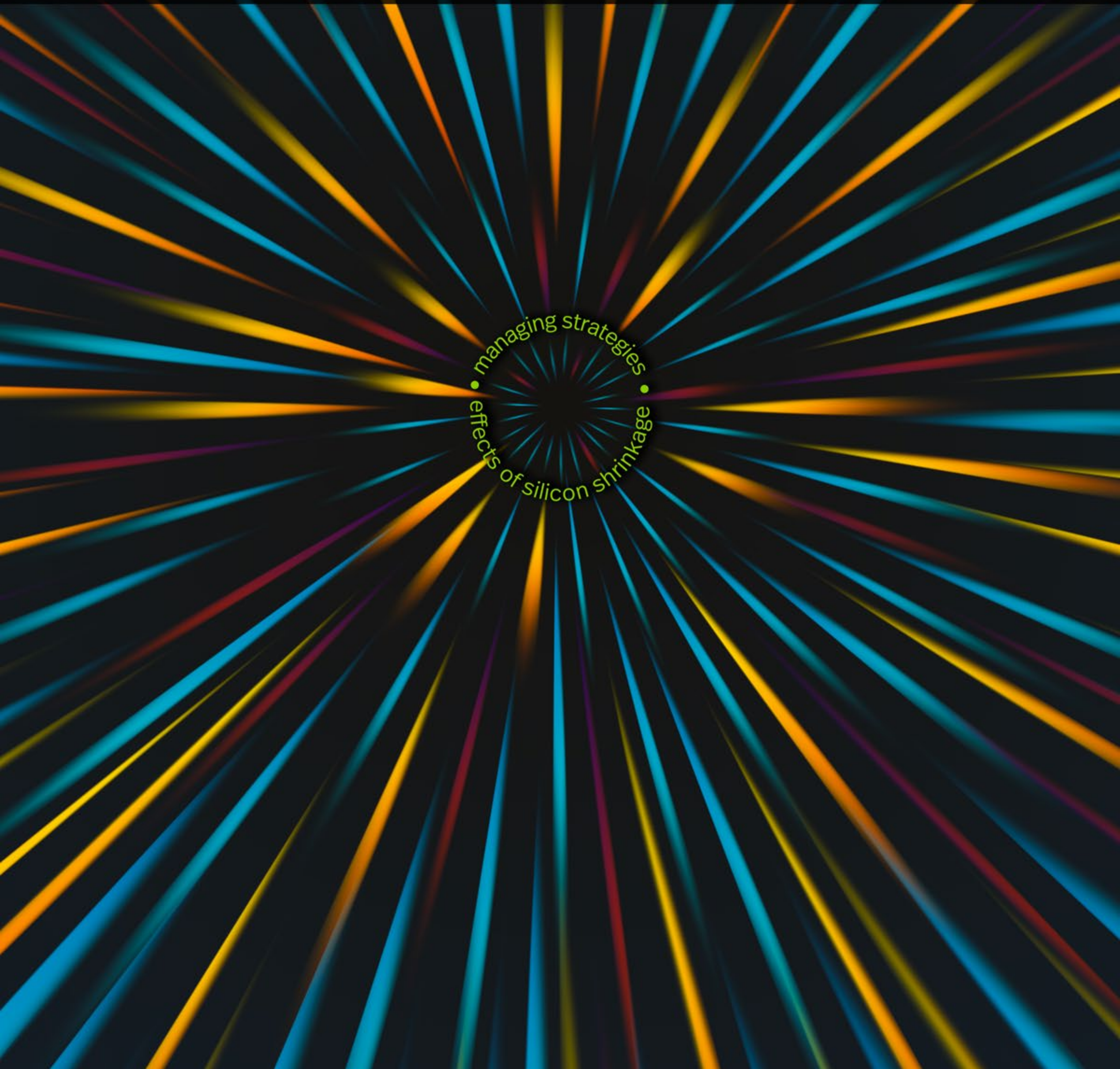
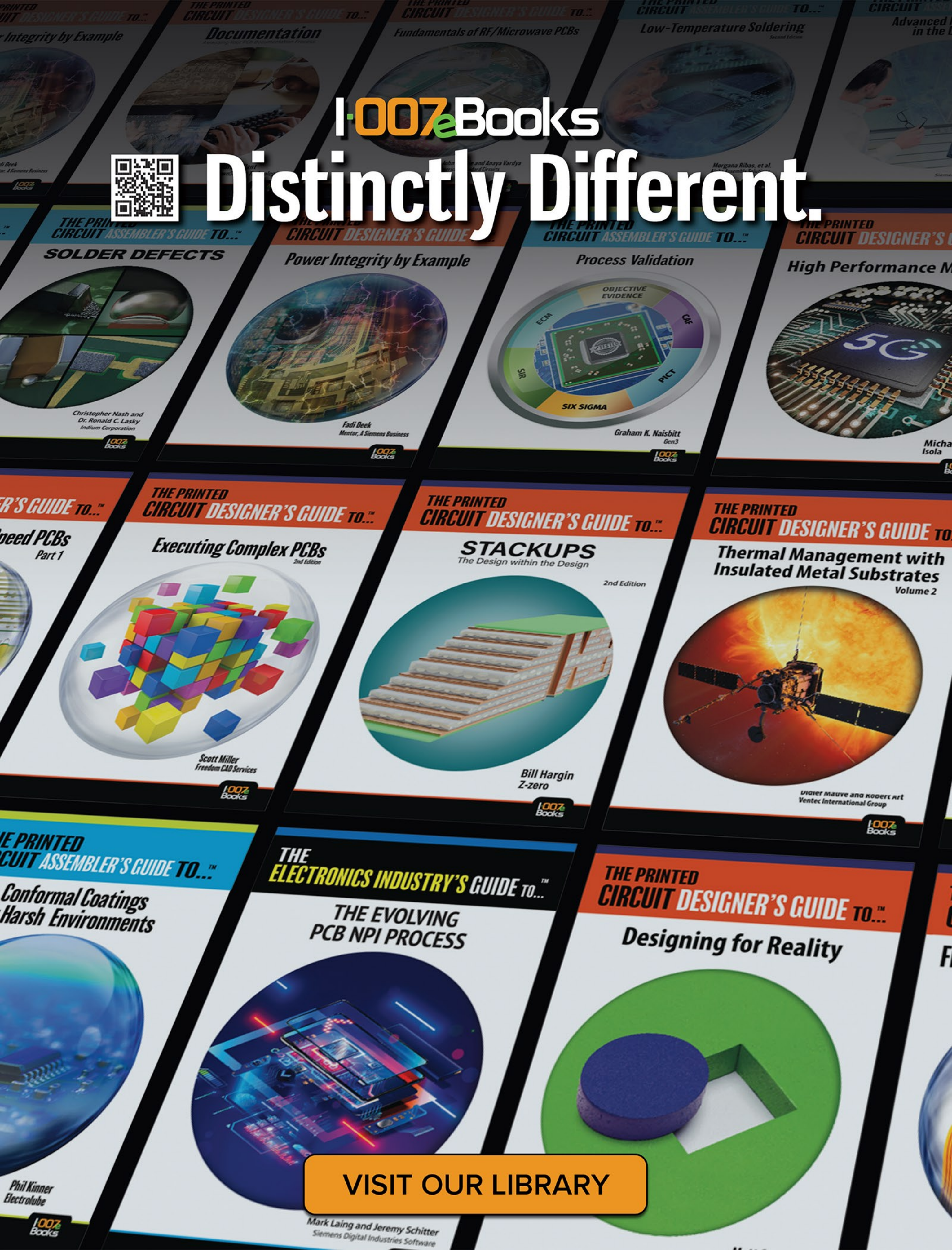
I-Connect007

FEBRUARY 2023

DESIGN007 MAGAZINE



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

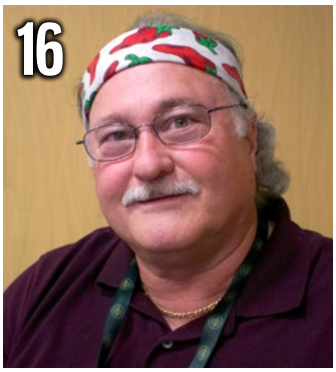


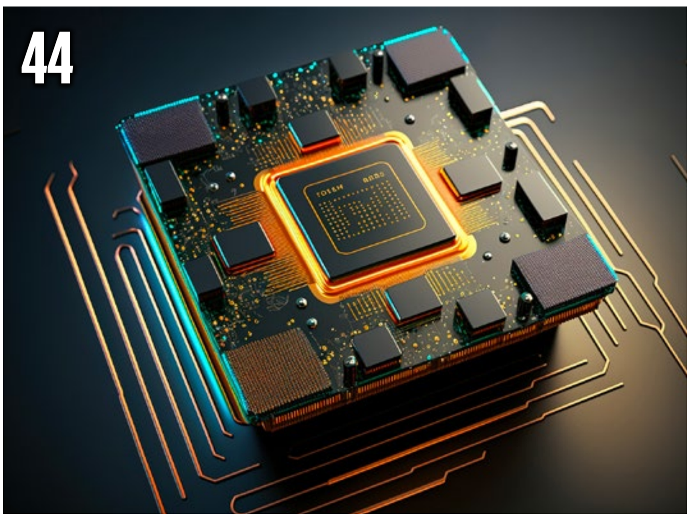
I-007eBooks
Distinctly Different.
THE PRINTED CIRCUIT ASSEMBLER'S GUIDE TO...™
SOLDER DEFECTS
Christopher Nash and Dr. Ronald C. Lasky
Indium Corporation
THE PRINTED CIRCUIT DESIGNER'S GUIDE TO...™
Power Integrity by Example
Fadi Deek
Mentor, A Siemens Business
THE PRINTED CIRCUIT ASSEMBLER'S GUIDE TO...™
Process Validation
Graham K. Naisbitt
Gen3
THE PRINTED CIRCUIT DESIGNER'S GUIDE TO...™
Executing Complex PCBs
2nd Edition
Scott Miller
Freedom CAD Services
THE PRINTED CIRCUIT DESIGNER'S GUIDE TO...™
STACKUPS
The Design within the Design
2nd Edition
Bill Hargin
Z-zero
THE PRINTED CIRCUIT DESIGNER'S GUIDE TO...™
Thermal Management with Insulated Metal Substrates
Volume 2
Ventec International Group
THE ELECTRONICS INDUSTRY'S GUIDE TO...™
THE EVOLVING PCB NPI PROCESS
Mark Laing and Jeremy Schitter
Siemens Digital Industries Software
THE PRINTED CIRCUIT DESIGNER'S GUIDE TO...™
Designing for Reality
Phil Kinner
Electrolube
VISIT OUR LIBRARY

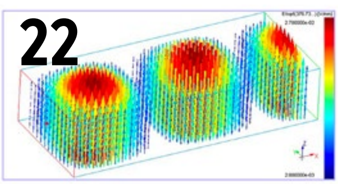
DESIGN007
MAGAZINE

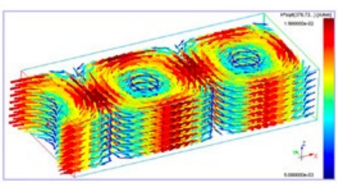
Shrinking Silicon: A Warp Speed Facilitator

The need for designers to understand EM effects is greater than ever as silicon continues to shrink, driving up signal speeds and rise times. In this issue, readers will learn the causes and effects of silicon shrinkage, including the need to better manage EM strategies—as well as signal integrity—on their PCB designs.





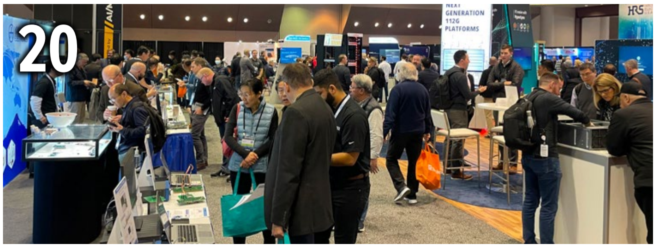




FEATURE INTERVIEWS

FEATURE COLUMNS

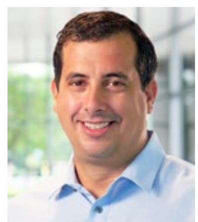
DESIGN007 MAGAZINE



SHORTS

COLUMNS

ARTICLES & INTERVIEWS



HIGHLIGHTS

DEPARTMENTS

Shrinking Silicon: A Warp Speed Facilitator

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

As we all learned by watching *Star Trek,* a lot of crazy things can happen at warp speed. Sure, it was great to get to Alpha Centauri in a hurry, but the Enterprise almost destroyed itself a few times when they put the pedal to the metal.

There's just no room for error at warp speed.

Now, many PCB designers are dealing with increasing signal speeds and rise times, and a parliament of other effects—some positive, some negative—thanks to shrinking silicon. Not quite warp speed, but a lot of unpredictable things can happen when the die get tiny.

Ask yourself: Do you really understand die shrinkage?

When we researched this topic, designers and design engineers told us that smaller die sizes have been driving up signal speeds for years. But now, with today's chipmakers making silicon increasingly smaller, things are getting, as one designer put it, "ridiculous."

Shrinking the signal channel drives up the speed of the signal—as one EE explained, like squeezing an ice cube until it shoots out of your hand. Field effects can't be ignored when the die gets tiny. Designers and design engineers must understand EM effects and all the trade-offs involved; they have to know how to propagate each signal without overshoot or undershoot.

Material selection plays a bigger role than ever. Seemingly simple concepts such as trace length can have a big impact on the design, and crosstalk is a potential bugaboo, always lurking around the corner.

Well, why does silicon keep shrinking? There are many benefits: Smaller die reduce the current and use less power, which translates into fewer thermal issues. They're also cheaper to produce. Like PCB fabricators cutting costs by squeezing more PCBs onto each panel, chipmakers can now stuff more chips onto every wafer.

It's a win-win for everyone but the designers and design engineers; they must deal with rise times that leave no room for error, even in a best-case scenario. Not only must designers manage signal integrity, but EM effects as well. This is where the world is headed; you don't want to be left behind.

So, in this issue, readers will learn the causes and effects of silicon shrinkage, including how to better manage EM strategies and signal integrity, as signal speeds and rise times continue their trek toward warp speed.

First, we start with a wide-ranging conversation with IPC instructor Kris Moyer, who teaches classes that touch on this topic. Kris explains all the challenges designers face with tiny die, and a few simple techniques that can save designers time and effort—if they know what they're doing.

Next, we have a conversation with NXP Semiconductor's Dan Beeker, who points out the need to have a firm understanding of the fundamentals of physics and EM effects when designing with shrinking silicon. Then signal integrity instructor Rick Hartley and columnist Barry Olney team up for a great article about displacement current and its role in EM energy propagation. We have a discussion with Dr. Todd Hubing, founder of LearnEMC and longtime EMC instructor, who points out the advantages and disadvantages of diminutive die, as well as a variety of mitigating tips and techniques for PCB designers and design engineers facing increased signal speeds and rise times.

We also have a host of columns from our regular contributors Matt Stevenson, John Coonrod, and Joe Fjelstad, as well as an interview with Jason Sciberras, who discusses the importance of offering optional parts in your bill of materials. With the supply chain issues we've had lately, this is quite a timely interview.

'Tis the Season for Trade Shows

Speaking of timely, I just returned from IPC APEX EXPO and DesignCon. I talked to a lot of designers, and plenty of process engineers who are eager to work with designers to help preclude DFM and DFA issues before they arise. It seems like everyone downstream from design is realizing the need to keep open lines of communications with the design team. I also talked with designers who are embedded with assembly teams in the name of DFA. Are we all finally starting to communicate?

Our *Real Time with...* IPC APEX EXPO video interviews are available here, and my DesignCon coverage will be published in upcoming issues of *Design007 Magazine* and Design007 Week newsletter. **DESIGN007**

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

Shrinking Silicon, Growing Signal Integrity Challenges

Feature Interview by the I-Connect007 Editorial Team

What happens when die sizes shrink? As IPC design instructor Kris Moyer explains, quite a bit. Shrinking silicon can mean rising signal speed and rise times, and traditional PCB designers may find themselves dealing with problems formerly only seen by RF engineers.

We asked Kris to discuss the pros and cons of silicon shrinkage and some of the techniques and trade-offs that PCB designers and design engineers need to understand as they find themselves entering the RF arena.

Andy Shaughnessy: This issue focuses on the effects of shrinking silicon on a board's signal integrity and EMI. So, what do PCB designers need to understand about die shrinkage?

Kris Moyer: Basically, the main thing that happens when you shrink the size of the die is that it shrinks the length of the channel of the transistors inside the die. What that effectively does is it increases the speed of the circuit, meaning it decreases the rise time or the fall time. Then you end up having to start treating your traces, geometries, and transmission lines almost as if they're RF designs.

We've heard for decades that RF designs are their own special little area of black magic, because we start dealing with all these waves and fields and so on. We say in digital design that it's the rise time and not the frequency. Which is the driving force, the key factor, that causes the need for all these high-speed designs? What is the frequency content?

Fourier's theorem says any wave form—square wave, triangle wave, sawtooth wave, or any wave form—can be recreated as a superposition of a sufficient amount of sine waves and cosine waves of sufficiently higher harmonics. Let's take a fundamental frequency, 1 kilohertz. And you have A1, A3, A5 and A10 kilohertz. You have all these harmonics, we

Kris Moyer

superimpose them, and you end up getting a square wave. Well, how square does that square wave need to be? This is the part that throws a lot of designers off.

When we talk about rise time, we're really talking about the time it takes that square wave, the digital signal, to change from a logic 0 to a logic 1. As the die shrinks, that time also shrinks. About 20 years ago, we were having rise times and fall times in the multiples of nanoseconds—five to 10 nanoseconds. It took that signal five to 10 nanoseconds to change from a logic 0 to a logic 1. I was just looking at one FPGA with rise times as fast as 0.25 nanosecond, and that's at 16 nanometers.

My friends who work in next-generation silicon at some of the big telecom companies are working in 5, 3, and 2 nanometer, and going sub 100 picosecond. Instead of 0.25 nanosecond, it's 0.1 nanosecond and 0.05 nanosecond rise times. They're such incredibly fast rise times that the number of harmonics we need to create a vertical square edge that transitions from A0 to A1 that fast means that the frequencies involved in that superposition in that Fourier series are up into the multiple gigahertz of frequency content. That means that you're in the RF frequency range.

Shaughnessy: Even if you're not an RF guy, there you are, dealing with RF.

Moyer: You're in the same frequencies as an RF type of design, where you have to deal with skin effect and lossy transmission line models. Very-high-frequency designs that used to be the arena only for the RF guys are now affecting digital and analog, though not so much for analog. But digital engineers who are dealing with these fast rise times now have to consider all of this high-frequency RF content.

I think a lot of engineers don't realize that with the Fourier series, that conversion, that content needs to be there to create a square wave of that fast of an edge. Basically, it says that frequency content exists. But I don't care what the clock frequency is. The frequency content exists in your square wave because the silicon has now shrunk, which means the DI/DT of that edge rate now contains this frequency content, whether you wanted it or not, and that is the single biggest issue that designers need to understand and fully conceptualize. In short, frequency content exists simply because the silicon has shrunk.

Shaughnessy: They're accidental RF engineers at this point.

Moyer: Pretty much. Just 20 to 30 years ago, maybe you had a frequency of 100 megahertz and you could still get away with pretty easy design rules. The other big problem that happens because of these edge rates is what we call the transitional electrical length or the distance that the signal will travel down the transmission line while it's actively changing from A0 to A1. That distance has also shrunk, and this is the other problem with digital design.

Historically we didn't have much problem routing digital signals, as long as the length of the trace that your signal is propagating down

is shorter than the distance that the signal travels during that switching edge. You don't have significant reflection problems because you are actively driving the signal you are actively putting energy into, or taking energy out of, that transmission line. That active driving will overcome any reflections or crosstalk, rise time, and all that. Well, again at A1 nanosecond edge rate and a dielectric constant of 4.0, that gives us about a 6-inch trace and that is fairly long on a circuit board at one-quarter of a nanosecond. You now have to start dealing with termination techniques, crosstalk avoidance, and all these technical problems, just because the switching edge got faster because the die has shrunk.

You now have to start dealing with termination techniques, crosstalk avoidance, and all these technical problems, just because the switching edge got faster because the die has shrunk.

Shaughnessy: So, it sounds like the "typical" PCB designer will need to be aware of all these potential pitfalls far before the design cycle starts. What can designers do to get ahead of this?

Moyer: Step one: Assume that if you have digital chips on your board, you have signal integrity issues and must apply the proper signal integrity and design analysis methodologies. Based on your stackup, determine your velocity of propagation and your transitional electrical length, and put in proper series termination. This is going to be a collaboration between the board engineer and the circuit engineer.

If your company doesn't have a dedicated signal integrity analysis engineer, the electrical engineer should be able to do the signal integrity analysis to determine the appropriate series termination resistor values. What are the parallelism requirements? Assume you're going to have those up front before you lay your first trace on the board. Have all your design rules created and defined, assuming signal integrity-related issues.

Nolan Johnson: As you say, it's not really black magic; it's all right there.

Moyer: There's a great textbook about this topic, written by Dr. Howard Johnson, called High-Speed Digital Design: A Handbook of Black Magic. He literally named his book "black magic," because in the early days, this was all not well understood. What do these structures look like? One of the things that our designers really need to understand is that we need to stop thinking about our board structures in two dimensions. We need to start considering them as 3D structures, considering the Z-axis for separation to planes. What about the fact that we have fringing capacitance coming off the edges of our traces? There is a vertical edge to our traces; it's not just a flat trace, even though that's how we draw it. You know, in our CAD tools we really need to start considering all the structures on our boards as 3D structures. What are the physics of those 3D structures and how are they going to behave at these high frequencies?

Shaughnessy: Who are the great instructors in this area? Who has written books on this?

Moyer: There are four main gurus here as far as authorship is concerned. I mentioned Howard Johnson already. Lee Ritchey of Speeding Edge is an outstanding guru and signal integrity instructor and author. Doug Brooks has written quite a few books on this, and Eric Bogatin has written and taught extensively about this.

Then there's Rick Hartley, who has also done many signal integrity classes, as well as professional development courses at IPC APEX EXPO. He's another excellent source. So those are five guys I would think of as gurus in signal integrity. I've got all their textbooks and I reference them all the time myself. And of course, come take any of the IPC courses in board design as well.

Johnson: Is there anything you'd like to add, Kris?

Moyer: I don't want to say it's new, but it is an expanding area of concern that is only going to keep growing as the values keep shrinking. This will be more pronounced and is definitely an area that all designers and engineers need to take seriously. They need to get a good handle on this, because the rules of thumb, the tribal knowledge from yesteryear, no longer holds sway. What was true 20 years ago is no longer true today.

Shaughnessy: Thanks for your time, Kris.

Moyer: Thank you. DESIGN007

BOOK EXCERPT

The Printed Circuit Designer's Guide to... Designing for Reality

Introduction

The printed circuit board industry is ever growing and changing. As a new generation of PCB designers spearheads innovation across a myriad of electronic device applications, individuals without those specialized skills are increasingly called upon to design boards in order to accelerate product development and drive down costs.

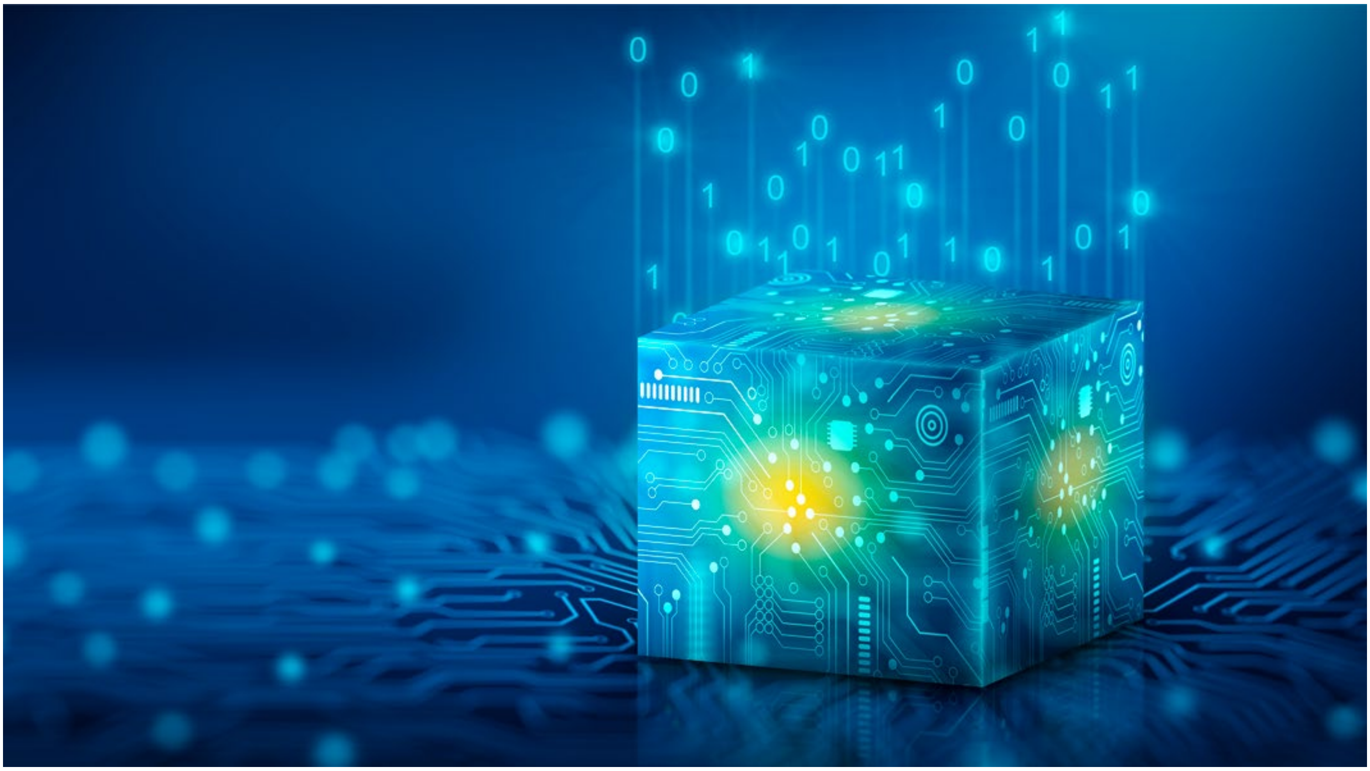
With more of us working to gain board layout experience on the job, Designing for Reality becomes a very pertinent discussion topic in the PCB industry. But what does that phrase even mean and why should we focus on such a topic? For those who may be new to PCB design or possess limited experience, it is crucial to recognize that creating a robust and manufacturable PCB design requires paying close attention to numerous details. There are a lot of unwritten rules, best practice techniques, and design requirements that vary by manufacturer. Learning and understanding these often-subtle factors will give designers a leg up creating realistic board designs and enable them to take their skills from novice up the ranks.

The goal of a PCB designer is to create a design that is free from defects that can "kill" the board or make it difficult to manufacture.

Realistic PCB designs should prioritize manufacturability and reliability of the PCB as well as meet the other design requirements. In order to do so, one must account for the production variables associated with individual manufacturing partners.

Whenever I have the opportunity to talk to a PCB designer or students, I encourage them to tour a PCB manufacturing facility to get a better understanding of how the board manufacturing process works. It is mind blowing. There are many more processes and steps required than one would imagine.

Download this title today!



Shrinking Geometries: Back to Fundamentals to Fight EMI

Feature Interview by Andy Shaughnessy
I-CONNECT007

When silicon shrinks, a variety of things can happen—some positive, some negative. But for PCB designers, the fight against EMI becomes more complex as signal channels shrink and rise times increase.

Dan Beeker is technical director at NXP Semiconductors, a veteran design engineer, and an instructor who has spent years helping students and customers battle EMI through building a better understanding of electromagnetic fields and field theory. In this interview, Dan explains what happens when silicon shrinks, how feature size controls signal speed, and why this marks the perfect time to return to the fundamentals of physics and field theory.

Andy Shaughnessy: Shrinking silicon is increasingly causing EMI issues for PCB designers and EEs. What sort of problems does shrinking silicon cause?

Dan Beeker: Smaller device geometries and higher current switching capabilities have thrust us all into the world of RF, HF, UHF, and microwave energy management. Rise times on even the lowest-tech devices now exhibit gigahertz impact. These changes directly impact product functionality and reliability. When IC technology was described as a percent of shrink from integer design rules, a circuit-based approach was usually close enough. Now that

IC technology is described in nanometers, that traditional approach completely falls apart. An EM field, physics-based approach is essential.

To make things worse, EMC standards have changed; we now have lower and higher frequency compliance requirements, much lower emissions levels allowed, and greater immunity required. The playing field and the equipment have completely changed. This really is a brand-new game. The challenges are not only about providing adequate power to the devices, but also managing the output signals. The smaller the transistor, the faster it turns on, and the bigger the impact it has on EMC and signal integrity.

Shaughnessy: What is the relationship between smaller silicon and EM fields, and what can these designers do proactively to fight EMI?

Dan Beeker

Beeker: The issue is driven by simple geometry. The smaller the transistor, the faster it switches, and the higher the frequency you must address. This is important both on the output side, which is generally acknowledged, and on the input power side as well. Most EMC issues I have seen are directly related to an improper power supply design. The problem gets even more difficult with today's technology, as the power supply requirements must first be addressed on the die. If the IC vendor has not done this properly, there is nothing you can do on the application side to correct it. You will find out the hard way if this is an issue once you try using the device. IC vendors do not tell you whether they have done this correctly.

Shaughnessy: If we add increased rise time and faster signals to this mix, how does material and material selection figure into the equation? Should OEMs select the material, or allow the fab to do so?

Beeker: Output signal characteristics are driven by the transistor geometries used to drive the pins. Assuming the input power is done correctly, impedance matching becomes the challenge. The material itself becomes a prime factor for very fast data rates (not switching frequencies) and timing coherence. Propagation times become less predictable in the dielectric, because the mix of glass vs. epoxy is not homogeneous, and the energy moves at a different relative velocity in each. As timing constraints become tighter, and data rates increase, this problem only gets more difficult to solve. Eyes close, and jitter reigns supreme.

Shaughnessy: What advice would you give designers and EEs who are starting to have EMI issues because of tinier silicon and increased speeds?

Beeker: The most important action for them is to go back to fundamental physics and make sure that the rules they use for design are based on the science. Current does not flow in a loop, and electrons do not carry the energy. EM field energy travels in the dielectric, and the job of the PCB designer is to create the pathways for controlling the fields. Layer counts cannot be driven by the trace density; they are driven by

the IC geometry and the EMC certification level required by the application.

Another important consideration is that the package geometries are changing as well. Smaller pin pitch and higher pin counts increase the difficulty and costs. It costs more to use smaller parts, both in PCB and assembly costs. One related factor that is not always considered is the cost of development tools for the newer, cheaper MCUs. Now you can get a 32-bit processor with lots of memory and I/Os for a fraction of the cost of the older 8-bit parts. However, the cost of debug tools and compilers is orders of magnitude more expensive. Add the issue of using a larger memory space, which may require an operating system (and a software team that can handle it), wow, so much for a "cheaper" part. The industry is finding out the hard way that, "System cost is not reduced by reducing IC geometries."

Shaughnessy: Is there anything else you would like to add?

Beeker: Electromagnetic fields travel in the space between the conductors, not in the conductors. The switching speed of the transistors determines the frequency of operation, not the clock rate. Signal and power connections must be one dielectric from ground for their entire length, including layer transitions.

There is no such thing as a noisy ground—just poor transmission line design.To quote Dr. Todd Hubing, "Thou shalt not split ground." Remember the Rules of Triplets: "You only need three components to contain EM energy: conductor, space (dielectric), and conductor. You only need three components to build electronic systems: conductors, spaces (dielectric), and switches. You can only do three things with electromagnetic field energy: store it, move it, or convert it to kinetic energy."

Any compromises to these rules will increase system noise and so must be done as carefully considered engineering decisions. As Ralph Morrison always said, "Buildings have walls and halls. People travel in the halls, not the walls. Circuits have traces and spaces. Energy and signals travel in the spaces, not the traces."

Shaughnessy: Thanks for your time, Dan.

Beeker: Thank you, Andy. And remember: It's all about that space! DESIGN007

DesignCon 2023 Opens With Strong Attendance

by Andy Shaughnessy

This year, the event was held Jan. 31–Feb. 2. The welcome reception Tuesday night was literally packed inside the Santa Clara Grand Ballroom. There must have been 150 people eating tacos and drinking IPAs and wine. It was so loud that you couldn't hear the person next to you. Good problem to have, though!

When the show opened Wednesday morning, it really felt like old times again. The aisles of the Santa Clara Convention Center were busy, and the classrooms for the technical conference were nearly full. AI was in the spotlight at DesignCon 2023. The Chiphead Theater hosted a panel on AI in EDA tools. One of the main challenges seems to be getting AI from proof of concept into production. Everyone agrees that AI has a role in EDA—how to get there is up for discussion. But EDA companies were demonstrating their tools' AI functionality during this show, particularly in simulation and analysis software.

This feels like a good omen for the rest of 2023. People are ready to get back to in-person trade shows and conferences. To use an overused turn of phrase, it seems like old times, but with the latest technology.

See my photos from the show.


NEW!
Community
MAGAZINE
WINTER 2023
IPC

Displacement Current: The Key to Electromagnetic Energy Propagation

Beyond Design

Feature Column by Barry Olney, IN-CIRCUIT DESIGN PTY LTD / AUSTRALIA
with Special Advisor Rick Hartley, R HARTLEY ENTERPRISES

The propagation of electromagnetic energy can be controlled in several ways depending on the medium the energy is traveling in. However, electromagnetic waves do not require a medium to propagate. This means that electromagnetic waves can travel not only through liquids, solids, and air, but also through the vacuum of space. What's more, they do not require electron current flow for the transfer of energy.

Electromagnetic energy can be guided in the following ways:

1. **Direct Current:** Conductors guide the energy flow.
2. **Alternating Current:** Conductors, coplanar and substrate integrated waveguides at high frequencies control the energy.
3. **Radio and Microwave Frequencies:** Waveguides and antennae guide the energy.
4. **Light Frequency:** Optical fiber channels, lenses/mirrors, and gravitational lensing control the energy path.

Continuing on from my previous column, "Forget What You Were Taught," let's take a closer look at how electromagnetic energy propagates at different frequencies[1].

Waveguides

A waveguide is a form of transmission line used to connect microwave transmitters and receivers to their antennas. They are metal tubes made of high-quality copper and brass. A waveguide can have a rectangular, circular, or elliptical cross-section. The rectangular section is most used for relatively short connections. Figure 1 depicts a plot in QWED software of the electric and magnetic field distribution along a rectangular waveguide. A transverse electromagnetic wave travels perpendicular to both the electric and magnetic fields.

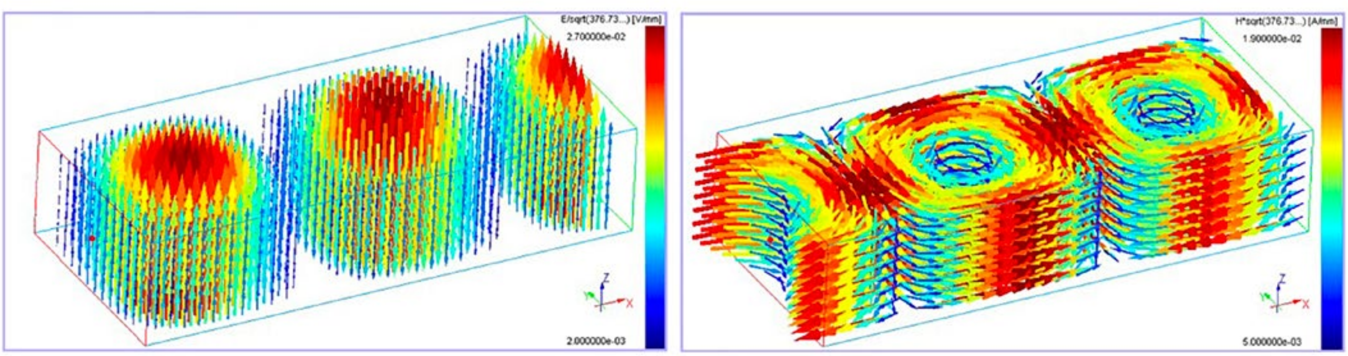


Figure 1: Electric field (left) and magnetic field (right) distribution along a rectangular waveguide. (Source: QWED)

Electric and magnetic fields, which are used for the transport of energy, are equal to zero on the metal surface of the waveguide. Therefore, these fields are confined to the waveguide's internal space, which minimizes losses. Without the physical constraint of a waveguide, wave intensities decrease according to the inverse square law as they expand uniformly in all directions. Waveguides act like conduits for high frequency displacement currents. All transmission lines function as conduits of electromagnetic energy when transporting pulses or high frequency waves.

Wireless Power

Most of us have Qi wireless chargers for our smart devices—who could live without them? The big advantage to using these chargers is that you do not have to constantly plug in cables to charge your phone. No wires—no current—but 15W of power.

Qi wireless charging uses resonant inductive coupling between the sender (the charging station) and the receiver (the mobile device) as in Figure 2. Figure 3 shows the Qi receiver coil in an iPhone 12. The two coils act as a transformer when a compatible device is placed on a charging station.

Transformers

A transformer works by electromagnetic induction (or mutual inductance). This occurs when two electrically isolated coils are in close proximity, such that one's magnetic field couples to the other. When an alternating current is applied to the primary coil, a fluctuating magnetic field is generated, which causes electromotive force in the secondary coil. This varying electric field creates displacement current in the secondary winding. Adding an iron core to the transformer improves the efficiency by directing the electromagnetic field so that it couples directly into the sec-

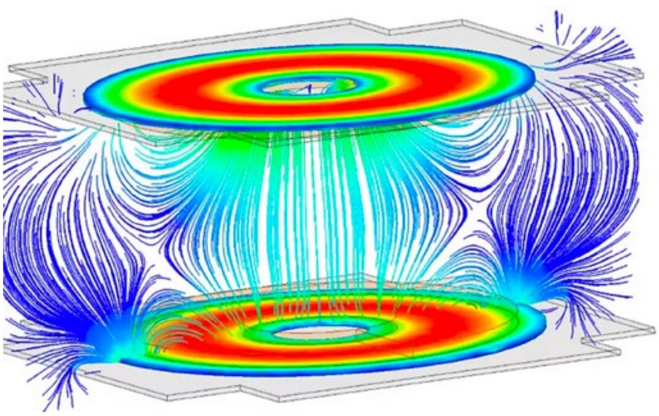


Figure 2: Qi wireless charging simulation (simulated in Ansys Maxwell).

Figure 3: Apple iPhone 12.

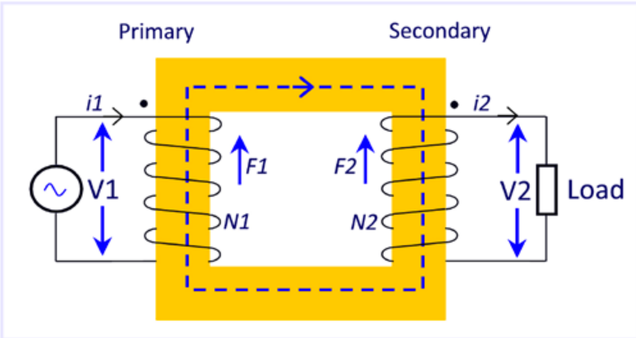


Figure 4: Basic transformer with primary and secondary coils surrounding an iron core.

ondary winding rather than radiating. Just as waveguides and traces guide electromagnetic energy, so does the core.

Transformers completely isolate the primary from the secondary. Transformers transfer the electric energy into magnetic energy (primary windings) and then back to electric energy (secondary winding). The transformer's core captures >90% of the magnetic energy and delivers it to the secondary windings.

AC Coupling of High-speed Serial Links

A capacitor is typically placed in series with both differential signals of high-speed SERDES serial links to remove common mode voltage differences between ICs or different technologies (Figure 5). Any capacitor placed in series with the signal path tends to pass the high-frequency AC portions of the signal, while simultaneously blocking the low-frequency DC portions. These capacitors are essential to a variety of high-speed interfaces. And, as the next generation of designs target data rates of 56Gbps and above, it becomes increasingly important to characterize channel transitions accurately to ensure a high confidence of success.

However, capacitors block electron flow. A capacitor in a circuit causes equal and opposite charges to appear on the plates, charging the capacitor and increasing the electric field between the plates. No actual charge is transported between its plates. Nonetheless, a magnetic field exists between the plates as though a current were present. One explanation is that displacement current flows in the dielectric and this current produces the magnetic field in the region between the plates.

This idea was conceived by James Clerk Maxwell in his 1861 paper "On Physical Lines of Force, Part III" in connection with the displacement of electric particles in a dielectric medium. Maxwell added displacement current to the electric current term in Ampère's Circuital Law. In his 1865 paper "A Dynamical Theory of the Electromagnetic Field," Maxwell used this amended version of Ampère's Circuital Law to derive the electromagnetic wave equation. This derivation integrates electricity, magnetism, and optics into one single unified theory. The displacement current term is now seen as a crucial addition that completed Maxwell's equations and is necessary to explain many phenomena, most particularly the existence of electromagnetic waves.

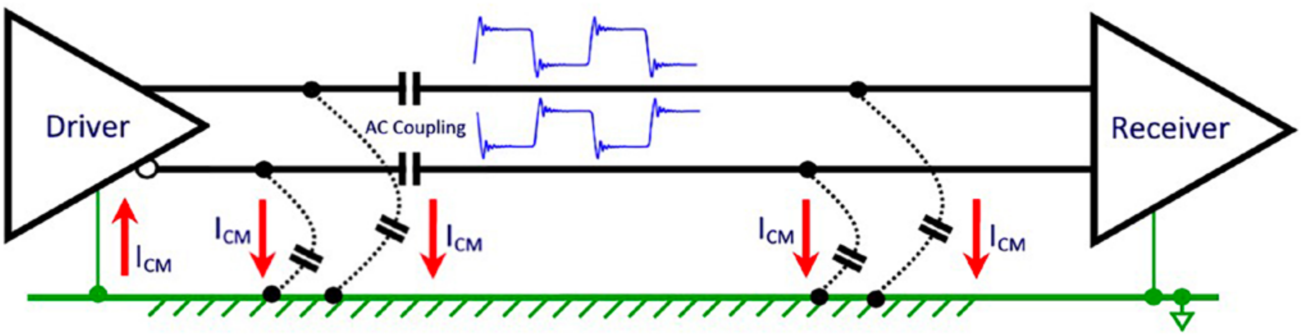


Figure 5: Common mode current (I_{CM}) is the displacement current.

Light

Maxwell described light as a propagating wave of electric and magnetic fields. More generally, he predicted the existence of electromagnetic radiation—coupled electric and magnetic fields traveling as waves at the speed of light.

Displacement current plays a vital role in the propagation of electromagnetic radiation, such as light and radio waves, through empty space. A traveling, varying magnetic field is associated with a periodically changing electric field that may be conceived in terms of a displacement current. Maxwell's insight on displacement current, therefore, made it possible to understand electromagnetic waves as being propagated through space completely detached from electric currents in conductors.

Displacement current plays a vital role in the propagation of electromagnetic radiation, such as light and radio waves, through empty space.

The displacement current density term, appearing in Maxwell's equations, is the quantity ∂D/∂t that is defined in terms of the rate of change of D, the electric displacement field. Displacement current density has the same units as electric current density, and it is a source of the magnetic field just as the actual current is. However, it is not an electric current of moving charges, but rather a time-varying electric field. This implies that a changing electric field creates a magnetic field, even with no charged particles in motion. In physical materials (as opposed to vacuum), there is also a contribution from the slight motion of charges bound in atoms, called dielectric polarization.

Displacement current explains how electromagnetic energy propagates but is really just a fudge. Scientists have a creative way of accounting for what they do not comprehend: They add a constant. For instance, astrophysicists cannot explain why the universe is expanding when it logically should be contracting due to the attraction of gravity. They then created "dark energy" to account for the force of expansion. Displacement current is just another "unexplained phenomenon" that accounts for current. Few theories in physics have caused as much confusion and misunderstanding as that of displacement current.

There are two types of current:

1. ***Conduction current*** is the net flow of charges at DC. This is what we traditionally think of as current flow.
2. ***Displacement current*** is the rate of change of the electric displacement field. It is not electron current flow but rather a time-varying electric field that creates a magnetic field along a transmission line mimicking current flow.

Transmission Lines

Dan Beeker stated that: "Field energy moving through a space is the current flow in a transmission line. The magic here is the displacement current flowing through the dielectric at the wave-front, along the transmission line. Fields do all the work. Current flow is a measure of moving field energy through a space. Current flow occurs in the space between the conductors that bound the dielectric[2]."

Ralph Morrison summed it up beautifully: "Light energy can be directed by lenses; radar energy can be directed by waveguides and the energy at power frequencies can be directed by copper conductors. Thus, we direct energy flow at different frequencies by using different materials. We have learned how to control where we want the field energy to go.

"The Laws of Physics apply to everything in the universe. If we accept the concept that

electromagnetic fields carry energy in space, it must be true at all frequencies in all media. That is the law. If it is true for light, it must also be true for high-speed transmission lines, 60 Hz power, and at DC[3]."

Key Points

- The propagation of electromagnetic energy can be controlled in a number of ways depending on the medium the energy is traveling in.
- The propagation of electromagnetic energy does not require electron current flow for the transfer of energy.
- Waveguides act like conduits for high frequency displacement currents.
- Transformers transfer the electric energy into magnetic energy (primary windings) and then back to electric energy (secondary winding).
- Capacitors block electron flow. However, displacement current flows in the dielectric and this current produces the magnetic field in the region between the plates.
- The displacement current term is now seen as a crucial addition that completed Maxwell's equations and is necessary to explain many phenomena, most particularly the existence of electromagnetic waves.
- Displacement current plays a vital role in the propagation of electromagnetic radiation.
- A traveling, varying magnetic field is associated with a periodically changing electric field that may be conceived in terms of a displacement current.
- Electromagnetic waves are propagated through space completely detached from electric currents in conductors.
- Displacement current is not an electric current of moving charges, but rather a time-varying electric field.
- There are two types of current: Conduction current and displacement current.
- Field energy moving through a space is the current flow in a transmission line. The magic here is the displacement current flowing through the dielectric at the wave-front, along the transmission line.
- The laws of physics apply to everything in the universe. DESIGN007

References

1. "Beyond Design: Forget What You Were Taught," by Barry Olney, *PCBDesign007 Magazine,* Nov. 2022.
2. "Electromagnetic Fields for Normal Folks," presentation by Dan Beeker at AltiumLive 2017, Oct. 3, 2017.
3. "Laws of Physics," by Ralph Morrison, reprinted in *Printed Circuit Design & Fab Circuits Assembly Magazine,* April 2021.

Resources

- "How Does a Transformer Work: 9 Answers You Should Know," by Sneha Panda, LambdaGeeks.com, 2023.
- "Waveguide," Wikipedia.org, Dec. 5, 2022.
- "Displacement current," Wikipedia.org, Nov. 4, 2022.
- "Short-Slot Waveguide Hybrid," Micowaves101.com, 2008.
- "Basics of waveguide theory," by Christian Wolff, RadarTutorial.eu.

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

Rick Hartley is the principal engineer at R Hartley Enterprises and has been in the industry for over 50 years. He is one of the primary consultants for PCB manufacturing and design companies. Rick has also conducted classes worldwide on EMI, signal integrity, and various other electrical topics for the last 30 years.

Medical Technology: How PCBs Help Save Lives

Connect the Dots

by Matt Stevenson, SUNSTONE CIRCUITS

PCBs power all sorts of innovative devices, everything from virtual reality headsets to drones, but our industry isn't all about fun and games. Advances in practical technology have also changed the face of manufacturing and logistics. But none of these breakthroughs have been more life changing for the average person than in the areas of health, wellness, and medical treatments.

PCBs have become the backbone of all effective, efficient, and safe personal health medical devices. Electronics and computer technology are enabling medical breakthroughs that help identify and treat illnesses more quickly and accurately, as well as provide advanced monitoring for patients during recovery.

New obstacles to innovation pop up every day. To keep health services technology moving into the future, PCB manufacturers need to operate transparently and focus on a commitment to quality.

PCB Industry Adapts to Meet New Technology Challenges

Even before the global challenges presented by COVID-19, there existed growing stress on medical services in almost every corner of the world[1]. When the pandemic disrupted supply chains and workflows, PCB manufacturers had to adapt quickly to help create advanced medical technology that would assist providers in diagnosing and treating patients while resources were stretched thin.

Worldwide shortages caused by the pandemic meant that even existing technology faced new levels of demand. Devices such as ventilators have been in short supply since the early stages, and the list continues to grow[1]. To ease the pain for health services providers, products need to be rapidly manufactured and made available to meet urgent demands.

To ease the pain for health services providers, products need to be rapidly manufactured and made available to meet urgent demands.

With shorter production windows and increased raw materials costs, it becomes even more important to achieve first-pass production yields. Offshoring had been a viable option for simpler board designs prior to the pandemic, but persistent issues with supplier reliability, unpredictable delivery windows, and increased QA challenges have caused electronics manufacturers to re-evaluate this practice.

From smaller MRI machines and new DNA testing technology to innovative wearable tech, these devices require complex, often multi-layer PCBs. This leads many health industry electronics manufacturers to seek a domestic alternative to help them with more complex circuit designs on smaller, miniaturized boards.

Working with a manufacturing partner closer to home reduces risk for manufacturers. Transition plans, phase-ins, phase-outs, and revision control often demand immediate attention, which was already difficult to rely on with offshore manufacturers because of time zone differences or language barriers.

During prototype design, manufacturers need effective communication and coordinated effort to ensure they get the quality boards their projects need to succeed. Close collaboration with a domestic PCB manufacturer offers more transparency during the manufacturing process, accelerates issue resolution, increases yield, and—most importantly—improves PCB quality.

Commitment to Quality Fuels Innovation

Quality medical devices don't materialize by magic. Producing them takes a lot of coordination and effort. Electronics manufacturers serving the healthcare industry need their devices to work properly, and they need PCB manufacturing partners they can trust.

Manufacturers that obtain International Organization for Standardization (ISO) certifications send a message to the health services industry that they adhere to requirements, specifications, and guidelines that help consistently ensure processes are focused on quality. The ISO 9000 family and ISO 9001 certifications are built on standards achieved by deploying and maintaining a Quality Management System (QMS).

A QMS focuses on key principles during every phase of production. These principles continuously guide the manufacturer to find new ways to improve processes and products. These principles include:

- **Customer focus:** Exceed customer expectations and create value in every aspect of the production process.
- **Committed leadership and engaged personnel:** Every day, every level of manager reinforces mission, vision, strategy, policies, and processes throughout the organization.
- **Holistic process approach:** Treat each production component as part of a larger production ecosystem, managing individual processes as part of the interrelated whole.

- **Evidence-based decision-making:** Focus on cause-and-effect relationships during production, relying on facts and evidence to improve decision-making and outcomes.
- **Relationship management:** Understand how customers, suppliers, business partners, and employees contribute with continued improvement.

PCB manufacturers are also providing the support and expertise necessary to reliably streamline manufacturing of medical technology—saving time and money for electronics manufacturers in the process. A trusted PCB manufacturing partner makes this happen by:

- Integrating design for manufacturing (DFM) methodology into workflows
- Making rapid prototyping possible with in-depth knowledge of components and processes
- Providing access to experts who can analyze designs and recognize flaws
- Giving quick access to support staff at the manufacturer
- Knowing best practices to maximize yield and functionality

Medicine and Technology: A Life-Changing Partnership

Together, medicine and technology continue to innovate and advance in ways that increase everybody's quality of life. New monitoring devices and miniaturized diagnostic technology are revolutionizing how medical personnel access more and better patient data—helping them safely deliver better care to more patients.

Everybody jokes that it's the 21st century and we don't yet have the flying cars that science fiction movies promised us—but we truly do live in an age of wonderful and advanced technology. It is rewarding to be part of a PCB industry that helps create technology that improves and saves lives every day. DESIGN007

References

1. "Medical Device Shortages During the COVID-19 Public Health Emergency," the U.S. Food & Drug Administration, FDA.gov, Dec. 12, 2022.

Matt Stevenson is vice president at Sunstone Circuits. To read past columns, click here.

Are You Offering Options in Your Bill of Materials?

Interview by the I-Connect007 Editorial Team

In this interview, Saline Lectronics (an Emerald EMS company) President Jason Sciberras talks about PCB designers offering packaging options in the bill of materials. As Jason explains, mil/aero manufacturers like Saline can't make many changes to a design without getting recertified, so including approved packaging options in the BOM from the start is a great way to go. Are you offering options in your BOM?

Andy Shaughnessy: It seems like designers are getting more adept, through necessity, at designing with a back-up plan in mind. We're hearing more about having to redesign as much as a quarter of the board because of supply chain issues. Sometimes they're trying to find two or three components that are available to make up for the one that they wanted, so they end up using more real estate. Are you running into that?

Jason Sciberras: We do a lot of military and aerospace work, so those changes just aren't possible. They would have to go through recertification and the changes are difficult. The shift that's coming in the industry is that designers are getting smarter.

We're starting to see some of them designing alternate packages right into the circuit boards so that if this ever comes back, we'll all think we're getting toward the backend of this thing and eventually it will get better here. But when we come into the next one—because there's

Jason Sciberras

always a next one—we will be better prepared than we are today.

In the last 20 years, we've seen similar cycles that affected the micro lead-frame chip carriers (MLCC), and we had an IC issue also. This one has lasted longer and has affected more products than anything we've ever seen before. The best thing about this one is that the entire world is going through it. We have great relationships with our customers; they're working with us to solve these problems. They understand what's going on. As a team, we've been very successful at keeping them going in some fashion without interruption.

Nolan Johnson: It seems like designing for alternate packages or multiple choices would be a good design practice even when the supply chain is operating smoothly.

Sciberras: I'm hoping that's here to stay. We're seeing customers ask, "What are the five or six critical components that would most likely impact my ability to keep the line up and running? What kind of solutions can I design into the layout right up front so that if, God forbid, we come down this path again, we know what to do about it?"

Shaughnessy: Do they call this out in the assembly notes? Where do they notate that there is an alternate if needed?

Sciberras: There are a couple of different ways you can handle it. As long as you're specific on your bill of materials, we're not going to switch back and forth. With the type of customers we're working with, traceability is critical; I can't avoid traceability.

Shaughnessy: One of our designer friends says that maybe we should have been double- and triple-sourcing parts all along. We probably should have been, but we didn't need to. You just knew the part was there. A lot of companies now say every component must have a backup.

Sciberras: We see a lot more options on a bill of materials today than we have in the past. We have some customers who are real traceability guys who say, "It's got to be this exact part." They've learned to rewrite their design so that they can get them through certifications with multiple AVLs. In the past, they just thought it was a challenge they didn't want to deal with.

Saline has a component engineer on staff. We're able to quickly identify alternates and tell customers what the exact differences are. Their engineers are just as busy as our engineers. If we can give them a really tight, narrowed-down piece of information, there's a higher success rate getting the customer to approve that and keep moving forward.

Johnson: What's a good best practice in this situation for the design team? Should they be specifying as much as possible their own alternates? You just mentioned that you can identify alternates for them. Do you prefer seeing a plan A, an ideal bill of materials, then

you pick the alternates and refer back, or would you prefer the customer to identify the alternates?

Sciberras: A little bit of both. When we're talking about some of the ICs that maybe are programmed, and they've got software that's very specific to their system, how they write that software may need to have some options in it so that it works with multiple manufacturers. At the design stage, when you're writing that software to do it in a way that allows you to be effective, a lot of the companies are outsourcing that software design. They might not have the information that they need to have in order to change it down the road.

> At the design stage, when you're writing that software to do it in a way that allows you to be effective, a lot of the companies are outsourcing that software design.

It requires thinking right up front about how many options there are on my most critical components. Making a good plan at the start is ideal; there are a lot of pieces, and it starts with the design. Then you work with your supplier; what does that communication look like? It will be multiple pieces, not just one magic piece that solves the entire problem.

Johnson: That definitely requires the design team. Dropping down into the more passive parts, how critical is that for them to be second sourcing those for you?

Sciberras: It depends on how critical and able they are to accept alternates. If you are pretty open on caps and resistors, you can give me a specification of what you're looking for, and you can list the manufacturer as "any," and that gives us a lot of room to make decisions for them. It gives us a lot of capacity to give them what they need when they need it, but there are other places.

If you have one AVL on your BOM, that's probably a bad idea to start because I can find you alternates if somebody else makes something. There are some of these very critical components that, right up front, where nobody else makes that exact style. You should probably consider having alternate options, maybe a different layout: "How else can I do this Bluetooth module?"

Johnson: That makes the bill of materials spec particularly important because they can say, "I don't care." For example, a capacitance value and tolerance—it's on them to communicate piece-by-piece to the exact specification you need.

Sciberras: Designers are engaging us earlier in the process than we've seen in the past. We've always been pretty early on in the design phase just because of the length of the relationship. They understand it, but now, because they've been burned, they're willing to engage the contract manufacturer sooner. That's important because we have tools available to us where I can scan their entire bill of materials and say, "Hey, this part is not really recommended for new designs." Sometimes, the designers are not paying attention to those same details.

Johnson: Thanks for speaking with us, Jason.

Sciberras: Thank you. DESIGN007

Test Vehicles for PCB Electrical Material Characterization

Lightning Speed Laminates

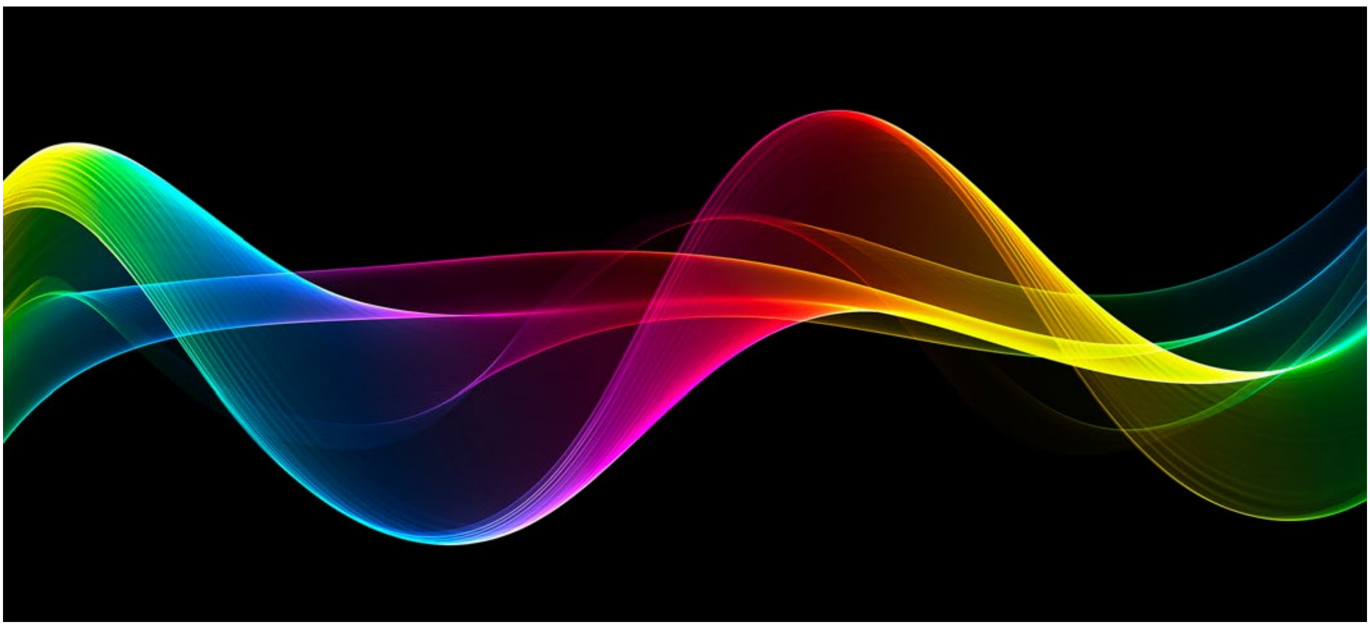
by John Coonrod, ROGERS CORPORATION

Electrical characterization of high frequency circuit materials can be done in a variety of ways. Some characterization methods use a fixture to test the raw dielectric material, while other methods use circuits. The Dk and Df results shown on material datasheets are commonly obtained by a fixture test method. However, when comparing results from circuit evaluations, the Dk results from the fixture tests occasionally do not agree with the Dk results of the circuit tests.

The reasons for these differences vary and are often situation dependent. In general, however, a fixtured test method will not have the same variables which will impact the Dk extraction, as compared to a circuit test. When using a circuit test, the Dk extraction will be affected by various aspects of PCB fabrication processing, such as conductor width control, copper thickness variations, final plated finish, and copper surface roughness.

A typical fixture test method will not have these PCB variations; instead, it may have variations due to entrapped air, fixture alignment variations, electrical coupling differences, and some tolerancing effect due to the machining of the fixture components. Additionally, the electromagnetic fields applied to the material under test can vary across test methods. The different field orientation also can be problematic when evaluating anisotropic circuit material. Most circuit materials used in the PCB industry are anisotropic, meaning that the Dk is not the same on all three axes of the material.

If the electrical characterization to be done is related to a PCB being built in large volume, the best test method and/or best test vehicle will be the one that is most like the actual cir-



cuit being built in volume production. If the electrical characterization is a project that will consider many different types of circuit materials for qualification for future projects, it may be best to use the same fixtured test method for all materials. If the same test method is used and is well understood, the evaluation should be relatively impartial to the materials being tested. However, since different circuit materials have different processing needs for the circuit fabrication process, it may not be a good comparison to evaluate different circuit materials using the same circuit test method.

There are many test vehicle options available for circuit test methods. Some of the more common circuit test vehicles include ring resonators and transmission line circuits with different lengths; antenna patch radiators, 180-degree hybrids, edge-coupled filters, and delay lines are additional options.

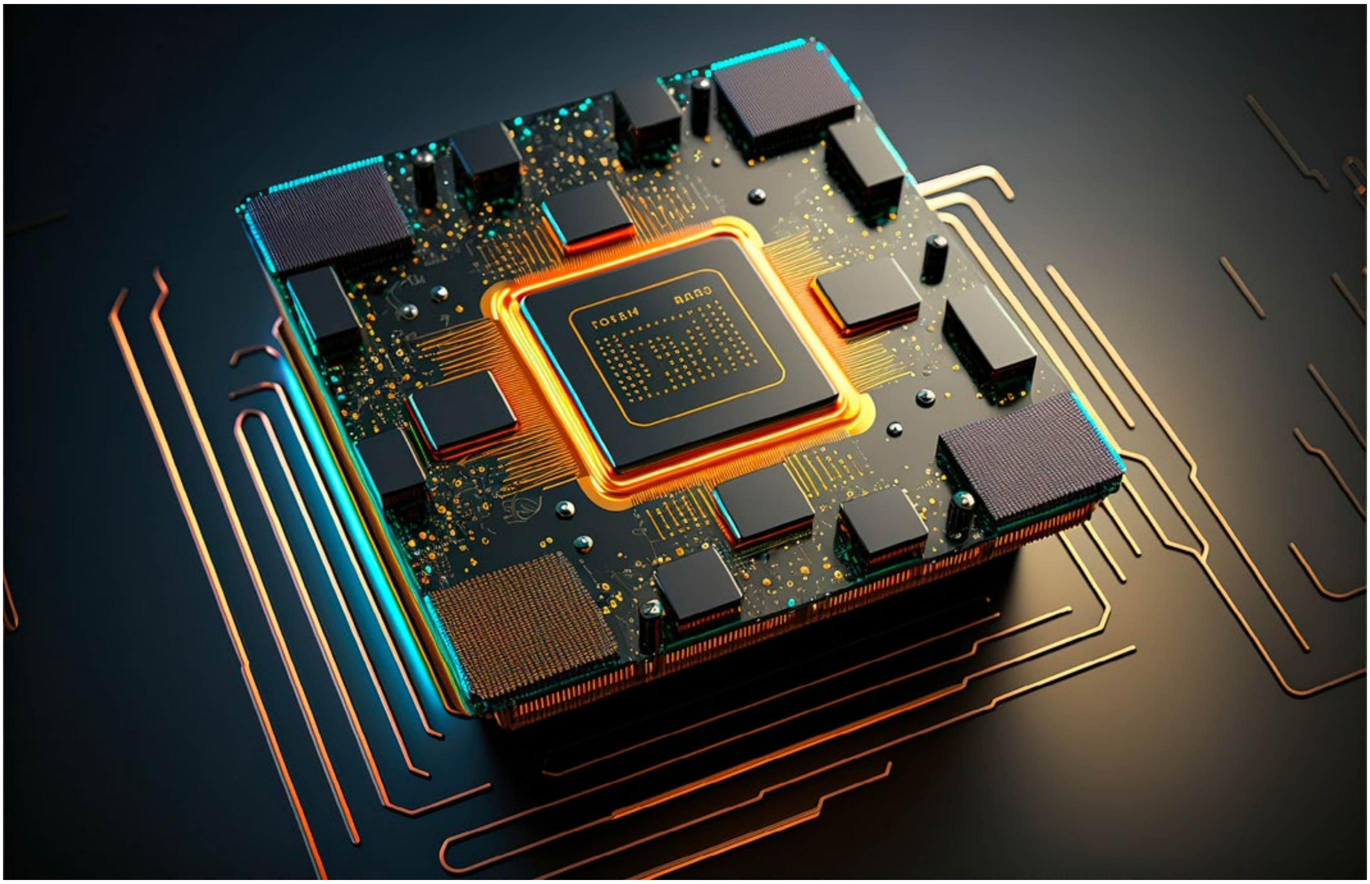
Ring resonators have been used for material characterization for many years and with good success. But over the past several years, and with more applications using millimeter-wave (mmWave) frequencies, obtaining accurate results when using ring resonator test vehicles has become problematic. Wavelength can be a simple way to think about these challenges, as related to lower and higher frequency. At mmWave frequencies, wavelengths are very small, and the propagating wave will be more sensitive to circuit anomalies, which can cause RF disturbances. These small circuit anomalies may be normal for circuit fabrication and may not cause any difference in RF performance when operating at lower frequencies with longer wavelengths.

Many ring resonators are gap coupled; differences in the gap coupling can influence the ring resonator's center frequency, which impacts the Dk extraction accuracy. The RF differences due to the gap are usually related to etching and copper thickness variation. If the same ring resonator design is used for two circuits built at different times and one circuit has thicker copper than the other, the circuit with the thicker copper will have more fields fringing in air than the circuit with thinner copper. Air has a very low Dk and having more fields in air will naturally lower the extraction of the Dk value. The lower Dk result will be related to the circuit fabrication process rather than the circuit material. The conductor trapezoidal shape and its normal variation from circuit to circuit can also impact the RF performance due to affecting the fields in the gap coupled areas. Again, at lower frequencies, these small differences due to coupling will be less significant, but at mmWave frequencies the differences can be substantial.

There are also copper surface roughness effects to consider for Dk extraction. The copper surface in this case refers to the copper surface at the substrate-copper interface as the laminate is made. When the copper is rough, it will slow the wave propagation and alter the RF performance. The copper surface roughness is not perfect and there is normal variation in roughness from batch to batch of the same copper foil. For a microstrip ring resonator, the copper roughness will not be the same for the signal plane as the ground plane, due to this normal variation in roughness. At lower frequencies, the differences in roughness are less significant than at mmWave frequencies. The copper roughness and its variation are other issues to consider for the test vehicle being used at higher frequencies.

There are many test methods and test vehicles that can be used to characterize high frequency circuit materials. Designers should contact the material supplier to inquire how the published Dk was obtained, as well as ask for test method recommendations that are appropriate to the RF design being considered. DESIGN007

John Coonrod is technical marketing manager at Rogers Corporation. To read past columns, click here.



Shrinking Silicon, EMI, and SI

Feature Interview by Andy Shaughnessy
I-CONNECT007

As IC features continue to shrink, the PCB designer's job gets more interesting—signal speeds and rise times are increasing, they're encountering EMI and signal integrity issues once only seen in the RF world.

Dr. Todd Hubing is a longtime EMC instructor, president of LearnEMC, and a professor emeritus of the Electrical and Computer Engineering program at Clemson University. I asked Todd to discuss the challenges that shrinking silicon can present for traditional PCB designers, as well as the opportunities and benefits of smaller chip features.

Andy Shaughnessy: Shrinking silicon is causing an increase in EMI and SI issues for PCB designers and EEs. What sort of problems does shrinking silicon cause?

Todd Hubing: When the features of an IC shrink, transition times tend to be faster, and the high-frequency content of digital signals tends to increase. When newer, faster ICs are used in products that were designed with the older versions of those ICs, those products can suddenly start failing to meet EMC or SI requirements even though there was no nominal change in the design. ICs with smaller features can also be more susceptible to damage by voltage transients, though that largely depends on the mitigation features built into the design.

Shaughnessy: What is the relationship between smaller silicon and EM fields, and what can these designers do to proactively fight EMI and SI?

Hubing: Field coupling directly from an IC is not directly affected by smaller silicon. Field coupling depends more on the currents pulled through the inductance of the lead-frame. Shrinking the features on the silicon can cause those currents to be higher or lower at any given frequency depending on the application.

To protect against future silicon feature-size changes, PCB designers should always proactively control the transition times of any signal that could ultimately be the source of crosstalk or radiated emissions. They shouldn't rely on "slow" devices to meet EMI and SI requirements.

Todd Hubing

Shrinking the features on the silicon can cause those currents to be higher or lower at any given frequency depending on the application.

Shaughnessy: If we add increased rise time and faster signals into this mix, how does material and material selection figure into the equation? Do you think OEMs should select their material, or allow the fab to do so?

Hubing: I believe laminate selection is strictly an SI issue, and then only for the fastest digital signals. Choosing whether to let the fabricator select the laminate depends on the many factors, but, in my opinion, that decision is unlikely to be impacted by silicon shrinkage.

Shaughnessy: What advice would you give designers and EEs who are starting to have EMI and SI issues because of tinier silicon and increased speeds?

Hubing: Control your transition times. Use a series resistor for capacitive loads. Use controlled-transition-time logic for matched loads.

Shaughnessy: Is there anything else you'd like to add?

Hubing: Generally—again, in my opinion—shrinking silicon is having a positive impact on both EMI and SI. More is being done with less power and smaller package sizes. If transition times are proactively controlled, newer devices tend to be better options for both EMC and signal integrity.

Shaughnessy: Thanks for your time, Todd.

Hubing: Thank you, Andy. DESIGN007

MilAero007 Highlights

Northrop Grumman, NASA to Shape Future ▶

Integration of Uncrewed Autonomous Systems Northrop Grumman Corporation is collaborating with NASA to develop and test solutions for integrating large, uncrewed aircraft systems into the National Airspace System (NAS).

Heart Aerospace Selects Siemens Xcelerator for New Electric Airplane ▶

Electric airplane maker Heart Aerospace has selected Capital from the Siemens Xcelerator portfolio to support its E/E system design, development, and certification of zero emission electric aircraft. The Capital tool's strong compliance functionality can help Heart Aerospace to leverage automation and digital data continuity to facilitate faster regulatory compliance.

Raytheon Intelligence & Space Awarded Missile Track Custody Development Contract ▶

Raytheon Intelligence & Space has been awarded a prime contract to develop a prototype Missile Track Custody system for the U.S. Space Force. MTC is the service's first Medium Earth Orbit missile tracking system.

US, Japan Sign Space Collaboration Agreement at NASA Headquarters ▶

During an event hosted by NASA Administrator Bill Nelson and Deputy Administrator Pam Melroy at the agency's Headquarters in Washington recently, representatives from the United States and Japan gathered to sign a pact that recognizes a mutual interest in peaceful exploration.

Satcom Direct Installs Prototype Terminal for Inmarsat's Jet ConneX Inflight Broadband Service ▶

Satcom Direct, the business aviation solutions provider, has completed the installation of the first Plane Simple Ka-band antenna on the company's SD Gulfstream G550.

DARPA Kicks Off JUMP 2.0 Consortium Aimed at Microelectronics Revolution ▶

DARPA, along with the Semiconductor Research Corporation (SRC) and industry and academic stakeholders, is kicking off the Joint University Microelectronics Program 2.0 (JUMP 2.0), which expands on the original collaboration aimed at accelerating U.S. advances in information and communications technologies.

Heat Shield Inspections Underway on Artemis I Orion Spacecraft ▶

Inside the Multi-Payload Processing Facility at NASA's Kennedy Space Center in Florida, engineers and technicians conduct inspections of the heat shield on the Orion spacecraft for the Artemis I mission.

IPC Welcomes New Director of North American Government Relations ▶

IPC announces the addition of Jeffrey Goldberg as director of North American government relations to its staff at IPC's offices in Washington, D.C. In this role, Goldberg will help lead IPC's federal advocacy work in collaboration with IPC's Government Relations Committee and IPC's Vice President of Global Government Relations Chris Mitchell.


The RF Specialists
• ISO 9001:2015 Certified
• AS 9100 Rev D Certified
• MIL-PRF-31032-C Qualified
• MIL-PRF-55110 Qualified
• ITAR Registered

DFM 101
Final Finishes: OSP

Article by Anaya Vardya
AMERICAN STANDARD CIRCUITS

Introduction

One of the biggest challenges facing PCB designers is not understanding the cost drivers in the PCB manufacturing process. The next final finishes to discuss in this series is OSP. As with all surface finishes there are pros and cons with the decision of which to use. It is a combination of application, cost, and the properties of the finish. OSP is RoHS-compliant as there is zero lead content in the finish.

Final Finishes

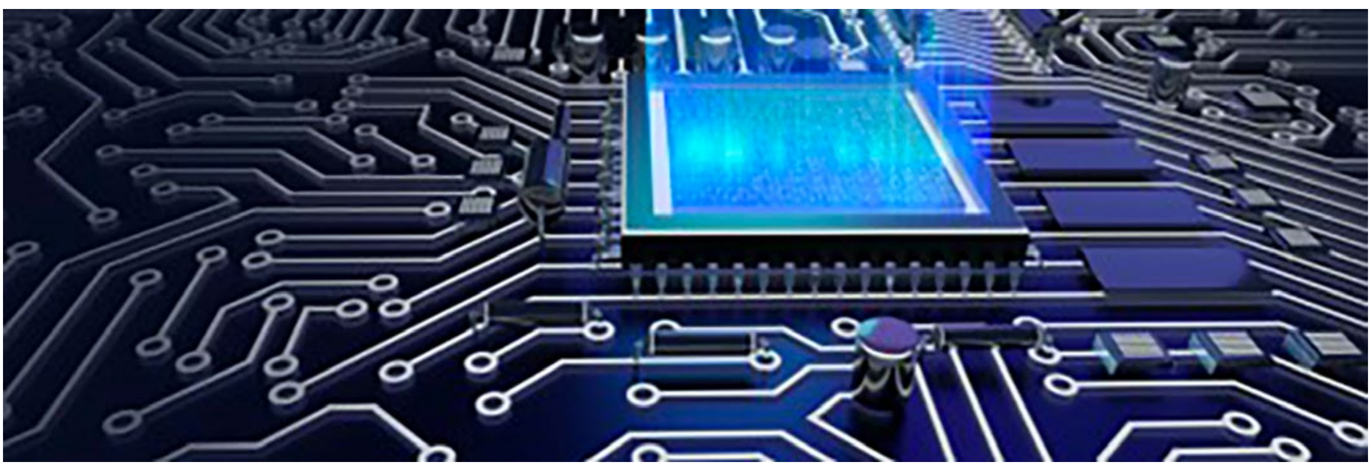
OSP (Organic Solderability Preservative)

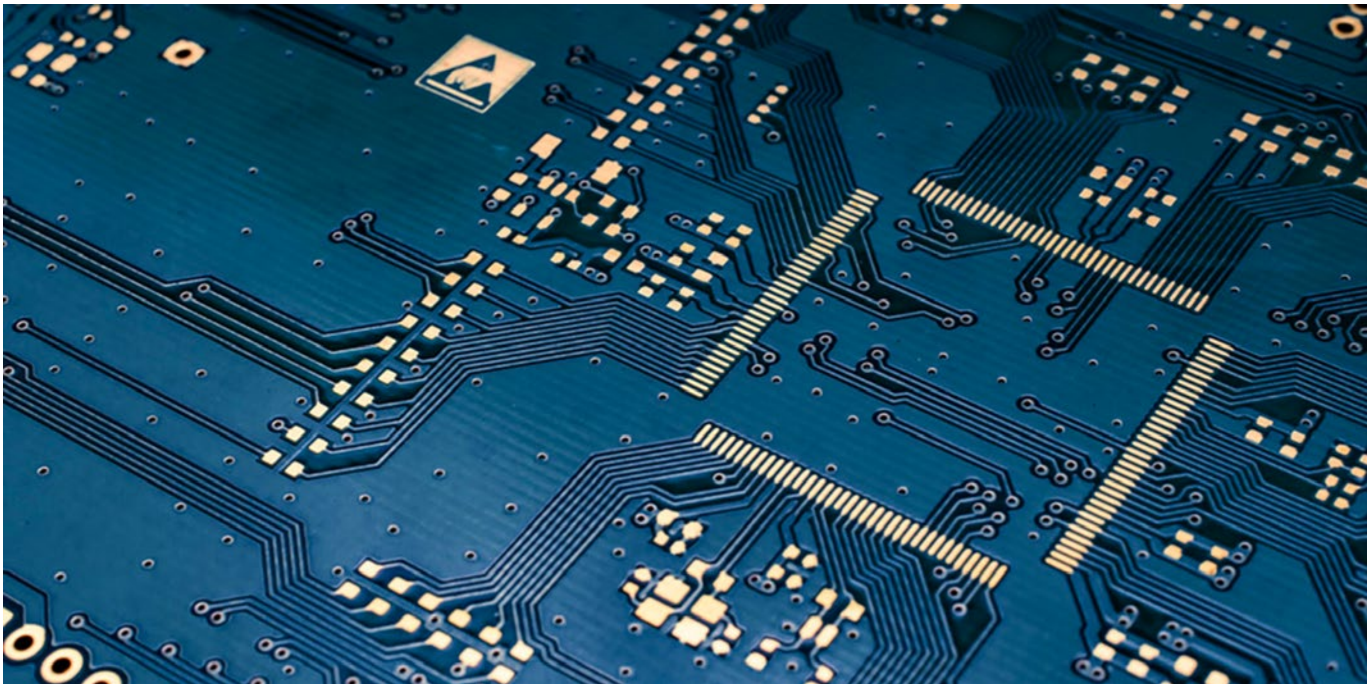
OSP is a thin organic coating, typically 5.9 to 11.8 μin [0.15 to 0.30 μm] thick, designed to prevent the oxidation of copper to maintain solderability over an extended period. Compared with other surface finish technologies, OSP is somewhat different. It uses a chemical process to produce an organic film on the bare copper surface which acts as a barrier to copper oxidation. OSP is organic, not metallic, and its cost is lower than most surface treatment technologies.

As with all surface finishes, the primary purpose is to protect the solderable surfaces on the PCB from oxidation and to aid in assembly soldering. This process coats a very thin coating of an organic material that inhibits copper oxidation. It is so thin that it is nearly impossible to see and measure. The organic material is removed by the assembly flux. Boards that have been OSP coated will have bright copper pad coloration. OSP is specially designed for mixed metal applications, such as electroless nickel immersion gold (ENIG). The OSP selectively deposits on copper while leaving gold connectors or metallic heat sinks free of contamination.

There are a couple of common OSP finishes widely used in the industry:

ENTEK CU-56: This is used for assemblies that will only go through a single reflow process. This finish is not being utilized much anymore





because of the use of mixed technology boards (SMT and through-hole).

ENTEK CU-106A: This is the most prevalent version primarily due to the ability to survive multiple thermal assembly operations. PCBs that have multiple surface finishes can use the CU-106A(X) finish.

Pros and Cons of OSP

Pros

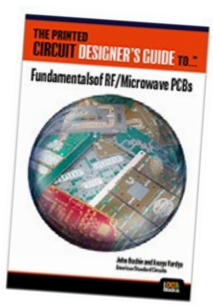
- Inexpensive and typically used in fine pitch surface mount designs
- Environmentally friendly
- Provides a coplanar surface
- Lead-free (Pb-free)
- Mixed metal applications

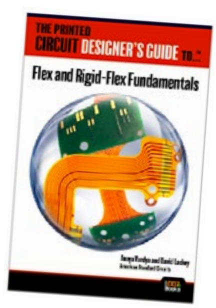
Cons

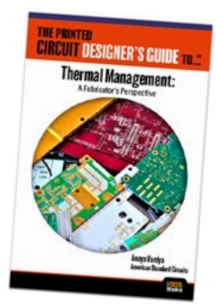
- Press fit pin insertion
- Handling sensitive
- Limited shelf life
- In-circuit test (ICT) difficulties
- Exposed copper after assembly
- Multiple reflow cycles and solder paste removal are problematic

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

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







Flex007 Highlights

Designers Notebook: Flexible Circuits for In-line SMT Assembly Processing ▸

Incorporating surface mount components directly onto a flexible circuit's etched copper land patterns is not unlike the assembly process used for rigid circuit boards. To maximize robotic assembly efficiency and increase throughput of the flexible circuit, however, the circuit design engineer will need to provide a format that includes all features required for in-line assembly processing.

MKS Continues to Increase Performance for CapStone Flex PCB Drill System ▸

MKS Instruments, Inc., a global provider of technologies that enables advanced processes and improve productivity, at IPC APEX 2023 announced the launch of SPOT ON™, an optional feature upgrade available for the CapStone™ laser via drilling system. SPOT ON increases system uptime through early warning of spot quality deviation and increases the CapStone's yield performance by reducing leading causes of via quality degradation.

Rogers Appoints Senior Vice President of Global Operations and Supply Chain ▸

Rogers Corporation announced the appointment of Larry Schmid as Senior Vice President, Global Operations and Supply Chain. In this role, Schmid will direct the operational and supply chain organizations across Rogers' global operations, including in the U.S, China, Belgium, Germany, England, South Korea, and Hungary. He will also implement Rogers' ongoing operational excellence initiatives to improve financial performance.

Flexible Thinking: A Patently Innovative Resolution ▸

The month of January is named for the Roman god Janus, a two-faced deity whose role is to watch over doorways, the comings and goings of individuals, and the passage of time. In January, individuals often make resolutions to themselves; though following through on these resolutions may be short-lived, a new year often prompts us to be mindful of the need to make some improvements in our lives—whether personal or professional.

IPC Debuts First Issue of *IPC Community* at IPC APEX EXPO 2023 ▸

On January 23, at IPC APEX EXPO 2023, IPC, in partnership with IPC Publishing Group, launched an industry-specific quarterly publication (a digital publication with a special print edition for show participants), "IPC Community." The publication celebrates member success while sharing the important work being done within the association to better serve its members and the global electronics manufacturing community.

U.S. Debut of Ventec's New Range of PCB Laminates at IPC APEX EXPO 2023 ▸

Ventec's expanding range of PCB materials is designed to meet the technology needs of the PCB manufacturing industry. With laminates specifically developed for use in demanding conditions, Ventec's products provide reliability and high-performance, supported by Ventec's fully controlled and managed global supply chain to ensure dependable delivery even in an unpredictable worldwide landscape.

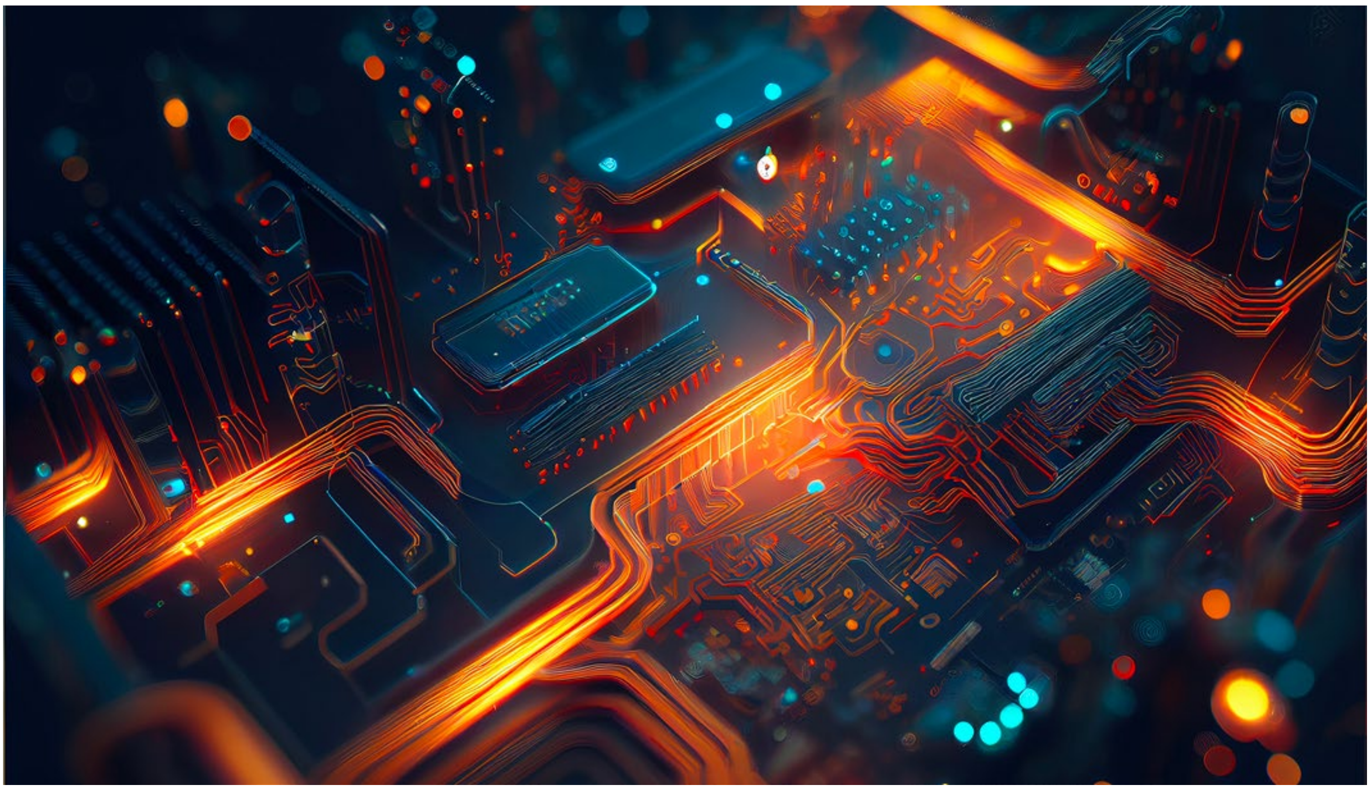
A Once and Future Idea

Flexible Thinking

by Joe Fjelstad, VERDANT ELECTRONICS

Electronic assemblies are typically comprised of a mix of discrete resistors, capacitors, inductors, and the like, along with numerous integrated circuit chips, each chip having a certain function or range of different functions. In such assemblies, there are also several different connectors and/or sockets that allow for the assembly to be connected to other assemblies. Making interconnections between and among these many and various active and passive devices is the job of the circuit designer. To date, a broad range of IC packaging and electrical interconnection techniques have been used in such assemblies, especially at the higher end.

As folks try to extend the runway of Moore's Law, these include various multichip module (MCM) structures along with system-in-package (SiP) offerings, both precursors to the currently much talked about range of heterogeneously integrated packaging solutions found in virtually every electronics engineering magazine. These package assembly devices are still in need of and typically mounted onto an interconnection substrate, most commonly a printed circuit board. Actual interconnection continues to be carried out using a soldering process, though efforts have been underway for some time (by myself and others) to eliminate soldering




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

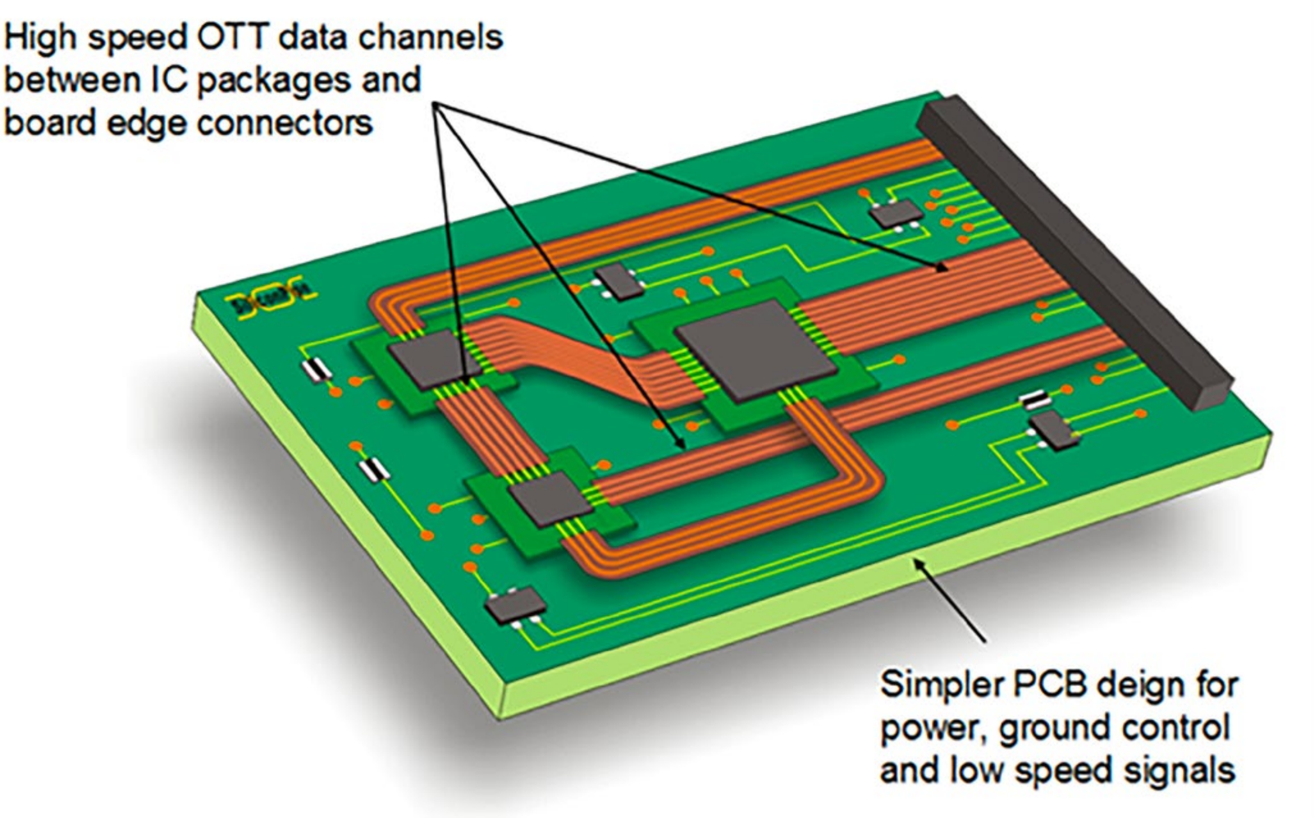


Figure 1: An "ahead of the curve" 3D interconnection concept, the OTT concept proposed partitioning circuit design to create both a simpler PCB and more controllable high-speed interconnections between ICs. In 2005, the method illustrated was shown capable to data rates of greater than 20 Gbps over distances of greater than 75 cm through two connectors, while meeting specified bit error rate requirements.

in favor of more environmentally friendly and less costly methods.

That aside, with ongoing advances in semiconductor technology design, packaging and interconnection innovations, and improved manufacturing technologies, the feature size of integrated circuits has continually decreased. Correspondingly, the operating speed of integrated circuits has increased with each new generation. Many in the electronics industry hope and expect that this trend will continue into the foreseeable future, though most acknowledge that there are limits to the materials and physics of semiconductor technology. One thing that is well known and understood is that for the highest operating speed, it is desirable to have short interconnection paths between electronic elements, both on an individual integrated circuit and between the terminals of interconnected chips, whether in an MCM, SiP, or discrete package.

The challenge of design is that there are often conflicting requirements. What is an advantage to one area of production can be a detriment to another, or even negatively impact the performance of the design itself. One such example is pad pitch. As termination pitches continue to shrink, from an assembler's perspective it remains desirable to have a large pad pitch, for a larger pad pitch will improve yield and reduce the overall cost of assembly at the board level. Unfortunately, the large pads typically mean a larger package and a greater capacitive loading at the termination, thus a larger PCB and greater cost for the elements to be interconnected.


Hmm, what is recommended **minimum distance for copper to board edge?**

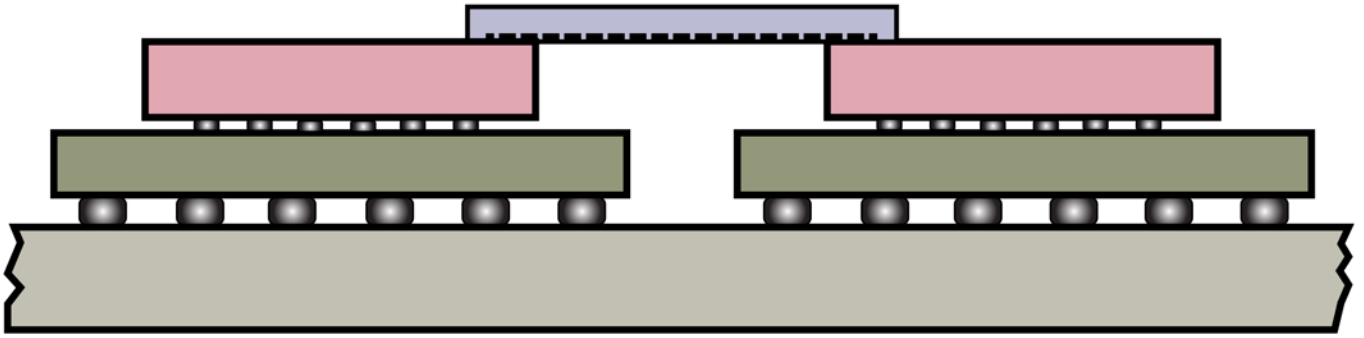


Figure 2: In contrast to the earlier approach to OTT design, the current generation can be used to interconnect ICs having TSV technology from the backside, further improving performance and allowing for the interconnection of two or more chips within a package or between packages.

There are, however, ways to reduce the I/O requirement at the board level; by using a flexible circuit to make interconnections to terminations on the upper surfaces of the chip packages, high-speed signals can be lifted out of the PCB into the space above the board. I described this approach at my previous start up, Silicon Pipe, in the early 2000s, calling it "OTT" technology, meaning off (or over) the top.

OTT technology was invented to circumvent the problem of dealing with noise and crosstalk in the PCB. As the march into the third dimension of electronic interconnection using heterogeneous interconnection solutions continues, the OTT solution was among the first proposed 3D interconnection solutions. Figure 1 offers a graphical representation of that radical approach.

The basic concept of OTT is now being conceptually exploited as "bridge" technology and employs through-silicon via (TSV) technology that allows chips to be interconnected—not just vertically (stacked) but also laterally between chips. This is illustrated in Figure 2. The advantages are obvious: This layout ensures the shortest path routing of critical signals and clocking between chips.

In summary, the OTT concept proposed by Silicon Pipe some 20 years ago is finally finding practical application in today's heterogeneous integration tool chest, offering an alternative means to getting higher performance while staying within the boundaries marked by the limits of physical science and package assembly technology. Some ideas are just born too early and need the future to catch up to them. **DESIGN007**

Joe Fjelstad is founder and CEO of Verdant Electronics and an international authority and innovator in the field of electronic interconnection and packaging technologies with more than 185 patents issued or pending. To read past columns or contact Fjelstad, click here. Download your copy of Fjelstad's book Flexible Circuit Technology, 4th Edition, and watch his in-depth workshop series "Flexible Circuit Technology."

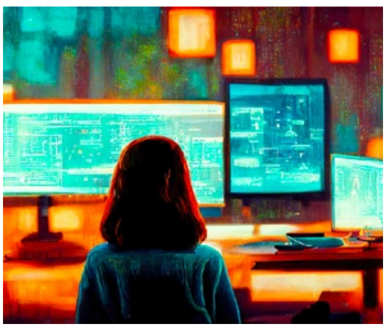
TOP TEN EDITOR'S PICKS

Connect the Dots: The Recipe for Customer Service Success

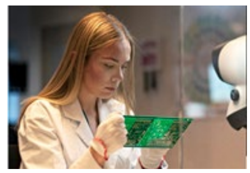
At Sunstone we concentrate on the current best practices for keeping our employees safe while keeping quality work as a top priority, and providing a remote work option for our employees has been one such effective strategy. It brings me great joy to report that, so far, we have effectively maintained both the health and employment of all our employees. As I was recently reflecting on this, I decided to ask Al Secchi, our global customer support and sales manager, what he has learned professionally these past two years.

Altium Focuses on Design Education

Altium keeps its eyes on the designers of the future. The company has been working with colleges and universities for years, providing free seats of Altium Designer for the next generation of PCB designers and design engineers. They offer a variety of other educational programs as well, including Upverter classes and a design competition that aims to address environmental change. Here, Rea Callender, Altium's VP of education, discusses its educational programs.



Fresh PCB Concepts: Designing Controlled Impedance PCBs



Controlled impedance traces are necessary on some PCBs for high-speed signal transmission. The goal with controlled impedance traces is to design the proper propagation delay into the trace. On boards with an antenna, impedance matching is required for a reliable signal. Other boards may have components that require a specific propagation delay in order to perform calculations.

The Battle of the Boards

Last year, IPC held its first-ever design competition at IPC APEX EXPO in San Diego. PCB designers from around the world competed in a series of heats during the months before the show, culminating in a showdown on the show floor between the top three finalists. This year, the competition is back for its sophomore year. I asked Patrick Crawford, manager of design standards and related programs for IPC, to "layout" the details on the design contest.

Scaling Beyond Silicon

Technology has always invoked radical changes, but unlike today, there used to be one major revolutionizing technology trend at a time. The world is becoming increasingly connected, more automated and more intelligent, driven by generational drivers—hyperscale computing, 5G, artificial intelligence and machine learning (AI/ML), industrial IoT (IIoT), and autonomous vehicles.

Tim's Takeaways: Threading the Needle Through Advanced Packaging

When I first started designing circuit boards, components were less dense and more generic in both size and shape. I remember being told to leave enough room at the top of my through-hole 14-pin DIPs for a bypass cap. When I asked for the size, shape, or part number, I was told that it wasn't important; I just needed to create a generic part with a 0.4" x 0.1" rectangle using 62-mil pads spaced 300 mils apart from each other.

Advanced Packaging Means Advanced Routing Issues


Kris Moyer

In today's ever-shrinking world of electronics designs, the use of BGA parts with very fine pitch features is becoming more prevalent. As these fine-pitch BGAs continue to increase in complexity and user I/O (number of balls), the difficulty of finding escape routes and fan-out patterns increases.

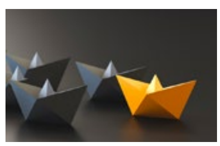
Keysight Introduces Signal Integrity Simulation Software for Hardware Engineers

Keysight Technologies, a leading technology company that delivers advanced design and validation solutions to help accelerate innovation to connect and secure the world, introduces the Electrical Performance Scan (EP-Scan), a new high-speed digital simulation tool that supports rapid signal integrity (SI) analysis for hardware engineers and printed circuit board (PCB) designers.

Freedom CAD Services Hire Chief Strategy Officer

Dan Amiralian will be responsible for building the capabilities to fulfill the requirements of turnkey electronic product development, by augmenting Freedom's 20 years of complex PCB design experience. Dan has been in the electronics industry for over forty years predominately in EMS market. His prior employments include Efficiency Products International, Fairchild, Cybermation, Distron, Virtex, and Bedrock Automation. He was president and owner of New Age EMS before selling the company in 2016.

Everyone Wants Change: Who Wants to Lead the Way?



Nolan Johnson recently met with Alun Morgan, technology ambassador at Ventec, and Ventec COO Mark Goodwin to discusses the industry's determination to cling to outdated processes and standards, and some potential consequences. To maintain efficiency and keep pace with the market's newest entries in Asia, Alun and Mark believe that legacy companies in the West must be open to challenging conversations that will require questioning old practices.

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

THERMAL SYSTEMS

Sales Engineer SMT North Mexico

Rehm Thermal Systems, a leading German manufacturer of reflow soldering systems with convection or condensation and drying and coating systems, has produced energy-efficient manufacturing equipment for the electronics and photovoltaics industry since 1990. We also offer tailor-made applications related to the soldering, coating and hardening of modules.

Responsibilities:

- This position is responsible for expanding our customer network and maintaining existing customer relationships in the Northeast Mexico region. The Sales Engineer would work closely with the German headquarters and the General Manager Rehm Mexico to implement the sales strategy.
- A candidate's proximity to Monterrey, Mexico, is a plus.

Qualifications:

- An Engineering degree or comparable qualification with a strong technical background is required.
- Sales-oriented attitude, good communication skills and willingness to travel frequently within Mexico is essential.

We offer innovative products, a great dynamic work environment and exciting training opportunities in our German headquarters.

To learn more about Rehm Group please visit our website at www.rehm-group.com.

Please send resumes to: Mr. Luis Garcia at luis.garcia@rehm-group.com.

apply now

The Test Connection, Inc.

Sr. Test Engineer (STE-MD)

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

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

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

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

apply now

Career Opportunities

Europe Technical Sales Engineer

Taiyo is the world leader in solder mask products and inkjet technology, offering specialty dielectric inks and via filling inks for use with microvia and build-up technologies, as well as thermal-cure and UV-cure solder masks and inkjet and packaging inks.

PRIMARY FUNCTION:

1. To promote, demonstrate, sell, and service Taiyo's products
2. Assist colleagues with quotes for new customers from a technical perspective
3. Serve as primary technical point of contact to customers providing both pre- and post-sales advice
4. Interact regularly with other Taiyo team members, such as: Product design, development, production, purchasing, quality, and senior company managers from Taiyo group of companies

ESSENTIAL DUTIES:

1. Maintain existing business and pursue new business to meet the sales goals
2. Build strong relationships with existing and new customers
3. Troubleshoot customer problems
4. Provide consultative sales solutions to customer's technical issues
5. Write monthly reports
6. Conduct technical audits
7. Conduct product evaluations

QUALIFICATIONS / SKILLS:

1. College degree preferred, with solid knowledge of chemistry
2. Five years' technical sales experience, preferably in the PCB industry
3. Computer knowledge
4. Sales skills
5. Good interpersonal relationship skills
6. Bilingual (German/English) preferred

To apply, email: BobW@Taiyo-america.com with a subject line of "Application for Technical Sales Engineer".

apply now

IPC Instructor
Longmont, CO

This position is responsible for delivering effective electronics manufacturing training, including IPC certification, to adult students from the electronics manufacturing industry. IPC Instructors primarily train and certify operators, inspectors, engineers, and other trainers to one of six IPC certification programs: IPC-A-600, IPC-A-610, IPC/WHMA-A-620, IPC J-STD-001, IPC 7711/7721, and IPC-6012.

IPC instructors will primarily conduct training at our public training center in Longmont, Colo., or will travel directly to the customer's facility. It is highly preferred that the candidate be willing to travel 25–50% of the time. Several IPC certification courses can be taught remotely and require no travel or in-person training.

Required: A minimum of 5 years' experience in electronics manufacturing and familiarity with IPC standards. Candidate with current IPC CIS or CIT Trainer Specialist certifications are highly preferred.

Salary: Starting at $30 per hour depending on experience

Benefits:

- 401k and 401k matching
- Dental and Vision Insurance
- Employee Assistance Program
- Flexible Spending Account
- Health Insurance
- Health Savings Account
- Life Insurance
- Paid Time Off

Schedule: Monday thru Friday, 8–5

Experience: Electronics Manufacturing: 5+ years (Required)

License/Certification: IPC Certification–Preferred, Not Required

Willingness to travel: 25% (Required)

apply now

Career Opportunities

CHEMCUT

Technical Sales Manager

Objectives

Provide sales leadership and management for a regional sales territory. Responsible for retaining current customers as well as developing and attracting new customers and markets. Responsible for selling current and new products, keeping abreast of new technologies, market trends, and customer product needs.

Essential Functions and Responsibilities

- Develop and service assigned geographic region
- Actively and consistently seek new customers
- Visit customers and potential customers to develop relationships, deliver sales presentations, follow up on leads, and close sales
- Provide technical support and product recommendations in person, by email, and phone
- Manage major accounts; establish long-term, ongoing relationships with key individuals
- Provide feedback to Chemcut as well as sales peers regarding competition, pricing, and marketing opportunities

Qualifications

- Bachelor's degree in mechanical, electrical, chemical engineering or related fields
- 3-5 years of field sales experience with technology driven industrial products
- Well-developed sales and customer relations skills
- Ability to make decisions and evaluations to determine customer needs
- Ability to travel up to 50% of the time
- Excellent oral and written communication skills
- Knowledge of target market industries

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

apply now

Prototron Circuits

Sales Representatives

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

Reasons you should work with Prototron:

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

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

apply now

Career Opportunities

Career Opportunities

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

EMA Design Automation®

Technical Marketing Engineer

EMA Design Automation, a leader in product development solutions, is in search of a detail-oriented individual who can apply their knowledge of electrical design and CAD software to assist marketing in the creation of videos, training materials, blog posts, and more. This Technical Marketing Engineer role is ideal for analytical problem-solvers who enjoy educating and teaching others.

Requirements:

- Bachelor's degree in electrical engineering or related field with a basic understanding of engineering theories and terminology required
- Basic knowledge of schematic design, PCB design, and simulation with experience in OrCAD or Allegro preferred
- Candidates must possess excellent writing skills with an understanding of sentence structure and grammar
- Basic knowledge of video editing and experience using Camtasia or Adobe Premiere Pro is preferred but not required
- Must be able to collaborate well with others and have excellent written and verbal communication skills for this remote position

EMA Design Automation is a small, family-owned company that fosters a flexible, collaborative environment and promotes professional growth.

Send Resumes to: resumes@ema-eda.com

apply now

Career Opportunities

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

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

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

ARLON ELECTRONIC MATERIALS

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

- Engineering
- Quality
- Various Manufacturing

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

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

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

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

apply now

Career Opportunities

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

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

Career Opportunities

Career Opportunities

U.S. CIRCUIT

Plating Supervisor

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

Competitive benefits package.
Pay will be commensurate
with experience.

Mail to:
mfariba@uscircuit.com

apply now

APCT
Passion | Commitment | Trust

APCT, Printed Circuit Board Solutions: Opportunities Await

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

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

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

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

apply now

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

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

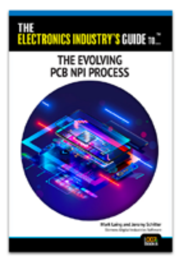
This book provides the reader with a clearer picture of what to know when selecting which material is most desirable for their upcoming products and a solid base for making material selection decisions. **Get your copy now!**

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: **SHELLY STEIN**

COVER IMAGE: **ADOBE STOCK © ИГОРЬ ЖУКОВ**

DESIGN007 MAGAZINE

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

February 2023, Volume 12, Number 2
DESIGN007 MAGAZINE is published monthly
by IPC Publishing Group, Inc., dba I-Connect007

ADVERTISER INDEX

Problems solved!
PCB007 MAGAZINE
NOVEMBER 2022
PCB007 MAGAZINE
DECEMBER 2022
PCB007 MAGAZINE
JANUARY 2023
SMT007 MAGAZINE
Cybersecurity & Counterfeiting
START PLANNING NOW WITH OUR IPC APEX EXPO PRESHOW COVERAGE
DESIGN007 MAGAZINE
DECEMBER 2022
DESIGN007 MAGAZINE
JANUARY 2023
the ADVANCED future
DESIGN007 MAGAZINE
FEBRUARY 2023
SMT007 MAGAZINE
JANUARY 2023
SMT007 MAGAZINE
FEBRUARY 2023
It's All About Strategy
SUBSCRIBE NOW