superior customer service

QUALIFICATIONS:

- 5-7+ years of related experience in the manufacturing sector or equivalent combination of formal education and experience
- Excellent oral and written communication skills
- Business-to-business sales experience a plus
- Good working knowledge of Microsoft Office Suite and common smart phone apps
- Valid driver's license
- 75-80% regional travel required

To apply, please submit a COVER LETTER and RESUME to: Fernando Rueda, Americas Manager

fernando_rueda@kyzen.com

apply now

MACHINES FOR PRINTED CIRCUIT BOARDS

Field Service Engineer

Location: West Coast, Midwest

Pluritec North America, ltd., an innovative leader in drilling, routing, and automated inspection in the printed circuit board industry, is seeking a full-time field service engineer.

This individual will support service for North America in printed circuit board drill/routing and X-ray inspection equipment.

Duties included: Installation, training, maintenance, and repair. Must be able to troubleshoot electrical and mechanical issues in the field as well as calibrate products, perform modifications and retrofits. Diagnose effectively with customer via telephone support. Assist in optimization of machine operations.

A technical degree is preferred, along with strong verbal and written communication skills. Read and interpret schematics, collect data, write technical reports.

Valid driver's license is required, as well as a passport, and major credit card for travel.

Must be able to travel extensively.

apply now

Career Opportunities

Career Opportunities

Career Opportunities

Printed Circuits, a fast-growing printed circuit board fabricator, offers:

- Excellent opportunities for advancement and growth
- Dynamic manufacturing environment
- Excellent health, dental and other benefits
- Annual profit-sharing plan
- Signing bonus
- Additional incentives at the leadership level
- Clean facility with state-of-the-art manufacturing equipment
- Highly collaborative corporate and manufacturing culture that values employee contributions

Laminator Technician

Nature of Duties/Responsibilities

- Layup cover lay
- Layup rigid flex
- Layup multilayer/CU core boards
- Oxide treat/cobra treatment of all layers/CU cores
- Shear flex layer edges
- Rout of machine panel edges and buff
- Remove oxide/cobra treatment (strip panels)
- Serialize panels
- Pre-tac Kapton windows on flex layers (bikini process)
- Layup Kapton bonds
- Prep materials: B-stage, Kapton, release sheet
- Breakdown: flex layers, and caps
- Power scrub: boards, layers, and caps
- Laminate insulators, stiffeners, and heatsinks
- Plasma cleans and dry flex layers B-stage (Dry)
- Booking layers and materials, ready for lamination process
- Other duties as deemed necessary by supervisor

Education/Experience

- High school diploma or GED
- Must be a team player
- Must demonstrate the ability to read and write English and complete simple mathematical equations
- Must be able to follow strict policy and OSHA guidelines
- Must be able to lift 50 lbs
- Must have attention to detail

Wet Process/Plating Technician

Position is 3rd shift (11:00PM to 7:30AM, Sunday through Friday)

Purpose

To carry out departmental activities which result in producing quality product that conforms to customer requirements. To operate and maintain a safe working environment.

Nature of Duties/Responsibilities

- Load and unload electroplating equipment
- Fasten circuit boards to racks and cathode bars
- Immerse work pieces in series of cleaning, plating and rinsing tanks, following timed cycles manually or using hoists
- Carry work pieces between departments through electroplating processes
- Set temperature and maintains proper liquid levels in the plating tanks
- Remove work pieces from racks, and examine work pieces for plating defects, such as nodules, thin plating or burned plating
- Place work pieces on racks to be moved to next operation
- Check completed boards
- Drain solutions from and clean and refill tanks; fill anode baskets as needed
- Remove buildup of plating metal from racks using chemical bath

Education and Experience

- High school diploma or GED required
- Good organizational skills and the ability to follow instructions
- Ability to maintain a regular and reliable attendance record
- Must be able to work independently and learn quickly
- Organized, self-motivated, and action-oriented, with the ability to adapt quickly to new challenges/opportunities
- Prior plating experience a plus

apply now

Career Opportunities

MivaTek Global

Field Service Technician

MivaTek Global is focused on providing a quality customer service experience to our current and future customers in the printed circuit board and microelectronic industries. We are looking for bright and talented people who share that mindset and are energized by hard work who are looking to be part of our continued growth.

Do you enjoy diagnosing machines and processes to determine how to solve our customers' challenges? Your 5 years working with direct imaging machinery, capital equipment, or PCBs will be leveraged as you support our customers in the field and from your home office. Each day is different, you may be:

- Installing a direct imaging machine
- Diagnosing customer issues from both your home office and customer site
- Upgrading a used machine
- Performing preventive maintenance
- Providing virtual and on-site training
- Updating documentation

Do you have 3 years' experience working with direct imaging or capital equipment? Enjoy travel? Want to make a difference to our customers? Send your resume to N.Hogan@MivaTek.Global for consideration.

More About Us

MivaTek Global is a distributor of Miva Technologies' imaging systems. We currently have 55 installations in the Americas and have machine installations in China, Singapore, Korea, and India.

apply now

eptac

TRAIN. WORK SMARTER. SUCCEED.

Become a Certified IPC Master Instructor

Opportunities are available in Canada, New England, California, and Chicago. If you love teaching people, choosing the classes and times you want to work, and basically being your own boss, this may be the career for you. EPTAC Corporation is the leading provider of electronics training and IPC certification and we are looking for instructors that have a passion for working with people to develop their skills and knowledge. If you have a background in electronics manufacturing and enthusiasm for education, drop us a line or send us your resume. We would love to chat with you. Ability to travel required. IPC-7711/7721 or IPC-A-620 CIT certification a big plus.

Qualifications and skills

- A love of teaching and enthusiasm to help others learn
- Background in electronics manufacturing
- Soldering and/or electronics/cable assembly experience
- IPC certification a plus, but will certify the right candidate

Benefits

- Ability to operate from home. No required in-office schedule
- Flexible schedule. Control your own schedule
- IRA retirement matching contributions after one year of service
- Training and certifications provided and maintained by EPTAC

apply now

Career Opportunities

The Test Connection, Inc.

Rewarding Careers

Take advantage of the opportunities we are offering for careers with a growing test engineering firm. We currently have several openings at every stage of our operation.

The Test Connection, Inc. is a test engineering firm. We are family owned and operated with solid growth goals and strategies. We have an established workforce with seasoned professionals who are committed to meeting the demands of high-quality, low-cost and fast delivery.

TTCI is an Equal Opportunity Employer. We offer careers that include skills-based compensation. We are always looking for talented, experienced test engineers, test technicians, quote technicians, electronics interns, and front office staff to further our customer-oriented mission.

Associate Electronics Technician/ Engineer (ATE-MD)

TTCI is adding electronics technician/engineer to our team for production test support.

- Candidates would operate the test systems and inspect circuit card assemblies (CCA) and will work under the direction of engineering staff, following established procedures to accomplish assigned tasks.
- Test, troubleshoot, repair, and modify developmental and production electronics.
- Working knowledge of theories of electronics, electrical circuitry, engineering mathematics, electronic and electrical testing desired.
- Advancement opportunities available.
- Must be a US citizen or resident.

apply now

Test Engineer (TE-MD)

In this role, you will specialize in the development of in-circuit test (ICT) sets for Keysight 3070 (formerly HP) and/or Teradyne (formerly GenRad) TestStation/228X test systems.

- Candidates must have at least three years of experience with in-circuit test equipment. A candidate would develop and debug our test systems and install in-circuit test sets remotely online or at customer's manufacturing locations nationwide.
- Candidates would also help support production testing and implement Engineering Change Orders and program enhancements, library model generation, perform testing and failure analysis of assembled boards, and other related tasks.
- Some travel required and these positions are available in the Hunt Valley, Md., office.

apply now

Sr. Test Engineer (STE-MD)

- Candidate would specialize in the development of in-circuit test (ICT) sets for Keysight 3070 (formerly Agilent & HP), Teradyne/ GenRad, and Flying Probe test systems.
- Strong candidates will have more than five years of experience with in-circuit test equipment. Some experience with flying probe test equipment is preferred. A candidate would develop, and debug on our test systems and install in-circuit test sets remotely online or at customer's manufacturing locations nationwide.
- Proficient working knowledge of Flash/ISP programming, MAC Address and Boundary Scan required. The candidate would also help support production testing implementing Engineering Change Orders and program enhancements, library model generation, perform testing and failure analysis of assembled boards, and other related tasks. An understanding of stand-alone boundary scan and flying probe desired.
- Some travel required. Positions are available in the Hunt Valley, Md., office.

Contact us today to learn about the rewarding careers we are offering. Please email resumes with a short message describing your relevant experience and any questions to careers@ttci.com. Please, no phone calls.

We proudly serve customers nationwide and around the world.

TTCI is an ITAR registered and JCP DD2345 certified company that is NIST 800-171 compliant.

apply now

Career Opportunities

American Standard Circuits
Creative Innovations In Flex, Digital & Microwave Circuits

CAD/CAM Engineer

Summary of Functions

The CAD/CAM engineer is responsible for reviewing customer supplied data and drawings, performing design rule checks and creating manufacturing data, programs, and tools required for the manufacture of PCB.

Essential Duties and Responsibilities

- Import customer data into various CAM systems.
- Perform design rule checks and edit data to comply with manufacturing guidelines.
- Create array configurations, route, and test programs, panalization and output data for production use.
- Work with process engineers to evaluate and provide strategy for advanced processing as needed.
- Itemize and correspond to design issues with customers.
- Other duties as assigned.

Organizational Relationship

Reports to the engineering manager. Coordinates activities with all departments, especially manufacturing.

Qualifications

- A college degree or 5 years' experience is required. Good communication skills and the ability to work well with people is essential.
- Printed circuit board manufacturing knowledge.
- Experience using CAM tooling software, Orbotech GenFlex®.

Physical Demands

Ability to communicate verbally with management and coworkers is crucial. Regular use of the telephone and e-mail for communication is essential. Sitting for extended periods is common. Hearing and vision within normal ranges is helpful for normal conversations, to receive ordinary information and to prepare documents.

apply now

ARLON
ELECTRONIC MATERIALS

Arlon EMD, located in Rancho Cucamonga, California, is currently interviewing candidates for open positions in:

- Engineering
- Quality
- Various Manufacturing

All interested candidates should contact Arlon's HR department at 909-987-9533 or email resumes to careers.ranch@arlonemd.com.

Arlon is a major manufacturer of specialty high-performance laminate and prepreg materials for use in a wide variety of printed circuit board applications. Arlon specializes in thermoset resin technology, including polyimide, high Tg multifunctional epoxy, and low loss thermoset laminate and prepreg systems. These resin systems are available on a variety of substrates, including woven glass and non-woven aramid. Typical applications for these materials include advanced commercial and military electronics such as avionics, semiconductor testing, heat sink bonding, High Density Interconnect (HDI) and microvia PCBs (i.e. in mobile communication products).

Our facility employs state of the art production equipment engineered to provide cost-effective and flexible manufacturing capacity allowing us to respond quickly to customer requirements while meeting the most stringent quality and tolerance demands. Our manufacturing site is ISO 9001: 2015 registered, and through rigorous quality control practices and commitment to continual improvement, we are dedicated to meeting and exceeding our customers' requirements.

For additional information please visit our website at www.arlonemd.com

apply now

Career Opportunities

U.S. CIRCUIT

Plating Supervisor

Escondido, California-based PCB fabricator U.S. Circuit is now hiring for the position of plating supervisor. Candidate must have a minimum of five years' experience working in a wet process environment. Must have good communication skills, bilingual is a plus. Must have working knowledge of a plating lab and hands-on experience running an electrolytic plating line. Responsibilities include, but are not limited to, scheduling work, enforcing safety rules, scheduling/maintaining equipment and maintenance of records.

Competitive benefits package.
Pay will be commensurate
with experience.

Mail to:
mfariba@uscircuit.com

apply now

APCT
Passion | Commitment | Trust

APCT, Printed Circuit Board Solutions: Opportunities Await

APCT, a leading manufacturer of printed circuit boards, has experienced rapid growth over the past year and has multiple opportunities for highly skilled individuals looking to join a progressive and growing company. APCT is always eager to speak with professionals who understand the value of hard work, quality craftsmanship, and being part of a culture that not only serves the customer but one another.

APCT currently has opportunities in Santa Clara, CA; Orange County, CA; Anaheim, CA; Wallingford, CT; and Austin, TX. Positions available range from manufacturing to quality control, sales, and finance.

We invite you to read about APCT at APCT.com and encourage you to understand our core values of passion, commitment, and trust. If you can embrace these principles and what they entail, then you may be a great match to join our team! Peruse the opportunities by clicking the link below.

Thank you, and we look forward to
hearing from you soon.

apply now

PUBLISHER: **BARRY MATTIES**
barry@iconnect007.com

MANAGING EDITOR: **ANDY SHAUGHNESSY**
(404) 806-0508; andy@iconnect007.com

TECHNICAL EDITOR: **PETE STARKEY**
+44 (0) 1455 293333; pete@iconnect007.com

EDITOR | COLUMNIST COORDINATOR: **MICHELLE TE**
michelle@iconnect007.com

CONTRIBUTING TECHNICAL EDITOR: **DAN FEINBERG**
baer@iconnect007.com

CONTRIBUTING TECHNICAL EDITOR: **HAPPY HOLDEN**
(616) 741-9213; happy@iconnect007.com

SALES MANAGER: **BARB HOCKADAY**
(916) 365-1727; barb@iconnect007.com

MARKETING SERVICES: **TOBEY MARSICOVETERE**
(916) 266-9160; tobey@iconnect007.com

PRODUCTION MANAGER: **SHELLY STEIN**
shelly@iconnect007.com

MAGAZINE LAYOUT: **RON MEOGROSSI**

AD DESIGN: **SHELLY STEIN, MIKE RADOGNA, TOBEY MARSICOVETERE**

TECHNOLOGY SPECIALIST: **BRYSON MATTIES**

COVER DESIGN AND TYPOGRAPHY: **SHELLY STEIN**

COVER IMAGE: **ADOBE STOCK © SASCHA BURKARD**

DESIGN007 MAGAZINE

DESIGN007 MAGAZINE®
is published by BR Publishing, Inc.,
942 Windemere Dr. NW, Salem, OR 97304

July 2022, Volume 11, Number 7
DESIGN007 MAGAZINE is published monthly,
by BR Publishing, Inc.

ADVERTISER INDEX

Accurate Circuit Engineering........................ 11
Altium.. 59
American Standard Circuits........................ 41
APCT.. 35
Burkle North America............................... 75
Cadence Design Systems.......................... 53
Candor Industries.................................... 5
Downstream Technologies.................... 15, 29
EMA Design Automation........................... 33
Flexible Circuit Technologies...................... 63
I-Connect007 eBooks.................. 2, 3, 31, 93
In-Circuit Design Pty Ltd........................... 77
IPC.. 19
MacDermid Alpha Assembly Solutions........... 71
Miltronix... 21
NCAB Group...................................... 47, 51
Precision Technologies............................. 27
Prototron Circuits................................... 57
Pulsonix... 67
Siemens Digital Industries Software.............. 7
Taiyo America.. 25
U.S. Circuit... 73
Ventec International Group........................ 23


Problems solved!
PCB007 MAGAZINE APRIL 2022
PCB007 MAGAZINE MAY 2022
GOLDEN ROI OPPORTUNITIES
PCB007 MAGAZINE
PLATING
Cybersecurity & Counterfeiti
DESIGN007 MAGAZINE MAY 2022
Designing in a Vacuum
DESIGN007 MAGAZINE JUNE 2022
DFM DESIGN WITH MANUFACTURING
DESIGN007 MAGAZINE JULY 2022
GIGO HURTS
SMT007 MAGAZINE JUNE 2022
COLLABORATION AND PARTNERSHIPS
SMT007 MAGAZINE JULY 2022
SECURITY
CMMC
TRIPLE WORD SCORE
TRIPLE LETTER THREAT
SUBSCRIBE NOW