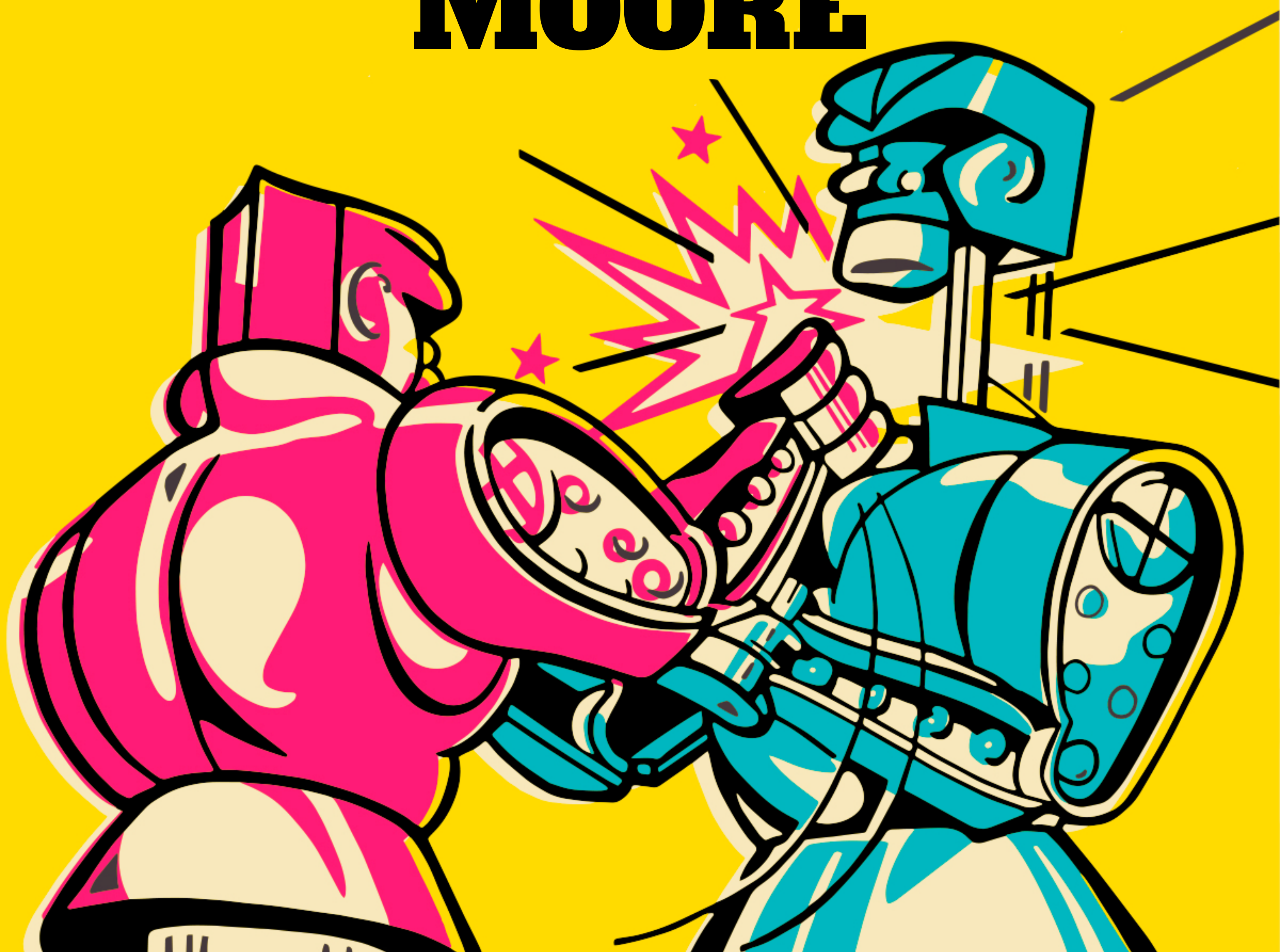
I-Connect007

NOVEMBER 2022

DESIGN007 MAGAZINE

MAXWELL VS. MOORE

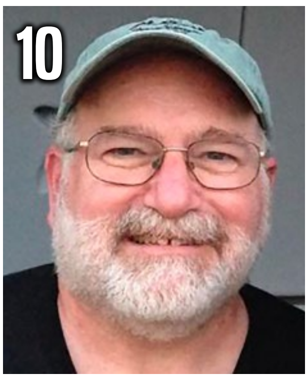


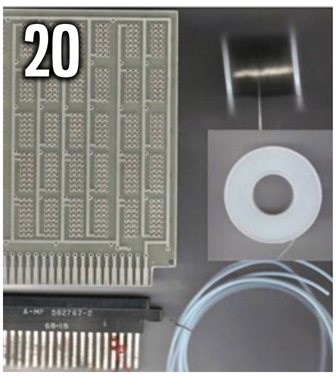

REALITY CHECK AHEAD
THE PRINTED CIRCUIT DESIGNER'S GUIDE TO...™
Designing for Reality
Matt Stevenson
Sunstone Circuits
I-007e Books
Learn how to create realistic designs and advance your skills!
Download
I-007e Books

DESIGN007
MAGAZINE

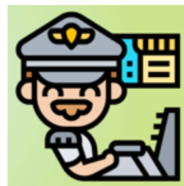
A Fight to the Physics

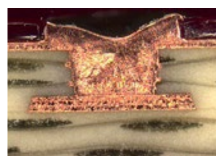
Lately, some design instructors have been advocating for a return to the fundamentals of physics. Are today's PCB designers spending too much time studying Moore's Law and not enough on Maxwell's equations? In this issue, our experts explain why designers need to understand concepts like field effects as well as circuit theory, and how the disciplines work together symbiotically.





DESIGN007 MAGAZINE





It's All About the Physics—or Is It?

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

Lately, we've heard quite a few design experts say, "PCB design is all about the physics. Designers should focus more on understanding the laws of physics and less on circuit theory."

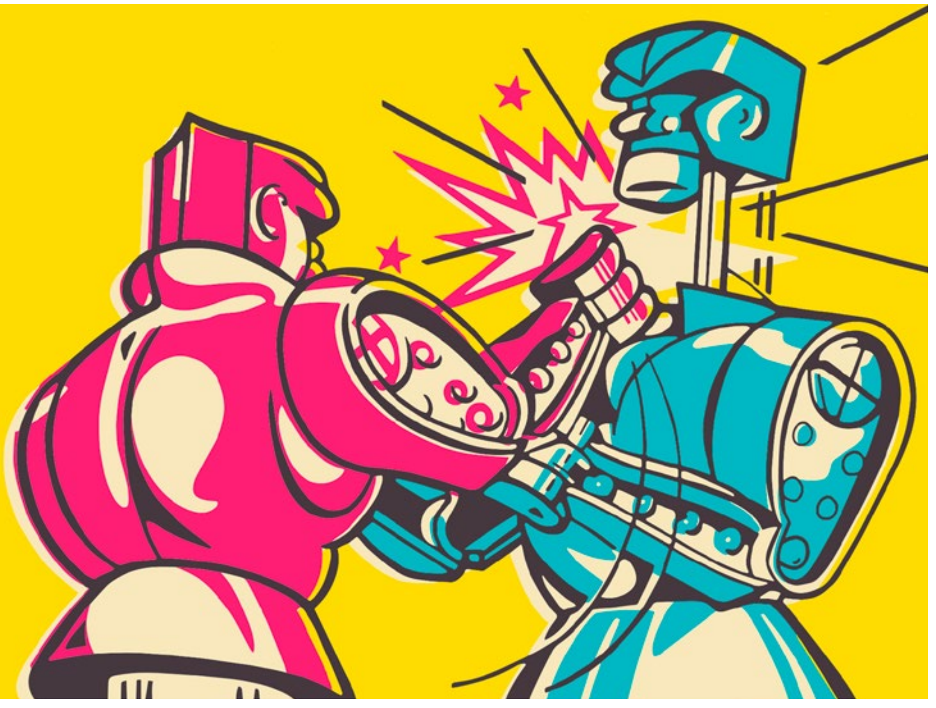
As feature sizes continue to shrink and we see more challenges popping up like thermal, it's no wonder that physics has become a hot topic in some corners of the design community. After a few conversations with designers and instructors, we knew we had a good topic. We wanted to address these questions:

- How does physics work in PCB design?
- How do the laws of physics fit in with circuit and transmission line theory?
- What should designers understand about physics and how does it apply to PCB design?

While putting this issue together, we investigated potential cover ideas. "What if we had James Maxwell and Gordon Moore boxing on the cover, in a Faraday cage match? Let's get ready to rumble!" That led us to the Rock 'Em Sock 'Em robots that now grace our cover.

Of course, Maxwell and Moore aren't really on opposite sides of the ring in the electronics discussion. They're more like two sides of the same coin. Today's designers need to understand the laws of physics and electrical circuit theory, both individually and as they work together.

The phrase, "It all goes back to the physics," implies that designers should return to the fundamentals of electrical and electromagnetic theory, led by pioneers such as Maxwell, Georg Ohm, and Joseph Fourier. But I'm curious: How many physics classes have you taken? Did you major in physics? I know many designers with bios brimming with various types of engineering diplomas, but only a handful seem to have physics degrees.



In the end, the smart designer today needs to know a little bit of everything, from mechanical engineering to RF and microwave. Physics is just one more tool in your toolbox. It can help you understand fields and the properties of the materials, as well as concepts such as how fields affect inductance.

Over the past few months, we noticed something: Physics can involve a lot of heavy math, such as Maxwell's equations. But as Eric Bogatin, who majored in physics, says, designers can learn the fundamentals of physics without necessarily mastering all the math. It might sound counterintuitive, but don't let the math keep you from studying physics.

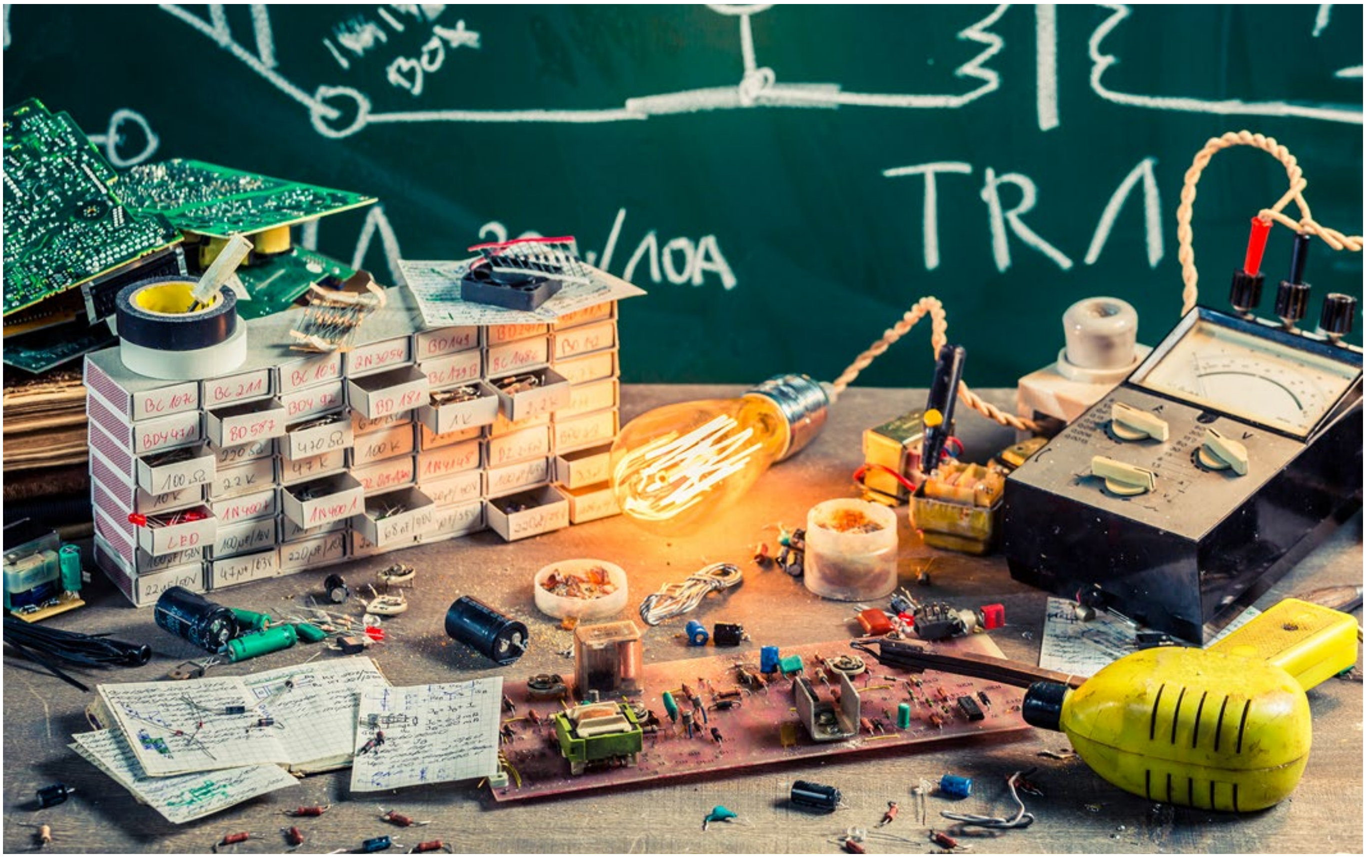
This issue is a great place to start. Here, you will learn why you can't ignore the physics of PCB design, especially as features continue to shrink. Our experts will discuss how the laws of physics (field theory, thermal effects, etc.) relate to solid PCB design practices. We also raise awareness of the need for greater understanding of the laws of physics and discuss a variety of resources related to physics and PCB design.

Our interview with Eric Bogatin leads this "ready to rumble" issue. He explains how a knowledge of physics can help designers master electrical engineering concepts, how these disciplines work together, and why it's so important to avoid letting the math "get in the way" when learning the laws of physics. Next, Happy Holden describes his brief interaction with Maxwell's equations in college—avoiding them, primarily. We have a great article by Douglas Brooks and Johannes Adam that uses physics to help illustrate why vias don't get hot. Columnist Martyn Gaudion discusses how analysis and measurements can help you out of a jam when the laws of physics don't seem to apply. Finally, Tamara Jovanovic brings us an update on today's curriculum, fresh from the halls of academia as she pursues her master's in electrical engineering.

We have columns from our regular contributors Barry Olney, Stephen Chavez, Matt Stevenson, Beth Massey, and Tim Haag, as well as an interview with Joe Clark and Mark Gallant. We also have another article by Anaya Vardya in his DFM101 series.

There's a lot happening in our segment of the industry, with four months of conferences and trade shows coming up. We'll be there, bringing you the news you need to know. See you next month. **DESIGN007**

Andy Shaughnessy is managing editor of *Design007 Magazine.* He has been covering PCB design for 23 years. To read past columns, click here.



The Physics of PCB Design

Feature Interview by the I-Connect007 Editorial Team

In this wide-ranging interview, Dr. Eric Bogatin discusses the relationship between physics and electrical theory, and why it's critical for designers and design engineers to understand the laws of physics. As he points out, the math is important, but designers shouldn't let the principles of physics "hide behind the math."

Eric discusses some points of physics that designers need to understand, the physics resources available, and why it's so important to have some understanding of Maxwell's equations, even if you don't have a strong math background.

Andy Shaughnessy: Eric, we often hear that designers needs to focus on physics, not just circuit theory. Since you have two physics degrees and you teach signal integrity, what do you feel most designers are missing in terms of the physics?

Eric Bogatin: First, let me understand the terms that we're going to use. When you say physics, are you talking about EM fields and Maxwell's equations? Let's talk about the terms.

Happy Holden: Other than Newton, James Maxwell helped define modern physics. He didn't discover magnetic fields, but he certainly showed the relationship. We're constantly talking about signal integrity problems, and Moore's Law is driving and challenging Maxwell's laws. They're both important.

Bogatin: Moore's Law is what drives the ever-finer feature sizes and transistors, the channels.

That means shorter rise times and more signal integrity problems, but when it comes to the SI problems and the impact the interconnects have on the signals, there are two ways of thinking about them: the circuit theory approach and the fields approach, which is really a transmission line analysis that's inherently distributed where the electromagnetic fields are important.


Eric Bogatin

Barry Matties: When someone says, "PCB design is really just about physics," is there more to that statement than what you just described?

Bogatin: When they say it's all physics, it's the electromagnetic fields described by Maxwell's equations and electromagnetic fields. They're all synonymous in this context. It's correct that the way signals interact with interconnects is all electromagnetic fields and the boundary conditions—all Maxwell's equations. You don't have to be a PhD student to learn how to solve Maxwell's equations, but you must understand a little bit about electromagnetic fields, how they interact, and how they propagate.

I had Professor Walter Lewin as a freshman at MIT, and I still vividly remember his lectures. Now there is a video series from his second semester freshman physics class, and he's got a million views. I use what I learned in his class almost daily. When I look at the videos, which are recorded 40 years after I took the class, I see he hasn't changed at all, and the videos are timeless. I always recommend them.

Holden: For people who are interested, are Lewin's courses suitable for those without electrical engineering degrees, but are interested in understanding the principles?

Bogatin: Yes. They're offered to freshmen. You don't need to be enrolled in electrical engineering classes. There's a little bit of math, but he goes through it slow enough that if you've had a little bit of calculus, you'd see it instantly. If you didn't, that's okay. It's only 25% equations. You still get the principles.

Holden: Maxwell is not the easiest subject. Both Nolan and I were in electrical engineering, but because of the difficulty and the flunk-out rate from fields theory, we chose coding and stayed away from the RF and the fields. I just didn't have the mathematical prowess to handle that.

Bogatin: You're right. If you go that next step and talk about electromagnetic fields, Maxwell's equations are differential equations, and you must understand some of that.

What I like about Walter Lewin's lectures is he emphasizes the principles and the behaviors and doesn't let the math hide it. My students bring me his videos all the time. Most of the YouTube videos on understanding electromagnetics are bad analogies or they're not even the right physics. You're not learning something you can use to leverage other things down the line.

Just for perspective, at MIT you get mechanics and electromagnetics in your freshman year. It's some math, but you get the heavy math in your junior and senior years. In my junior year, I took electromagnetics. We used John David Jackson's Classical Electrodynamics and it's incredibly heavy in the math.

Holden: Is the abundance of heavy math instruction the reason why a lot of your focus has been on the rules of thumb and simplification in your books and publications?

Bogatin: Yes. Much of my style comes from what I learned from classes at MIT. Not to say that there isn't rigor, but my professors emphasized the understanding part first, then the math, and I picked up on that. It is remarkable how far you can go with simple models to understand things. They have the math at their core. Math is the language of engineering and science. You must have that, but you don't need to have every conversation with math.

There's one approach I use called strategic simplification. You want to simplify the problem enough to understand the main points, answer important questions, and get to an answer quickly, but not so simple that you have degraded it, so it doesn't apply to real problems. How do you take complex problems, describe them in a simple way to get an answer quickly, while still having the core of the problem in the solution in the description with not so much math?

Having said all that, math is important. If you have the opportunity, get as much as you can and apply it. Do it as a student, because when you're a professional engineer, you don't always have that time.

Holden: Now with software, you must be cautious about just plugging in numbers without understanding what they're simulating or the repercussions, because you could come out with a totally wrong answer and have no clue that it doesn't make sense.

Bogatin: You're right. It's easy to get an answer these days with all the hardware and software tools that are out there. You push a button, and you get an answer, but if it's not set up the way it's supposed to be, you have no idea whether your answer is realistic. Does it make sense? Having that understanding of what to expect is my "Rule No. 9." This comes from the engineering judgment and the engineering intuition of knowing how things behave.

Back to your original question, how important is physics? It's fundamentally important. Every electrical engineer who will do anything related to signal integrity, power integrity, or EMI needs a good foundation in fields from which they can then build their intuition and apply it to using the hardware and the software tools.

Shaughnessy: You have a PhD in physics. How would you describe the interrelationship between physics and circuit theory?

Bogatin: I come at it from a physics perspective, which is a little bit more emphasis on the fundamentals of the phenomena, the electromagnetics, and the properties of the materials. That's my starting place. Then I can decide, "When is it good to approximate using circuit elements?" One of my differentiators in the industry has been my physics background rather than an EE background.

"When is it good to approximate using circuit elements?" One of my differentiators in the industry has been my physics background rather than an EE background.

The irony is that I'm in the EE department at University of Colorado-Boulder (CU). I give students some of the electromagnetic side of things when we talk about circuit elements. An inductor is a perfect example. An EE student learns inductors, capacitors, and resistors as ideal circuit elements with no idea about the physics of an inductor, and this idea of magnetic fields. If you only think of a little squiggly line—a coil—when you see an inductor, it doesn't give you that physical intuition. How do the geometry and materials affect the

magnetic fields and how does that affect the inductance? I wish more of my students had a physics background.

Shaughnessy: Where does this leave a designer? They've been primarily focusing on circuits and circuit theory. Now what?

Bogatin: Well, they must gain some understanding of electric, magnetic, and propagating fields. I have heard some analogies made at presentations about energy propagation, that the signals are the energy, and it propagates in the white space between the conductors. I'm not saying that's wrong, but it's very misleading. It's not about the energy flow; it's about the fields.

Yes, we have energy and fields, but the key difference is that when you talk about electromagnetic fields, we call them vectors because they have a direction, but they also have a sign. You can have a plus and minus direction or magnitude for the electric fields. That means you can get cancellation and reinforcement, but if you only think about energy, then you'll lose certain insight. How do you get energy to cancel out? How do you get waves to cancel out? You really need the intuition about how fields behave, not energy propagation. That's my one commentary about how to think about signals on interconnects.

Matties: Eric, how would you look at the current design resources in terms of their knowledge of this topic?

Bogatin: I speak to and do training with engineers onsite around the world. All the top experts, the guys working on 10 gigabit serial links and above, designing the highest-end circuit boards, are well versed in electromagnetics. They may have an EE background, but they've taken enough of the EM classes to have a good sense of how they behave so they can apply their engineering intuition.

Matties: But in percentages, does that high-end group represent 20% of all designers and the other 80% have knowledge gaps that need to be filled?

Bogatin: Yes, that's probably a good number. Some of these large OEMs have teams with a couple of SI experts. They're the guys who really understand Maxwell's equations and how to use all the tools. They've learned on the job, read books, and taken classes. They've learned using tools about the fundamentals, or they went back for a master's degree to learn electromagnetics. Then you have a lot of other engineers who can be trained on specific tasks.

Matties: The old guys are leaving the industry. Do you think we're attracting enough young designers?

Bogatin: "Enough" is a relative term. I get emails from companies every week asking if we have any SI engineers we can send their way. I post the requests on our department's Discord server and our alumni have a chance to respond. There are more openings in the field than we have trained engineers for.

Matties: It will probably be that way for some time.


Hmm, what is the recommended **minimum solder mask** width to be able to get a solder mask bridge **between two copper pads?**

Bogatin: Here at CU last year, we put together a master's program specifically in high-speed digital engineering that covers signal integrity, power integrity, and EMC. We'll take a student graduating with an undergraduate degree, put them through this two-year master's program, and arm them with the expertise to be a contributing engineer on a design team. For example, one course is based on one of my textbooks about high-speed digital engineering, and then they take electromagnetics for SI. They get that background of Maxwell's equations. I'm also working on a course for high-speed measurements, with high-speed scopes and probes, S parameters, TDR, and the best measurement practices in the high-speed world.

We'll take a student graduating with an undergraduate degree, put them through this two-year master's program, and arm them with the expertise to be a contributing engineer on a design team.

Shaughnessy: These graduates sound like more than a PCB design or layout person.

Bogatin: Yes. We want to generate hardware engineers, someone who could contribute to the team as a lead signal integrity expert in a hardware group, rather than a layout engineer. You don't need a master's degree to be a layout engineer. The more electromagnetics you understand, the more effective you will be because you're responsible for the physical design of the system, but being able to communicate with and talk with a good lead signal integrity or hardware engineer is more important than being the expert yourself.

There's so much a hardware engineer needs to know these days, especially just getting your master's degree; it's hard for one person to be the expert in everything.

Shaughnessy: What would you recommend studying?

Bogatin: I have three books that are really a good foundation. There's *Signal and Power Integrity: Simplified,* which is a great foundation book. I have a new one with Artech, *Bogatin's Practical Guide to Transmission Line Design* and *Characterization for Signal Integrity Applications.* That provides fundamentals of how to think about transmission lines.

Last year, I wrote *Bogatin's Practical Guide to Prototype Breadboard and PCB Design* for my class on printed circuit board design. It provides a strategy for doing rapid prototyping.

These three provide a good foundation and I strongly recommend them. To get more advanced, I recommend *Advanced Signal Integrity for High-Speed Digital Design* by Stephen H. Hall and Heck, who were both at Intel. If you whiz through my books, you've got the math, and if you want to see the real details, their book is a great follow-on.

Shaughnessy: There don't seem to be many PCB design books with a focus on physics.

Bogatin: The problem is that if you look at most books that include electromagnetics, 99% of them start with vector calculus, Maxwell's equations, differential and integral formulas, and solving Maxwell's equations. That's how we teach it here at CU because we're teaching it for generalists.

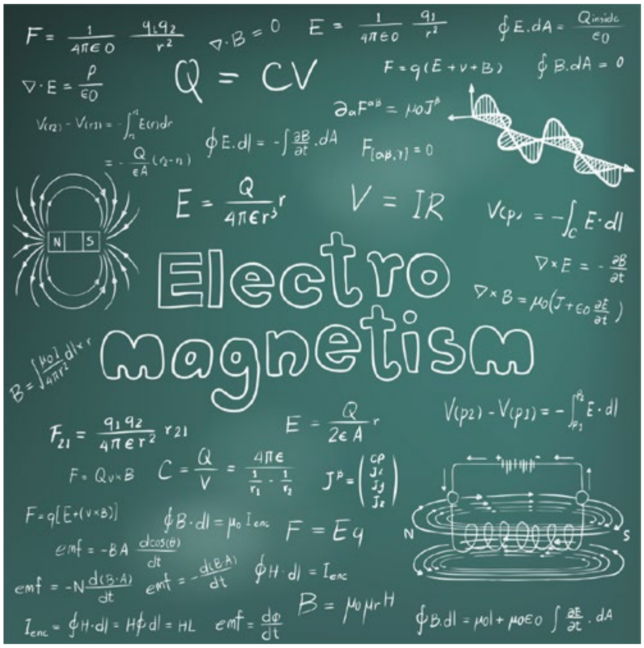
Shaughnessy: So, an EE grad and a physics grad can both be good PCB design engineers; they just come at this from different backgrounds.

Bogatin: I hope so. If I was a hiring manager looking for a junior engineer right out of school, I would equally consider an EE or a physics major undergraduate.

Shaughnessy: Were you interested in electromagnetics in high school? How did you end up focusing on it? What led you that way?

Bogatin: I went through physics at MIT. I learned the standard physics, mechanics, the EM, vibrational, then quantum mechanics, thermodynamics, all that stuff that every undergraduate gets, but I was really interested in astrophysics and cosmology. For graduate school, I debated, "What do I want to do?" This was in the early days of lasers, and I knew that whatever I did, lasers would be part of it. So, I went to University of Arizona, Tucson, which had an optical science center.

I learned about lasers, quantum optics, and I worked on a relativity experiment that became my dissertation. It was the best of all worlds. I worked on a cosmology experiment that used lasers on a benchtop in a lab. When I got my degree in quantum optics and cosmology, my mother said, "Where are you going to find want ads saying, 'Cosmologists wanted, apply here'?"



After grad school, I started at Bell Labs, and I was the token physicist with a bunch of chemists. That was how I got into this field. They were working on circuit board technology, building and selecting new materials and fabrication techniques. They needed to understand how their material selections influenced performance.

That was how I began applying the electromagnetics to figure out, "How do signals interact with the materials, the interconnects, and the decisions they made about designing the materials and photoimage materials and different copper platings?"

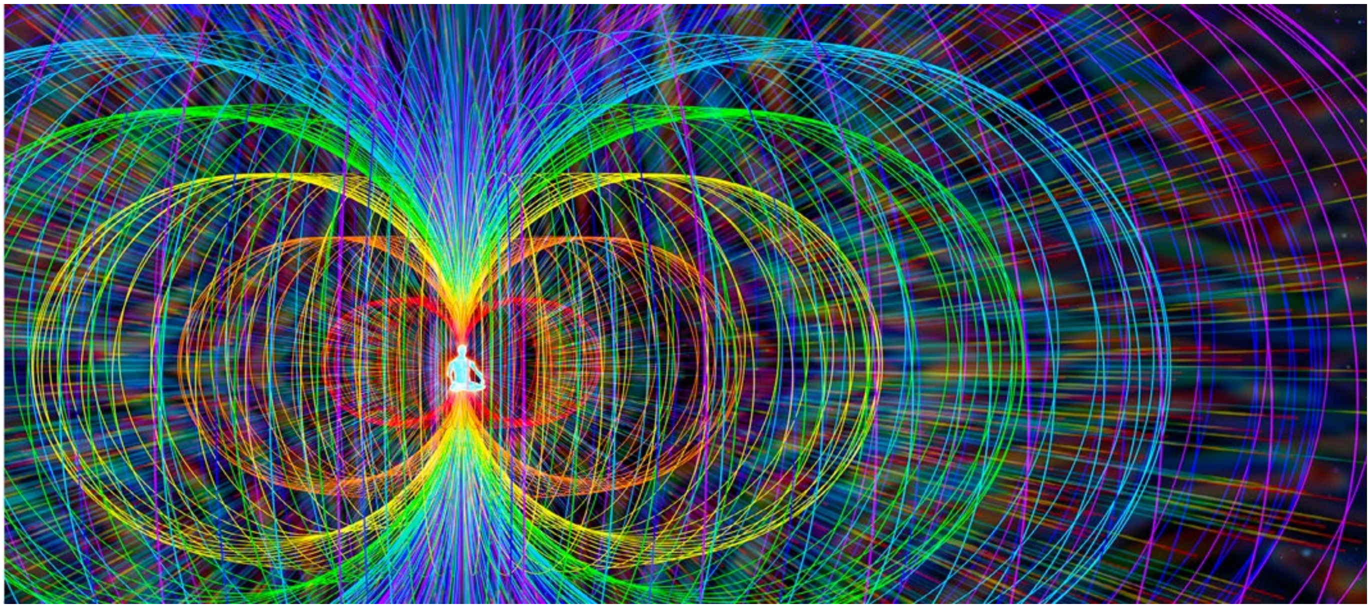
Every job I after that had a strong element of needing to understand the electromagnetics. I thought my physics background helped me get up to speed. I learned about signals and the electronics part of it. Now I teach electronics, so it worked.

Holden: Looking at your crystal ball, what can we look forward to if they continue to reduce this size of those devices down to the atom level and consequently their rise and fall times get increasingly shorter?

Bogatin: I think about that a lot. I'm not an expert on the semiconductor physics. Some of the impurity doping levels are at PPM levels. When you get to the size of a channel, the volume of a channel where you get only two or three doping atoms in there, how do you design a reproducible part where you have statistics start influencing the behavior of those elements? I believe there's a definite limit to Moore's Law as we design components today.

I think Intel is at the 4-nanometer node. That's 40 angstroms for the channel size, so maybe it will be in the 2-nanometer node soon, which may be the limit to where statistically you can't dope a channel enough. What's beyond that? I hear talk about growing in the third dimension and building multilayer active devices.

If I was a young physicist or engineer and I wanted to get into semiconductor devices, I would be looking at molecular electronics and quantum computing. I have a professor colleague here who's an expert in genetic com-



puting using DNA as computational elements. He designs strands of DNA that do computation and circuits. It's genetic circuits; he makes chemical reactions go through oscillations and feedback loops. He says it's not designed to replace digital logic. It won't be fast, but it can control biological systems. Maybe I would get into genetic circuits. There are many directions to head and I'm not sure which will be the horse that wins.

Shaughnessy: Eric, where is the cutting edge of technology now or in the next few years?

Bogatin: There are the high-end technologies, the high-speed serial links. The question always arises: When do we make the transition from electrons to photons? When will copper die? We did an issue on that in SI Journal last year. There are systems demonstrated at DesignCon at 224 gigabits per second with copper. You're pushing the limits, and 224 gigabits per second per lane is pretty incredible in copper. Will you do 512? Maybe that's where optical will be. Thinking about optical interconnects is something in the future.

A big challenge is power consumption. If I was looking for a direction, I would look at power consumption, thermal management, and how you engineer for lower power consumption circuits. That's becoming a limiting feature. There are a lot of these network processor chips. Google, NVIDIA, Facebook, even Uber have them. They have huge server farms, and they do some of their own chips. Some of these network processor chips take 500 amps of current. Even if it's not high speed, per se, there are still challenges in doing the power routing.

There are many important design challenges for high-current distribution. IoT is this huge thing, and in these designs, energy harvesting is important. Many of these are portable. You need energy storage, energy harvesting, and the low-power communications to some gateway or to the cloud directly. There's a lot of life in the low end that requires good, thorough engineering. We spend so much time talking about just signal integrity and the highest high-speed serial links, but gosh, there are a lot of tough engineering challenges throughout the entire spectrum of electronic products today.

Shaughnessy: This has been great, Eric. Thanks for doing this.

Matties: Yes, Eric, it's always a pleasure.

Bogatin: Thank you all for the opportunity. **DESIGN007**

My Experience With Maxwell

Feature Article by Happy Holden
I-CONNECT007

I was first introduced to James Maxwell in 1967 as a college student. I had to decide whether I would take the Maxwell fields course or the switching and coding course. Being a chemical engineering major with a co-major in control theory, I had heard about the trials and tribulations of the infamous Maxwell fields course.

After a lot of consideration, I decided to take the switching and coding course, since it was more related to computer theory, while the fields course was more related to RF, power generation/distribution, and communications.

In those days, our transistors, tubes and ICs were still pretty slow, except for radio, radar, etc. At that time, signal integrity in board layout was not an issue. I was using RTL, DTL and slow TTL logic on breadboards of non-plated through-holes with tinned-copper wire and Teflon® spaghetti tubing (Figure 1).

But after talking with students who had managed to successfully pass the fields course, I was awed by the mathematical rigors they had endured. I was astounded when these very same students found thermodynamics so difficult. Maxwell's eqations are not easy.

Maxwell's Equations

Born in Edinburgh, Scotland in 1831, Maxwell packed a variety of discoveries into his 48 years of life. He made advances in astronomy, the kinetic theory of gases, and optics. Maxwell was also something of a theologian, with deep knowledge of the Psalms.

But Maxwell is best known for his work in electromagnetism and the set of four partial differential equations that bear his name. Maxwell's equations, also known as Maxwell-Heaviside equations, explain how electrical and magnetic fields are generated, as well as how they interact with each other. He is also credited with being the first to calculate the speed of propagations of electromagnectic waves—the speed of light. Albert Einstein was one of Maxwell's biggest fans.

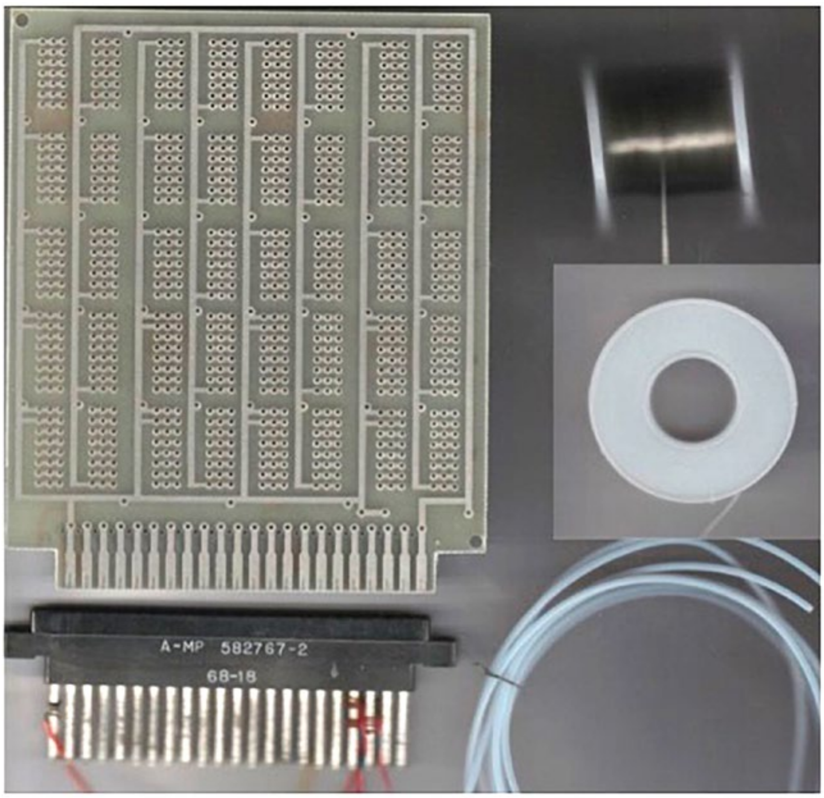


Figure 1: A non-plated through-hole breadboard for tinned wire with spaghetti tubing. If it works here, don't worry about Maxwell.

$$\nabla \cdot E = \frac{\rho}{\epsilon_0}$$

$$\nabla \cdot B = 0$$

$$\nabla \times E = -\frac{\partial B}{\partial t}$$

$$\nabla \times B = \mu_0 \left(J + \epsilon_0 \frac{\partial E}{\partial t} \right)$$

Figure 2: Maxwell's four coupled differential equations. (Source: Wikipedia)

It's not a stretch to say that every electrical and electronic technology that we use today was enabled by Maxwell's work 160 years ago.

Here's what some of the top minds of his day, and today, have to say about Maxwell's contributions:

"Maxwell's equations have had a greater impact on human history than any 10 presidents."

—Carl Sagan

"From a long view of the history of mankind—seen from, say, 10,000 years from now—there can be little doubt that the most significant event of the 19th century will be judged as Maxwell's discovery of the laws of electrodynamics."

—Richard P. Feynman

"The only things we really know are Maxwell's equations, the three laws of Newton, the two postulates of relativity, and the periodic table. That's all we know that's true. All the rest are man's laws."

—Dean Kamen

"Since Maxwell's time, physical reality has been thought of as represented by continuous fields, and not capable of any mechanical interpretation. This change in the conception of reality is the most profound and the most fruitful that physics has experienced since the time of Newton."

—Albert Einstein

"Maxwell's equations didn't just change the world. They opened up a new one."

—Ian Stewart. DESIGN007

Thinnest Ferroelectric Material Ever Paves Way For Efficient Devices

Advanced materials known as ferroelectrics present a promising solution to help lower the power consumed by the ultrasmall electronic devices found in cell phones and computers. Ferroelectrics are a class of materials in which some of the atoms are arranged off-center, leading to a spontaneous internal electric charge or polarization. This internal polarization can reverse its direction when scientists expose the material to an external voltage.

Unfortunately, conventional ferroelectric materials lose their internal polarization below around a few nanometers in thickness. This means they are not compatible with current-day silicon technology. This has prevented the integration of ferroelectrics into microelectronics.

But now a team of researchers from the University of California at Berkeley performing experiments at the U.S. Department of Energy's Argonne National Laboratory have found a solution that simultaneously solves both problems by creating the thinnest ferroelectric ever reported and the thinnest demonstration of a working memory on silicon.

In a study published in the journal Science, the research team discovered stable ferroelectricity in an ultrathin layer of zirconium dioxide just half a nanometer thick. The team grew this material directly on silicon. They found ferroelectricity emerges in zirconium dioxide when it is grown extremely thin, approximately 1-2 nanometers in thickness.

The researchers were also able to switch the polarization in this ultrathin material back and forth with a small voltage, enabling the thinnest demonstration of a working memory ever reported on silicon. Beyond the immediate technological impact, this work also has significant implications for designing new two-dimensional materials.

(Source: Argonne National Laboratory)

Forget What You Were Taught

Beyond Design

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

Ralph Morrison was a physicist who promoted the belief that electromagnetic energy flows in spaces, not the traces. That energy does not flow in the copper traces of a PCB, but rather the energy follows the traces acting as a waveguide and propagates through the dielectric material. This explains many electromagnetic (EM) effects such as radiation from outer microstrip layers and from stripline fringing fields, how components can be magnetically coupled, and why crosstalk is created by overlapping EM fields. But it also raises a few questions.

The simplistic approach to analyzing electronic circuits is to use the lumped element model. This methodology assumes that the attributes of the circuit—resistance, capacitance, and inductance—are concentrated into idealized electrical components connected by a network of perfectly conducting wires. However, that is not the case. As the frequency and rise time increase, these elements become distributed continuously through the substrate along the entire length of the trace. The copper trace and the adjacent dielectric materials become a transmission line, the skin effect forces current into the outer regions of the conductor, and frequency dependent losses impact the quality of the signal. The PCB trace now behaves as a distributed system with parasitic inductance and capacitance characterized by delay and scattered reflections. The behavior we are now concerned about, occurs in the frequency domain.

We were taught in circuit theory (and still are) that electric energy is made up of electrons moving in the conductors. Unfortunately, circuit theory does not consider the flight time of

the signal, energy dissipation, and reflections which are key issues of digital logic design. It incorrectly teaches that:

- The conductors carry the energy
- The load instantly affects the flow of energy
- The transmission line has no losses

So now, it is time to forget what you were taught and instead think in terms of electromagnetic field theory. We must focus on the movement of EM fields and not on the flow of electrons.

The first law of thermodynamics states: Energy can neither be created nor be destroyed; it can only be transferred from one form to another. When an electromagnetic wave propagates from the source, it transfers energy to objects in its path. Electromagnetic fields transport all the energy on a circuit board. This carrier wave can transport information, or it can supply energy to power an IC. If information is carried, then the propagation of the wave alters the timing of the data and must be added to the delay of the circuit. The energy supplied to a load is carried in the space between the connecting conductors and not in the conductors themselves. This concept applies at all frequencies including DC. EM fields can also move energy in free space but not as DC.

Electromagnetic energy travels at the speed of light in a vacuum or air but is delayed by the dielectric materials used to construct multilayer PCBs. The relative permeability or dielectric constant (Dk) of the surrounding materials impacts the velocity of propagation (v) at the speed of light (c).

Equation 1

$$v = \frac{c}{\sqrt{Dk}}$$

As the dielectric constant of a material increases, the velocity of propagation slows down. With a Dk = 4, the velocity is half the speed of light. If one uses different core and prepreg materials in the substrate, then each layer will have varying speeds depending on the dielectric constant. This will impact the timing of memory buses in particular.

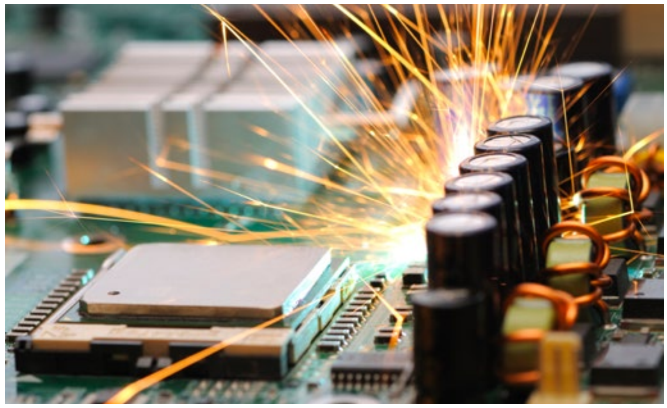


Figure 1: A trace disconnects when there is too much energy in the field.

What Happens at Low Frequency and DC?

Voltage differences can exist between points in space or between conducting surfaces. Electric fields exist at all frequencies including DC. In a field representation, lines of force start on a fixed amount of positive charge and terminate on the same amount of opposite charge. When the lines are close together, the forces are the greatest.

A small electric field within a conductor is required for current flow. When there is no current flow there is no field. The velocity of electrons in copper is extremely slow. In a typical geometry, the average velocity is less than 1 cm/s. This occurs because the electron density in a conductor is extremely high. What is important is the nature of the electric and magnetic fields.

At low frequencies (below 100 kHz), most of the field generated by a circuit returns to the source. This is a good example of where circuit theory can be put to good use. At DC, the fields are unchanging, and the energy is moving at about half the speed of light. We need to know the value of current flow to calculate voltage drop, power losses, and the magnetic field.

The fields associated with traces over a conducting plane are located under the traces. At frequencies below a kilohertz, the series inductive reactance between two points is lower than the resistance between these points. Therefore, at low frequencies current patterns are controlled by conductor resistance. The field required to move current in a solid plane at DC is very small.

Why Does a Thin Trace Overheat and Behave Like a Fuse?

The way I see it: electromagnetic energy moving through the dielectric induces current to flow in the conductors. The current is a result of the energy in the dielectric. It is an effect, not the cause. The trace and the plane bound the energy and thus it is the energy density between the conductors that determines the current density in the trace geometry. The wider and thicker the conductor (Figure 2), the more energy can be contained in the field. Thus, a larger cross-sectional area of copper is required to deliver higher energy at DC. However, the plane has sufficient capacity to support the energy flow. If the trace is too narrow, then the contained energy will dissipate as heat until the trace disconnects.

A DC power integrity simulator highlights areas of high current density, which will lead to higher temperatures. A low thermal resistance ensures that the heat is transferred through the material much faster. This resistance is directly proportional to the length of the thermal path and inversely proportional to the cross-sectional area and thermal conductivity of the thermal path.

For instance, if the load is shorted, then the EM field will try to maintain the output voltage by providing more energy. As the energy level of the field increases, it looks for the weakest link, which is the point of lowest resistance on the trace segment. The excess energy will then transform into thermal energy, vaporizing the trace at the weakest point.

The trace also cools by conduction through the substrate. But as the trace temperature increases, the dielectric material beneath degrades, passing through the stages of the glass transition to thermal decomposition and finally delamination. Deterioration of the underlying dielectric material will reduce the effective cooling capacity of the material and thus increase the temperature of the trace.

If there is current flow in a conductor, an electric field must exist. At DC, this field will be the same at any depth. However, a changing magnetic field associated with AC flow forces the current to bind to the perimeter of the conductors. This is known as the skin effect and applies to digital circuits, where currents flow very near the surface of the conductors. For a sine wave current at 1 MHz, the skin depth in copper is only 66 μm. This means that the current at this depth is the surface current reduced by 8.68 dB. At 1 GHz, the skin depth is 0.066 μm and decreases with higher frequency.

As frequencies and rise times continue to increase, the PCB designer needs to consider the path of electromagnetic fields through the substrate. To fully understand the operation of digital circuits, we must focus on the

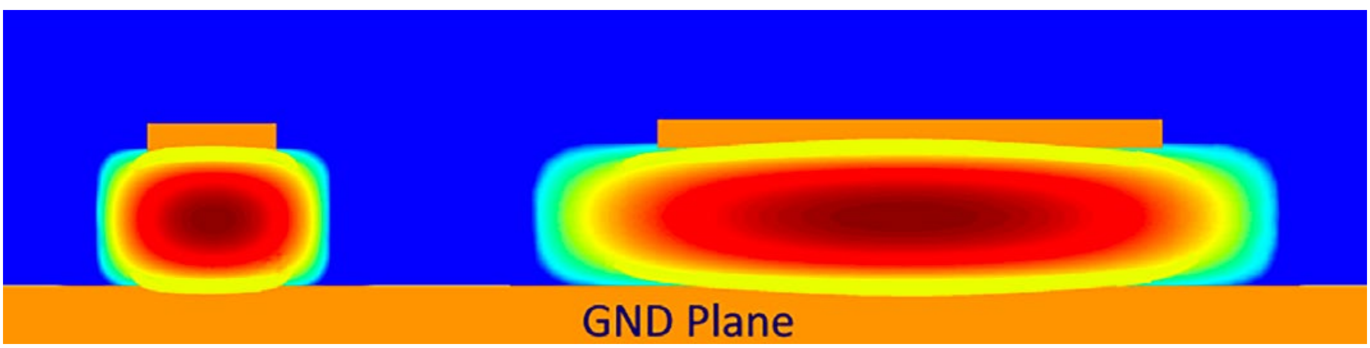


Figure 2: Small traces (left) confine less energy than larger traces (right).

movement of EM fields and not on the flow of electrons. This also applies to DC analysis of the power distribution network. Rather than considering the current capacity of a conductor, one should allow for the amount of field energy that the conductor can confine.

Key Points

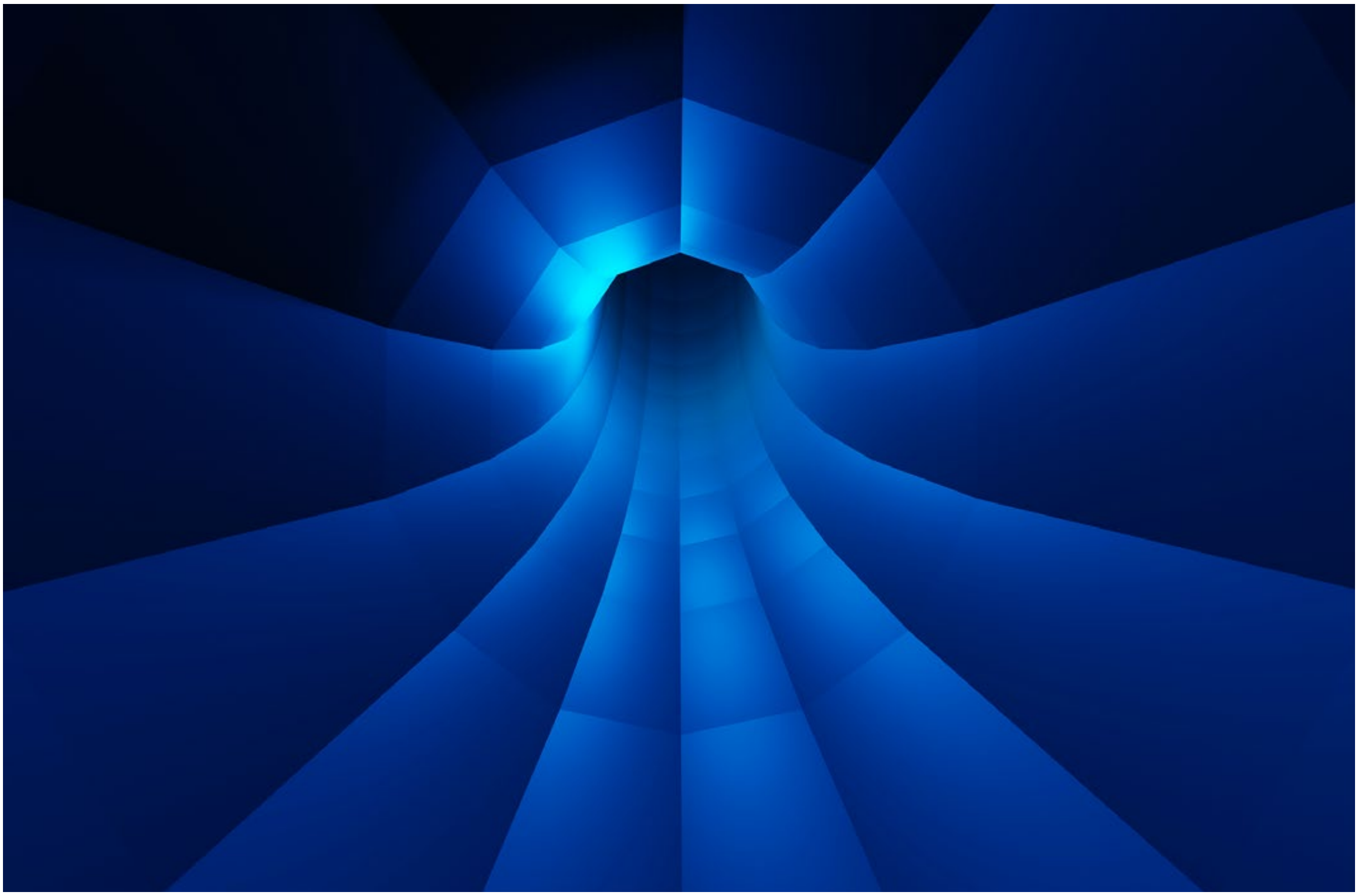
- Energy does not flow in the copper traces of a PCB, but rather the energy follows the traces acting as a waveguide and propagates through the dielectric material.
- The lumped element model assumes that the attributes of the circuit—resistance, capacitance, and inductance—are concentrated into idealized electrical components connected by a network of perfectly conducting wires.
- A distributed system disperses elements through the substrate along the entire length of the transmission line and is characterized by delay and scattered reflections.
- Forget what you were taught in circuit theory and instead think in terms of electromagnetic field theory. We must focus on the movement of EM fields and not on the flow of electrons.
- An EM carrier wave can transport information, or it can supply energy to power an IC.
- When information is carried, the propagation of the wave alters the timing of the data and must be added to the delay of the circuit.
- Electric fields exist at all frequencies including DC.
- The velocity of electrons in copper is extremely slow. In a typical geometry, the average velocity is less than 1 cm/s.
- The fields associated with traces over a conducting plane are located under the traces.
- At frequencies below a kilohertz, the series inductive reactance between two points is lower than the resistance. Therefore, at low frequencies, current patterns are controlled by conductor resistance.
- Electromagnetic energy moving through the dielectric induces current to flow in the conductors.
- The trace and the plane bound the energy and thus it is the energy density between the conductors that determines the current density in the trace geometry.
- The wider and thicker the conductor, the more energy can be contained in the field.
- If the trace is too narrow, then the contained energy will dissipate as heat.
- A DC power integrity simulator highlights areas of high current density, which will lead to higher temperatures.
- The trace also cools by conduction through the substrate.
- The skin effect applies to digital circuits, where currents flow very near the surface of the conductors. DESIGN007

Resources

1. Beyond Design by Barry Olney: The Wavelength of Electromagnetic Energy, The Big Bang-Lumped Element to Distributed System, Surface Roughness.
2. An interview with Rick Hartley discussing Ralph Morrison by Judy Warner.
3. "Fast Circuit Boards" by Ralph Morrison
4. "The Energy Density of Electromagnetic Waves" by Cadence.

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 or contact Olney, click here.

Electronics vs. Physics: Why Vias Don't Get Hot



Feature Article by Douglas Brooks, PhD and Johannes Adam, PhD
ADAM RESEARCH

Most of are aware that when we pass an electrical current through a trace (conductor), the trace will heat up. This temperature increase is caused by the I^2R power loss dissipated in the resistance of the trace. The resistance of a copper trace is mostly determined by its geometry (cross-sectional area), and there are lots of studies trying to look at the relationship between the current down a trace (of known size) and the resulting temperature of the trace[1].

But the situation is much more complicated than this. There are physical properties that exist that result in helping to cool the trace. These properties are usually a combination of conduction of the heat away from the trace through the material, convection of the heat away from the trace through the air, and radiation of the heat away from the trace. A stable temperature is reached when the I^2R heating equals the cooling—i.e., when the electronic and physical properties are balanced[2].

We have spent several (enjoyable) years collaborating on these effects. Douglas is an electrical engineer and understands electronics. Johannes is a thermodynamics physicist, and understands heat transfer. Between us we learned a lot about the thermal characteristics of vias.

Vias Don't Get Very Hot

Traces heat because of the current through their resistance, resulting in I^2R power losses.

This temperature increase is caused by what we call "Joule heating:"

Joule heating is the physical effect by which the pass(age) of current through an electrical conductor produces thermal energy. This thermal energy is then evidenced through a rise in the conductor material temperature, thus the term "heating." One can see Joule heating as a transformation between "electrical energy" and "thermal energy," following the energy conservation principle[3].

We are all aware that, in general, for a given trace an increase in current through the trace will correspond to an increase in temperature of the trace. When it comes to vias, the industry guidelines have generally been to size the cross-sectional area of a via to equal that of its parent trace. Then the via will be the same temperature as the trace. IPC formalizes this guideline in IPC-2152:

The cross-sectional area of a via should have at least the same cross-sectional area as the conductor or be larger than the conductor coming into it. If the via has less cross-sectional area than the conductor, then multiple vias can be used to maintain the same cross-sectional area as the conductor[4].

Our research discovered that this guideline is totally false. In our book[5] we report on experimental results where we put current through a 10-mil diameter, plated (1.0 ounce) via. When we put 4.75 A through the via, we recorded a via temperature of 64.5°C. When we put 8.6 A through it, the via temperature was only 44.5°C, some 20°C cooler. The difference? In the first case the parent trace was 27 mils wide. In the second case the parent trace was 200 mils wide. The much wider trace in the second case provided a significant heat sink for the via. The heat conducts away from the via so rapidly that the via simply cannot get much hotter than the parent trace. The implication of this, of course, is that many fewer vias are needed on our high-current-carrying traces.

In electrical engineering we understand that the temperature of a conductor is related to the current. But in the special case of vias, the physics of heat transfer dominates the situation, and the temperature of the via is not directly related to the electrical current.

Thermal Vias are Not Very Effective

Often, if we have a hot component on the top layer, we will put a copper pad under it to help spread the heat. Some people recommend dropping copper (filled) vias from that pad to "something" on the bottom layer of the board. That "something" might be a copper pad the same size as the top pad, or it might be a copper plane of some sort. Those vias are called "thermal vias," and the internet is filled with articles and advice about them.

But almost all the references say that multiple thermal vias (10s of them) are needed because each one is only marginally effective. It turns out there is a very good reason why thermal vias are not very effective. It has to do with physics.

A thermal via generally connects between two surfaces. Thermal conductivity between two surfaces is a function of, among other things, the parallel overlapping areas. The pad areas are, of course, much larger than the thermal via cross-sectional areas. So, the presence of the underlying pad has already provided a significant path for thermal con-

It turns out there is a very good reason why thermal vias are not very effective. It has to do with physics.

ductivity through the board material, and therefore temperature reduction, between the two pads. A substantial amount of cooling (temperature reduction) has already occurred before the first thermal via is applied[6]. The physics of thermal conductivity through a material has already accomplished a lot of what we needed. That is why the thermal vias have only a marginal effect on further cooling[7]. DESIGN007

References

1. See, for example, IPC 2152, "Standard for Determining Current Carrying Capacity in Printed Board Design," 2009, IPC.com.

2. See Brooks and Adam, *PCB Design Guide to To Via And Trace Currents and Temperatures,* Artech House, 2021, for a complete discussion of PCB trace heating and cooling.

3. We could say this is a transformation between electronics and physics. See simscale.com/docs/simwiki/heat-transfer-thermal-analysis/what-is-joule-heating.

4. IPC-2152, page 26. This was generally considered true by the industry and UltraCAD wrote some articles about 10 years ago saying the same thing.

5. See Reference 2, Chapter 8, for an extensive discussion of via temperatures.

6. See Brooks, Adam, and Castro, "Does a Heated Conductor Have Signal Integrity Implications?" *PCD&F,* March 2022.

7. We cover thermal vias in Reference 2, Section 8.7.

Dr. Douglas Brooks is the founder of UltraCAD Design and a longtime signal integrity instructor.

Dr. Johannes Adam is a thermo-dynamics physicist and founder of ADAM Research.

Does Physics Help with Electrical Engineering?

Illustration by Kingsley Neb

Two questions that are probably more important to ask: How does physics help with electrical engineering? And would it be beneficial to study both? First, how does it help?

Physics is a fundamental scientific discipline, maybe the most fundamental scientific discipline, depending on whom you ask! The study of physics builds a broad base of knowledge in mathematics and science as a way to understand how the universe works. Studying physics provides students with the kind of problem-solving and logic that, in turn, can be applied to technological advances.

If physics is the general, then electrical engineering is the specific. While earning a physics degree definitely opens doors to an array of job opportunities, adding electrical engineering to the mix will exponentially expand your options.

Electrical engineering provides practical skills. It takes the scientific knowledge and the mathematical complexities and transforms them into innovative ideas and new ways to design and build. But it's that knowledge of physics that helps the electrical engineer grasp the constraints inherent to a particular problem and allows him or her to develop a practical approach to achieving a solution.

(Source: Elmhurst University)

Leveraging Digital Automation to Accelerate PCB Design

Digital Transformation

by Stephen V. Chavez, SIEMENS EDA

What exactly is digital automation? I see it as the simplification of manual tasks that have been optimized in the digital world to the point where they require the least amount of effort to successfully do what they are required to do in the real world. We adopt digital technologies everywhere else in our lives, so it should be a natural progression to do so more fully in PCB design.

A good example of digital automation that comes to mind is something as simple as using an app on your cellphone to scan in a business card for collecting contact data information, and then collating this data digitally and providing a usable output. Another prosaic example is using a GPS to provide directions interactively to efficiently get you from point A to point B in the least amount of time. When you go to a restaurant and use your phone to place your order to quickly get a hot meal served to your table, most of this process flowed through the digital world. Think about all the items and processes that can now be tracked and optimized, and how these captured metrics can be used to increase efficiency and lower cost. Now, we all know PCB design is not quite that simple, but the concept of digital automation is basically the same. So, how do we leverage digital automation?

When it comes to leveraging digital automation in PCB design in today's industry, designing faster, better, and cheaper are critical imperatives in the face of several factors that challenge most engineering teams—ranging from addressing the law of physics in today's complex designs, the reduction of project bud-

gets and resources, and coming out of the dark cloud of COVID-19. Over these last two years, we adapted to the new virtual work environment, working with remote team members as each of us were basically in our respective physical isolation. Yet, many in the industry also had to adapt to functioning in multiple domains and collaborating and integrating with multiple disciplines in the quest for success. We have evolved in this digital world, and many have started leveraging digital automation.

Here are just a few examples of leveraging digital automation regarding printed circuit engineering:

- Process automation
  - Taking a manual process and streamlining it with software. This applies to the most basic things (e.g., CAD instead of mylar), but since everyone is on CAD, it really applies to automated steps on the journey from concept to manufacturing. Examples include levels of constraint-driven automation in routing, simplified component creation, data management, and integration between disciplines to facilitate collaboration, streamlined hand-off to manufacturing.
- Verification automation
  - Replace manual peer reviews. For example, schematic integrity, layout checks for manufacturability or performance.
- Automation enabled by AI
  - Apply AI to evaluate multi-discipline tradeoffs (e.g., power vs. thermal, performance vs. manufacturability), and recommend best options to improve decision making.
  - Apply AI for generative design, where the recommendations from the previous step are automatically executed, for example, placing place decoupling capacitors at optimal locations around an IC to provide the right amount of clean power.

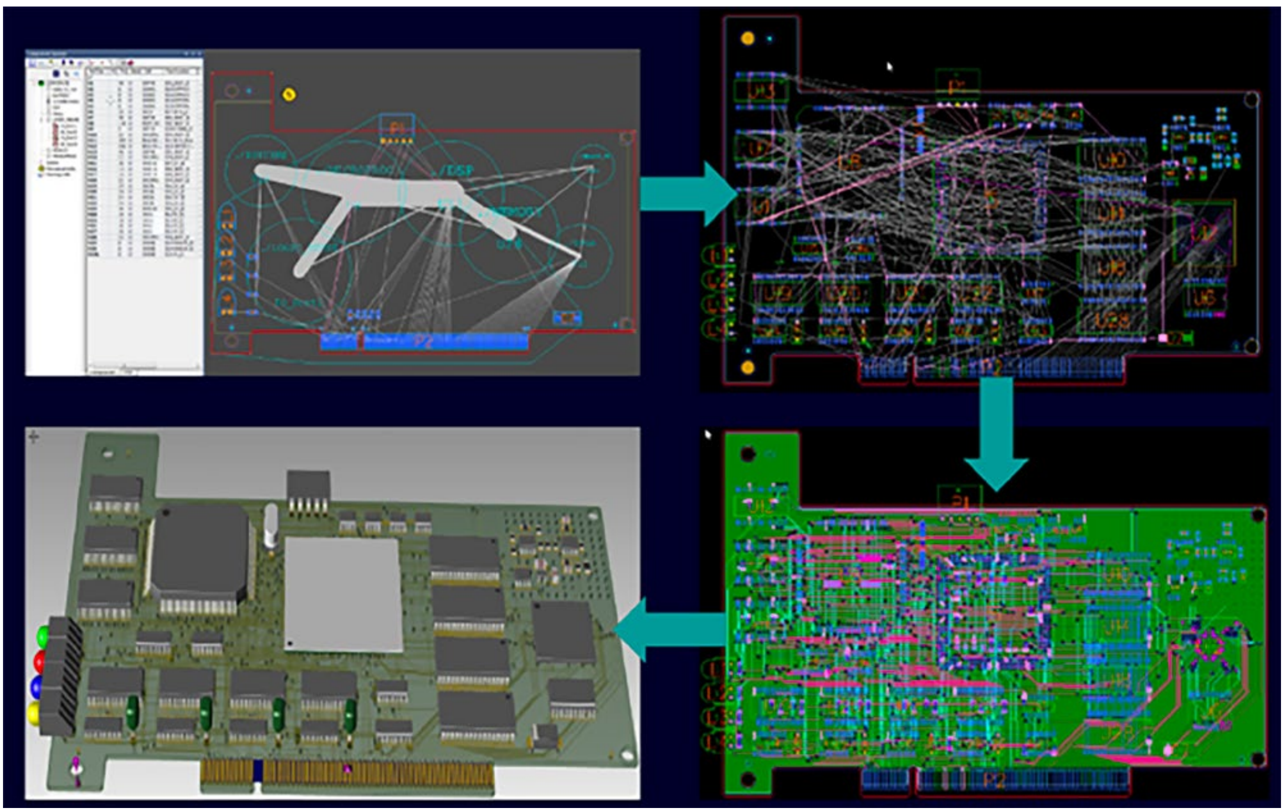


Figure 1: Digital automation is the only way to handle today's design complexity.

Digital automation for productivity and efficiency is the sought-after utopia of the PCB design and manufacturing processes. Today's electronic design automation tools are better suited and more capable than ever in arming engineering teams with tighter integrations, system model-based engineering, and verification automation to achieve this ideal approach in the never-ending quest to attain the highest level of repeated success from one design or project to another. Yet engineering teams continue to face complex challenges internally within their ecosystems and externally to their companies.

With the geopolitical and economic turmoil headlining the news today, combined with the slow recovery from this pandemic, supply chain disruptions continue and intensify the challenges for project teams to design their products while struggling to evolve internal legacy processes and dated methodologies to get to market as fast as possible. Engineering teams must now implement supply-chain resilience at the point of design.

Like many others in the industry, I have embraced our new working environment and have not let it stop me from evolving my skill set as a printed circuit engineer to include designing for supply-chain resilience as well as mastering my CAD tool. Yes, like so many others, roughly two years in physical isolation has me extremely eager to get back out there. Surfing threads on LinkedIn and YouTube, where anyone can easily find educational PCB design content, has gotten to a point of information overload due to the amount of content out there. Viewing content in the virtual world is not quite the same as physically attending an industry conference in person, where in-person, face-to-face interactions and collaboration amplify the potential for success.

I spent an entire week in Santa Clara, California, attending PCB West in October. Oh my gosh, it was crazy awesome. The excitement and buzz of those who attended, presented and or showcased on the show floor was the highest I have ever seen in in my lengthy career attending this conference. It was obvious that people were anxious and eager to get back to in-person interaction and collaboration. Everyone gathered in groups and talked about printed circuit engineering in one form or another. I was finally back in my element, and I know the feeling was mutual for so many others. We all had that same feeling of euphoria.

Among the many discussions I had at the conference, one recurring topic was automation. The gap between the old and the new generations of PCB designers continues to grow, and lack of education and training opportunities within the industry has caused a significant shortage of experienced and talented PCB designers. Those of us who have been around for more than 10 years and still have a lengthy runway in our career in PCB design have a great future ahead of us. Yet, as I see the pool of young EEs entering the PCB design ranks with a mindset shaped by growing up in the digital age, I am delighted to see that a digital virtual existence, automation, and optimization seem to come naturally to them. I have a good feeling for the up-and-coming printed circuit engineers, and I am confident that the industry will be in good hands.

In my PCB West presentation, "PCB Design Best Practices," I addressed my pillars of best practices. These are:

- Digital integration and optimization
- Engineering productivity and efficiency
- Digital prototype-driven verification
- System-level model-based engineering
- Supply-chain resilience

In this two-hour session, I explained that leveraging digital automation throughout each pillar increases the potential for overall success while reducing design cycle time. This increases multi-discipline collaborations and multi-domain integrations, and reduces project cost and risks. I shared my experiences in how leveraging digital automation allowed me and my teammates to achieve success faster than legacy methods and approaches to PCB design.

I was surprised to hear that many others agreed with me regarding the adoption and leverage of digital automation. We know we should take advantage of it, and we wonder why more in the industry are not doing so at a faster rate. For one reason or another, implementing and leveraging digital automation always seems to run into resistance in one form or another. Whether it's with implementing tool automation, process optimizations, or multi-discipline collaboration, digital automation has not been fully accepted and implemented.

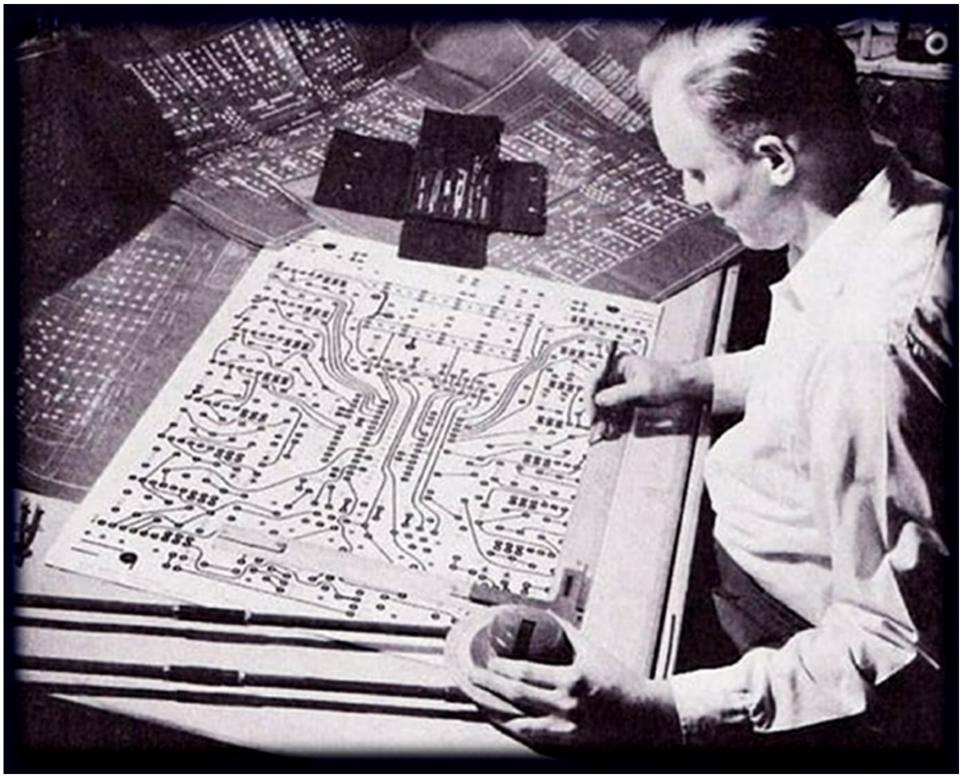


Figure 2: Rationale No. 6: It's more fun to draw a PCB with tape and mylar.[1]

Below are several typical rationales for not implementing and leveraging digital automation:

1. Internal company culture resistance to change (one of the biggest roadblocks to overcome).
2. Current process "works" and is standard (the manual approach).
3. Unfamiliarity with automated features in a tool.
4. Feeling less in control or don't trust a new tool.
5. No time to learn or train on new functions in a tool.

I can attest to all these perspectives firsthand as I have progressed in my career on many diverse, global engineering teams, spanning every market sector: commercial, aerospace, military, and medical.

Not every step in the design process can be automated, but we should always be on the lookout for opportunities to leverage digital automation when and where we can. There are many areas that can take advantage of and leverage digital automation, from multi-discipline collaboration, such as analog, digital, RF, and ME, to multi-domain integration, such as electrical, mechanical, systems, software, manufacturing, verification, and producibility. Today's engineering tools are ripe for this.

If you can reduce your design cycle time by the slightest percentage and get to market that much faster, isn't it worth it? I strongly believe it is. Time is money. Getting your product to market the fastest, with the best quality, the least amount of cost and risk, and within the shortest potential schedule can make all the difference. Believe me, leveraging digital automation in printed circuit engineering makes a difference.

In the coming months, I'll be discussing the five pillars of best practices in PCB design in detail in a podcast series. I'll also be sharing my opinions from my career experiences on the diverse topics of printed circuit engineering on my social media channels. DESIGN007

References

1. *A Manual of Engineering Drawing for Students and Draftsmen, 9th Ed.*, by French & Vierck, 1960, p. 487.

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

Novel Carrier Doping in p-type Semiconductors

The carrier concentration and conductivity in p-type monovalent copper semiconductors can be significantly enhanced by adding alkali metal impurities, as shown recently by Tokyo Tech researchers. Doping with isovalent and larger-sized alkali metal ions effectively increased the free charge carrier concentration and the mechanism was unraveled by their theoretical calculations. Their carrier doping technology enables high carrier concentration and high mobility p-type thin films to be prepared from the solution process, with photovoltaic device applications.

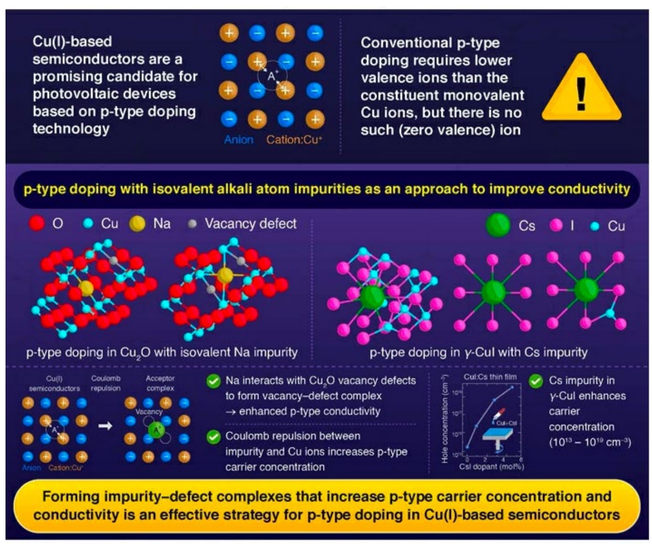
To propose a new carrier doping design for p-type doping in CuI, researchers from Japan and USA recently focused on the alkali impurity effect, which has been empirically used for hole doping in copper monovalent semiconductors, copper oxide (Cu_2O) and $Cu(In,Ga)Se_2$.

In a novel approach outlined in a study published in the Journal of the American Chemical Society, the team, led by Dr. Kosuke Matsuzaki from Tokyo Institute of Technology (Tokyo Tech), Japan, demonstrated experimentally that p-type doping with alkali ion impurities, which has the same valence as copper but larger size, can improve conductivity in Cu(I)-based semiconductors. The theoretical analyses show that the complex defects, which are composed of alkali ion impurity and vacancies of copper ions, are an origion of hole generation (p-type conductivity).

Based on the p-type doping mechanism to form acceptor-type Cu vacancy defect complex, the team investigated larger alkaline ions, such as potassium, rubidium, and cesium (Cs), as acceptor impurities in γ-CuI.

Indeed, the development could be a major leap forward for copper(I)-based semiconductors, and could soon lead to their practical applications in solar cells and optoelectronic devices.

(Source: Tokyo Institute of Technology)



Fitting Physics to Fact

The Pulse

Feature Column by Martyn Gaudion, POLAR INSTRUMENTS

Driving down cost has always been the goal of modern engineering. The phenomenal reduction in manufacturing cost we've enjoyed over the past 50 years is primarily the outcome of engineers whose goal it is to make a good, affordable product. Economic cost-effective production benefits us all, including consumers, defense, aerospace, and automotive sectors.

Composites

Composite materials are incredibly versatile; they blend the characteristics of the component parts and are engineered to result in a product with the benefits of both the composite parts with few of the disadvantages of the individual constituents. In our PCB world, the use of composites gives access to PCB substrates with as many or as few layers as desired and with a wide variety of flavors, from low cost to high reliability and high speed, whilst at the same time juggling the safety and regulatory challenges of the available fire-retardant properties.

Statistics

At a large scale, many of the variations of composite materials, from an electrical perspective, are small enough to be dismissed, but with feature dimensions and interconnect geometries shrinking, it's important to be aware of the inherent variation of the electrical properties of the substrate. These have been dealt with in much detail by the many authors of this column. However, the result for designers of inherent electrical differences dis-

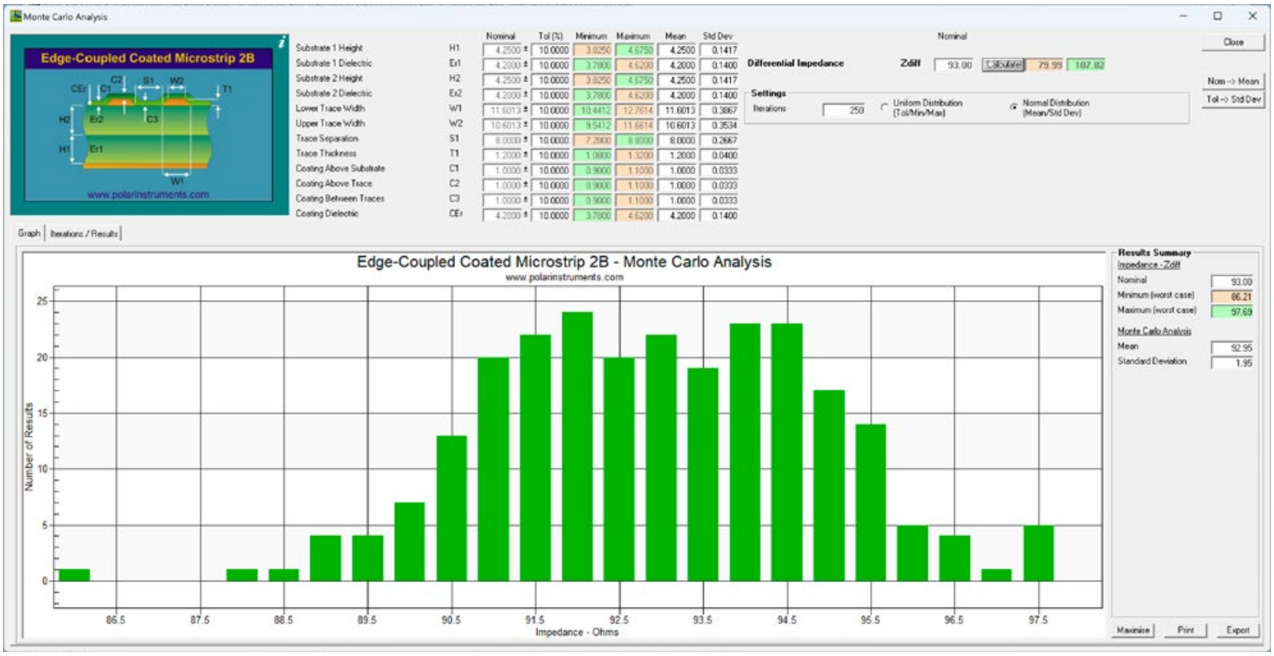


Figure 1: Monte Carlo of 250 transmission lines with a tolerance of 10% on all parameters.

tributed in the substrate is that as a designer you must look at the statistical effect on your design. How much variation is there and how much is that likely to affect prop delay, impedance, or insertion loss over a typical production run?

Monte Carlo Gives a Reality Check

It is easy to focus on one parameter and how that will impact the desired electrical characteristic, however, a far better approach is to set a tolerance for each primary dimension which may impact the circuit, then run a Monte Carlo analysis. This allows you to simulate a production run. I have used characteristic impedance of a 93-ohm differential pair in this example, but the Monte Carlo holds true for any characteristic which is impacted by a number of factors. In this example (Figure 1), I set each raw material dimension a 10% tolerance, and then simulated building 250 PCB transmission lines with a normally distributed, randomly selected set of parameters. This gives a far better impression of how production will vary over a batch of boards rather than simply focusing on one or two characteristic dimensions or looking at the worst case. It does allow you to experiment with which a particular dimension should be more tightly controlled to achieve a higher yield.

Figure 1 demonstrates that with a 10% tolerance on every characteristic over a production run, the majority of impedance results will fall well within 5-6%, but there will be outliers just out of the 10%.

In Figure 2 you can see that in this simulation the tightening of the trace separation specs significantly tightened the distribution with a significant peak on the nominal.

In Figure 3, purely for illustration, I pushed the dielectric constant tolerance out to an exaggerated ±20% and the more benign effects of Er, having only a 1 square root effect, show a minimal flattening when compared with the 10% illustration in Figure 1. These three illustrations demonstrate how a signal integrity engineer needs to balance the material properties of a base material with varying constituent properties and use statistics to see whether there is the potential for good yields based on the simulation. Making a good yielding design from materials which have inherent variabil-

Figure 2: Monte Carlo tolerance of 10% on all parameters but 5% on trace separation.

ity is one of the key areas where a design or SI engineer can add value to the production process and ensure a product is either profitable—if commercial—or within budget for contractual purposes.

When the Going Gets Rough

In the first part of this column, I looked at how an understanding of statistics helps maximise yields when working with composite materials. But copper is not a composite; it is a "pure"

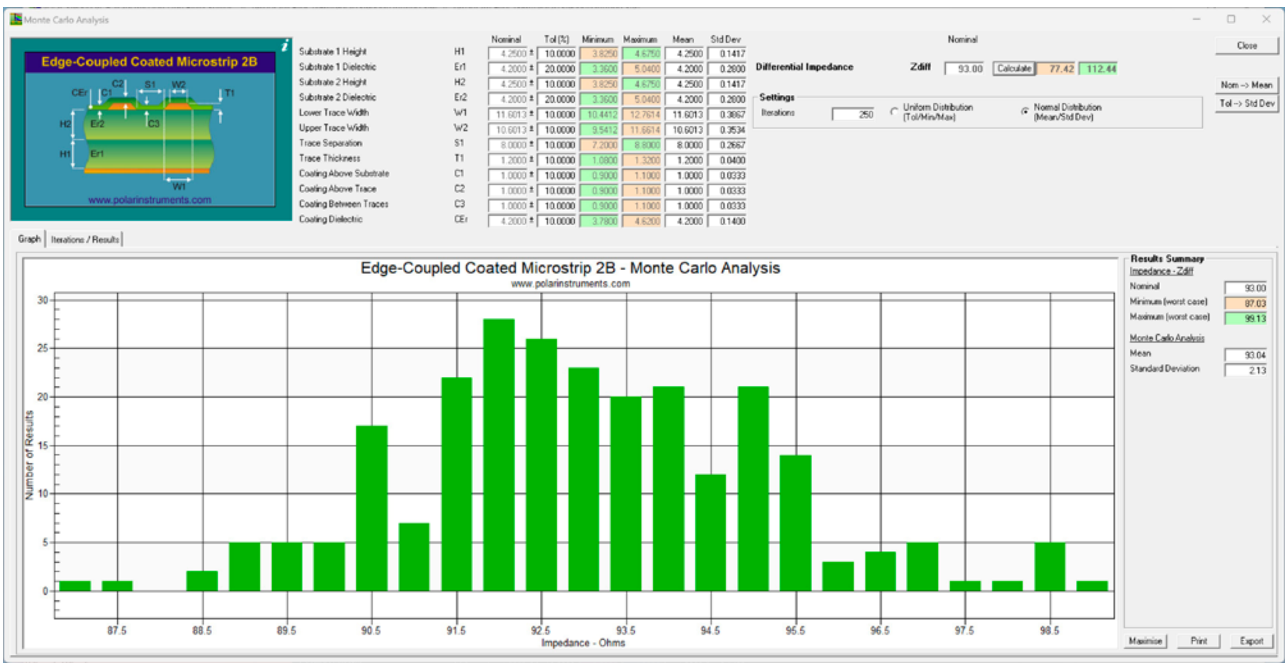


Figure 3: Monte Carlo tolerance of 10% on all parameters but 20% on Er.

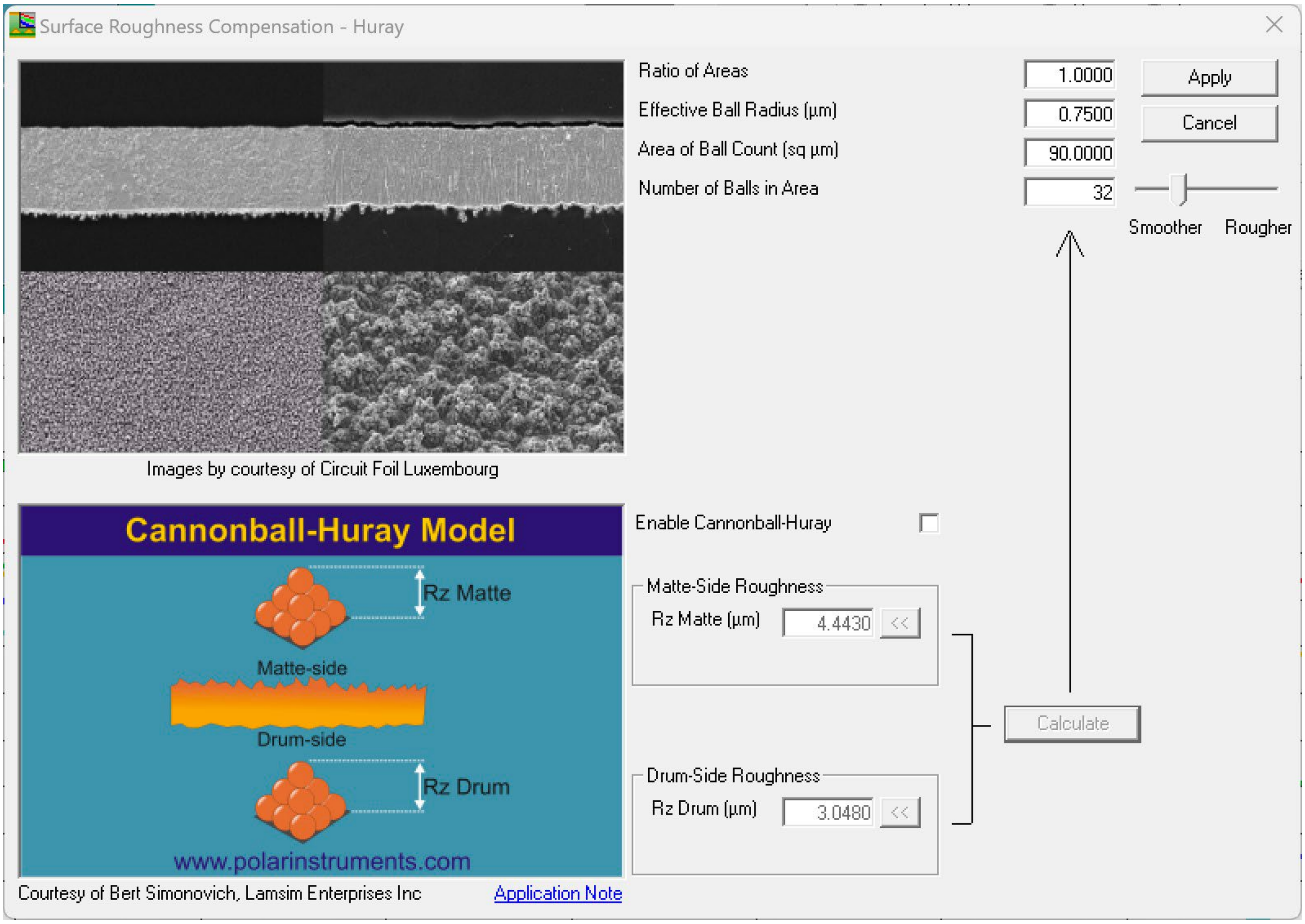


Figure 4: Surface roughness is difficult both to measure and model.

element. However, in high frequency designs the signal largely moves to the surface of the copper, and here it engages with the surface finish. For boards to be useful they must bond and survive assembly, possibly some rework, and then the environment that they will be used in.

This means that the copper must bond sufficiently to the epoxy in the prepreg to ensure that the board does not delaminate during either assembly or its working life. If you are using Gigabit speeds, then you will know that the roughness impacts insertion loss. The complex nature of electroplated surfaces means that it is far from practical to field solve the EM fields and associated losses at the surface, so empirical models are used to make predictions of loss owing to skin effect which are "good enough" for the SI engineer to work with when considering the loss budget.

Chemistry suppliers are gradually improving the pre-treatments in use to permit the use of ever smoother copper, which is easing this modelling situation, but most designs need to take care of roughness modelling. Designers new to the subject may think that they can obtain all the roughness data they need from a datasheet. However, more seasoned professionals will know that, depending on the stackup and the drill arrangements, only some of the surfaces are as in the datasheet; some will be pre-treated for bonding, and others will be plated as part of the plated through-hole process. This adds a layer of electrodeposited copper on the foil or core material and awareness of the stackup and the PCB fabrication process. It also adds the knowledge that maybe one fabricator prefers one approach over another and that for good modelling the SI engineer needs an understanding of both the specified materials and the stackup, as well as the approach of the fabricator. It's not as simple as sitting in a darkened room studying datasheets.

Conclusion

Good use of modelling tools, a knowledge of PCB fabrication, and, even better, a close relationship with your fabricator perhaps will help you answer why "the laws of physics" don't always seem to apply. The fact is that they do, but the materials we use and the geometries that are used mean that SI engineers, PCB designers, and PCB technologists need to know how material composition and interaction with the PCB fabrication process will impact a successful design. With careful use of simulation, your chances of a high yielding reliable design are increased whilst at the same time reducing your need for repeated prototype builds. **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
- You can also view other titles in our full I-007eBooks library.

New Tech Manages Two-Way Power Flow to Commercial Buildings

Researchers at Oak Ridge National Laboratory recently demonstrated a new technology to better control how power flows to and from commercial buildings equipped with solar, wind or other renewable energy generation.

"We are creating an electric grid of the future that allows renewable energy to be deployed in the most effective way," said ORNL's Madhu Chinthavali, who leads the research. "With this new grid interface architecture, operators can control energy flows much more meaningfully, even when power generation is decentralized."

Renewable energy is key to helping the U.S. electricity sector achieve national decarbonization goals. But they also add uncertainty to the electrical grid because they are unevenly available across the country and generate electricity intermittently. Developing and coordinating power electronic systems to incorporate these resources more easily is vital to creating a more resilient grid for reliable electricity.

Several industries could see significant benefits. The technology could be used by a builder or building owner to save money and energy, or it could be installed by a utility for enhanced power control and reliability. The team is moving to the next step in the research: substituting higher-power, commercial converters secured directly from industry.

This will demonstrate that the power electronics hub can manage the megawatts of power handled by electric utilities using components from commercial suppliers. The power electronics hub is an example of the type of technology developed in GRID-C that could be deployed with a potential consortium of partners.

(Source: Oak Ridge National Laboratory)

MilAero007 Highlights

A Focus on Process Control, Part 1 ▶

Michael Carano is a noted subject matter expert with respect to process control, electroplating and metallization technology, surface finishing, and reliability. So, it was only natural that we sat down to talk about mechanizing an existing facility given today's fickle environment.

Michael Carano: A Focus on Process Control, Part 2 ▶

In this second half of our conversation, Michael Carano discusses some of the metrics that fabricators need to consider before investing in new processes, especially process control technologies, and some of the challenges board shops face updating brownfield sites.

IPC Advanced Packaging Symposium: An Urgent Need to Support Global Efforts ▶

After two days of presentations, panel discussions, and impromptu hallway conversation at the IPC Advanced Packaging Symposium, one thing became clear: There is an urgency to support advanced packaging in all regions across the globe.

Copper Foil Market Worth US$17.32B by 2030 at 10.31% CAGR ▶

According to a Comprehensive Research Report by Market Research Future (MRFR), "Copper Foil Market Information by Product, Form, Category, Distribution Channel, and Region—Forecast till 2030," the market is estimated to grow at a 10.31% CAGR to reach USD 17,321.8 Million by 2030.

PCB Legislative Update: HR 7677 ▶

Electronic industry association leaders like IPC, PCBAA, and USPAE have been trekking to Capitol Hill almost weekly this year to reinforce the dire state of the industry and seek additional co-sponsors for HR 7677, a bill supporting the American printed circuit board industry.

Collins Aerospace Receives Milestone Certification for Combined Vision Systems ▶

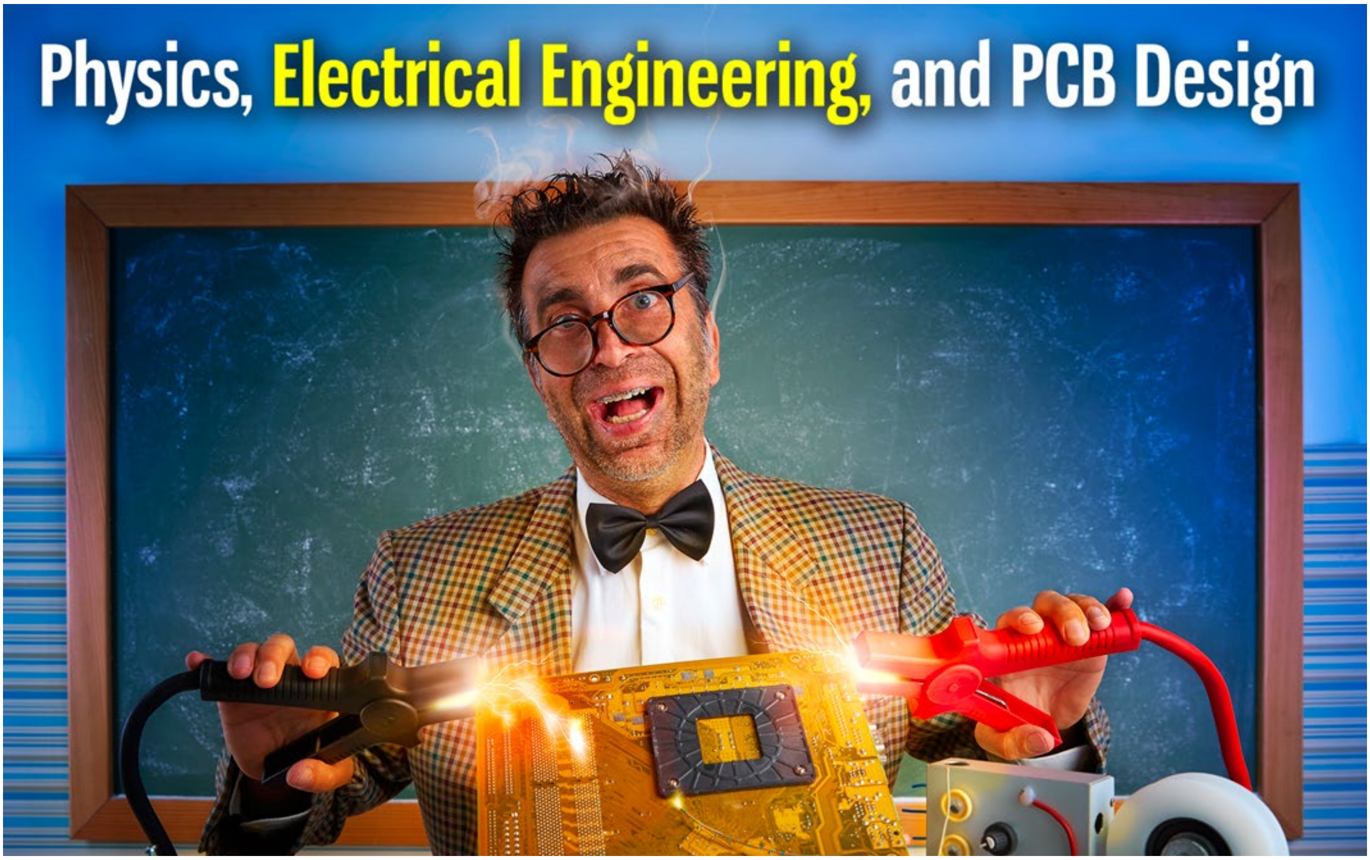
Collins Aerospace has achieved a technical standard order (TSO) for its combined vision system (CVS) for business aviation aircraft. The CVS provides clarity to pilots in all types of weather to navigate aircraft confidently and securely through low visibility situations.

Boeing-Built SES Satellites Send, Receive First Signals ▶

Two newly launched Boeing-built satellites are sending and receiving signals as they continue their journey to their orbital destinations. "Our unique dual-launch configuration was again successful on this mission," said Ryan Reid, president of Boeing Satellite Systems International.

Industry Well Represented on New U.S. Government Advisory Committee on Microelectronics Industry ▶

Two of the electronics industry's most far-sighted and innovative leaders have been named to the U.S. Department of Commerce's new Industrial Advisory Committee (IAC), which will provide guidance to the Secretary of Commerce on a range of issues related to CHIPS for America Act programs.



Physics, Electrical Engineering, and PCB Design

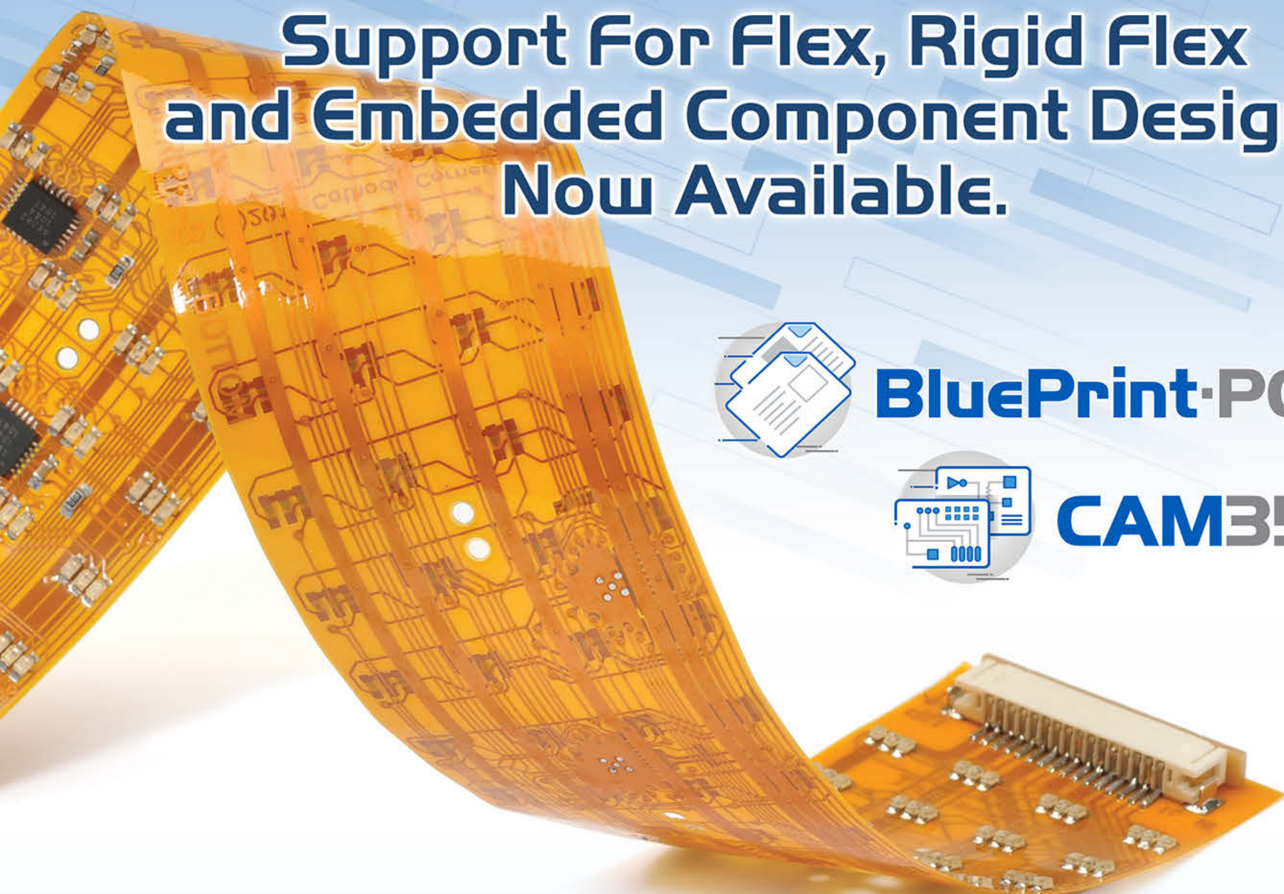
Feature Article by Tamara Jovanovic
HAPPIEST BABY

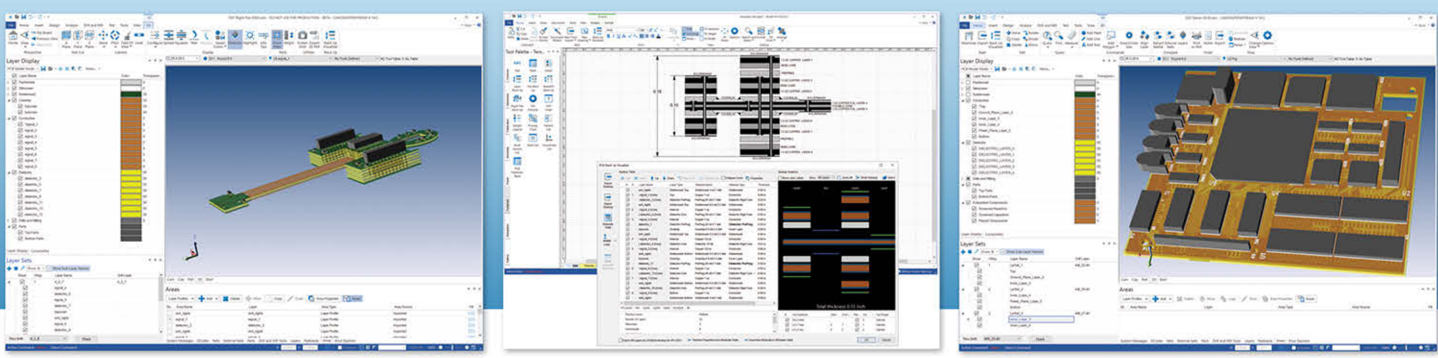
When I was a sophomore in college, I had an amazing professor for Physics II: Electricity and Magnetism. He made a series of complex topics fun to learn, and his personality and way of teaching were almost tailor-made for the way I like to learn. He explained new concepts through practical examples, and always kept students engaged throughout the class, making sure everyone understood the lectures. Physics II was an engineering prerequisite, and I didn't mind taking the class since I really enjoyed the material. However, I did find myself wondering a few times, "Will I ever use any physics in real life?"

It turns out that the answer to the question was yes. Through the rest of my undergrad and my master's program, I have used physics more than just occasionally. To put it plainly, physics is essentially the study of how everything works, and it is everywhere around us. Without realizing it, we use physics while doing simple tasks around the house. The laws of physics describe everything around us, from opening a jar, using our phones, and ironing clothes, all the way to understanding how the human body works, driving a car, natural disasters, and electronics manufacturing.

Electrical engineers use the laws of physics and mathematics to convert electrical energy and power into a circuit, device, or system. Whether it's semiconductors, circuit design, power distribution, grounding, or shielding, physics is woven through electrical design, and it helps solve complex problems even when we don't realize it.

Any circuit board design is a process. In my job, the very first step in PCB design is board planning with mechanical engineers. There is usually a specific industrial design that is established as baseline, and we have to work around that when incorporating mechanics and electronics. We discuss critical components and





parts, their size and potential location, and we make decisions on where they could be placed, as well as how specific features of the product can be implemented and executed. We also have to ensure that board shape and size can house all the components and circuits that need to be implemented. In the meantime, I start working on board schematics and design validation.

Physical strain on the PCB is another aspect of my job that involves physics. Engineers must select the optimal location for mounting and locator holes on the board. Mechanical engineers perform simulations to make sure that they aren't putting unnecessary strain on the board and ensure the board doesn't get damaged during assembly into the product. Once mounting hole locations are identified and the schematic is completed, an electrical/layout engineer can start working on layout.

In layout, even more physics concepts come into play. Boards are getting more complex every day, and EDA tools continue to evolve as well. PCB designers are lucky that a lot of the design rules involving physics and calculations have already been set in pre-fixed rules. These can be adjusted and changed, and absolutely should be tweaked in high-speed designs or specific signal designs. But as they are, any board designer should be able to design a good board without issues just by following these built-in specifications.

The placement of components is critical when designing around a specific IC. Orientation, distance from the part, ground planes and trace thickness all come into play. For optimal board design, a designer needs to make sure that the amount of current going through the signal can be supported by the trace width.

To quote Eric Bogatin's best practices for laying out a 2-layer board, it is best to use 6-mil wide signal traces, 20-mil wide power traces, and 13-mil drilled diameter vias. These guidelines can be applied in multilayer board designs, but note that a 6-mil trace can carry up to 1A of DC current, so if that's overkill for your own application, these parameters can be slightly adjusted to better suit your needs.

Another important aspect of physics that we see when designing PCBs is heat transfer and thermal relief. If temperature guidelines are ignored, a board can overheat and damage the components on it, as well as start misbehaving, and nobody wants that in their product. An engineer must know where to place thermal relief on the board, as well as how to implement it properly.

In addition, thermal relief can sometimes be a bit overdone, and this can drive up the cost of your project. I always spend a decent amount of time learning about the IC I am designing around, studying the datasheet, making sure I am implementing all the necessary subsystems and circuits, and ensuring that I am putting forward the most cost-optimized design.

Certain circuit designs require physics laws and concepts, especially depending on what you're trying to implement. For example, in any RF design, an engineer must consider the specifics for the part that's being used, as well as how to properly ground and shield the part and place critical components around it. Moreover, many countries have electromagnetic compatibility (EMC) standards that mass-market electronics must pass in order to be sold in those countries, and EMC is firmly in the physics wheelhouse. The ability to understand mathematical calculations comes in handy when trying to achieve EMC and signal integrity.

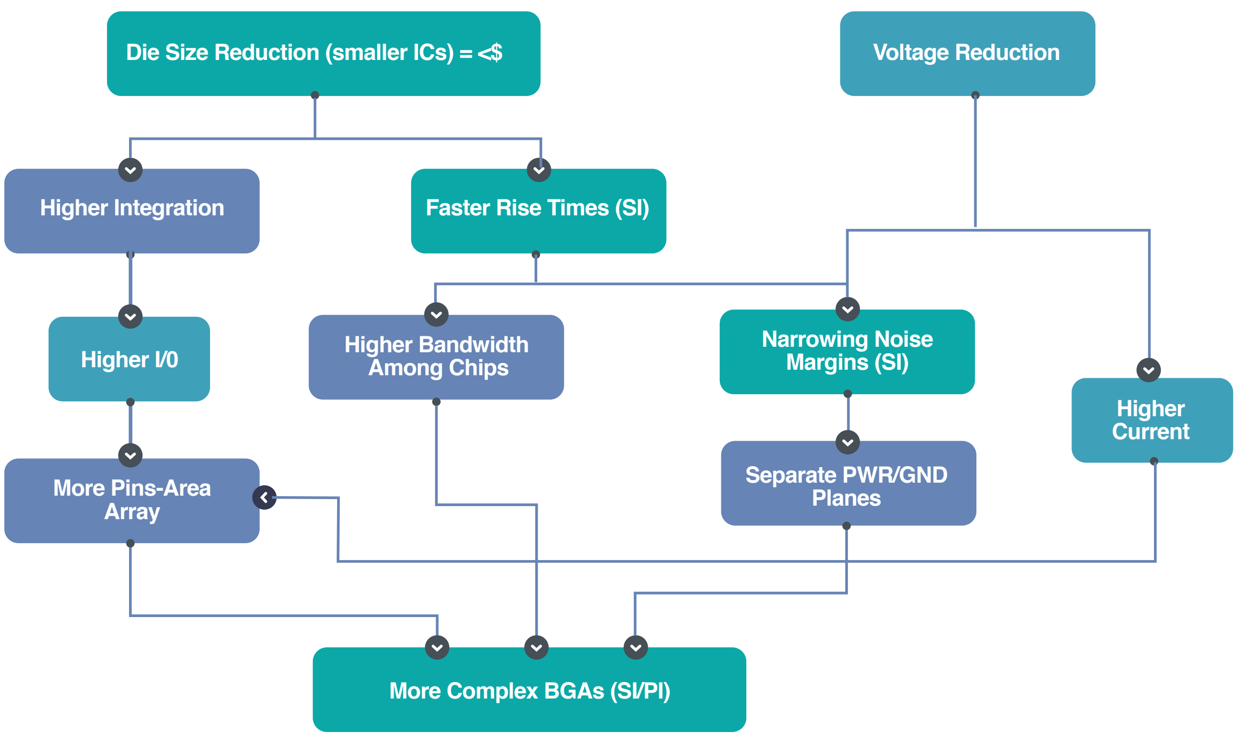
Physics and electrical engineering both play a big role in a successful PCB design. Prudent designers will continue to learn about both disciplines—and how they can affect one another—throughout their careers. **DESIGN007**

Tamara Jovanovic is an electrical engineer with Happiest Baby, a smart bed manufacturer in Los Angeles. She is currently working on her MSEE.

Maxwell vs. Moore

Moore's Law brought everyone into the SI arena of Maxwell's Laws

IC Technology Trends From Moore's Law



DFM 101 Final Finishes: ImmSn

Article by Anaya Vardya
AMERICAN STANDARD CIRCUITS

Introduction

One of the biggest challenges facing PCB designers is understanding the many cost drivers in the PCB manufacturing process. This article is the latest in a series that will discuss these cost drivers (from the PCB manufacturer's perspective) and the design decisions that will impact product reliability.

Final Finishes

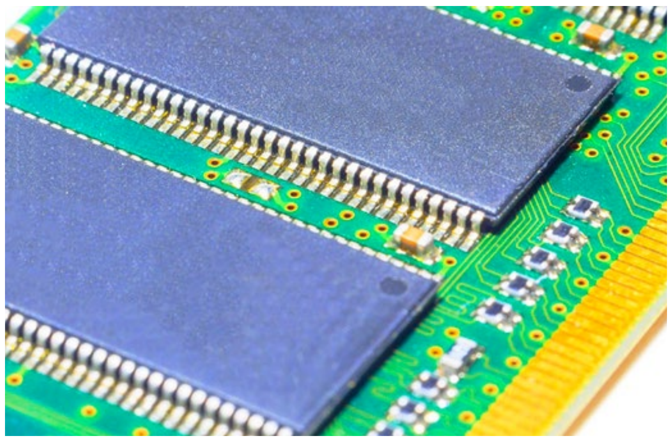
The primary purpose of a final finish is to create electrical and thermal continuity with a surface of the PCB. Final finishes provide a surface for the component assembler to either solder, wire bond, or conductively attach a component pad or lead to a pad, hole, or area of a PCB. Another use for final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

There are a number of 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 a number of considerations that should be part of any decision to choose a final finish:

- Lead (Pb) 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 (Pb)-Free Finishes

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.

Final Finishes

ImmSn (Immersion tin)

Immersion tin is a single layer metallic coating of 15.7 µin to 23.7 µin [0.4 to 0.6 µm] of immersion deposited Sn.*

This process coats a thin layer of tin directly on top of the copper surface. The tin produces an extremely flat surface for mounting of surface mount components with ultra-fine-pitch devices. This also provides a thicker, uniform surface that provides lubrication for press-fit pins. ImmSn is inexpensive and typically used for flat surface. ImmSn is OK for fine-pitch devices with solder mask dams greater than 5 mils [127 µm].

ImmSn is the top choice for press fit pin insertion, for the following reasons:

- Careful handling required
- Poor environmental resistance of intermetallic
- Not great for multiple reflow cycles

*Per IPC-4554 Nominal pad size of 0.060" x 0.060" (1.5mm x 1.5mm)

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 *The Printed Circuit Designer's Guide to... Flex and Rigid-Flex Fundamentals*; and author of *The Printed Circuit Designer's Guide to... 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.

Managing Solder for Fewer Heat Sink Failures

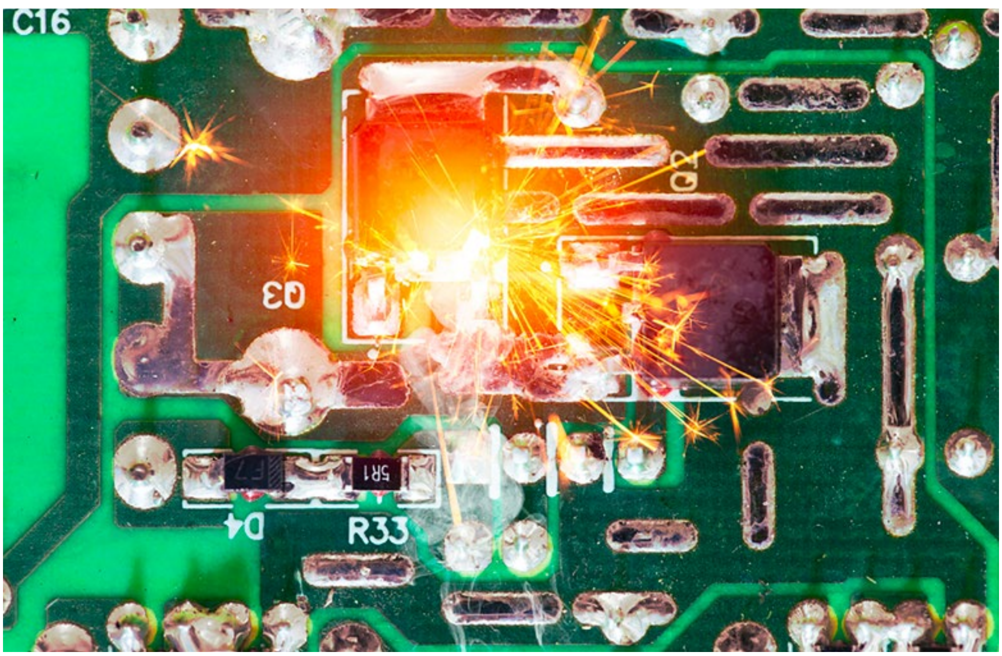
Connect the Dots

by Matt Stevenson, SUNSTONE CIRCUITS

Heat sink failures can be difficult to detect, especially when the failure rates are low. But even if the volume of failures is low, those costs quickly run into thousands of dollars. One of the primary causes of heat sink failure is inconsistent soldering of thermal pads. Given the cost of reliability problems, finding a path to improvement is crucial.

Uncontrollable, Unpredictable Solder

The standard, manufacturer-recommended approach for heat sinking is to place large copper pads on both sides of the board under the chip. These pads are then connected with vias to conduct heat, and full solder paste coverage is used to get the best thermal contact. Unfortunately, this approach can cause problems.



The issue is that solder is unpredictable once melted, which puts every board at risk of a poor heat sink connection. This problem is so pervasive that you might be tempted to accept it as the unavoidable cost of doing business. However, there are ways to solve this challenge. There are two main mechanisms that cause these failures: solder wicking through vias and solder movement under large pads.

Solder Wicking Through Vias

If the landing pad is covered in tiny holes, molten solder can run through them to the back of the board. This results in less solder connecting the pad, with the remaining solder distributed unevenly. The wicking will vary from board to board, causing unpredictable consistency and poor reliability.

Common vias are not precision components, and the amount of copper plated onto them varies. Some may be wider than intended and others may be plated partially or fully shut. Identically designed boards that went through the same manufacturing process may now have significantly different thermal responses.

Solder Movement Under Large Pads

It is important to consider the unpredictability of solder movement during reflow. If the board is warped or there is not enough solder under the slug, capillary action can pull the solder to one side of the chip. Attempt to correct this by applying more solder, and the chip might float right off the signal pins. Excess solder can overflow the pad, sending solder balls out to short or bridge other areas of the board.

The varying amount of solder pulled through the vias only exacerbates the problem and hinders attempts to correct for it by adjusting the amount of paste used.

Finding the Solution

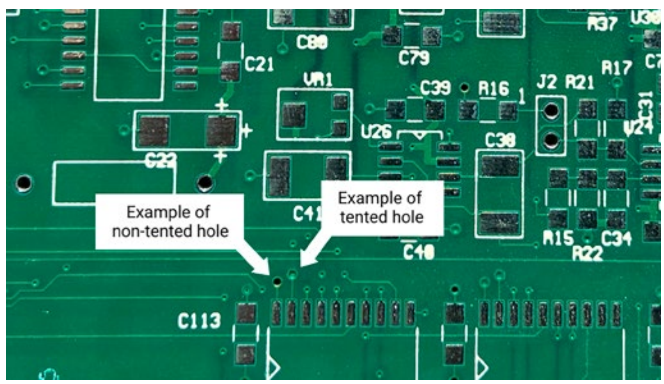
A proper solution requires fixing two different problems, but both are related to solder. First, solder must be prevented from wicking through vias and ending up on the wrong layers of a PCB. Second, solder must be prevented from moving past its area of application.

The solution can be approached with two different methods:

- First, apply solder mask over the landing pad and open circular "islands" for paste application. If the solder will not behave in a large area, break that area up into an array of smaller areas. Since solder mask restricts the paste to its area of application, this reduces the amount of solder connecting the chip to the board, increasing the consistency. The circular solder paste apertures release the solder more reliably than those with sharp corners, which helps prevent loose solder balls.
- Second, surround the "islands" with small (~12 mil or smaller) vias that are tented and covered with solder mask. Removing the vias from the immediate area being soldered and tenting them prevents any stray solder from wicking down to the other side of the board while still providing good thermal transfer to the pads underneath. Add these vias as close as possible to the islands. The solder mask tenting will block any solder that wicks onto an exposed via due to manufacturing tolerances.

When implementing this solution, keep in mind a few key rules:

- Make sure the pad under the chip is a solid copper plane, so it will spread out the heat.
- To give maximum coverage to the heat slug, use a hexagonal packing pattern for the solder islands.
- The solder mask tents should be the same size as the resist mask opening on these islands. This ensures that adequate solder is present to bond to the chip.
- Thermal spokes will reduce thermal conductivity, so don't use them on any layer of the vias.
- Vias should be solder mask tented, not plugged or filled. The epoxy in solder mask plugged vias may not cure fully and will tend to expand and erupt if subject to enough heat.



Implementing the Design

The most efficient way to implement this is to make it part of your package definition—making it uniformly applied to all similar parts. Unfortunately, many CAD packages do not support the direct addition of vias and copper planes to package designs. One workaround is to create square SMD pads to act as the top and bottom copper sinks, as well as small plated through-hole pads to act as the vias. These can then be configured as unconnected pins in your component definition. You may also have to create custom paste and solder masks for these pins.

If you are running into failures that may be caused by solder problems with thermal pads, try out this solution for your next project. We think you will be happy with the results. The benefits speak for themselves: fewer failures, a more consistent product, and happier customers. DESIGN007

Matt Stevenson is the VP of sales and marketing at Sunstone Circuits. To read past columns, click here.

Download *The Printed Circuit Designer's Guide to... Designing for Reality* by Matt Stevenson.

IDTechEx Reports on Five Key Trends for Tomorrow's Electric Car

The new IDTechEx report, "Electric Cars 2023-2043," provides a deep dive into future automotive markets and identifies five key trends:

Advanced Li-ion Battery Cells & Packs

Li-ion batteries based on graphite anodes and layered oxide cathodes have come to dominate large parts of the electric vehicle markets. However, as they start to reach their performance limits and as environmental and supply risks are highlighted, improvements and alternatives to Li-ion batteries become increasingly important.

Power Electronics

In automotive power electronics (inverters, onboard chargers, DC-DC converters), key advancements are being made to improve powertrain efficiency, allowing for either battery pack capacity reduction or improved range. One of the key avenues to achieving greater efficiencies is the transition to silicon carbide MOSFETs and high voltage vehicle platforms at or above 800V.

Electric Motors

Electric motor markets are still evolving today with new designs improving power and torque density. These are not just incremental improvements either, with developments such as axial flux motors and various OEMs eliminating rare-earths altogether.

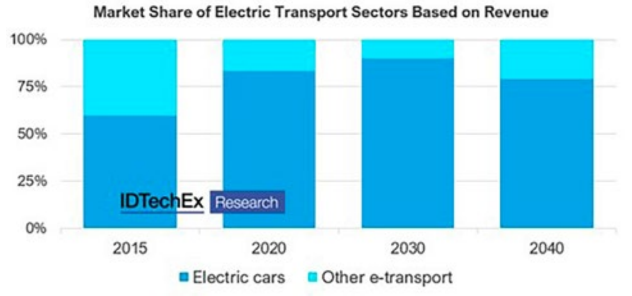
Fuel Cells

Opportunities for fuel cells in car markets are limited, although markets are still growing, underpinned by government support. The deployment of fuel cells within vehicles is not a new concept. Major OEMs have invested large sums over the past 30 years in advancing the technology.

Autonomy

"Autonomous vehicle" is an umbrella term for the six levels as defined by the SAE. Today, most new cars are arriving with the option of level 2 functionality, and the industry is technically ready for level 3 once regulatory hurdles clear.

(Source: PRNewswire)



Encapsulation Resins: PU vs. Epoxy

Sensible Design

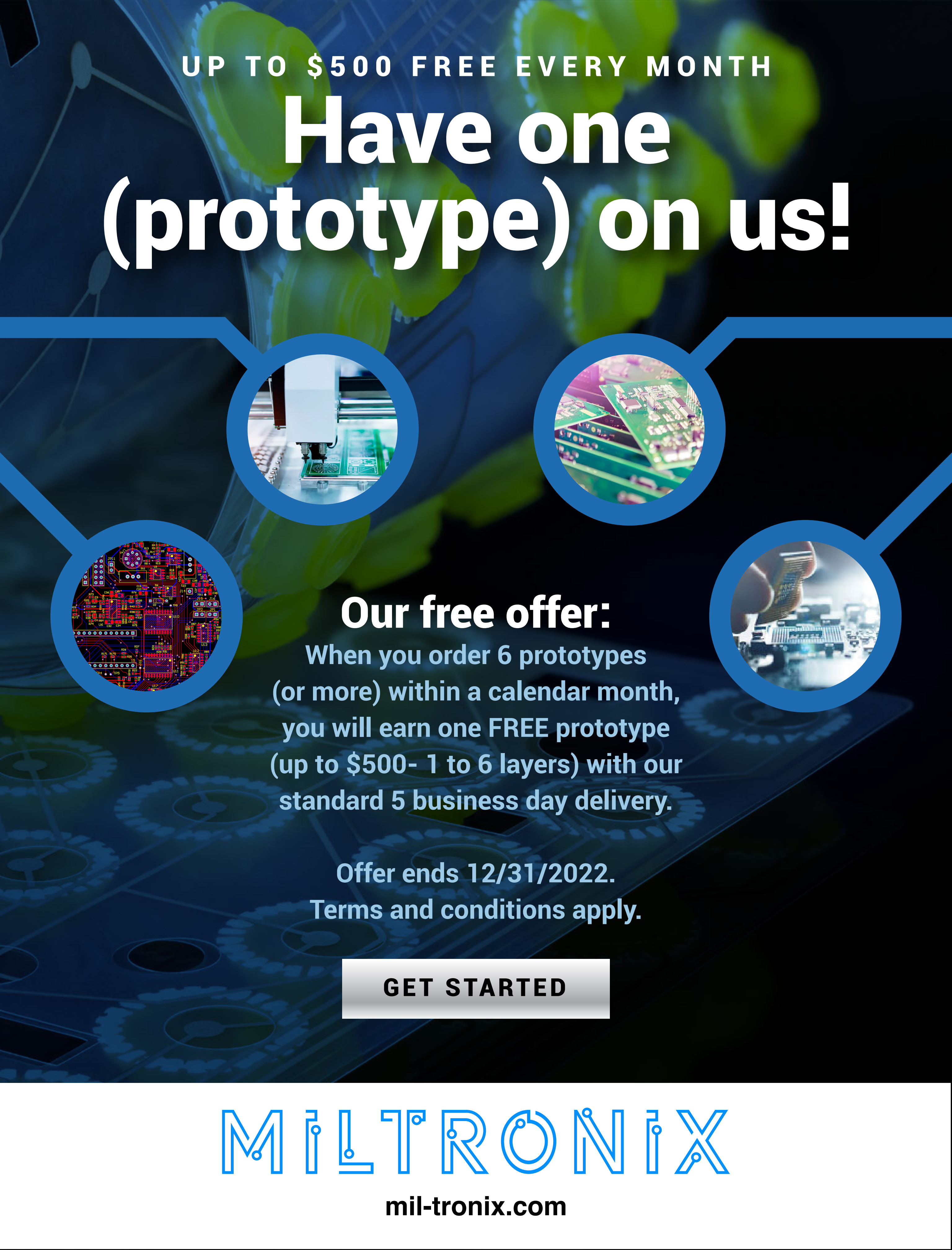
by Beth Massey, ELECTROLUBE

Most resin systems in use today are extremely complex products with elaborate chemical formulations. Process characteristics and final properties are usually adjusted by the manufacturer to suit the customer's requirements, and this necessitates a great deal of skill, knowledge, and experience. Yet resin technology is an often-overlooked area in electronics manufacturing. Over the past few years there have been several major developments in material formulation and utilisation that have resulted in extremely effective new products being brought to the market. Epoxy resins are the long-time favourites for potting and encapsulation, but with formulators rapidly reaching the limits of what can be achieved with these materials, they have turned to polyurethane chemistry. This month, I will look at polyurethane-based products that are fast becoming the resins of choice for those requiring high performance with cost-effective pricing.

What are the key differences between PU and epoxy encapsulation resins?

The main differences are flexibility and hardness; epoxies tend to be much more rigid and tough than polyurethanes, offering excellent sealing and ingress protection to the assemblies they are encapsulating, while polyurethanes are usually a little softer and more flexible, which can often make them more forgiving when considering stresses on delicate components during thermal cycling.

These differences can also influence the temperatures the resins are suitable for. For instance, epoxies are not typically suitable for temperatures below -40°C as they become too brittle, while polyurethanes can be suitable for temperatures as low as -60°C. At the top end of the temperature scale, polyurethanes are only suitable up to approximately 130°C—or 150°C in exceptional cases—whereas epoxies can be pushed higher, above 200°C.


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

Have you ever been convinced that an epoxy would be the best solution, but a PU resin won the day?

This isn't something we typically see when we work with a manufacturer from the outset. Usually, it is clear once we have collaborated with them to identify their key requirements and end-use parameters to assess which chemistry would be best. In fact, we are often able to narrow it down to one or two specific resins to take to the testing stage. However, it is something that we have seen in the past when a manufacturer has chosen a resin for themselves without any of our specialist input and find that their resin choice is not working well in the field.

By way of an example, an EMS company was potting remote vehicle unlocking units for an OEM and had self-selected an epoxy for their units. Once the units reached the thermal cycling stage of approval testing, it was found that many units were failing, and further investigation revealed that this was because components were being pulled off the board. What happened was the rigid, inflexible epoxy was adhering well to the assembly but expanding and contracting to a different degree than the board and components due to a CTE difference. This was causing so much stress that the solder joints were breaking. We recommended that they test one of our resins instead that is a tough but flexible polyurethane, and they found when they used this particular resin, they saw no failures during thermal cycling as this was able to flex better with the assembly during rapid temperature changes.

Do you have a most popular resin and why?

This is a tricky one, many of our resins are very popular, but if I had to highlight specific resins it would be our general purpose ER2188 and UR5604 because they are excellent "all-rounder" resins. ER2188 is a tough and rigid epoxy, while UR5604 is a more flexible polyurethane, although still tough enough for many environments. Both are flame retardant, to UL94 V-0 level, and have a good level of thermal conductivity to allow for heat dissipation away from hot spots in the assembly. Unlike many other resins with good thermal conductivity, they have excellent flow, allowing them to be used even in tightly spaced units that need the resin to flow into small gaps. Both resins offer excellent electrical performance and dielectric strength, and excellent general environmental protection. All these properties make them an excellent starting point for many applications.

In the early days of potting and encapsulation, epoxy resins were certainly the materials of choice. But with epoxy resin technology maturing over recent years and most of the exciting developments in resin technology now taking place in polyurethane chemistry, use is gradually switching across. Polyurethane resins are now becoming more dominant and taking market share from epoxies to an increasing extent. DESIGN007

Beth Massey is head of encapsulation resins at Electrolube. To read past columns from Electrolube, click here.

Additional content from Electrolube:

- *The Printed Circuit Assembler's Guide to... Conformal Coatings for Harsh Environments* by Phil Kinner
- "Coatings Uncoated!" a free micro webinar series
- You can also view other titles in our full I-007eBooks library

Analyzing Gerry Partida's 'Significant' Microvia Reliability Paper

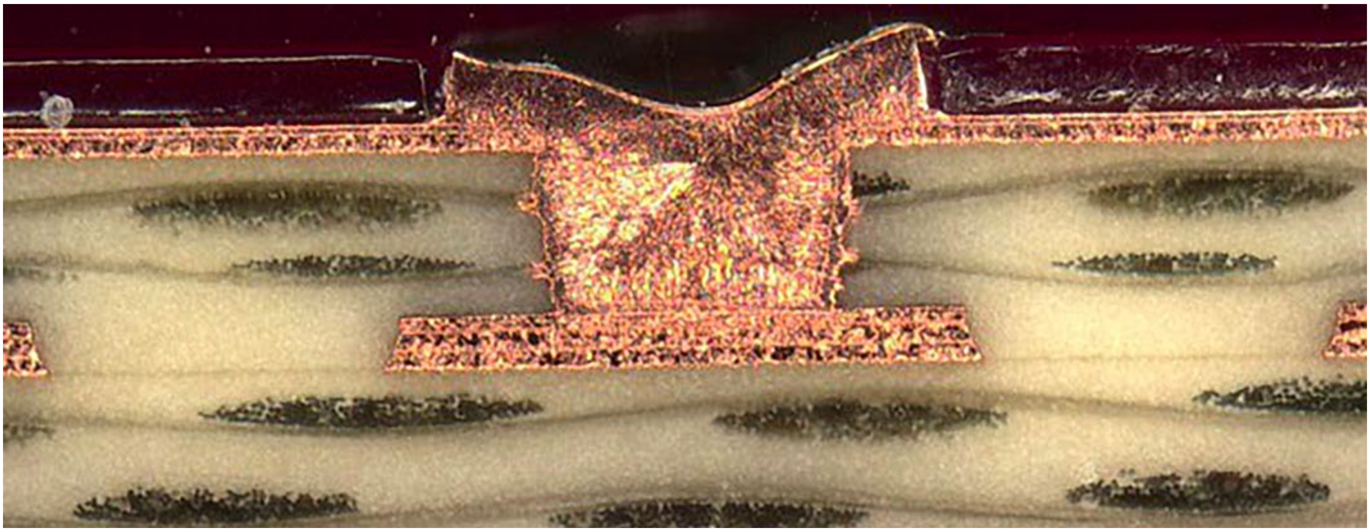
Article by Happy Holden

I-CONNECT007

Gerry Partida, vice president of technology at Summit Interconnect, authored a technical paper, "Next Progression in Microvia Reliability Validation—Reflow Simulation of a PCB Design Attributes and Material Structural Properties During the PCB Design Process," at IPC APEX EXPO 2022, and it's worth revisiting. This significant paper on microvia reliability validation provides a summary of what's been happening in the microvia fabrication arena, especially regarding the issue of latent defects in stacked microvias. This paper is available for download here.

Partida offers a thorough look at the controversy initiated in 1996 when latent electrical defects began showing up in field-deployed boards using microvias. My familiarity with this topic dates to 2000, when IPC organized round robin testing of microvia technologies. All looked good, and HDI technology took off. As evolution would have it, the technology grew and diversified. But all the new variants were not necessarily tested like the initial round in 2000.

As Gerry summarizes, "Today, the industry is facing a similar challenge with microvia reliability, especially after reflowing of the PCB at assembly, during rework, or operating in the field. As with the shortcomings of electrical testing in the past, the industry designed PCBs with microvias without evaluating the thermal properties of the material or the geometries in the design. Fabricators produced the finished goods and evaluated the finished PCB to established performance standards such as IPC-6012. When difficult-to-detect failures occurred post assembly, a test method IPC-TM-650 2.6.27 was established and a caution was added to the IPC- 6012 Rev E in Section 3.6, Structural Integrity. The testing of a D coupon via IPC-TM-650 2.6.27 did validate that the finished PCBs were safe for assembly, but it did not stop a fabricator from building a bad design."



Many studies have shown that the failures seem to be initiated at the interface of the microvia target pad and the plated microvia interface—a plain "butt joint." As the elevated temperature of lead-free assembly reflow is employed, the stress on this joint exceeds what it can accommodate, unlike tin-lead reflow that occurs at a lower temperature. What caught us unprepared is that after reflowing, the joint appears to be okay and will pass other performance testing, waiting until it fails in the field. This explanation seems to be validated by research that iMEC in Europe has conducted for the European Space Agency. Their finite element analysis of the microvia structure shows the maximum stress at this location and the stress increases as you stack the microvias and increase the temperature.

But the high point of the paper and presentation is a new way to do stackup validation and predictive pre-production simulation of reliability. Gerry explains that there has never been a method to simulate a PCB design validating that the material selection, dielectric thickness, microvia size, and configuration (single, stacked, or staggered microvias) could survive 6x reflows. But, as with the evolution of electrical test and the use of software to validate design and final test, we now have software that will validate the structural integrity of microvias during PCB stackup—before a design has been approved and placed into the fabrication process.

Gerry explains how Avishtech's Gauss Stack simulation software can provide the industry a way to validate and fabricate a microvia design with confidence and validate that the PCB has met the structural requirements by testing to IPC-TM-650 2.6.27. In this paper, he demonstrates real cases where this software has identified structural issues with a microvia design and how this software can provide modification of the PCB design geometries that will result in a working stackup.

Gerry walks us through how the software considers the Z-axis CTE of laminate materials. He provides examples of stackups that appear to be fine but would fail in lead-free reflow or soon after. Gerry contends, "With predictive engineering and reliability simulation, it is now possible to screen out an unreliable stackup and change to a material type that allows design geometries to pass the required reflow test requirements. It is also possible, through simulation, to determine that the microvia design must be modified by increasing the microvia diameter, staggering the microvias or changing the prepreg selection."

Summary

To conclude his paper, Gerry reiterates some of the advantages of utilizing reliability simulation at the beginning of the design cycle, including designing in reliability, reducing the number of redesigns and marginal designs, and creating design attributes that are "based on science and not tribal knowledge."

If you're a designer or fabricator dealing with microvia failures and reliability, you're in luck. Gerry will be back at IPC APEX EXPO 2023 to present a new paper, "Microvia Reliability Stacking and Staggering for a Successful Design," during the Technical Conference.

Gerry's abstract proposes to demonstrate how it is possible to use three or four stacked microvias, provided that the design considers the reliability influences of material expansion, pitch, microvia diameter, reflow temperature, and length of the stacked microvias. Stay tuned. I'll see you in January. **DESIGN007**

Happy Holden has worked in printed circuit technology since 1970 with Hewlett-Packard, NanYa Westwood, Merix, Foxconn, and Gentex. He is currently a contributing technical editor and columnist with I-Connect007, and the author of *Automation and Advanced Procedures in PCB Fabrication,* and *24 Essential Skills for Engineers.* To read past columns, click here.

What More Do We Need to Know?

Tim's Takeaways

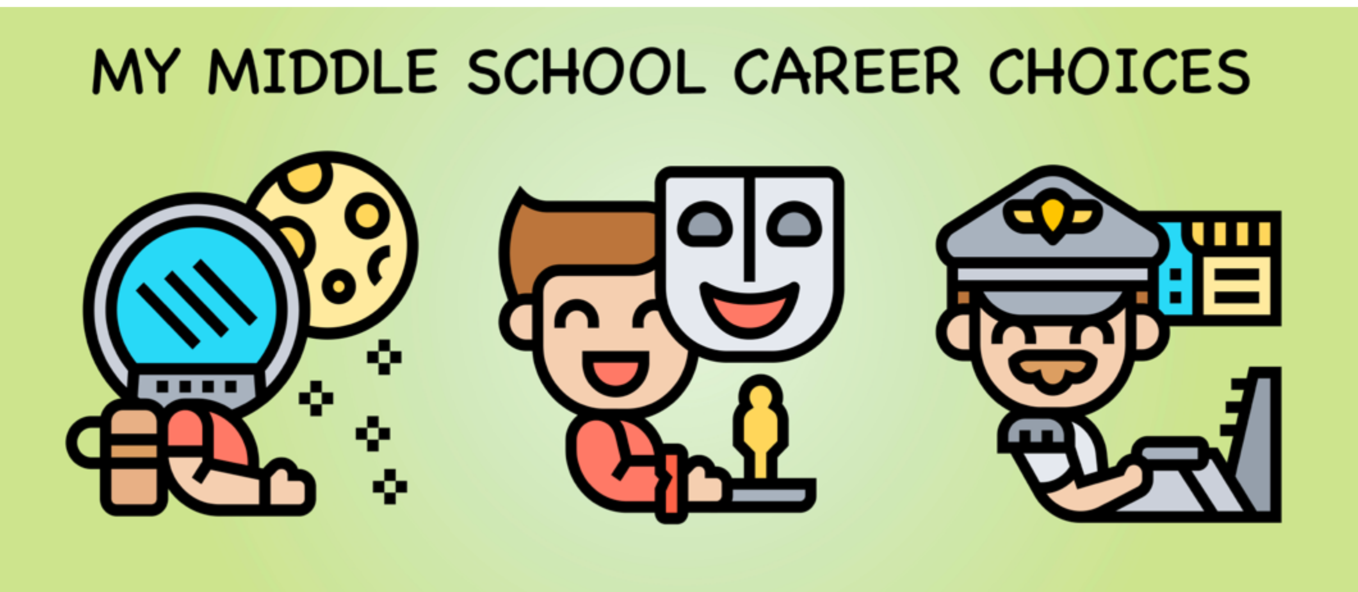
by Tim Haag, FIRST PAGE SAGE

Although it's been more years than I care to admit, I still remember very clearly the class on careers I was required to take in junior high school. On a table in the front of the class were several boxes filled with all sorts of different job cards that a student would search through to learn about different professions. Each card listed the schooling and experience required for that particular job, its expected responsibilities and duties, and an estimated salary. Our assignment was to choose a handful of these cards that interested us, study them, and then list the reasons why we would or wouldn't pursue those jobs as a career.

As you might have guessed, I blew the assignment.

To be fair, the concept of preparing for a career simply wasn't real for me at that point in my life, and I certainly didn't have any goals in mind for my future. Looking back on it now, I realize that this class was intended for students like me who needed to discover what the future held for them, but I missed its intentions entirely. Instead, my focus in those days was band class, biking all over town with my friends, and girls—and not necessarily in that order. So, if I remember correctly, to pull at least a passing grade I gave the careers assignment a half-hearted attempt and pulled cards for an airline pilot, astronaut, and an actor. Things that sounded fun but were not very realistic.

I really wish now that I had pulled the job cards for a profession in design, electronics, or engineering. If so, then perhaps I might have chosen a path to PCB design much earlier than I did. However, I would be willing to bet that if I could go back in time and look through those



boxes once again, that I would never find a specific card on PCB layout and design. Even though most people today are familiar with a circuit board, it is rare to find anyone outside our industry who understands their complex process of conception, creation, and manufacturing. And (way) back when I was in school, that entire process would have been even more of a mystery.

When I eventually started laying out circuit boards, many of the designers at the time came from a wide variety of career paths within our industry. Some came with a background in electrical engineering, while others started out as technicians, graphic artists, and even one goofy kid who started out in a PCB photography lab. (That was me in case you didn't realize it). PCB layout CAD systems were just coming into their own, which gave me an advantage due to my comfort level with computers.

At that time, many designers were still working on drafting boards, or taping out their layouts on a light table. Eventually those methods were completely replaced by CAD systems, which have had their own rapid growth as evolving design requirements require continual enhancements and upgrades. Now, except for legacy designs that are still built from film stored in a vault, tape-up designing is relegated to the hobbyist. But design tools and methods aren't the only aspects of our industry which have changed over the years.

It has become essential for PCB designers to have a solid foundation of education and training to be successful in the industry. Entry-level designers today typically have a much better understanding of electronics than what we had when I first started, and those requirements are always growing.

It isn't enough anymore to simply understand how to arrange the circuitry so that it works; it is also important to understand why that circuitry works the way that it does, along with the many conditions that can change that. In this edition of *Design007 Magazine,* you will see a lot of discussion about the need for designers to not only understand electronics, but also the underlying physics that is involved in the performance of circuitry. This issue would make for some interesting reading and researching for a middle-scholar studying a career card.

There is a lot to be said about the physics of electronic circuitry, but I'm going to leave that discussion to the experts in this magazine. However, it raises a question: How many other disciplines should layout designers explore to enhance their skill sets for success in our industry? Here are some areas that come to mind:

- Fabrication
- Assembly
- Component engineering
- Supply chain
- Marketing

Fabrication

Most of us who lay out circuit boards probably have a good understanding of how a PCB is fabricated. But as with any subject, there is always more that can be learned to help design a better board. Do you understand the capabilities and limitations of your fabricator(s)? Do you understand the nature and characteristics of the materials being used and how these will affect your design, especially in high-speed applications? Is it better to increase the layer count of the board for greater signal integrity, or reduce the count to minimize the cost? Understanding these various points may well end up making the difference between success or failure in your design.

Assembly

As with fabrication, there is a lot more to understand about PCB assembly than simply maintaining the correct distance between components. Most designers have a firm grasp of the basics of design for manufacturability (DFM), but do you go deeper into these subjects by applying the correct DFT, DFA, DFF, and other DFX rules? Do you know that the DFM rules will change depending on what soldering method is used to build the board? Have you placed your parts to also allow for easy access of cables, switches, and other human interfacing parts during system assembly? Many boards are laid out with adequate part spacing but fail because a connector was rotated incorrectly for its mating cable.

Component Engineering

When you begin a new circuit board layout, do you jump right in, or do you verify that the parts being used are correct? There are many factors that can affect a part, including its availability and cost, to say nothing of its electrical performance. Often a design will come through with parts that are out of date because the schematic was built using copied sheets from an older design. PCB designers need to check for all these potential problems and understand how to resolve any that they find.

Supply Chain

Speaking of component price and availability, do you keep your finger on the pulse of what parts should or shouldn't be used on a board? A circuit board job can come to a screeching stop during assembly because the parts they rely on are no longer available. PCB designers should at least have resources, like their manufacturing partners, that they can use to verify the parts before they finalize the design.

Marketing

Do you know the purpose of your design, where it will be used, and how? These details can impact the operation of the board which could change how it should be designed. Also, simply understanding the product's schedule and requirements could help you to plan for and hit design benchmarks that will help marketing to be more successful.

But Wait, There's More

These are only a small sampling of the topics that today's designers should include in their repertoire of knowledge, and there are many more that we haven't even touched on. For instance, a designer should be fluent in the CAD tools being used and have a solid understanding of how they interreact with other systems and third-party tools. This can greatly enhance your design productivity and ultimately the success of the project you are working on.

All these additional topics may seem like a lot for a designer to grasp, but one of the defining characteristics of our industry is the need to continually grow in our knowledge and skills. PCB designers must stay on the forefront of new ideas and technology, and in that regard, I believe the ability of designers to learn and adapt is no different now than when I first started.

Sure, the specific knowledge base and skill sets required now are much more advanced than they were 40 years ago. After all, we were just coming up to speed on PCB CAD systems and surface mount components. Today's concerns about the physics that affect high-speed design performance were not ideas that most designers would have even considered let alone designed for. But just as we triumphed over the new requirements confronting us back then, it is my firm belief and expectation that today's designers will expertly adapt and grow with the new responsibilities facing them.

It is a genuine pleasure to see the changes that are spreading throughout our industry, and the growth that is occurring. It really is exciting to see where the next great steps in our industry will take us. Until next time, keep on designing. **DESIGN007**

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

Flex007 Highlights

The Chip Shortage Leads to Innovation ▸

The chip shortage is by no means over, with estimates expecting it will last into 2023. Some could see it taking even longer, such as Intel CEO Pat Gelsinger, who expects it to see shortages into 2024 due to those now impacting electronics production equipment. But if there's any bright spot to be had, it's that a crisis often leads to long-term solutions.

Schweizer Electronic Inks Patent Transfer Agreement with China Subsidiary ▸

Schweizer Electronic AG and its subsidiary Schweizer Electronic (Jiangsu) Co. Ltd. based in Jiangsu, China (SEC) concluded a Contribution and Patent Transfer Agreement on 10 October 2022.

IBIDEN Group Switches to 100% Renewables in Southeast Asia ▸

IBIDEN Co., Ltd. is pleased to announce that in August 2022 its production subsidiary, IBIDEN Philippines, Inc., converted its electricity consumption to renewable energy sources.

ILFA Invests in the Expansion of its PCB Lamination Capacities ▸

During the LAUFFER Technology Days, ILFA Managing Director Thomas Michels and LAUFFER Managing Director Christof Lauffer announce a close cooperation, which is sealed with a joint letter of intent.

Zhen Ding Posts 31% Revenue Growth in September ▸

Zhen Ding Technology Holding Ltd, a Taiwan-based company primarily engaged in the design, development, and manufacturing of printed circuit boards, has posted sales of NT$20.27 billion ($634 million at $1:NT$31.98) for September 2022, up by 31% year-on-year (YoY) and by 22.6% from the previous month.

Ventec Expands U.S. Team with New Director OEM Sales & Business Development ▸

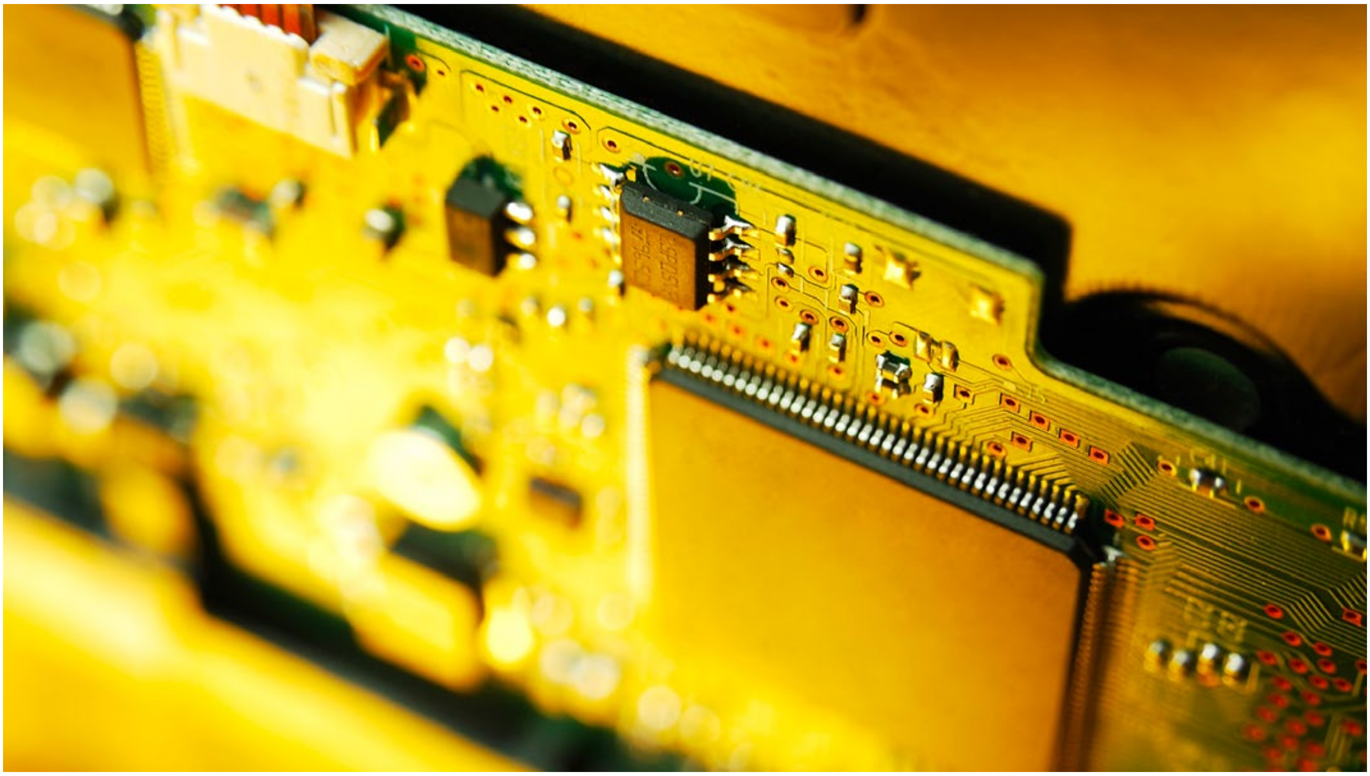
Ventec International Group Co., Ltd. is pleased to announce that Chad Wood has joined the company's US team. As director OEM Sales and Business Development, Chad will drive business development, sales and support services for OEM customers predominantly in the Western region of the USA.

DuPont, Rogers Provide Update on Pending Merger ▸

DuPont de Nemours, Inc., and Rogers Corporation provided the following update on regulatory approval of the pending merger of Rogers and Cardinalis Merger Sub, Inc., a wholly owned subsidiary of DuPont, which was previously expected to close in the third quarter of 2022.

Market and Tech Convergence: Electrically Conductive Inks ▸

It's no secret that electronics products are being asked to do more, pushing the boundaries of technological capabilities. Next-gen consumer products like flexible wearables and foldable mobile devices are proving themselves to be more than a fad, with start-ups and major companies alike booming on the market for the past few years.



DownStream Flexes in Rigid-Flex

Interview by Andy Shaughnessy
I-CONNECT007

During PCB West, I caught up with DownStream Technologies co-founder Joe Clark and Senior Product Marketing Manager Mark Gallant. We discussed some of their latest tool updates, including a greater focus on bringing post-processing functionality, such as interlayer analysis capability, to rigid-flex circuits. Joe also offered a look at global design trends going into 2023, as more engineers take on PCB designer roles while senior designers are retiring.

Andy Shaughnessy: Joe, give us an update on what's been going on with DownStream for the last year.

Joe Clark: Last year was a record year for us, where we experienced over 10% growth even in the middle of COVID. We're on track to repeat that again in 2022. Much of that is driven by our major customers adopting design-for-manufacturing and intelligent design formats. A lot of the analysis that used to be done at the end of the design process has been moving "upstream," and that has implications on the tool capabilities, i.e., it's got to be easy to use and have low cost of ownership. It's hard to believe that we've been doing this for 20 years.

Shaughnessy: Who knew 2002 was a great time to start a company?

Clark: It was actually the worst time to start a company. But our vision was that design-for-manufacturing analysis would become a mainstream mission-critical requirement, and that the analysis would naturally move upstream. If you remember back when we were PADS,

Joe Clark

we acquired HyperLynx because our belief was that designers should be doing signal integrity checks during the design creation cycle, not at the end; we were right. We then spun out of Innoveda, took the CAM350 product with us, and added DFM analysis capabilities along with support for intelligent formats (ODB++, IPC-2581) to support in-cycle DFM analysis. We were right again.

Also, as we worked with customers, it became clear to us that another mission-critical part of the process not being addressed was the designers' need to create detailed documentation for their PCB designs. At the time, they all used their CAD tools, but they were not created to do documentation. PCB documentation is not a "nice to do" thing; it is a necessary thing to do as part of a good new-product design process. Working with customers, and again taking advantage of the shift to neutral intelligent design formats, we developed BluePrint to address the needs for creating quality, adaptable PCB documentation.

Shaughnessy: Most designers love doing layout, but they hate post-processing and documentation creation.

Clark: They hated doing it, especially documentation, in large part due to the nature of the existing tools. They were simply too difficult to use in the design cycle. We focused on providing tools of high functionality combined with ease of use. For PCB documentation, we developed a tool to automate it, to leverage the intelligence of the design to allow users to create, in a very automated way, comprehensive PCB documentation to their particular requirements. Believe me when I say that PCB doc requirements vary greatly company to company and often even division to division. Our BluePrint tool is fully customizable.

Shaughnessy: What is your focus as we head into 2023?

Clark: There are new trends coming, and our most recent focus is on rigid-flex. Working with companies like Microsoft, we realized that there are really no tools to address a host of new challenges: multiple stackup regions, the potential problems from bending a circuit, inter-layer dependencies, and other such technological obstacles.

Working with our customers helps us to define and develop new capabilities that support new technologies like rigid-flex designs. It is a great example of a disruptive technology. It requires a whole rethink of how we document, present, bring the data in, interface with manufacturing, and what's required for doing true design-for-manufacturing analysis.

Mark Gallant: We've had customers approach us with their design challenges in the rigid-flex design world. They needed a way to analyze their designs and weren't finding a lot of solutions—even basic trace fracture analysis like detecting vias in a bend area. They didn't have and could not find tools that provided such capability. Also, in rigid-flex, the interlayer—the interstitial of layers—needs analysis capabilities too. For example, any rigid-flex design requires a metallization layer to shield the rigid-flex circuitry. Well, that metallization layer must make contact with the trace layer below it. You must create exposures on the solder mask between them to allow contact with ground on the trace layers.

Through working with one of our customers, we developed a very complex set of analyses that allows them to do layer-to-layer analysis in rigid-flex. This is where some of the weaknesses in other tools are. If you put a metallization layer on top of a solder mask on top of a trace layer, and you have to bond them, you need to have mask openings and make sure those openings are the right size to prevent glue squeeze-out. That's an example of where we've been focused in phase one.

Shaughnessy: We haven't seen much written about rigid-flex interlayer analysis.

Gallant: It's really helped a lot of our customers that are doing complex flex. But again, the rigidized flex, where they have a piece of FR-4 that exists on top of or in combination with other layer materials, requires a layer-to-layer specific analysis. These are examples of the complex challenges that you won't encounter in a traditional trace analysis DFM.

I call this phase one because we're working to expand our analysis to create even more efficient tools to analyze rigid-flex designs. As customers work with our tools, we're fortunate that they're giving us the feedback to help us create the next generation of DFM tools for rigid-flex.

Shaughnessy: You talked about phase one. What is phase two?

Gallant: There's much more that we can do for rigid-flex. We already provide analysis for vias, bends, and trace corners in bend areas. We currently analyze for other potential fracture situations such as traces with corners or width transitions in a bend area. There's the I-beaming effect—having two layers with parallel traces, one on top of the other. In rigid-flex, that's a fracture potential. Rigid-flex designers need to be able to detect the condition where you have an I-beam and you need to separate traces layer to layer. That's just one example of how we can expand our analysis. As customers start to implement our tools, we're betting they'll come back looking for further enhancements to our analysis capabilities.

Mark Gallant

Shaughnessy: It sounds like you're becoming an analysis tool company.

Clark: Don't tell anyone. It's been part of our growth and responding to the needs of our customers.

Shaughnessy: What other trends do you see in PCB design right now?

Clark: One big challenge that many companies have is their traditional PCB designers are retiring, and companies are asking, "What do we do now"? We see that the trend is for more and more of the EEs, the upstream engineers, to be doing more analysis and layout and we believe this trend will continue and even accelerate. This is leading us to create easy-to-use cloud-based tools. We're looking at not just up-porting our tools to a cloud environment, but rethinking our tools, and making them available as a web-based application. There's a lot happening not only technologically, but also economically, as these printed circuit board experts retire and move on. This trend is driving what we're working on.

Shaughnessy: What do you see at a macro level? We're officially in a recession, but some managers are saying, "If this is a recession, I'll take it."

Clark: That's a very good question. With trends, you do your best analysis of what you're seeing and what you're hearing. I'm the finance guy, and I assumed that with COVID we would all see a slowdown in business. I thought demand would drop off, and we'd see some budgets re-evaluated and some dip in the business. But it never happened.

2020 and 2021 were record years for us, and 2022 is on record again for growth of over 10%. In 2023, we don't know what will happen. We were developing a very healthy business in Russia, for example, but that business is pretty much gone because of the circumstances there. Overall, all aspects of our products and our territories have seen growth, especially in the U.S. and China. Sometimes it seems that everything I learned in business school is out the window.

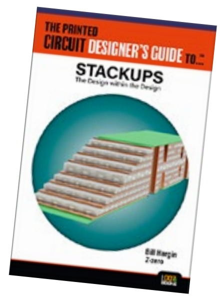
Shaughnessy: I think a lot of other finance guys in this industry would agree with you. Great talking with you, Joe. Good to see you, Mark.

Clark: Always a pleasure, Andy. DESIGN007

BOOK EXCERPT

The Printed Circuit Designer's Guide to... Stackups: The Design within the Design

by Bill Hargin, Z-zero



Chapter 3: Cutting Your Losses

In this chapter, we'll briefly discuss signal loss as it relates to your stackup and a process you can use to select the best laminate for your application while avoiding under- or over-designing.

For today's faster rise times in high-speed PCB applications, the lossy effects of transmission lines dramatically influence signal quality. As Figure 3.1 shows, these effects—both attenuation and rise-time degradation—vary directly with length at higher frequencies.

Two important mechanisms that absorb energy from the signal need to be considered: dielectric loss and conductor loss caused by the "skin effect" and copper roughness.

Dielectric Loss

Dielectric loss increases with frequency, and with a material's loss tangent or dissipation factor (Df). Standard FR-4 materials are considered to be in the high-loss category, with loss tangents ranging from 0.02–0.03. For a higher price, lower loss materials are available, with Df values for ultra-low loss and "extremely-low loss" materials currently trending below 0.0025 for several high-end laminate.

Continue reading.

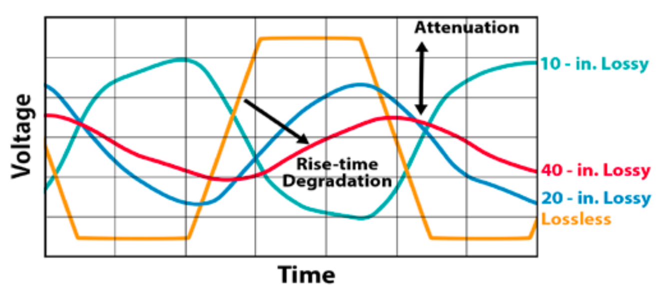
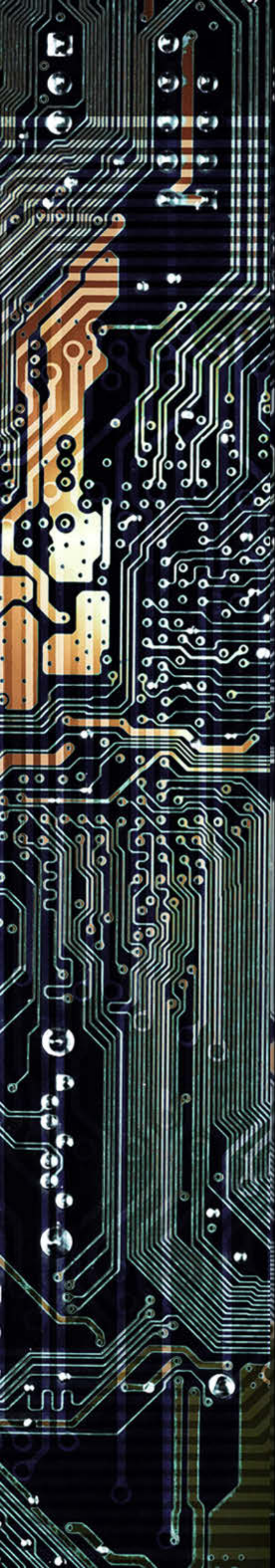


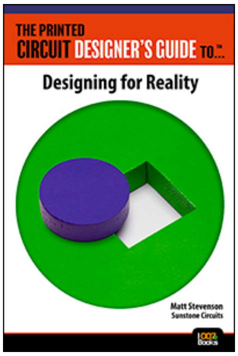
Figure 3.1: Rise-time degradation and attenuation (loss)for a lossless interconnect (orange), as well as lines of 10–40 in. through a lossy dielectric.



TOP TEN

EDITOR'S PICKS

Sunstone and I-007eBooks Launch Book on Designing for Reality



I-007eBooks is excited to announce the release of the latest title in its series for designers, *The Printed Circuit Designer's Guide to... Designing for Reality.* This book covers both written and unwritten rules for how to create a realistic, manufacturable design.

Fresh PCB Concepts: How Large Does That Pad Really Need to Be?


Ryan Miller

Annular ring is the amount of copper left and surrounding the hole after processing. This measurement is taken from the edge of the hole to the edge of the land at the thinnest part. To ensure a robust design, we must design efficient annular ring for PCB interconnections.

EMA Helps Ease Designers' Supply Chain Woes


Chris Banton

Supply chain issues are continuing to cause disruptions in our industry, though lead times have dropped from astronomical to merely troublesome. In this interview, Chris Banton, EMA Design Automation's director of marketing, explains how Cadence's software has evolved as designers' needs have changed in the past few turbulent years.

Some Relief, But Hold Off on the Party

To help PCB designers and design engineers get a clearer picture of the stress points in the industry, particularly from a company that deals directly with EMS providers, we reached out to CalcuQuote CEO Chintan Sutaria with a list of questions. The following Q&A explores trends in the PCB supply chain and heady advice for dealing with long lead times and counterfeit parts.

Dana on Data: My Holiday Season Data Wishlist

Every year children often start creating their holiday gift list before Halloween. So, I thought that it would be a great idea to provide my holiday present request list to the PCB industry this month. My fundamental wish is simple: I wish to make it easier for designers to output designs and for PCB fabricator front-engineering teams to spend less time reviewing data so they may release production tooling into their factories faster.

Elementary, Mr. Watson: Is the Tail Wagging the Dog?

I recently had the opportunity to work on a rather critical PCB design project during what should have been the final design review.

Tim's Takeaways: Success Begins With a Little Confidence


Bruno Gaido, Family Photo

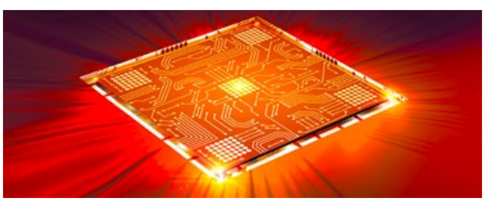
Bruno Gaido was a young radioman-gunner portrayed by Nick Jonas in the 2019 movie "Midway." An early scene shows a Japanese bomber trying to sink the USS Enterprise by crashing into it with his plane.

Book Excerpt: 'An Introduction to *The Printed Circuit Designer's Guide to... Stackups*'

To give readers a sample of *The Printed Circuit Designer's Guide to... Stackups—The Design within the Design,* by Bill Hargin, we are providing the book's introduction.

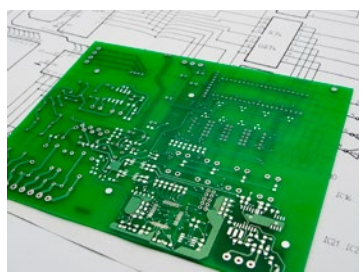
Q&A: The Learning Curve for Ultra HDI

For this issue on ultra HDI, we reached out to Tara Dunn at Averatek with some specific questions about how she defines UDHI, more about the company's patented semi-additive process, and what really sets ultra HDI apart from everything else.



Connect the Dots: The ABCs of Clean Schematics

The production team is always excited when the first shipment of boards for a new electronic device comes back from the PCB manufacturer. Anticipation builds as the engineer connects the first set of components, puts everything together, and gets ready for that first test.



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

Career Opportunities

Find industry-experienced candidates at jobConnect007.

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

all4-PCB®
WORLDWIDE

Technical Sales Representative

We are an established distributor that represents manufacturing equipment and specialty consumables for the PCB manufacturing industry as well as other markets. All4-PCB represents products from suppliers in both Asia and Europe.

The objective of the position is to maintain and further develop the manufacturing consumable product business in the PCB industry. Excellent and well-organized communication flow between our principles and the customer base is required. We are looking for a dynamic, results-orientated sales personality with a technical background, capable of understanding the technical applications of the products.

A generous commission structure is available on top of solid base salary.

Responsibilities

- Grow existing accounts by maintaining relationships with clients
- Manage operation of accounts through responding to customers, forecasting, inventory management
- Generate new leads and tackle existing leads to contribute to business growth
- Attend trade shows and relevant conferences
- Supporting sales network in North America. Travel is required.

Qualifications

- A technical background in chemistry or engineering is beneficial. Min. 2-year degree.
- Proficient in Microsoft Office
- Strong organizational, communication and analytical skills
- Strong understanding of full sales process
- Experience utilizing customer relationship management software
- US citizenship or green card is needed and a valid driver's license

Apply to: Torsten.Reckert@all4-pcb.us.

apply now

FLEXIBLE CIRCUIT Technologies

Application Engineer

Flexible Circuit Technologies (FCT) is a global supplier providing design, prototyping and production of flexible circuits, rigid flex circuits, flexible heaters and full assembly services.

Responsibilities

- Gain understanding for customer/specific project requirements
- Review customer files, analyze - application, design, stack up, materials, mechanical requirements; develop cost-effective design to meet requirements
- Quote and follow-up to secure business
- Work with CAD: finalize files, attain customer approval prior to build
- Track timeline/provide customers with updates
- Follow up on prototype, assist with design changes (if needed), and push forward to production
- Work as the lead technician/program manager or as part of FCT team working with an assigned application engineer
- Help customer understand FCT's assembly, testing, and box build services
- Understand manufacturing and build process for flexible and rigid-flex circuits

Qualifications

- Demonstrated experience: flex circuit/rigid-flex design including design rules, IPC; flex heater design +
- Ability to work in fast-paced environment, broad range of projects, maintain sense of urgency
- Ability to work as a team player
- Excellent written and verbal communication skills
- Willing to travel for sales support and customer support activities if needed

Competitive salary, bonus program, and benefits package. Preferred location Minneapolis, MN area.

apply now

Career Opportunities

Career Opportunities

Career Opportunities

INDIUM CORPORATION®

Supplier Quality Manager

Headquarters, New Hartford, NY

JOB SUMMARY:
The Supplier Quality Manager is responsible for maintaining and improving the quality of Indium Corporation's supplier base as well as compliance with identified quality standards and risk mitigation. This position will work cross-functionally with Supply Chain, Operations, and our suppliers. The role will ensure that the quality levels of all Indium Corporation suppliers and products meet customer requirements while supporting the company's growth, vision, and values.

REQUIREMENTS:

- Bachelor's degree in business, supply chain or a science-based discipline
- Minimum 3 years in a supply chain role supporting or leading supplier quality
- Obtain and/or maintain International Automotive Task Force (IATF) auditor certification within first 3 months of employment
- Able to work independently or lead a team, as needed, to meet goals
- Excellent oral and written communication skills
- Knowledge of quality standards
- Proficiency in MS Office

apply now

KOH YOUNG AMERICA

Technical Service & Applications Engineer

Full-Time — Midwest (WI, IL, MI)

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 is expanding the team with an Applications Engineer to provide helpdesk support by delivering guidance on operation, maintenance, and programming remotely or on-site.

Responsibilities

- Provide support, preventive and corrective maintenance, process audits, and related services
- 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
- 75% domestic and international travel (valid U.S. or Canadian passport, required)
- Able to work effectively and independently with minimal supervision
- Able to readily understand and interpret detailed documents, drawings, 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

KYZEN
Where Science and Care Converge.

Regional Manager

Midwest Region

General Summary: Manages sales of the company's products and services, Electronics and Industrial, within the States of KS, MO, NE, and AR. 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

pluritec®
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

Manncorp™

SMT Field Technician
Hatboro, PA

Manncorp, a leader in the electronics assembly industry, is looking for an additional SMT Field Technician to join our existing East Coast team and install and support our wide array of SMT equipment.

Duties and Responsibilities:
- Manage on-site equipment installation and customer training
- Provide post-installation service and support, including troubleshooting and diagnosing technical problems by phone, email, or on-site visit
- Assist with demonstrations of equipment to potential customers
- Build and maintain positive relationships with customers
- Participate in the ongoing development and improvement of both our machines and the customer experience we offer

Requirements and Qualifications:
- Prior experience with SMT equipment, or equivalent technical degree
- Proven strong mechanical and electrical troubleshooting skills
- Proficiency in reading and verifying electrical, pneumatic, and mechanical schematics/drawings
- Travel and overnight stays
- Ability to arrange and schedule service trips

We Offer:
- Health and dental insurance
- Retirement fund matching
- Continuing training as the industry develops

apply now

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

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

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

Career Opportunities

Career Opportunities

Career Opportunities

Career Opportunities

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

Educational Resources

I-007eBooks The Printed Circuit Designer's Guide to...

NEW BOOK!

Designing for Reality by Matt Stevenson, Sunstone Circuits

Based on the wisdom of 50 years of PCB manufacturing at Sunstone Circuits, this book is a must-have reference for designers seeking to understand the PCB manufacturing process as it relates to their design. Designing for manufacturability requires understanding the production process fundamentals and factors within the process that often lead to variations in manufacturability, reliability, and cost of the board. Speaking of making better decisions, **read it now!**

Thermal Management with Insulated Metal Substrates, Vol. 2

by Didier Mauve and Robert Art, Ventec International Group

This book 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. **Add this essential book to your library.**

High Performance Materials

by Michael Gay, Isola

This book provides the reader with a clearer picture of what to know when selecting which material is most desirable for their upcoming products and a solid base for making material selection decisions. **Get your copy now!**

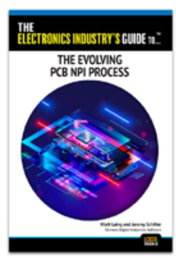
Stackups: The Design within the Design

by Bill Hargin, Z-zero

Finally, a book about stackups! From material selection and understanding laminate datasheets, to impedance planning, glass weave skew and rigid-flex materials, topic expert Bill Hargin has written a unique book on PCB stackups. **Get yours now!**

THE ELECTRONICS INDUSTRY'S GUIDE TO... The Evolving PCB NPI Process

by Mark Laing and Jeremy Schitter, Siemens Digital Industries Software



The authors of this book take a look at how market changes in the past 15 years, coupled with the current slowdown of production and delivery of materials and components, has affected the process for new product introduction (NPI) in the global marketplace. As a result, companies may need to adapt and take a new direction to navigate and thrive in an uncertain and rapidly evolving future. Learn how to streamline the NPI process and better manage the supply chain. ***Get it Now!***

Our library is open 24/7/365. Visit us at: I-007eBooks.com

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: **MIKE RADOGNA**

COVER IMAGE: **ISTOCK PHOTO © CSA PRINTSTOCK**

DESIGN007
MAGAZINE

DESIGN007 MAGAZINE®
is published by IPC Publishing Group, Inc.
3000 Lakeside Dr., Suite 105N, Bannockburn, IL 60015

November 2022, Volume 11, Number 11
DESIGN007 MAGAZINE is published monthly
by IPC Publishing Group, Inc., dba I-Connect007

ADVERTISER INDEX

Problems solved!
PCB007 MAGAZINE
AUGUST 2022
PCB007 MAGAZINE
SEPTEMBER 2022
A NEW ERA IN MANUFACTURING
PCB007 MAGAZINE
OCTOBER 2022
FOCUS ON IMAGING
Cybersecurity & Counterfeiting
DESIGN007 MAGAZINE
SEPTEMBER 2022
DESIGN007 MAGAZINE
OCTOBER 2022
TO UHDI & BEYOND!
DESIGN007 MAGAZINE
NOVEMBER 2022
MAXWELL VS. MOORE
SMT007 MAGAZINE
OCTOBER 2022
SMTAI 2022 PASSPORT
SMTA International
MD&M
EV Keynote
Young Professionals
New Technology
Technical Papers
SMT007 MAGAZINE
NOVEMBER 2022
Workflow Management
SUBSCRIBE NOW