DECEMBER 2019

I-Connect007

DESIGN007 MAGAZINE

What You Need to Know

FLEX007
SEE INSIDE!

OUR LIBRARY IS ALWAYS OPEN

Registered members of **my I-Connect007** have 24/7 access to our ever-growing library of eBooks, which now includes a wide range of topics from DFM to Complex PCBs.

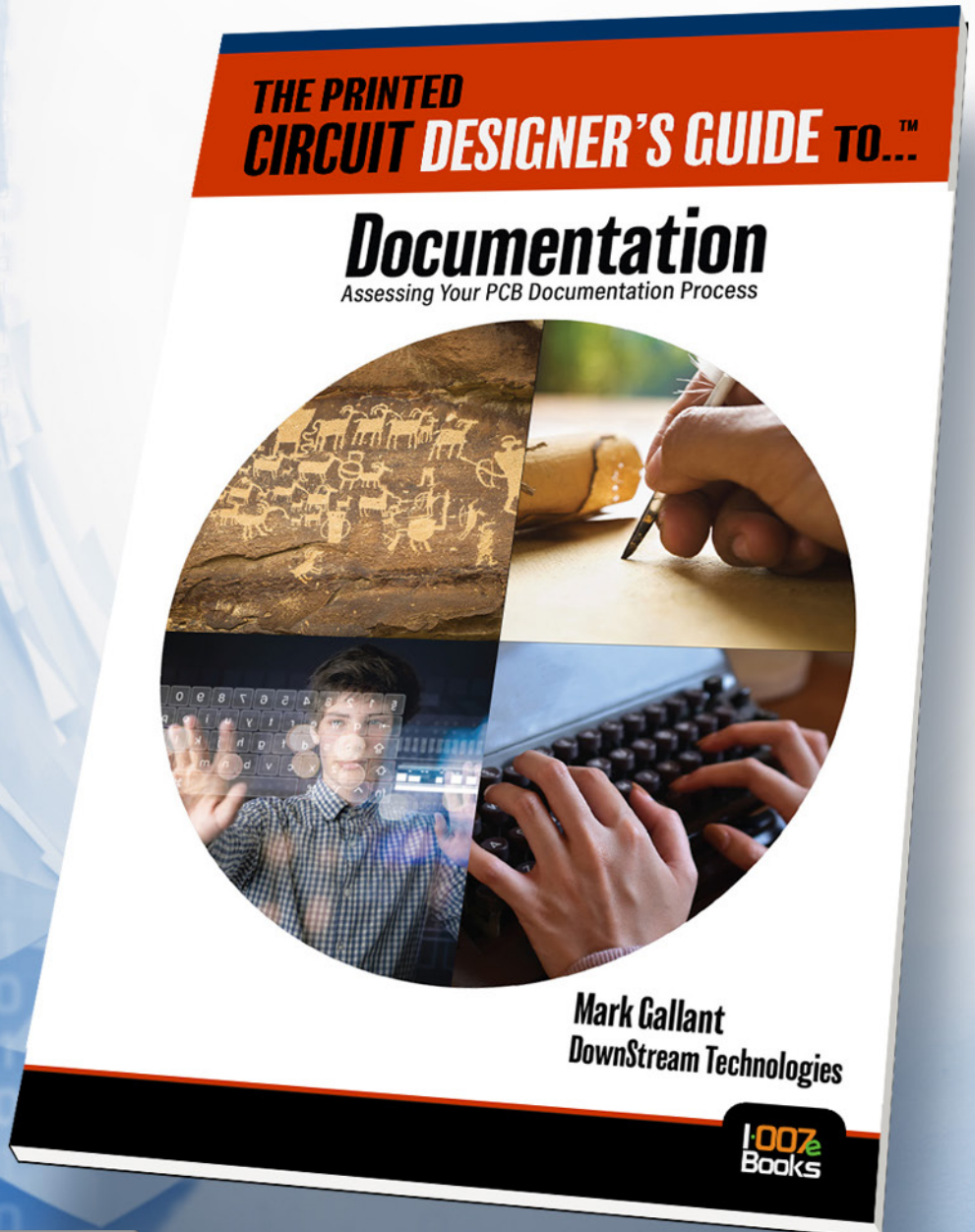
Enrich your mind and expand your knowledge! Download these popular titles with just a click.

They're pretty awesome!

I-007eBooks.com

Your life just got better.

This book will teach you how to save time and money, reduce frustration, and make your job easier!



I-007e Books

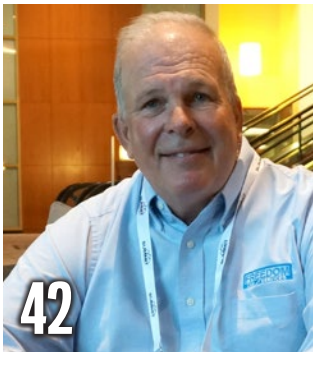
I-007ebooks.com/doc

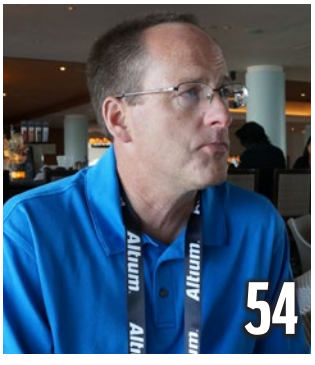
DOWNLOAD NOW

DESIGN007 MAGAZINE

What You Need to Know

Think about this for a minute: If someone asked you what you need to know to be a great PCB designer, what would you say? Where would you even begin? This month, we asked our expert contributors to discuss what they believe other designers and design engineers need to know as we move forward into 2020.

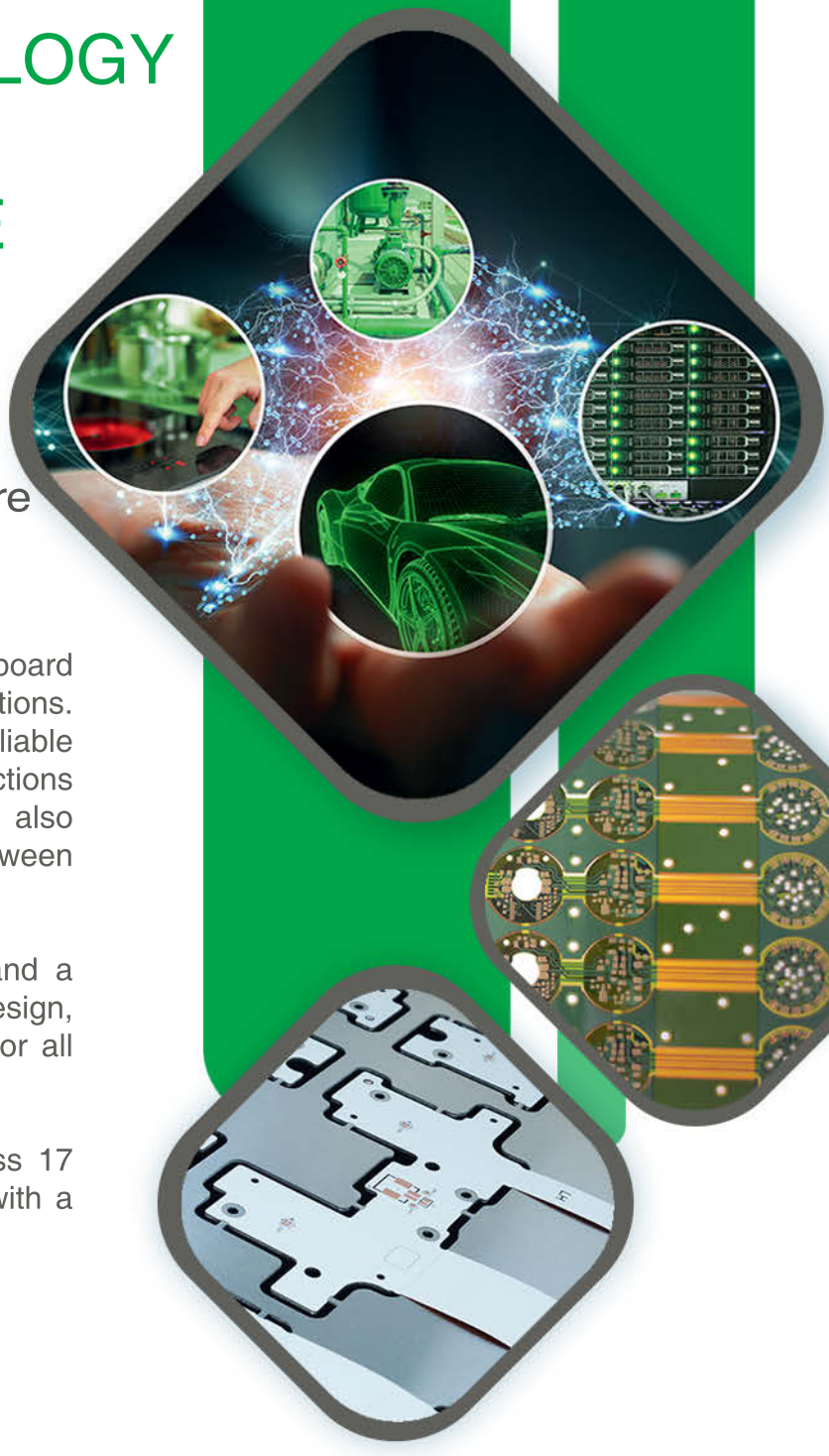




FEATURES:

DRIVING TECHNOLOGY TO NEW EXCELLENCE

Multilayer, HDI, Bendable,
Flex/ Rigid Flex,
High Frequency PCBs and more



ABOUT CML

CML are the leaders for printed circuit board (PCB) manufacturing and sourcing solutions. CML's mission "We care about reliable connections," apart from the reliable connections of our PCBs, paramount importance is also placed on quality and the relationship between CML, our customers and suppliers.

CML provides a wide product portfolio and a range of services from consultancy, design, engineering, manufacturing and logistics for all markets - convenient and reliable.

CML has a global network, located across 17 international cities and a local presence with a multilingual and culturally diverse team.

Contact our USA office today

Chris Minard
chris.minard@cml-america.com
+1 650 269 4823

For information on our other offices, please visit our website:
http://www.cml-globalsolutions.com

CML
More than a manufacturer

DESIGN007 MAGAZINE

SHORTS:

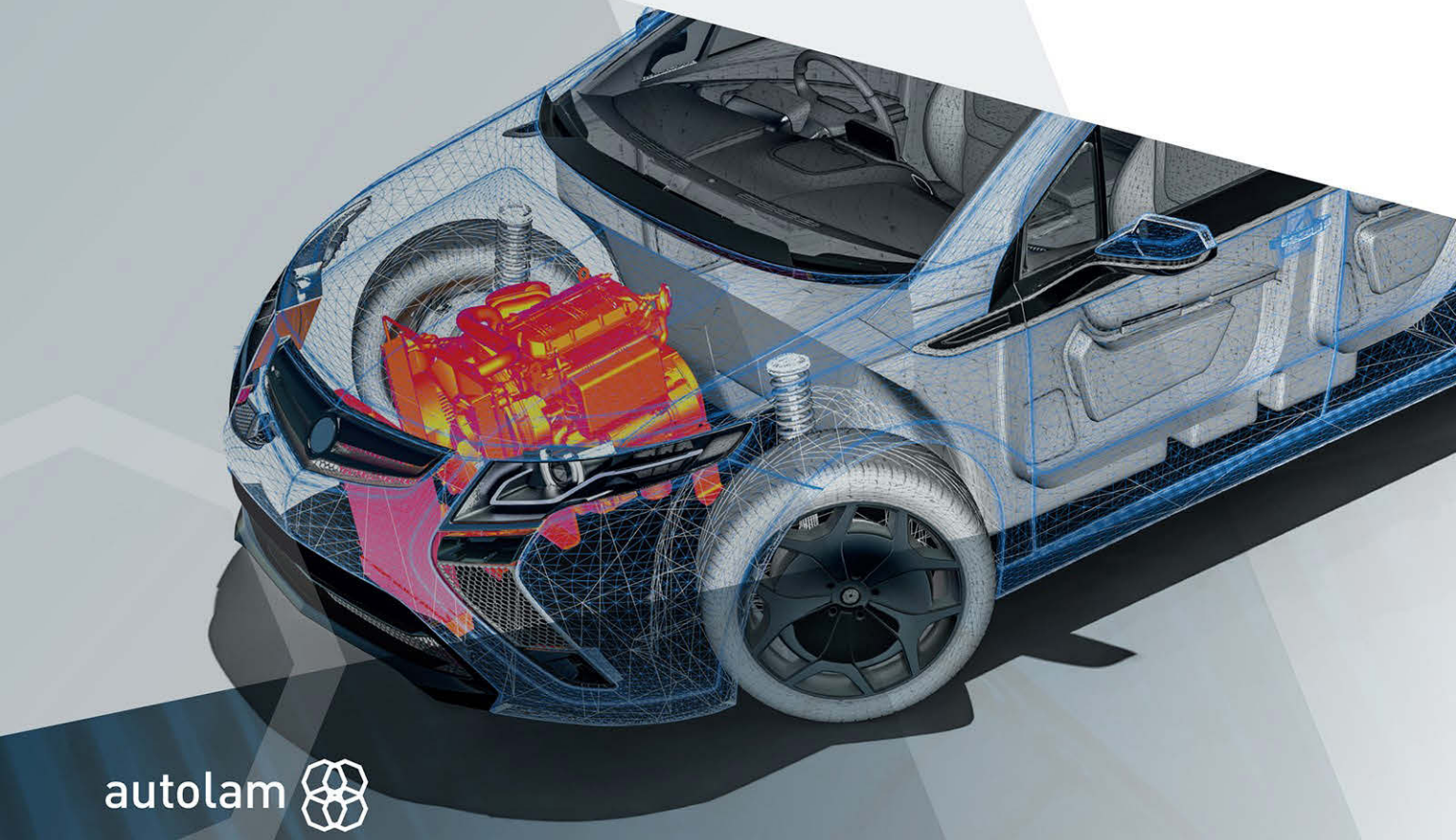
DEPARTMENTS:

COLUMNS:

HIGHLIGHTS:

ventec
INTERNATIONAL GROUP
騰輝電子

autolam: Base-Material Solutions for Automotive Electronics



autolam

Automotive electronics technologies are evolving at an increasing rate. Paying attention to the properties of materials at the substrate level is the first step towards achieving the most stringent performance targets of today's automotive manufacturers. autolam offers the solutions demanded by the diverse and unique requirements of automotive applications today and in the future.

IATF 16949:2016 CERTIFICATION

venteclaminates.com

Visit us: DESIGNCON® 25TH ANNIVERSARY 2020 WHERE THE CHIP MEETS THE BOARD

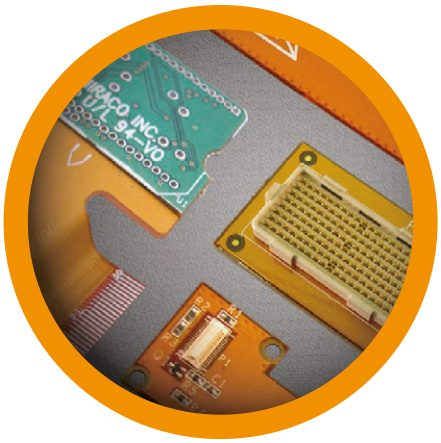
APEX EXPO® IPC 2020

FLEX007

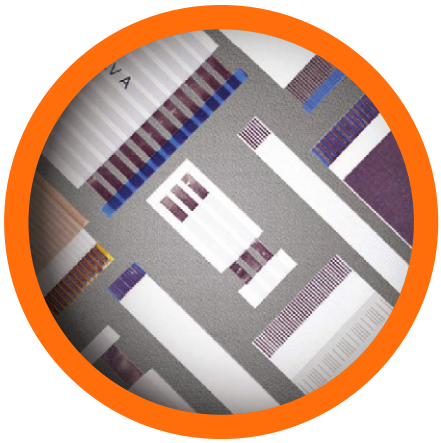
Focusing on Flex

There's so much innovation going on in the flex world that it can be difficult to make sense of it all. In this issue, we provide a snapshot of the flexible circuitry segment as we head into 2020. We cut through the noise and tell you what you need to know about flexible and rigid-flex circuits.

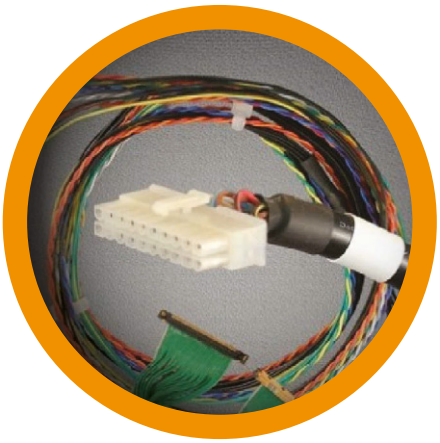
30+ YEARS EXPERIENCE. MORE THAN 30 PATENTS. JUST ONE GOAL.



Flexible Printed Circuits



Flat Flexible Cable



Wire Harnesses

Providing a quality, cost-effective solution on time has been our goal since day one. We achieve this through unparalleled customer service and by designing, sourcing and finishing interconnection systems for our customers worldwide.

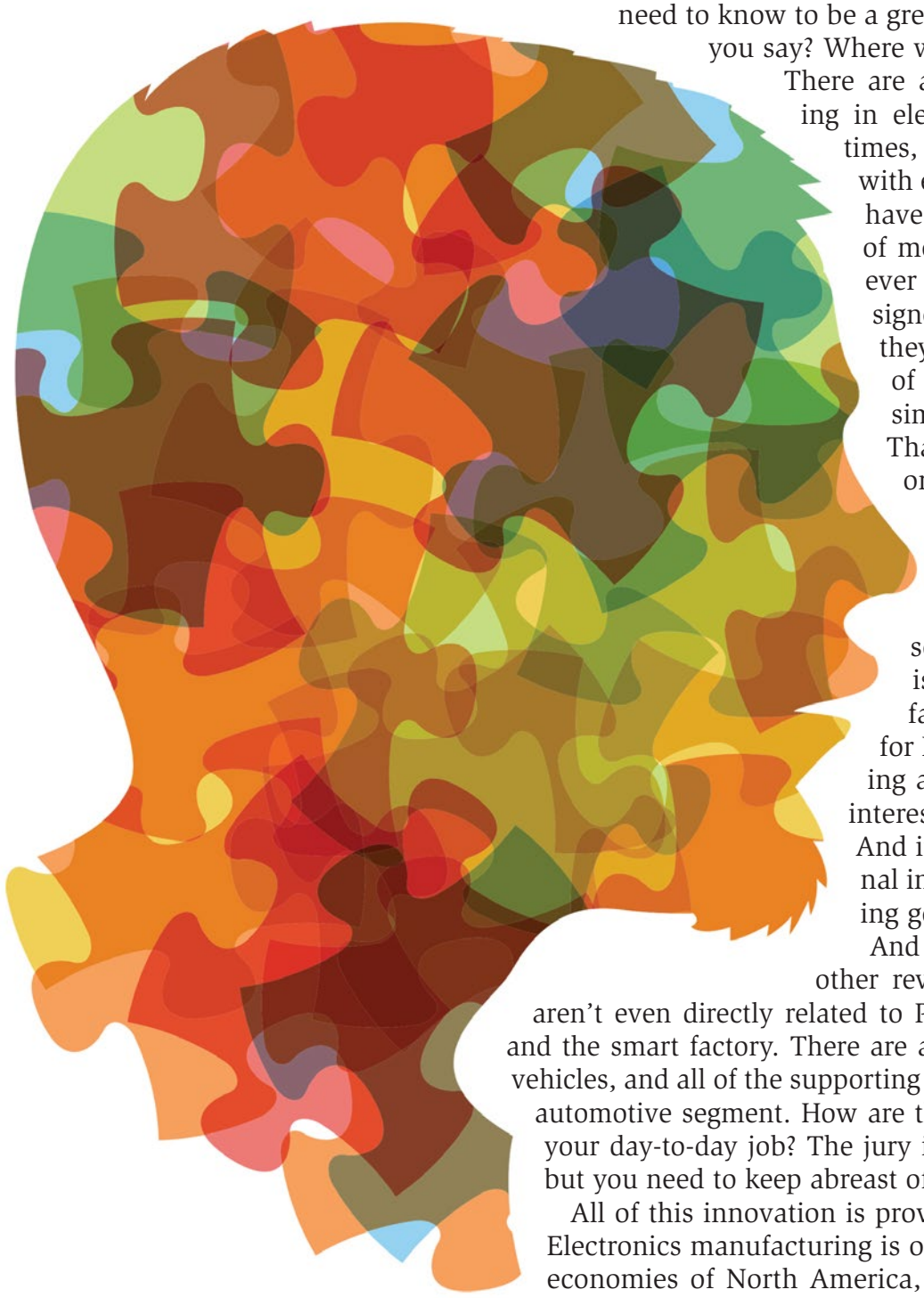
Miraco, Inc.
Manchester, NH
miracoinc.com
603-665-9449

MIRACO

What You Need to Know

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007



Think about this for a minute: If someone asked you what you need to know to be a great PCB designer, what would you say? Where would you even begin?

There are a lot of cool things happening in electronics technology, but, at times, it can be difficult to keep up with everything. Designers need to have at least some understanding of more types of technology than ever before. Fortunately, PCB designers are a lot like detectives; they like getting to the bottom of things, solving puzzles, and simplifying complex issues. That's a good thing because it's only getting more complex, or more interesting, if you will.

At the board level, even if you're not dealing with RF right now, you should know something about some of the issues that RF engineers are facing. The same holds true for HDI; even if you're not working at that level, it's in your best interest to be familiar with HDI. And if you're not dealing with signal integrity issues yet, as the saying goes, you will be soon.

And then there are all of these other revolutionary technologies that aren't even directly related to PCB design, such as 5G, IoT, and the smart factory. There are also autonomous and electric vehicles, and all of the supporting technologies surrounding the automotive segment. How are these concepts going to affect your day-to-day job? The jury is still out, for the most part, but you need to keep abreast of these ideas just the same.

All of this innovation is proving to be great for business. Electronics manufacturing is one of the engines driving the economies of North America, Europe, and Asia. The U.S.

economy continues to hum along, adding over a quarter-million jobs in November—the highest since January—and unemployment has dropped to 3.5%, tying the 50-year record low. It's not just a great time to be in this industry; it's a great time for employees, period. On the other hand, employers are getting accustomed to having positions open for months at a time, with some managers joking that they just can't fire anyone right now. And as we find out in this issue, it's a tough time for recruiters.

This month, we asked our expert contributors to discuss what they believe other designers and design engineers need to know as we move forward into 2020. We start out with an article by John Watson, CID, of Legrand North America, who explains what design advice he would give a younger version of himself if he could travel back in time in a DeLorean. Then, Taylor Rouse of Aerotek, a high-tech staffing company, discusses the ins and outs of today's engineering job market and offers some job-hunting hints that may sound counterintuitive at first.

Next, Linda Mazzitelli of Altium updates us on the convergence of ECAD and MCAD software tools, as well as virtual and augmented reality. Scott Miller of Freedom CAD shines the light on designing today's complex PCBs and why communication with your fabricator is critical for advanced board designs. And Roger Beers of Quantel focuses on a topic that doesn't get much coverage: Are ISO standards strangling innovation instead of helping?

We also have columns from our regular contributors, including Barry Olney of iCD, Stephen Chavez of the IPC Designers Council, John Coonrod of Rogers Corporation, Bob Tise and Matt Stevenson of Sunstone Circuits, Mark Thompson of Prototron Circuits, and Phil Kinner of Electrolube.

It's hard to believe that DesignCon and IPC APEX EXPO are only seven weeks away. Are you ready for 2020? Stick with Design007 Magazine and the rest of the I-Connect007 publications. We'll bring you the information you need to know every day.

Have a great holiday season! **DESIGN007**

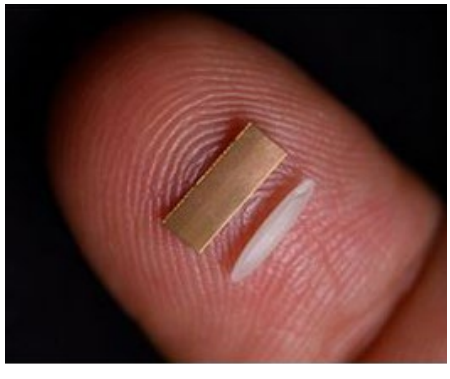
Andy Shaughnessy is managing editor of *Design007 Magazine.* He has been covering PCB design for 19 years. He can be reached by clicking here.

Tiny Intel EMIB Helps Chips 'Talk' With Each Other

Most chips in today's smartphones, computers and servers are comprised of multiple smaller chips invisibly sealed inside one rectangular package.

How do these multiple chips communicate? An Intel innovation called EMIB (embedded multi-die interconnect bridge) is a complex multi-layered sliver of silicon no bigger than a grain of rice. It lets chips fling enormous quantities of data back and forth among adjoining chips at blinding speeds: several gigabytes per second.

Today, Intel EMIBs speed the flow of data inside nearly 1 million laptops and field programmable gate array devices worldwide. That number will soon soar and include more products as EMIB technology enters the mainstream. For example, Intel's Ponte Vecchio processor, a general-purpose GPU the company unveiled Nov. 17, contains EMIB silicon.



To meet customers' unique needs, this innovative technology allows chip architects to cobble together specialized chips faster than ever. And compared with an older, competing design called an interposer—in which chips inside a package sit atop what is essentially a single electronic baseboard, with each chip plugged into it—tiny, flexible, cost-effective EMIB silicon offers an 85% increase in bandwidth. That can make your tech—laptop, server, 5G processor, graphics card—run dramatically faster. And next-generation EMIB could double or even triple that bandwidth.

(Source: Intel)

What You Need to Know About a PCB Designer Career

John Watson

Feature by John Watson, CID
LEGRAND NORTH AMERICA

I have been in the electronics field either as a technician or on the PCB design side for nearly 40 years. My first PCB design was done on Tango PCB software, which later became Protel and finally Altium. My tool of choice for most of my career has been an Altium product of some kind.

Fortunately, my career has grown just as ECAD software tools began hitting their stride, with each software release adding new tools and features. If you were starting a career in electronics or PCB design today, I could easily see how you might find the software selection confusing.

We are at the end of yet another phenomenal year. I must say that I have never personally seen such an extended period of growth. Fortunately, there is no sign of a slowdown, but instead, expansion is expected to last for years down the road. However, it's not about looking at where we came from, but rather where we are going.

With that in mind, I was asked to discuss what I think PCB designers need to know. If I could find a stable source of 1.21 gigawatts to charge up my flux capacitor, get my Delorean up to 88 miles per hour, and go back in time, here's some of the advice I would give my younger self.

Never Stop Learning

Why? Because the industry never stops changing. Those that continuously stay in that state of learning are the ones who succeed. The old saying, "Knowledge is power," often attributed to Sir Francis Bacon, has never been truer. Fortunately, there is never a lack of things to learn.

Don't ever become apathetic about learning. Make a point of keeping a running list of ideas or subjects that you want to research, and then purposely set aside time in your week strictly to study and learn about these things anywhere you can. Everything you're looking for is documented somewhere; you just need to find it. It has never been easier; with endless resources now online, it is not a question of access.

I see people who have stopped learning all of the time, even in our industry. I was recently consulting at a local company regarding its

Create Comprehensive PCB Documentation to Drive Fabrication, Assembly and Inspection

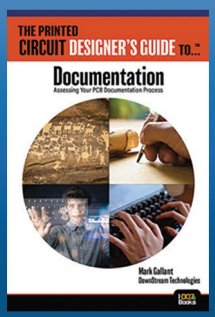
BluePrint-PCB®

BluePrint automates the PCB documentation process, creating comprehensive electronic drawings, successfully driving the procurement, fabrication and assembly process.

- 3D technology allows visualization of imported CAD data as a 3D PCB
- Lowers documentation cost and time
- Increases documentation detail & accuracy
- Shortens PCB design cycles
- Reduces documentation time by 60-80%
- Improves manufacturing & simplifies inspection

Download your free e-book "The PCB Designer Circuit Designer's Guide to Assessing Your PCB Documentation Process"

CLICK HERE TO DOWNLOAD



This book, written by Mark Gallant of DownStream Technologies, explains how an automated documentation solution like BluePrint-PCB® can eliminate post-processing errors and speed up time to market.

This book is a must-read for any PCB designers or design engineers who would like to adopt 21st-century PCB documentation processes.

For more information visit downstreamtech.com or call (508) 970-0670

DownStream Technologies

design architecture. I watched as an individual length-tuned several address lines manually by pulling out a calculator and "running the numbers." That was a pretty old-school way of accomplishing that task. He was unaware of Altium's length-tuning feature, which is driven by design rules. It was unfortunate because here was someone who had simply stopped learning, and advancements in the industry and tools left him behind. And it doesn't take long to get left behind.

Instead, when a new ECAD software tool is released, go through the release notes and learn all you can about the new features. Determine how they can be used to help in the PCB design process.

Be a Stickler for Details

Paying attention to detail is a crucial skill for designers who want to avoid errors and maintain efficiency and integrity in their designs. And paying attention to details requires discipline. Too many times, I saw projects that went entirely off the rails when we knew there were issues, and we proceeded forward with the intention of "fixing them later."

It takes discipline to know all of the slightest details of a design and how they are all interconnected. It takes discipline to compartmentalize an entire project and understand the requirements for each stage.

But most of all, it takes discipline to proceed forward only when those details have been checked and double-checked. Discipline means that you have dedication and commitment to the integrity of the design.

Paint With a Broad Brush

When I attend PCB design conferences, I find that there is an incredible amount of interest in HDI and high-speed classes. Of course, those are the "cool" modules to learn, but the electronics industry involves so much more than just HDI and high-speed design.

For example, the development of autonomous vehicles, a huge growth industry, is leading various areas of innovations across multiple technologies. Of course, these vehicles have high-speed circuitry, but they also require

a tremendous amount of power—high-current circuits—which is an entirely different animal. A designer who is proficient in multiple specialty areas of PCB design is a significant asset to any company. It is the designer who paints with a broad brush who finds more opportunities opening up, often becoming the company's "go-to" designer.

Identify and Use All of Your Resources

Identify and take advantage of any and all available resources. There are endless PCB design resources—from mentors to websites and organizations—that cover the PCB design and design engineering segments. Here are a few of my favorites.

IPC

IPC is one of the best resources for the PCB design community. Since 1957, IPC, a not-for-profit organization, has provided the standards to guide the electronic industry through never-ending change. The unique part about IPC is that it's useful for everyone in this industry: designers, board manufacturers, assembly companies, suppliers, OEMs, and anyone else involved with PCBs.

Along with the many available standards, IPC also offers a variety of training courses, including their certification programs specifically for PCB designers: the CID and the CID++, both of which I highly recommend. I regret

that I waited so long to get my CID certification, so don't make the same mistake.

I-Connect007

At my company, Legrand North America, I lead a team of 60 designers in the building control division. It is my responsibility to teach and mentor these designers. One of the primary resources that I recommend for my team members is *Design007 Magazine* and the other I-Connect007 magazines, sites, and newsletters. I have found I-Connect007 to be on the cutting edge of the electronic industry. The editors and contributors provide a smorgasbord of industry news, technical articles, and some of the best training materials/ebooks available.

Altium

I would be remiss to leave out Altium and all of the resources they provide. The list of Altium resources includes their video library, blogs, forums, technical books, white papers, and conferences dedicated solely to educating PCB designers. There is never a lack of Altium materials to read. I might add that the team at Altium is creating a resource community that is not vendor-specific, focusing on the bigger picture of PCB design, not the ECAD software you're using.

Conclusion

Finally, I would tell my younger self, "Don't let anyone tell you what your potential will be." Remember: Misery loves company. It seems that the miserable people who gave up on their dreams a long time ago are usually the ones telling you what you can't do. Never, under any circumstance, let anyone do that. The only person that can limit you is, well, you.

Your future and your continuing success are not dependent on your education level, where you grew up, or your age. The career path you decide to follow doesn't come from the outside, but rather from the inside. And no one can take that away from you. **DESIGN007**

John Watson, CID, is a senior PCB engineer at Legrand North America and a PCB design instructor.

IDTechEx Research Asks How Autonomous Driving Will Shape the Future of the Mobility Sector?

Autonomous driving is shifting the existing automotive supply chain from the traditional system of OEMs and suppliers to a collaborative ecosystem comprising OEMs, mobility service providers, software and hardware solution providers, and infrastructure providers. We have recently seen competitors joining hands and forming some unlikely-sounding alliances to reduce the cost of autonomous driving development, as well as to share resources and capabilities. In July 2019, BMW Group signed a long-term strategic cooperation with rival company Daimler AG that begins with brining Level 4 capability to market in the mid-2020s. The long-term collaboration aims to extend to encompass a scalable platform for autonomous driving where they can share components, validation efforts and other competencies.

Mobility services enabled by autonomous driving technologies, which allows fleet operators to get rid of the biggest operation cost—the human driver—will offer a cheaper alternative to purchasing and owning a private car. In the next two decades, mobility-as-a-service (MaaS) will grow rapidly to meet the increasing travel demand and in the meantime gradually replace private driving, according to IDTechEx's latest report titled "Autonomous Cars and Robotaxis 2020-2040." IDTechEx forecasts that in a moderate scenario, shared autonomous cars could account for up to 4 trillion miles travelled on the road—about 30% of the total travel demand—by 2040, and global passenger car sales are expected to peak in 2031.

(Source: IDTechEx)

The Big Bang: Lumped Element to Distributed System

Beyond Design

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

The simplistic approach to analyzing electronic circuits is to use the lumped element model. This methodology assumes that the attributes of the circuit—resistance, capacitance, and inductance—are concentrated into idealized electrical components connected by a network of perfectly conducting wires. However, in reality, that is not the case.

As the frequency and rise time increase, these elements become distributed continuously through the substrate along the entire length of the trace. The copper trace and the adjacent dielectric materials become a transmission line, the skin effect forces current into the outer regions of the conductor, and frequency-dependent losses impact on the quality of the signal. The PCB trace now behaves as a distributed system with parasitic inductance and capacitance characterized by delay and scattered reflections. The behavior we are now concerned about occurs in the frequency domain. In this month's column, I will discuss the difference between the lumped element model and the distributed system (Figure 1).

In my previous column, "The Frequency Domain," we saw that impedance is defined in both the time and frequency domains. In the time domain, the impedance of a resistor (R) can be represented by a relationship between voltage and current (Ohm's Law). Similarly, an ideal capacitor (C) has a relationship between the stored charge and the voltage across its plates. And the behavior of an ideal inductor (L) is defined by how fast the current traveling through it changes in the time domain.

We group these three elements (RLC) in a category called lumped circuit elements, in the sense that their properties can be lumped into

Figure 1: (a) Lumped element; (b) distributed system.

Candor
Circuit Boards

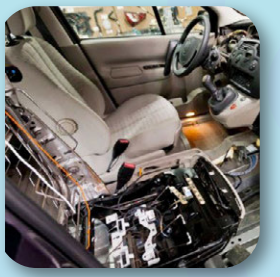
Innovative solutions for complex circuit board manufacturing

Click to see what we bring to each sector of the industry:

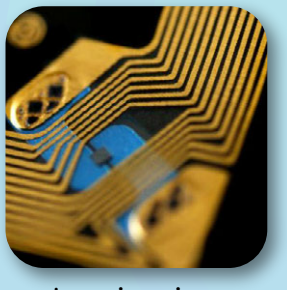
RF/Microwave PCBs

Power Electronics

High-Speed Data



Automotive Electronics



Lamination

Startups

Our technology produces what our customers dream!

www.candorind.com | (416) 736-6306

a single point. This is quite different from the properties of an ideal transmission line, which also consists of these three elements, but they are distributed continuously through the dielectric materials along its length. The distributed model is used when the wavelength becomes comparable to the physical dimensions of the circuit, making the lumped model inaccurate. This typically occurs at high frequencies, where the wavelength is very short. However, it can also occur on very long, low-frequency transmission lines, such as high-voltage power lines. The three primary elements now include distributed capacitance, inductance, and conductance (G).

The lumped element model completely fails at one-quarter wavelength (a 90° phase change), with not only the value but the very nature of the component itself being unpredictable. Due to this wavelength dependency, the distributed system model is used mostly at higher frequencies.

It is important to realize that the terms lumped and distributed are not properties of the system itself. These properties are related to the size of the circuit, compared to the wavelength of the voltages and currents passing through it. So, a resistor is, or isn't, a lumped element (even though it is usually meant to be one), depending on the frequency of the applied signals.

Lumped systems are described by ordinary differential equations because, due to the small size of the system (compared to the wavelength), the spatial derivatives can be neglected and we only need to consider time derivatives. On the other hand, for distributed systems, we need to take electromagnetic wave propagation into account to get spatial as well as time derivatives, which leads to partial differential equations in the frequency domain.

A transmission line can be represented by an infinite number of segments, incorporating series resistive and inductive elements with shunt capacitive and conductive elements, as in Figure 2. And because of the restricted velocity of propagation in the medium, the signal does not know what the termination is at the end of the line. It can only see the impedance of the line, which, by design, should be matched to the driver.

What forms the electromagnetic field in the transmission line?

This is a question that even Google can't answer (until now). Here's how I see it: An electric field is produced when voltage is applied across an IC output driver. When a signal varies this voltage, there is a surge of current that produces a magnetic field. This electromagnetic energy then transmits the signal, at about half the speed of light—limited by the dielectric medium—down the length of the transmission line following the trace. The energy radiates into the surrounding dielectric material and couples into nearby structures, creating a distributed system of parasitic elements. The electromagnetic fields are not restricted to the multilayer substrate, and, if adequate care is not taken, may emit radiation causing electromagnetic interference.

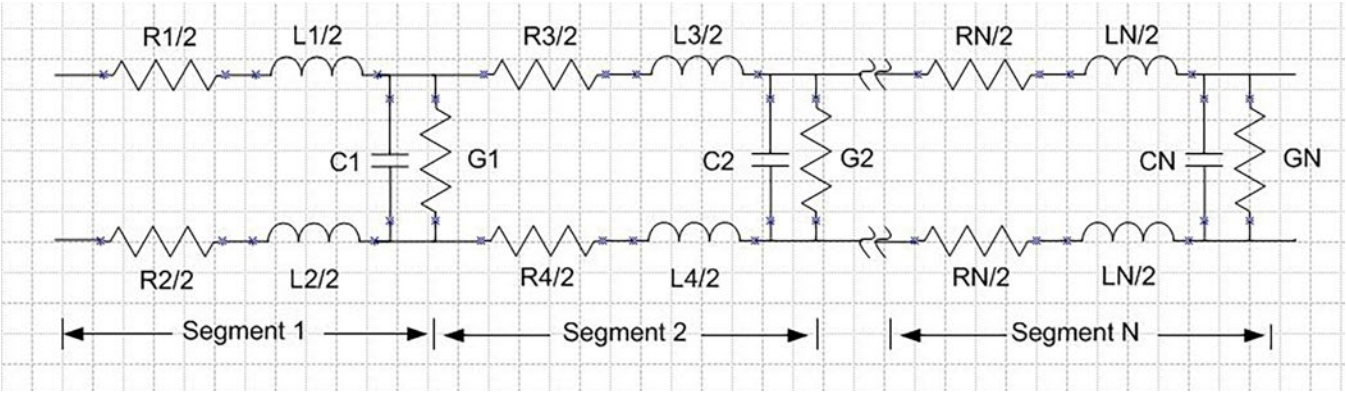


Figure 2: Transmission line represented by a series of R-L-C-G elements.

The distributed system model is more accurate but also more complex than the lumped element model. The selection of the model is dependent on the accuracy required in a specific application. There is no clear-cut demarcation in the frequency at which these models should be used, although the changeover is usually somewhere in the 100–500-MHz range. An often-quoted rule of thumb is that traces longer than one-tenth (0.1) wavelength will usually need to be analyzed as a distributed system, which is about where the two curves start to deviate noticeably in the example of Figure 3.

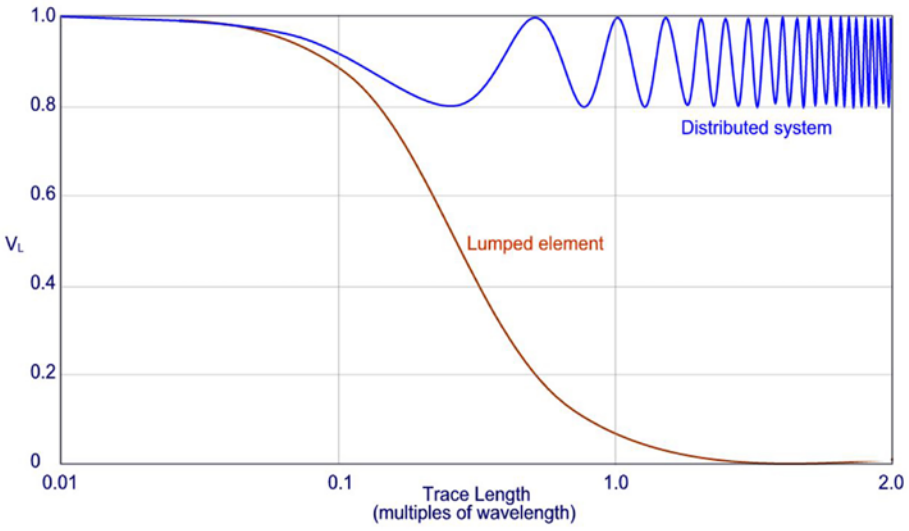


Figure 3: Lumped vs. distributed element analysis of a transmission line.

The technology employed and the physical scale of the design is also significant as miniaturized circuits can use the lumped model at a higher frequency. PCBs using plated through-hole technology are larger than equivalent designs using surface-mount technology. Hybrid integrated circuits are smaller than PCB technologies, and monolithic integrated circuits are smaller again. ICs can use lumped designs at higher frequencies than printed circuits, and this is done in some radio-frequency devices. This choice is particularly significant for handheld devices because lumped element designs generally result in a smaller product.

To illustrate the difference between the regimes of the analytical treatment of the transmission line, the two models are compared in a simulation for increasing lengths of transmission line (Figure 3). The analysis shows the behavior of the load voltage (V_L) using lumped and distributed element calculations for a lossless transmission line. The frequency dependence is shown in the form of the trace length being a multiple of the wavelength.

Depending on the signal rise time, the distributed model for transmission lines starts deviating from the simplified lumped element model between a trace length of 0.01 and 0.1 of the wavelength of the signal. This simulation uses a load impedance that is matched to the impedance of the transmission line, so the reflections are negligible.

Although a lossless distributed model is a good representation of a typical low-frequency transmission line, at high frequencies the conductor and dielectric losses also need to be considered.

Key Points

- The lumped element model assumes that the attributes of the circuit are concentrated into idealized electrical components connected by a network of perfectly conducting wires
- As the frequency and rise time increase, these elements become distributed continuously through the substrate along the entire length of the trace
- Electromagnetic energy radiates into the surrounding dielectric material and couples to nearby structures creating a distributed system of parasitic elements
- The distributed model is used, at high frequencies, when the wavelength becomes comparable to the physical dimensions of the circuit
- The lumped element model completely fails at one-quarter wavelength (a 90° phase change)

- The distributed system model is more accurate but also more complex than the lumped element model
- Traces longer than one-tenth wavelength will usually need to be analyzed as a distributed system
- The technology employed and the physical scale of the design is also significant as miniaturized circuits can use the lumped model at a higher frequency
- Lumped element designs generally result in a smaller product
- The distributed model for transmission lines starts deviating from the simplified lumped element model between a trace length of 0.01 and 0.1 of the wavelength of the signal

Further Reading

- B. Olney, "Beyond Design: Controlled Impedance Design," *The PCB Design Magazine*, May 2015.
- B. Olney, "Beyond Design: Transmission Lines—From Barbed Wire to High-speed Interconnects," *The PCB Design Magazine*, May 2014.
- B. Olney, "Beyond Design: Effects of Surface Roughness on High-speed PCBs," *The PCB Design Magazine*, February 2015.
- B. Olney, "Beyond Design: Common Symptoms of Common-mode Radiation," *Design007 Magazine*, May 2018.
- Arthur Anderson, "Transmission Lines: From Lumped Element to Distributed Element Regimes," All About Circuits, November 28, 2015.
- E. Bogatin, *Signal and Power Integrity: Simplified*, Prentice Hall, 2008.

Editor's note: All figures drawn by Barry Olney.

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 www.icd.com.au. To read past columns or contact Olney, click here.

APEX EXPO IPC 2020

Celebrating 20 YEARS OF EXCELLENCE IN ELECTRONICS

MEETINGS & COURSES: **February 1-6**
CONFERENCE & EXHIBITION: **February 4-6**
SAN DIEGO CONVENTION CENTER | CA

www.IPCAPEXEXPO.org

ELEVATE THE EXCELLENCE OF ELECTRONICS

Join Us to Celebrate 20 Years of the IPC APEX EXPO!

The pursuit of excellence in electronics is year-round. But during IPC APEX EXPO 2020, the focus of the electronics industry will be on how collectively, we can elevate all aspects of our industry and the products we create.

Together, we'll celebrate the 20th Anniversary of IPC APEX EXPO, explore innovative ideas and share our experiences, all with an eye toward a future driven by success.

Plan now to elevate your excellence in San Diego at IPC APEX EXPO 2020.

Register by December 19 to get your 20% advanced registration discount!

ipcapexexpo2020.ipc.org | #ipcapexexpo


Taylor Rousse

What You Need to Know: The High-tech Job Market

Feature Interview by Andy Shaughnessy
I-CONNECT007

I met with Taylor Rousse, an engineering recruiter for Aerotek, a high-tech staffing company, at PCB Carolina to discuss the demands she sees in different industry segments and how it varies region to region. Taylor also offers advice for designers or electrical engineers in the job market, including tips on writing that perfect résumé and the return of a counter-offer.

Andy Shaughnessy: Tell us a little bit about the company and why you're here.

Taylor Rousse: I'm an engineering recruiter, and I specialize in hardware, software, electrical, and test. Aerotek is the largest privately-held staffing company in the U.S. and in the world. There are a lot of different divisions, including customer service on the non-technical side; on the technical side, we also have engineering, clinical, and sciences. I specialize in the engineering division, so I only recruit people with engineering skill sets.

Shaughnessy: What are some of the trends you see in the industry right now?

Rousse: From an aerospace and defense standpoint, that is where I see the most business and recruits coming out of nationally. There aren't enough people to fill the jobs that they have within the aerospace and defense space. People who have security clearance are in extremely high demand, and not even from a compensation standpoint, but from a benefits standpoint too; they can demand what they want in a role. From an embedded system perspective, a lot of students coming out of school with a computer science degree are interested in the cloud and application software, and there is more demand than ever for embedded, C, and C++. FPGAs are absolutely off the chart. I'm looking for an entry-level FPGA designer right now, and the company is very flexible with what they'd offer for the right person.

Embedded, C, and C++ are a necessary skill set that's going to be around, especially when it comes to people who are coming out of school, because the company gets a chance to bring them up with training in the ways they

YOUR PCB PORTAL TO THE WORLD

DELIVERING GLOBAL-SCALE PCB SOLUTIONS

15
Subsidiaries Worldwide

80
Leading PCB Engineers

50
Countries Supplied

40
Thousand Different Items Annually

When you work with us, you have effortless access to a local partner who understands your short and long-term objectives, motives, environment, and concerns, while at the same time enjoying our global presence and the wide commercial advantages it brings.

VIEW CASE STUDIES

FINELINE
EXCELLENCE IN PCB

fineline-global.com

would like them to work. With FPGA design, you can get into the aerospace and defense space 100% and go for it. If a company gives you the chance to come on without a clearance, take that job.

Shaughnessy: I've heard that getting clearance can take a year or so.

Rousse: Yes, and there are different levels. There's interim clearance, secret clearance, and top-secret clearance. I work as part of what's called the Strategic Recruiting Center, which is a very niche part of Aerotek that allows us to have exposure to opportunities across the U.S. I have direct eyes on roles in Virginia, Florida, New York, Colorado, etc. If someone says, "I want to get back to my family in New York," I can say, "I have direct connections to people in New York." We have ongoing visibility everywhere.

If a company gives you the chance to come on without a clearance, take that job.

Shaughnessy: You're based in North Carolina

Rousse: Yes, we're based in Cary, North Carolina, and our corporate office is in Hanover, Maryland.

Shaughnessy: Do you see any differences regionally? Is it easier or harder to place certain job skills in one area or another?

Rousse: Geographically, people from Upstate New York are coming here. The areas around Syracuse, Buffalo, and Rochester are a little bit tougher. I see a lot of pickup in the defense space in Colorado, Virginia, and Maryland. In North Carolina, the defense space has not been as tapped into as much, but Virginia; Maryland; Washington, D.C.; New York; Colorado; and Florida are picking up. It's the C/C++ systems engineering. I see a ton of hardware design roles for some of the bigwigs in defense contracting.

Shaughnessy: It has been a long time since I've seen anybody at a trade show carrying their résumé around. With 4% unemployment, when customers come to you, do they want to make a lateral move?

Rousse: It's a combination of a lot of things. I don't have a ton of people reaching out to me because people have jobs. Not only do they have jobs, but these companies are fighting to keep them, to make sure the work environment is a good one so that they don't want to leave. People have jobs, and then their employers are working to make it a happy place to work.

When I reach out to someone who's passively on the market, I try to give them insight into the market and find out what their dream job or ideal situation is. Even if they're content right now, I offer to keep an eye out. I'll build relationships with people who aren't looking. Someone might say, "I want to get into medical devices," so I'll look for that, and when something comes up, I reach out. For the most part, the market is good right now.

Shaughnessy: You say good, but it seems like, from your perspective, you would call it bad.

Rousse: For me, it's bad (laughs)! It's about getting smarter with the way that you recruit because you can't put your sales hat on, make 50 cold calls, and put the numbers to the game; it's about going to industry events, getting to know people, and remembering their faces, what they asked for, and what they need. There's a trust factor there. If someone is super happy in their job and making good money, why are they going to jump for you who represents a job they may not trust? That's a huge part.

Shaughnessy: The devil you know is better than the devil you don't know.

Rousse: Exactly. Counter-offers are another huge thing we see. If someone's passively on the market, and they get another job, if their

company doesn't counter-offer right away, I would be extremely surprised; pay attention to those. We have an article that we put out on why counter-offers aren't effective and that your manager now knows that you're looking for a job. It all goes back to whether money is the reason that you're looking in the first place.

More than ever, we have customers and clients come up to us, saying, "We need FPGA designers who have Verilog. We can't get any résumés, and if we do get some, nobody is qualified." And we have more customers than candidates coming to us. If you're a person who is curious, we've preached not pushing it onto people. I'd be happy to be an eye into the market on what's going on, what kind of companies are looking, and, if I can't help you with that company, I'll still be happy to give you a referral.

Shaughnessy: That's good. Of the people you place, how many are electrical engineers or designers?

Rousse: At least 50%. We have a software component as well, and we probably have equally as many positions in the software space. I see a ton of electrical roles.

Shaughnessy: What advice would you give to somebody who is thinking of making a job change, even if they want to make a lateral move or move back home?

Rousse: First and foremost, put side projects or hobbies on your résumé. I was raised to think that you don't put your hobbies on your résumé.

Shaughnessy: That's what I've always heard.

Rousse: But I've had managers say they make connections when there's a hobby list. It shows that this person has the initiative and social capacity to go out and do other things in the world and personalizes them. I've had a few managers say, "They don't have any hob-

Figure 1: Taylor Rouse, center, with Aerotek's Zach Rich, left, and Emily Vlkojan-Reece at PCB Carolina.

bies," so we have to backtrack to figure that out. Side projects are huge. If you've been out of work for a while, whether you're a designer, manager, etc., do your own design projects on the side, even if it's you're fiddling with your design tool at home; that shows that you have a vision. Just because a company didn't pay you to do it doesn't mean you weren't keeping your skills fresh in some way.

I usually outline it as professional experience, being companies that you've worked for and what you've done there, and then you can simply list side projects and personal experience in the same way you share your professional job experience. Patents are huge too, and mention articles and papers that you've published. From a documentation standpoint, it's super important to have those skills. And, of course, include your education information and any classes or certifications. Experience is more important than a degree.

Shaughnessy: I've always heard you shouldn't have a long résumé unless you're a professor with a CV and can't help it. In general, is it better to keep it to one page if possible?

Managers spend about five seconds looking at a résumé to decide whether or not they want to hire that person, so that emphasizes how important it is to keep the bread and butter on the first page.

Rousse: Yes. Managers spend about five seconds looking at a résumé to decide whether or not they want to hire that person, so that emphasizes how important it is to keep the bread and butter on the first page. Most people scan the page, and this is how it should be laid out: name, summary, technical skills, education, and then professional experience. I would say make it detailed, but as short as possible; don't get artistic with it. Some people use Adobe Photoshop to do their résumés, and I can't tell you how many times I have heard a manager say, "I hate this creative résumé."

Shaughnessy: I once submitted a résumé early on in my career on lime green paper because I heard if you had different color paper, it would stand out. Is that worth a try?

Rousse: Maybe if you're going for a graphic design job or something in the arts, but I'd say it's more distracting than anything, and a manager will wonder about the rationale behind it, in addition to the content on the résumé.

Shaughnessy: Is there anything you want us to add?

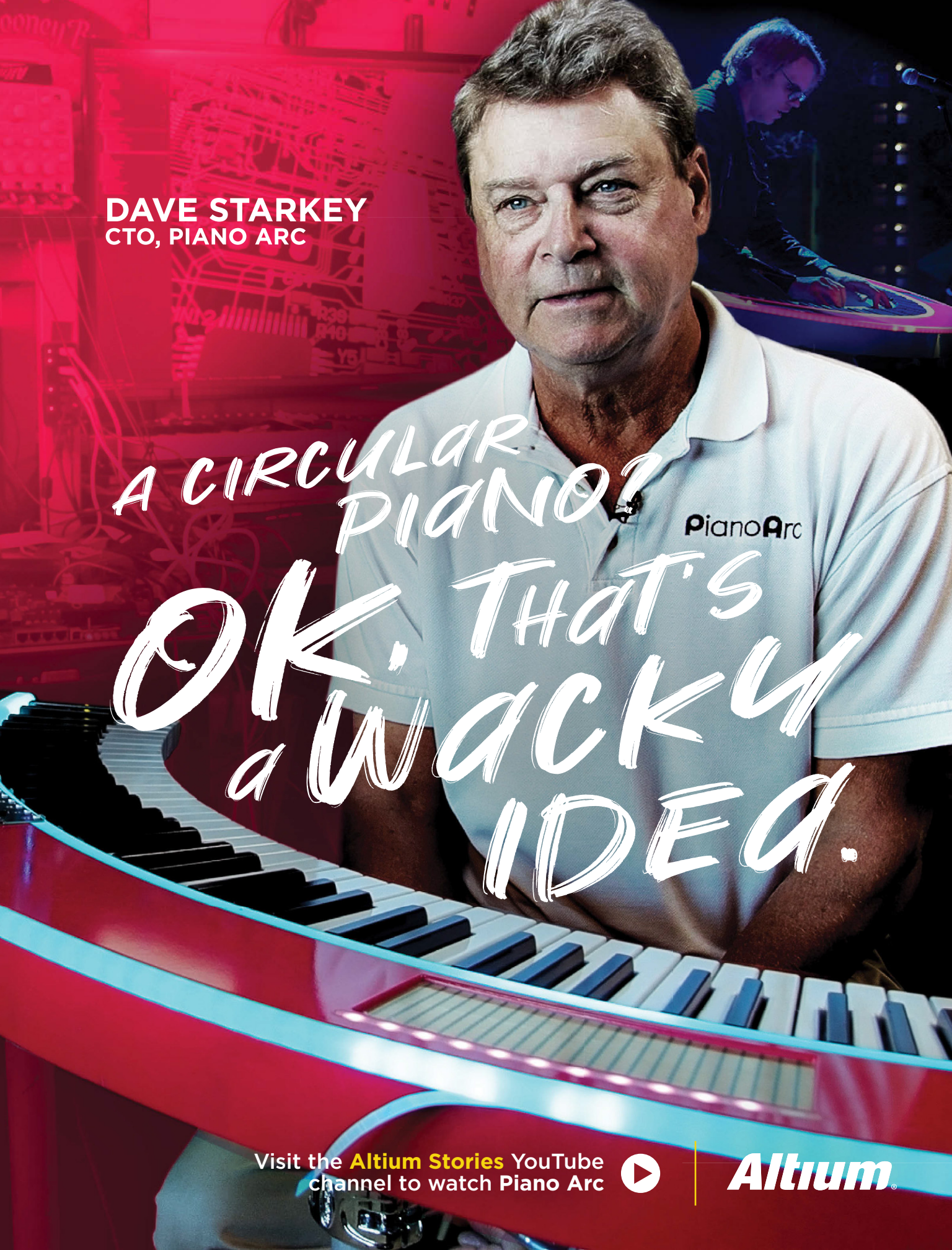
Rousse: If a recruiter reaches out to you, you can ultimately tell the good ones from the bad ones. You want to interact with people who will stay in touch with you and seek to understand your personal life and goals. If someone asks you about that, I would push you to entertain them, hear them out, pick one you trust, and hold them close. It goes a long way. You never know what could happen. And if someone tells me, "I'm not looking or interested," a month later, they might say, "Sorry, I was busy when you reached out last time." You may want help to outsource the design team later on, so don't automatically dismiss recruiters even if you're not looking for anything at that time.

Shaughnessy: You never know.

Rousse: Absolutely.

Shaughnessy: Thanks for your time, Taylor.

Rousse: Thank you, Andy. DESIGN007


DAVE STARKEY
CTO, PIANO ARC
A CIRCULAR PIANO?
OK, THAT'S A WACKY IDEA.
PianoArc
Visit the Altium Stories YouTube channel to watch Piano Arc
Altium

Closing Out 2019 on a Positive Note

The Digital Layout

by Stephen V. Chavez, MIT, CID+, IPC DESIGNERS COUNCIL

Wow! I can't believe its December already. The IPC Designers Council Chapters had another very active year that was filled with great industry content from various industry events and chapter activities. From chapters that remain very strong to an old legacy chapter being resurrected and a new chapter formed, local chapters have continued to be active in their respective regions. We finish 2019 by highlighting the Silicon Valley Chapter located in California and the Research Triangle Park (RTP) Chapter located in North Carolina.

Silicon Valley Chapter, California

Chapter Leader: Bob McCreight

The Silicon Valley Chapter held its final chapter meeting on October 24 in Milpitas, California. Thanks to Sierra Circuits for recording the meeting and their complementary slides. Much appreciation goes out to Footprintku and Zuken for their roles in hosting and sponsoring the meeting. The combined team did a great job, and we look forward to working with them again for next year's Q4 meeting. Overall, we had 35 attendees. The venue was excellent and the lunch was quite enjoyable. The topic presented by Scott Nance was very informative and contained lots of great PCB design material. I would also like to give a shout-out to Optimum Design Associates for their involvement. The next chapter meeting will be on February 13. More details will come in early 2020.

Research Triangle Park (RTP) Chapter, North Carolina

Chapter Leader: Tony Cosentino

The RTP Chapter hosted PCB Carolina 2019 on November 13 at the North Carolina State University McKimmon Conference and Training Center (Figure 1). The event was a great

Figure 1: PCB Carolina 2019 show floor.

While others struggle just to keep pace, we stay ahead of the curve by constantly investing in the equipment, people and resources needed to be the best total PCB solutions provider in the industry.

VIEW OUR CAPABILITIES

EAGLE electronics Inc.

www.eagle-elec.com

Figure 2: Keynote.

Figure 3: Evening reception.

success. As the current president of the chapter, I commend our officers and volunteers for their tremendous efforts to bring such a large event to this region.

The event included an awesome keynote address about IoT by Tom Snyder of RIoT Labs and Gaëlle Fages of Bayer Crop Science, along with breakfast (Figure 2). Afterward, the show floor opened with 78 vendors supporting PCB design, PCB software, PCB fabrication, PCB assembly, component and material suppliers, and much more. The first part of the event included eight technical sessions, followed by a large buffet lunch. The second part of the event included eight more technical sessions. All of the technical session details are located on the PCB Carolina website, and the actual presentations will be posted soon. The third part of the event included an evening reception with more food and drinks (Figure 3).

The RTP Chapter arranged two half-day soldering workshops through Circuit Technology Incorporated (CTI), which occurred during PCB Carolina. CTI's IPC Master Instructor Angel Deluna taught hands-on soldering workshops in accordance with IPC J-STD-001, IPC-610, and IPC-7711. And in conjunction with PCB Carolina, IPC offered a Tech Ed Course the day before the event on design for excellence by Dale Lee of Plexus Corporation. Further, EPTAC held IPC Designer Certification Programs for CID and CID+. CID certification was taught by CID Instructor Kelly Dack, and CID+ certification was taught by CID+ (Advanced) Instructor Dave Seymour (Figure 4).

The RTP IPC Designers Council offers this event to bring designers and engineers together to keep the local community connected and offer opportunities for continued education. We aspire to attract younger talent to this industry by engaging local universities and community colleges because they are our future. This year, we worked with IPC to bring 50 local high school students from five schools who are involved in the STEM program to PCB Carolina. STEM is a curriculum based on the idea of educating students in four specific disciplines (science, technology, engineering, and math). The students were in a structured environment called the Owl Project. During lunch, a panel of industry engineers talked with the students, and then the students were chaperoned through the 78 vendors at PCB Carolina.

This year's event was up in registration from last year's event by 14.6% (does not include STEM students). I hope to see new faces at PCB Carolina 2020 on November 11, so start making your plans now to attend.

For 2020, the chapter will hold elections at our January chapter meeting. I have been the RTP Chapter president for the last 10 years and

Figure 4: Classroom.

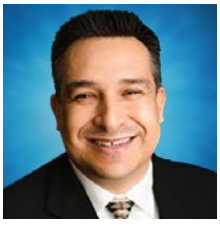
I plan to step down. I will remain active in an ancillary position to support the chapter as required. I publicly announced my plan to step down as president of the chapter at PCB Carolina during the keynote. I have publicly endorsed Randy Faucette (current VP) as the new president, and we are working to bring a new face to the VP position.

IPC CID/CID+ Certification Success

We continue to have successful IPC CID and CID+ certification courses as we close out 2019. Our most recent class successes were held in many diverse locations throughout the U.S. and abroad. Several recent classes to mention over these last few months ranged in locations from Santa Clara, California (PCB West), to Schaumburg, Illinois; Anaheim, California; Dallas, Texas; Raleigh, North Carolina (PCB Carolina); and Scottsdale, Arizona; and the last class of the year was held in Manchester, New Hampshire. Congratulations again to all those who have successfully achieved their IPC CID or CID+ certification. These certification courses are an excellent source of professional development.

And if you are not yet CID/CID+ certified, I highly recommend these certification courses as a path for continued education in PCB design. For 2020 CID and CID+ certification schedules and locations, contact EPTAC Corporation to check current dates and availability. Note: Dates and locations are subject to change, and a minimum enrollment of seven students is required for a class to be held.

The IPC Designers Council is an international network of designers. Its mission is to promote printed circuit board design as a profession and to encourage, facilitate, and promote the exchange of information and integration of new design concepts through communications, seminars, workshops, and professional certification through a network of local chapters. **DESIGN007**



Stephen Chavez, MIT, CID+, is a member of the IPC Designers Council Executive Board and chairman of the communications subcommittee. To read past columns or contact Chavez, click here.

What You Need to Know About ECAD/MCAD Convergence

Feature Interview by the I-Connect007 Editorial Team

The I-Connect007 Editorial Team spoke with Linda Mazzitelli just a few weeks after she joined Altium. We asked her to discuss the paper she presented at the conference, as well as her work with ECAD/MCAD collaboration and augmented reality (AR) and virtual reality (VR).

Andy Shaughnessy: Linda, it's good to see you again. I hear that you have a new job. Can you tell us about your role?

Linda Mazzitelli: I recently joined Altium, in the position of solution architect primarily for the Nexus, ConcordPro, and Altium 365 products. My prior role at PTC, in addition to ECAD product manager, also included being the partner liaison to the ECAD vendors. My new role at Altium will include essentially doing that in reverse (i.e., partnering with the PLM vendors from the Altium perspective).

Shaughnessy: Can you tell us about the presentation you gave at AltiumLive in San Diego, California?

Mazzitelli: I spoke about ECAD MCAD collaboration. Right now, the industry still largely uses a file-based transfer method between the two domains (DXF, IDF, or IDX). Altium is taking a different approach with cloud-based implementation, where it's more of a "managed" push-pull. It starts with one side proposing/creating changes, selecting a "push" button, adding notes for the recipient, and proposing those updates to the other domain. Those changes can then be reviewed and either accepted or rejected by the person on the other end, after which the response is sent back to the initiator, all with no physical files to manage.

I describe the process as being similar to having a conversation. Whoever initiates the "conversation," the domain initiating/proposing changes, has to get final confirmation that they were accepted or rejected. Because this is done through a cloud-managed application, it enables users to eliminate file-based transfers and not worry about who chooses which file and whether they are read in the right order. It is all managed behind the scenes automatically and eliminates virtually any potential for error.

IT-988G SE / IT-988G

IT-988G

- Halogen Free
- Tg = 190° C
- Td = 405° C
- Dk = 3.46 @10GHz*
- Df = 0.0025 @10GHz*
- Available with low Dk Glass - IT-988G SE

IT-988G SE

- Halogen Free
- Tg = 190° C
- Td = 405° C
- Dk = 3.24 @10GHz*
- Df = 0.0014 @10GHz*

Ultra high speed > 56 Gbps per channel, NRZ (PAM2), PAM4 applications

Insertion Loss - Measured

ITEQ products approach the properties of PTFE. ITEQ offers best in class performance.

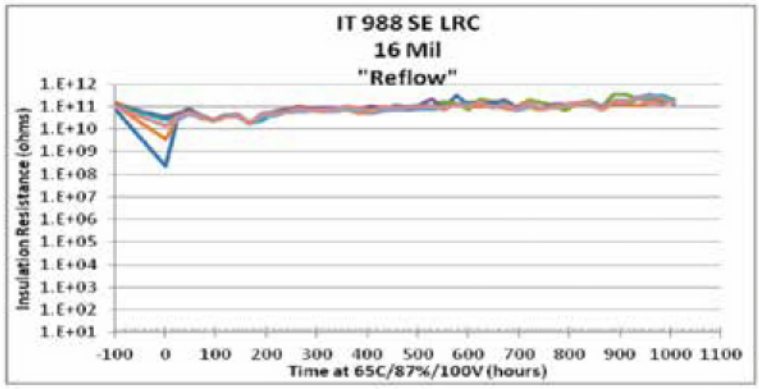
Sequential Lamination

7 Lamination cycle data

Lamination	DMA	DSC	TMA	T300 with CU	Solder Dip PCT: 1h @ 121°C	Td 2wt% / 5wt%
1	213	187 / 187	182	> 60	> 60	408 / 435
2	216	194 / 199	193	> 60	> 60	417 / 438
3	214	186 / 192	185	> 60	> 60	417 / 442
4	216	193 / 194	184	> 60	> 60	424 / 443
5	217	194 / 199	190	> 60	> 60	418 / 442
6	218	191 / 197	188	> 60	> 60	403 / 436
7	218	190 / 197	194	> 60	> 60	425 / 444

No thermal degradation observable after 7 lamination cycles

HDPUG MRT-6 LRC CAF Data



IT-988G SE

- High Tg / Lead Free / Halogen Free
- Ultra Low Loss Laminate & Prepreg
- 100G / 400G Solution
- Lower Dk (<3.24 @ 10GHz)
- Ultra low Df (<0.0014 @ 10GHz)
- Very Stable Dk-Df across frequency
- HDPUG MRT-6 - LRC and HRC passed all thermal requirements
- HDPUG MRT-6 - LRC and HRC passed CAF requirements
- Proven thermal stability beyond 7 lamination cycles and suitable for HDI applications

ITEQ has the leading solutions for High Speed Digital applications with the extremely cost-effective IT-170GRA1 and IT-968 to the ultra high speed IT-988G and IT-988G SE for the highest end designs beyond 56 Gbps.

*53% resin content

ITEQ

Shaughnessy: Originally, you were a designer, so you've been through ECAD.

Mazzitelli: Right, but the collaboration process has been tough to manage because it has always involved manual intervention. Until now, there has never been a quick and accurate way to manage the data going back and forth.

Shaughnessy: There's never been a good format. Each side wants different pieces of data, so it's not a conversation; they're talking past each other.

Mazzitelli: Right, and file-based methodologies still do not fully support ECAD requirements in MCAD, such as cavities, etc. Also, if two people are making changes at the same time to the same thing, who wins? When it's managed in the cloud, you have better coordination and collaboration versus having to rely on people to read-in the right files in the right order.

Dan Feinberg: Especially if you're making changes to the same thing, and they don't know what each other is doing.

Mazzitelli: Altium has an initiative here to make that work, and I think it's going to give them a competitive advantage. They are working with SolidWorks, Creo, and Inventor right now.

Shaughnessy: You have also been into data management for quite some time. What are the big issues you see in data management?

Mazzitelli: When you work for a PLM vendor, data management is part of the vocabulary. But coming into an ECAD vendor and company, the messaging is going to be interesting to position. As a whole, designers have been doing things the way they've been doing them for 40 years, and it works. So, it's not necessarily the board designers who are going to push to have this technology; it's more at the IT-level and the executive-level/management because they're the ones that want a single source of truth for all of their data.

Shaughnessy: You have all this data, which is great, but managing it is a whole thing in itself.

Mazzitelli: Every designer typically has their own naming conventions for boards. For example, I might call mine "Linda_Placement1, 2, 3, and 4." When I finish my placement and am going to route, I now label them, "Linda_Route1, 2, etc." Everybody does studies. So, even though I may be at "Linda_Placement5," I may think, "Well, 2 was better, so I'm ultimately going to go back to that." But what if I go on vacation? If you're gone and somebody has to pick up your design, what are they going to think? They're going to go to the latest one, which may not be the best one, but you forgot to communicate that before you left.

With Nexus and other data management products in the industry, this now takes the version control, or the work in process, out of the hands of the users; they don't have to worry about it. Every time they save, it creates a new version of the design. Then, when you're ready to release to prototype or production, you have all of your data under revision control so that the entire enterprise design team knows that it's the final version. Ultimately, this will make things easier, but it's still an education process in the industry.

Shaughnessy: We've gone from not having enough data to having almost too much data to manage, and it's all about making the data accessible.

Mazzitelli: Yes. And many designers don't see the value in data management because everybody looks at things from their own perspective. But data management exposes the data to the enterprise, including component managers, product managers, test engineers, and marketing people. It enables the entire infrastructure of a company to have access to the data almost on demand. Right now, if somebody needs a status on a particular PCB, they still need to walk over to the designer and ask where they are on the board. This could be virtually eliminated if the designs were managed and exposed within that enterprise system.

APCT

Passion I Commitment I Trust

Is This The Agent You've Chosen To Manage Your Offshore Supply Chain!?

APCT Global is not a Broker! We're a brick & mortar, PCB manufacturing company; one of the largest, independently owned PCB manufacturers in North America. We have a team of professionals to manage your account.

Make The Right Call - Trust The Experts at APCT Global

APCT GLOBAL

Contact APCT Global Today
203.284.1215 | APCT.com

Global Printed Circuit Board Solutions

APCT.com

APCT Santa Clara HQ	APCT Anaheim	APCT Orange County	APCT Wallingford	APCT Global
408.727.6442	714.921.0860	714.993.0270	203.269.3311	203.284.1215

Shaughnessy: One of the things you've been interested in and worked with is AR and VR. Where do you think those are headed?

Mazzitelli: When I was working at PTC, they were all about AR and VR, especially regarding factory and industrial controls. Where that's going as an industry is interesting. Now, you have the ability to overlay instructional information on any device. For example, with a camera, if I was pushing the shutter and it wasn't working, in an AR or a VR mode, I could put on my goggles and have all of the instructions on how to take it apart and troubleshoot it.

Feinberg: Not only that, but you can have the designer of the camera, or the tech service person who's half a world away, standing next to you and explaining what to do. I had a chance to try some of the stuff with Microsoft; they're doing a huge amount of work with extended reality (XR). With XR, there are many applications in different industries.

Mazzitelli: And the other value AR can provide is in replacing product manuals. As the people with many years of knowledge are retiring, the challenge becomes how to capture their knowledge so that it can be passed down to the next generation of people who will need it. With AR, you capture that and give them the information they need to be able to both run and troubleshoot issues by expanding on the physical world in an AR or a VR environment.

Feinberg: If you think about it, we are now over 100 years since the first VR, which was black-and-white movies.

Mazzitelli: I never thought of it that way, but that's interesting.

Feinberg: The first case of VR was the first audio recordings. Then, one of the very first movies showed a train coming right at you; people thought it was real and ran. My wife does a lot of semiprofessional photography, and we had a demo four years ago that was underwater. They called it VR, and it was augmented to the point that you were in it. This whale came up behind my wife, and I said, "Look at that." She turned around and screamed because it was so real. I also built a new computer that is aimed strictly at XR that has 100x the power of the big IBM computers.

Andy Shaughnesy and Linda Mazzitelli.

Shaughnessy: And some people say you could use it for training surgeons, so you could study surgery without having to cut anyone open.

Mazzitelli: Exactly.

Feinberg: Let's focus on the positive aspects of the technology and figure out the positive and negative effects of 5G. With 5G, once you take out the lag, your surgeon can be in Moscow, doing DaVinci robotic surgery on somebody in LA.

Mazzitelli: And looking at our industry, training is key for PCB design engineers. In the last 10 years, there have been very few new people in our industry. But I'm seeing younger PCB designers now, and AR and VR could be a big part of their training; they embrace new technology.

Shaughnessy: We appreciate your time, Linda. It's been great talking to you.

Feinberg: I enjoyed it too.

Mazzitelli: Thank you. **DESIGN007**

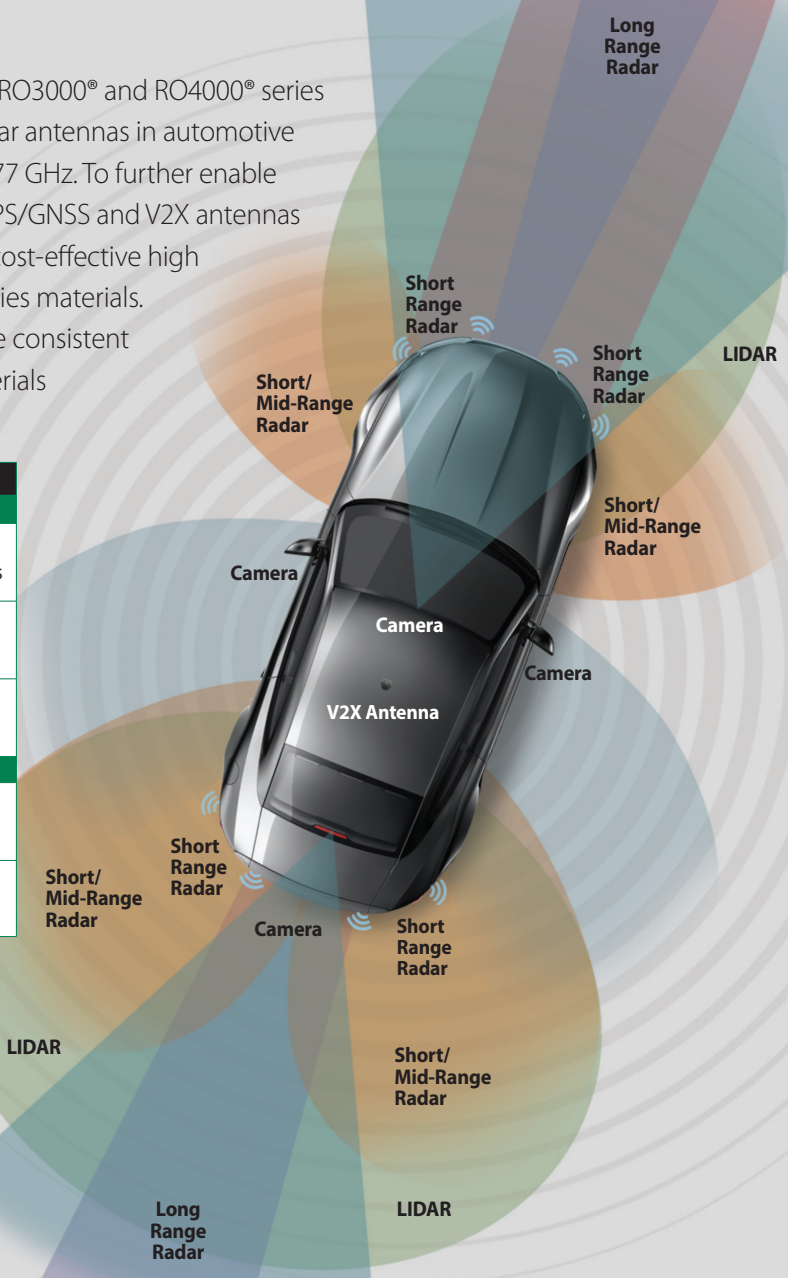
Rogers' Laminates: Paving the way for tomorrow's Autonomous Vehicles

Autonomous "self-driving" vehicles are heading our way guided by a variety of sensors, such as short and long range radar, LIDAR, ultrasound and camera. Vehicles will be connected by vehicle-to-everything (V2X) technology. The electronic systems in autonomous vehicles will have high-performance RF antennas. Both radar and RF communication antennas will depend on performance possible with circuit materials from Rogers Corporation.

High-performance circuit laminates, such as RO3000® and RO4000® series materials, are already well established for radar antennas in automotive collision-avoidance radar systems at 24 and 77 GHz. To further enable autonomous driving, higher performance GPS/GNSS and V2X antennas will be needed, which can benefit from the cost-effective high performance of Kappa™ 438 and RO4000 series materials. These antennas and circuits will count on the consistent quality and high performance of circuit materials from Rogers.

Material	Features
RADAR	
RO3003G2™ Laminates	Best in class insertion loss / most stable electrical properties for 77 GHz antennas
RO4830™ Laminates	Cost-effective performance for 77 GHz antennas
RO4835™ Laminates	Stable RF performance for multi-layer 24 GHz antennas
ANTENNA	
RO4000 Series Circuit Materials	Low loss, FR-4 processable and UL 94 V-0 rated materials
Kappa™ 438 Laminates	Higher performance alternative to FR-4

To learn more visit:
www.rogerscorp.com/autonomousdriving



ROGERS CORPORATION
Advanced Connectivity Solutions

USA - AZ, tel. +1 480-961-1382 • EUROPE - BELGIUM, tel. +32 9 235 3611 • **www.rogerscorp.com/acs**

Test Vehicles for Material Evaluation

Lightning Speed Laminates

by John Coonrod, ROGERS CORPORATION

There are many different types of PCB designs and constructions that can be used as test vehicles to evaluate electrical properties. If the test vehicle will be used for an evaluation that is specific to an application, it should be as similar as possible to the application. That may sound like common sense, but I have seen many companies use the same test vehicle for every evaluation they perform. However, if an evaluation is being done that is not specific to an application, but for general comparisons—such as evaluating the electrical performance of different circuit materials—there are multiple things to consider for the test vehicle. A proper test vehicle to compare different circuit materials would be a design and construction that takes into consideration the different material properties and have the least amount of PCB fabrication variables that can impact the results.

It is obviously desirable for a company to show it has well-defined and very consistent testing procedures for its electrical evaluation program used to evaluate different circuit materials. To do this will probably cause the evaluation procedures to be more complicated, but, in the end, the results will be more meaningful for the materials being evaluated. For example, it probably makes sense to have a different set of evaluation procedures for materials which are significantly different, for example, FR-4 materials would have a different evaluation procedure than high-frequency materials.

For evaluating high-frequency materials, there are many different test vehicles used. Some of the more common test vehicles are ring resonators, transmission lines, and delay lines. The circuit construction should also be a consideration; often, the ring resonators are used as a microstrip. The transmission lines and delay lines are also commonly microstrip, but sometimes, they are a stripline construction. If the evaluation is purely electrical performance related to the material only, the simplest construction and PCB fabrication process should be used. If the evaluation is a combi-

nation of electrical performance and how the materials are used in the PCB fabrication process, the constructions will probably need to be more complex and designed to be specific to the process concerns of the PCB fabricator.

The evaluation circuit used to characterize materials electrically should be a simple construction with a simple circuit design. It is highly recommended to evaluate the material at lower frequencies and with thick substrates. This will minimize some of the variables that can make the electrical evaluation less accurate. Basically, it is difficult to obtain accurate results at high frequencies. Also, when using thinner materials, the influence of etching control has more effect on the electrical results. Typically, when a thinner circuit is used, it will have a narrower conductor to maintain a controlled impedance and the variation of the conductor width due to etching control at the PCB fabricator that will have a greater influence on the electrical performance of the circuit as opposed to a thicker substrate with a wider conductor.

In general, I am not in favor of ring resonators; however, at lower frequencies and with a thicker substrate, the ring resonator can be a good test vehicle. The ring resonator should be based on a double-sided circuit (microstrip), loosely coupled. Lower-order nodes should be tested; test at frequencies below 6 GHz. If the circuit is a double-sided, non-plated through-hole (PTH), bare copper circuit, then this is a pretty good test vehicle, which will have minimal electrical variation due to PCB fabrication variables.

I have seen many companies use stripline circuits for their test vehicles. Due to the nature of this construction, these circuits are more influenced by PCB fabrication variables than is the simple test vehicle and construction I recommended (microstrip). In the case of stripline, typically, a core material and prepreg are used to make the stripline construction. These materials are very often dissimilar in electrical properties, which can confuse the electrical results of the evaluation. Additionally, the thickness control of the prepreg layer due to circuit fabrication can have a significant impact on the electrical properties of the circuit. The PTH vias, which are used to connect the top and bottom ground planes of the stripline, can also influence the electrical performance of the circuit. If stripline is necessary for the test vehicle, again, a thicker test vehicle being used at a lower frequency is advantageous to avoid some inaccuracies in the electrical evaluation.

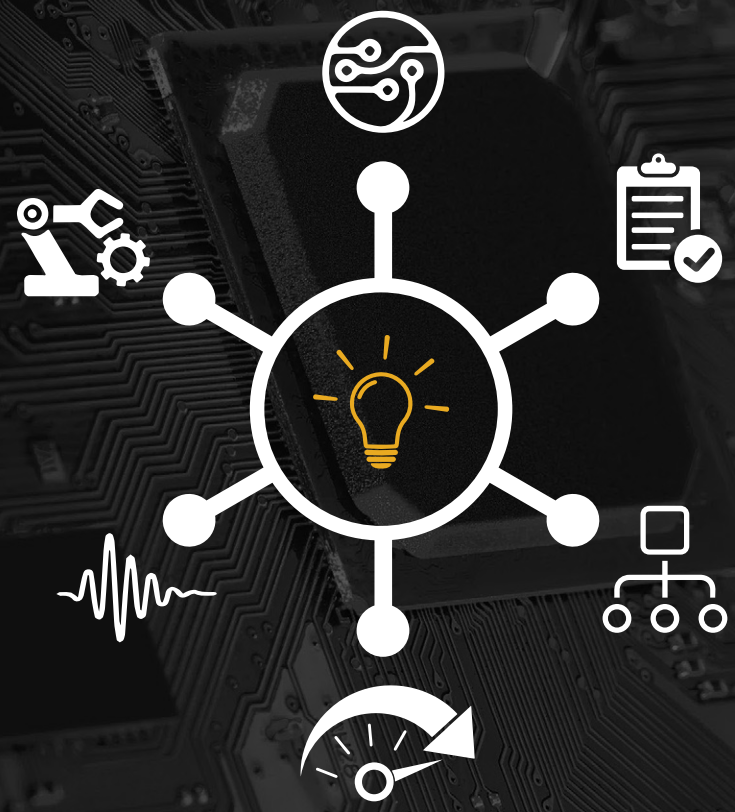
These very simple guidelines I gave for the test vehicle and construction would be good for a materials-only evaluation. This evaluation would simply be comparing different materials with the same test vehicle, side-by-side. However, it is common that the material evaluations need to be on thinner materials, higher frequencies, complicated multilayer PCBs, etc. When that is the case, the designer needs to consider the effects of PCB fabrication influences on the material properties.

A multilayer board using PTH technology means the final conductor thickness will have some normal variation. For a gap-coupled ring resonator, the copper thickness variation can impact the electrical results and cause inaccuracies in determining the Dk or Df values related to the circuit material. Some test vehicles are more sensitive than others to the copper thickness variation. As mentioned, the gap- coupled ring resonator can be affected by normal copper plating thickness variation, and so will a grounded coplanar waveguide (GCPW). However, a microstrip transmission line circuit is much less affected by copper thickness variation for its electrical performance. Additionally, final plated finishes and their normal thickness variations can also impact the accuracies of the test vehicle.

The bottom line is the engineer who defines the test vehicle (circuit design and construction) needs to put a lot of thought into how the normal PCB fabrication variables can impact the test vehicle in a way that influences electrical evaluation accuracy. **DESIGN007**

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

RISE ABOVE THE AVERAGE



UPGRADE YOUR KNOWLEDGE– Gain exclusive industry insights from the experts and achieve PCB design success with our growing library of free e-books, videos, blogs, and more.

I'm Ready to Learn ›

EMA Design Automation®

Scott Miller

What You Need to Know: Designing Complex PCBs

Feature Interview by the I-Connect007 Editorial Team

The I-Connect007 editorial team sat down with Freedom CAD's Scott Miller to talk about the industry's demand for more increasingly complex PCBs, and the challenges this presents. They also discuss Freedom CAD's in-house training programs, the company's recent book authored by Scott, and why communication is such an important tool in a PCB designer's toolbox.

Barry Matties: First, tell us a little bit about Freedom CAD and what you do.

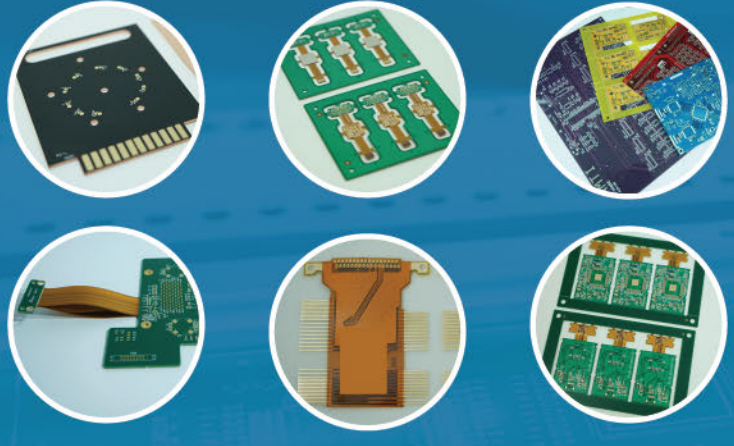
Scott Miller: Freedom CAD was founded in 2003 by a husband and wife, Lou and Lauren Primmer. In May of 2003, they acquired Plexus' Nashua, New Hampshire, design center, and started Freedom CAD Services with 12 employees. They were fortunate to be able to bring some of the best designers from Plexus with them to start the business. Since then, we have fluctuated between 40 and 60 employees, depending on how the demand for our services has swung. Our focus is on PCB design, servicing all of the aspects of PCB design, electrical engineering, mechanical engineering, and signal, power, and thermal analysis. However, we're best known in the market as a PCB layout specialist for complex, rule-driven designs. We also provide turnkey manufacturing solutions through strategic partners to provide customers with the ability to get from what I call the "CAD to the lab" with one-stop shopping.

Matties: In the years that you've been doing this, you've seen the complexity of the boards increase.

Miller: Dramatically. And the funny thing is that everybody's perspective of complexity is big. Big boards can be very complex. However, the irony is that small boards can also drive a lot of the complexity because you have less space to work with. Everybody's trying to make things smaller and more power-efficient. We're used to being challenged by large, complex boards. We still do those types of designs with multiple high pin count FPGAs and

Prototype/Production PCB Manufacturing & Assembly

- Founded in 1992
- Extensive experience with complex PCBs
- Multilayer boards
- Flex and rigid-flex
- HDI
- MCPCBs
- Quick turnaround
- One-stop PCB assembly
- Worldwide shipping & rush delivery



Contact us today for a speedy quote!

Shin Yi PCB

www.sy-pcb.com.tw/en

+1-437-987-6541
shinyih.pcb@gmail.com

+886-222013456
shinyihl@seed.net.tw

multiple banks of DDR, but now we see more of these smaller boards that are also complex and tight. They are similar to a system on a chip, but still a PCB. They can be challenging, with the HDI technology and high-component densities, and are similar technologies to what you find on a larger-scale board, but now you're confined to a much smaller footprint to work within. The cellphone is a perfect example of advancing technology that has driven so much capability into a handheld device.

Matties: Have you seen a shift in the type of customers as well?

Miller: With the IoT or wearables industry, there's a whole other realm of customers that you wouldn't have thought of as good candidates for PCB design and layout. But our core customers still come from the military/aerospace and telecom/datacom sectors. We see much more activity out of the automotive industry, too, and there's a growing demand from the AI industry.

With the IoT or wearables industry, there's a whole other realm of customers that you wouldn't have thought of as good candidates for PCB design and layout.

Matties: So, your customer list must be growing as well.

Miller: It's a challenge, but it's a good challenge because you want to make sure that you're working with the right customers. As we are identified as a solution for customers, we try to work our way through to make sure they are a good fit for us, and we're a good fit for them, more importantly.

Matties: What makes a customer a good fit?

Miller: Typically, for us, they're a technology-driven customer looking for an expert solution, not just to connect the dots design house. If it's a company that has a lower technology, then we're probably not the best fit for them. Our usual customers are looking for a value-adding supplier—someone who is going to provide feedback and make recommendations and suggestions, such as working through optimized stackups or making placement recommendations, and all of the things that can make the design more efficient and help to get it right the first time.

Matties: You recently published *The Printed Circuit Designer's Guide to… Executing Complex PCBs*. That book has been wildly successful with a lot of downloads and positive reviews. What feedback have you received, and how is that affecting you?

Miller: It has been great by helping us get more exposure and opening the doors to new customers. Some of the downloads are from customers that already know us, but it has helped to raise our credibility with them. The feedback has been that it was very well-written and hit on the hot points. Readers felt that they got something out of it, and the book provided something that wasn't in the marketplace before. That's the feedback I've received. It's not just in the design itself, but it's how you communicate, share data, and do things efficiently and effectively.

Matties: It wasn't a book about you, but about helping the industry because you are the experts and have been doing this for many years.

Miller: The origins of Freedom CAD go back to the earliest days of PCB design when laying out a PCB was done by putting tape on Mylar. So, we have been through all the evolution of PCB design. We wanted to share these years of experience in the book.

Matties: With that topic of complex PCBs, a lot of companies are now moving into that realm

where maybe they didn't have to do that before, and there are design challenges. Do you see any instances where people are coming to you with their designs just for analysis and review?

Miller: We are. I think the visibility from the book and some of the other marketing that we've been doing is opening customers' eyes who may not have been thinking about going outside for a design review. This past week, we received a request to perform a review from a customer that had an existing design to perform a review. They were going to take the design into volume production, and they came to us and asked us to do a review of it, to add test points, and to look for improvements for manufacturability.

Andy Shaughnessy: What are some of the biggest challenges that you see as a company and for your customers?

Miller: If you're familiar with the PCB design industry, it's no surprise that it's a graying industry. The majority of the North American PCB designers are age 50 and above, with lots of experience, but also closing in on retirement. One of the biggest challenges is to identify and develop talent that can replace those who are going to exit the industry for retirement or whatever the reason.

Shaughnessy: And it seems to be changing a little. Do you see more young people in the design community?

Miller: As PCB design becomes more visible, younger people are becoming interested. As one of the largest and more visible design companies, we are approached by young people who are interested to know if we have trainee positions. Some of them are individuals with parents who were or are designers, but some become interested as they hear and learn about the PCB design career path. I do see things changing, but more older people are leaving the industry than younger people coming into it.

Shaughnessy: You mentioned training. Do you work with schools?

Miller: No, we have a homegrown training program. We've run four different training classes where we have hired interns to teach them the process of PCB design. We have a syllabus and a proven methodology for training them on all aspects of layout. We start with the basics of silkscreens and assemblies and then help them understand schematics, routing, and placement. We take them through the whole process. We have been successful to the point where most of our young designers are the ones we brought in and trained.

Shaughnessy: You can't assume that everybody knows about that.

Miller: Right. The value to trainees is they are partnered and mentored by some of the best designers in the business. They are exposed to a lot of experience and knowledge. In addition, one of the tools we have within our company to help promote shared learning is an email alias called the "Designer" alias. This alias copies our entire design team and is like an intracompany helpdesk. If a designer is tackling something they haven't dealt with before, they can ask questions of the team. As we learn a new tip or trick, the designers can share it via Designer alias to the 30 or so designers that we have. And if they find a way to do something or there's a new technology or technique that's either board-related or fabrication-related, we share it. To be part of that environment as a young designer can be more valuable than a classroom only experience.

Nolan Johnson: As you're recruiting and talking to young engineers and designers, what are some of the attributes you look for? For example, do you look for specific job skills or a general attitude?

Miller: I'll steal a phrase from a former employee who worked for us, Robert Jardon; it's aptitude and attitude. We look for somebody who has an ambition and is seriously interested in

learning the art of PCB design. Then, do they have the aptitude to pick it up and the attitude to dig in, learn, and keep learning? There's a lot to learn. Our best designers, even with years of experience, continue to learn every day. It's not, "I'm now a senior PCB designer. I know what I need to do." The best designers recognize that to be the best and add the most value in their companies, they need to continue learning.

Shaughnessy: We certainly need more mentors in the design world.

Miller: We need to mentor younger people to replace those who are going to retire. We need to make the mentor-trainee connection so we can teach them not only the design skills of PCB layout but also professional communication and problem-solving skills. Mentoring helps to facilitate a broader level and faster pace of learning.

Matties: Is there anything that we haven't talked about that you feel like we should be sharing with the industry?

Miller: What has become an interesting trend is that PCB layout is more than just physical design skills; it is influenced by signal and power integrity simulations and thermal analysis. There are lots of challenges that go into a successful design, and the upfront analysis and simulations enable the designer to be much more likely to be successful. The days of designing, measuring, figuring out what we did wrong, and fixing don't cut it today; that's not efficient, and you can miss your market window if the board design needs to be redone because of crosstalk or voltage drop. We perform more signal and power integrity simulations and thermal analysis as part of our design process now than ever before. Five years ago, it used to be mostly signal integrity for high-speed designs, but now it often requires power integrity and thermal analysis.

Matties: We were talking about design review, but maybe there's a pre-design review meeting

where people can also come to your organization before they embark on their journey.

Miller: Yes, particularly with complex PCB designs. They require some detailed planning before you even start the design to get the results you expect. Floorplanning the design is important; it's a collaborative effort between the design engineer and the layout designer when we do these complex designs. It requires design rules and guidelines, crisp mechanical data, placement guidance, stackup selection, and the use of HDI. All of these factors need to be carefully reviewed before the layout begins.

Matties: And collaboration is something that people have been talking about for many years but seem not to follow through with. Do you see more collaborative efforts out there?

Miller: Absolutely, even to the point where we work with a number of customers where their own layout team works on a design, passes it to us, and then we work on it, pass it back to them, and then they work on it. We have truly tried to become an extension of our customer's team in a seamless way, and it works.

Matties: When I look at the reasons for coming to a design service like yours, it's because you have a broader pool to draw from for that knowledge rather than relying on one designer. As you pointed out, there's so much to know today, and no one designer can know at all.

Miller: A perfect example is that most companies may develop one or two backplanes a year unless they're a specialist in backplane technology. We do probably 25–30 backplanes each year because of the number of customers with which we work. As a result, we are aware of the unique requirements and challenges of backplane design because we design backplanes much more frequently than most companies. Another example is that we've dealt with a lot of customers who have migrated from DDR3 to DDR4 and were not familiar with the different routing requirements. We've been doing DDR4 since early in its evolution due to some of our customers being early adopters. These are just a couple of examples of how we are exposed to leading-edge technologies early in their evolutions.

Matties: How has the additive process changed the design approach? Are we designing the same way when you design a board for additive?

Miller: We worked with a company to do some of the additive process designs for their testing, and it is little difference in terms of the design impact. The major difference is how fine they can control the line widths and spacings because it's adding copper versus removing it. With the additive process, there's no trapezoidal effect on the traces. And they can add a much thinner layer of copper than a one-quarter ounce or even less.

Matties: In your design work, do you see more people going to that additive process?

Miller: Not yet, predominantly because it still has some certifications and qualifications to get through.

Matties: But there is certainly a trend to move in that direction.

Miller: Right, it fits the smaller packaging. You can go down to one-mil lines and control it accurately.

Matties: I would think that as everything gets tighter, more of the power integrity and other attributes you mentioned come into play.

Miller: To some degree. From a design standpoint, the physical characteristics don't change it, but from a technological standpoint, you can do things differently with thinner lines, smaller spaces, and better control.

Matties: And this is where the education of a design service comes in because you are doing this where it might be new for somebody else, and you have already mastered the learning curves.

Miller: Exactly. And fortunately, we're involved up front with one of the additive companies to help them develop their process.

Matties: That's a big advantage. Do you work closely with fabricators?

We work with most North American fabricators to validate our stackups for DFM, and when we manage the prototyping for our customers, we work directly with them.

Miller: We do. The design isn't complete until the board is built, and the OEM puts it on the bench and figures out whether it works or not. We work with most North American fabricators to validate our stackups for DFM, and when we manage the prototyping for our customers, we work directly with them. The intimacy between the design and fabricator continues to get closer and closer because of the challenges around stackups; as mentioned in our book, picking the right stackup is the cornerstone of a successful design.

Matties: We commonly hear that designers need to know the manufacturing process, and then a smaller group will say that they don't need to be the manufacturing experts. What sort of feedback should a designer look for from a fabricator?

Miller: Describing the requirements in a clear manner is important. A lot of times, designers will say, "I need 100 ohms on these layers and 90 ohms on these layers. I need to use blind or buried vias. I need to carry this amount of power. I need a material that can deliver 40-gigabit data rates," and so on. The fabricators will come back with their recommendations. This is critical because we start with something that falls within the capabilities of the fabricator. As the design progresses, we may need to refine the stackup to address unforeseen challenges in the layout that require changes. For instance, we may need to shuffle the stackup due to the results of the signal or power integrity analysis. When we do those kinds of things, we need to go back to the fabricators and make sure that we aren't doing something that's going to cause a DFM problem. Our connection and communication with the fabricator are vital.

Matties: Regarding post-production, is there any feedback that you look for after the fact from even the OEM or end-user?

Miller: There's always feedback. Even though we go through a DFM process during the design process, we'll specifically ask our customers to go back to their PCB fabricator and PCBA assembler for their feedback. We recognize that each shop has their unique capabilities, so we try to get that feedback and incorporate it before we ship the design. When there is a requirement to do a re-spin on a board design, we like to collect the feedback from the fabricator or assembler regarding yield issues and see if we can make any changes for yield improvements on future spins.

Matties: When you're working that close up front, it's almost more of a predictive engineering approach than a DFM approach.

Miller: To some degree, it's predictive engineering. You're trying to do what you can to make yourself successful up front, which is also a part of our book.

Matties: It's had some great reviews, and thank you for putting that out there for the industry. Scott, we certainly appreciate your time today.

Miller: Thank you, Barry. I appreciate it.

Scott Miller is the author of *The Printed Circuit Designer's Guide to… Executing Complex PCBs*. Visit I-007eBooks.com to download this book and other free, educational titles. **DESIGN007**

Nanotec Develops Flat External Rotor Motor With High Speed

With the DF20, Nanotec now offers a flat external rotor motor with a diameter of just 20 mm. It is available with two different windings for 12 and 24 V and is equipped with digital Hall sensors for simple control.

The DF20 features a rated power of 5 W and a rated speed of 5,200 rpm. The open design without rotor housing ensures optimal heat dissipation–even at high speeds. Thanks to its compact construction with flat ribbon cable, this brushless DC motor is ideal for applications with limited space, such as robot grippers, medical pumps or prosthetics. An adapter board is available for test purposes as an accessory.

The DF20 complements Nanotec's product family of external rotor motors, which includes motors with diameters of 32 and 45 mm.

(Source: Nanotec)

U.S. CIRCUIT
33rd
ANNIVERSARY
1985 ★ 2018

Since we first began in 1985, delivering the highest quality products possible has been our first priority. Our quality control systems are certified and we renew our commitment to quality each and every day.

Delivering quality products on time, combined with timely and accurate communication from our customer service team is how we create lasting relationships with our customers.

Take a look at our quality »

COMMERCIAL • MILITARY • MEDICAL • BAREBOARD

U.S. CIRCUIT

U.S. Circuit, Incorporated
2071 Wineridge Place
Escondido, CA 92029
+1 (760) 489-1413
www.uscircuit.com

ISO 9001:2008 REGISTERED

DEPARTMENT OF DEFENSE · UNITED STATES OF AMERICA

ITAR International Traffic in Arms Regulations Registered

Fabrication Notes and the Design Process

Connect the Dots

by Bob Tise and Matt Stevenson, SUNSTONE CIRCUITS

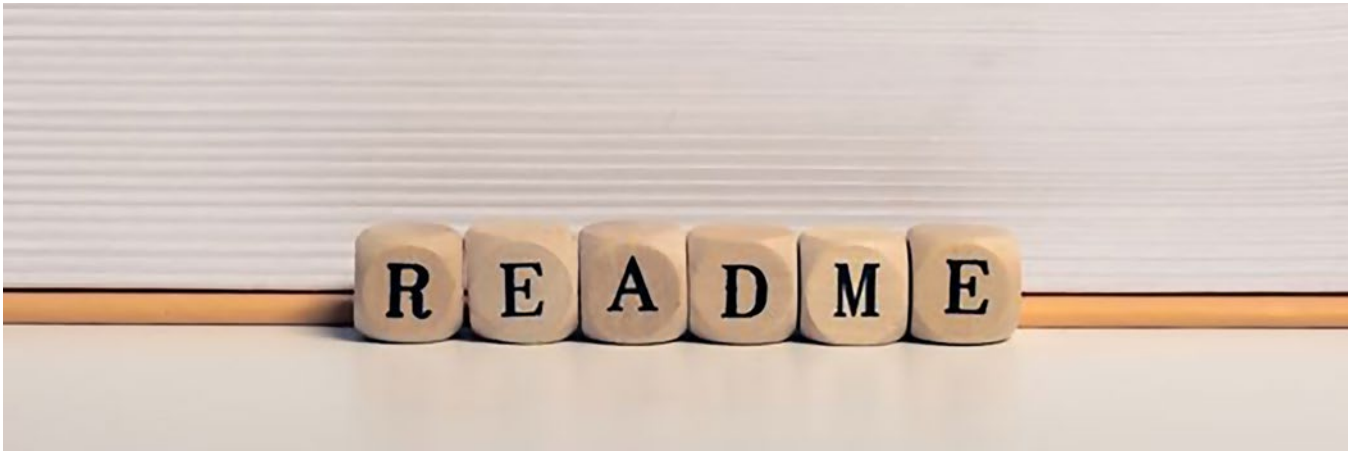
When is it appropriate to add full fabrication notes instead of just a simple README file, and when should you just send the Gerber file for fabrication? Each of those approaches to documentation—or lack thereof—may be appropriate, depending on which stage you're at in your design process.

Test Boards and Others

For test boards, bench tools, or other boards that will never be used by anyone but you, you'll probably use a prototype or run-as-sent service. For these boards, and in these scenarios, you likely won't have notes for UL marking, electrical testing, destructive testing, cross-section analysis, and so on. The extra documentation you create at this stage may be minimal or none; in fact, you might send in the Gerber file alone or attach a README file that says, for example, "Nonfunctional circuit for mechanical fit only." This will clue the fabrication house that the DFM checks they normally run—checks this board will fail—are not necessary. Occasionally, we also see a fully documented fabrication drawing with a large note telling us to *"ignore fabrication notes in conflict with manufacturer's standard processes."*

Early Product Development

When you're developing a new product, you can expect to go through several iterations of your boards to prove out your subsystems before arriving at something that starts to resemble a finished product. At this stage of development, when you're ordering boards, you might want more control over elements like thickness or layer stack-up than you took with your test boards. Because you're still in early development, keeping development moving along may be your priority. You might add an electrical test to make sure the board works as designed, but you're not adding in a lot of process controls or special handling instructions that might slow development or add costs. Documentation becomes a little more robust here than what you needed for your test boards, but just enough to get what you need to continue development.



First Time vs. Next Time

Deliver optimized designs the first time. By shifting manufacturing knowledge into the design process, concurrent DFM is now a reality, streamlining the design-to-manufacturing release process.

Shift left with Valor® NPI.

Mentor®
A Siemens Business

mentor.com/npi

Nearing Product Release

Once you get close to product release, your boards need to be pretty well-defined. At this stage, your documentation should include a full set of notes with specific callouts, material specification, silkscreen and solder mask, copper weight and plating thickness, test and inspection requirements, and so on. As you move into the final rounds of development and then production, you'll need to add more documentation, including detailed manufacturing requirements and inspection criteria to meet commercial standards. At this stage, you need full fabrication notes because quality is critical here; missing important documentation can be expensive and severely impact the quality of your final product.

But Wait, There's More!

Depending on what you're designing, you might need even more documentation. If you're designing for the transportation, medical, defense industries or another highly regulated industry, there's a lot more documentation you'll need. In these scenarios, the sky's the limit in terms of what standards you might have to meet and the accompanying documentation you need to supply. Any of the following may apply to the specific product you're building: destructive testing, IPC-6012 Class 3 or Class 3A, ITAR, FDA requirements, etc. When you get into these areas, your notes may take up a full sheet in your drawing package and refer to design and manufacturing specifications that are hundreds of pages long. These documents can cover not only quality manufacturing processes, but also things like materials sourcing, record keeping requirements, test specifications, labor and purchasing, ethics, handling, packaging, and any number of standards you may have to meet to bring your product to market.

Create the Right Notes for the Right Stage of Your Design

As your boards get more complex and complete, so should your fabrication notes. And while documentation itself is important, it is equally critical to create the right level of documentation for the development phase you're in. Often, developers add a full documentation set along with an order for low-cost test boards. PCB manufacturers can't deliver the same level of service for inexpensive test boards as they do for production boards, and it will save you time and money to set your expectations—and documentation—appropriately for what you're ordering.

Some prototype or run-as-sent services will automatically delete anything that isn't a Gerber file, and others might see pages of fabrication notes for a batch of test boards and stop the order to review the notes and call or email you to discuss what the need really is. Often, if someone really just needs some test boards but sent along a full set of canned fabrication notes, the end result is that the order for test boards is delayed while the PCB manufacturer vets the requirements with the customer.

To help ensure that your PCB manufacturer can get the boards you need in the timeframe, and at the price point you need for your stage of product development, create and send only the fabrication notes that are relevant for that stage. You'll save yourself time and money and get your product to market faster. **DESIGN007**

Bob Tise is an engineer at Sunstone Circuits, and **Matt Stevenson** is the VP of sales and marketing. To read past columns or contact Tise and Stevenson, click here.

Bob Tise

Matt Stevenson

Superior resins engineered for challenging environments

In Encapsulation Resins our comprehensive range meets the increasing demands of the electronics industry. Whether it's epoxy, polyurethane or silicone resin systems, our products are designed to protect and insulate printed circuit boards (PCBs) and electronic components from the threats of harsh and challenging environments.

Encapsulating the entire device in resin offers protection against moisture, vibration and rapid changes in temperature, thus offering superior performance under extreme conditions.

Whether for general purpose or tailored for individual requirements, every electro-chemical solution is within our spectrum. Our specialist approach to problems has established us as a key provider and developer to the technical electrical industry, both commercial and domestic.

Isn't it time you discovered how Electrolube can serve you?

Scan the code to discover our full spectrum of superior Encapsulation Resins and electro-chemical solutions.

Tel: 800-474-1472

www.electrolube.org

ELECTROLUBE

THE SOLUTIONS PEOPLE

Electronic & General Purpose Cleaning	Conformal Coatings	Encapsulation Resins	Thermal Management Solutions	Contact Lubricants	Maintenance & Service Aids

Roger Beers

Are ISO Procedures Strangling Innovation?

Feature Interview by Andy Shaughnessy
I-CONNECT007

Andy Shaughnessy recently spoke with Quantel's Roger Beers, who voiced his concerns about the struggles engineering departments face when dealing with new technologies within ISO procedures. While great for repeatable processes, Roger says ISO in its current state may be preventing engineering growth.

Andy Shaughnessy: Roger, give us a quick background about yourself and how you got into PCBs and your position at Quantel USA.

Roger Beers: I grew up on the southern Oregon coast. My first computer was the Commodore VIC-20; next, I moved up to "the pet" and then the Commodore 64. The Commodore was my introduction to the software world and later into the engineering world. That's where you learned how to tinker, read Compute!, use someone else's code to make games just to play, etc. My interest in electronics, computers, and software all came about through the old Commodore days. Currently, I'm an electrical engineer on laser control systems for Quantel Laser USA.

Shaughnessy: What would you say is your sweet spot for your company?

Beers: We make a lot of high-precision lasers, such as LIDAR for defense, medical, and self-driving cars. We have high-power lasers for military, industrial, and scientific applications, or whatever the application is. A lot of research laboratories come for the high rep rate.

Shaughnessy: That's a fun area to be in. We talked earlier, and you had an interesting take on ISO certification. Tell us more about that.

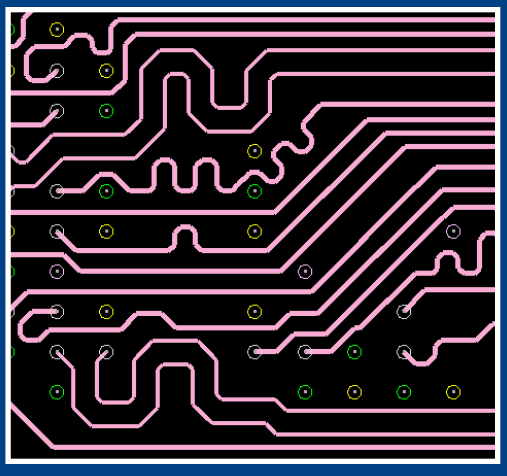
Beers: ISO is basically "document what you say, and say what you do." Right now, with the new ISO coming out, it's all about risk-based analysis and mitigating risks for the company (stockholders). From a marketing standpoint, it's a selling tool as well, but to me, it should all be about a quality system. The big thing I see with a quality system is how do you keep up with technology when your ISO procedures are written in such a way that you can't move ahead with new software because you'd have to tweak documentation to keep up with it? I think management could be holding back engineering if you do everything according to ISO procedures. There is a fine line between keeping procedures vague and not so detailed but still driving consistent and best design practices.

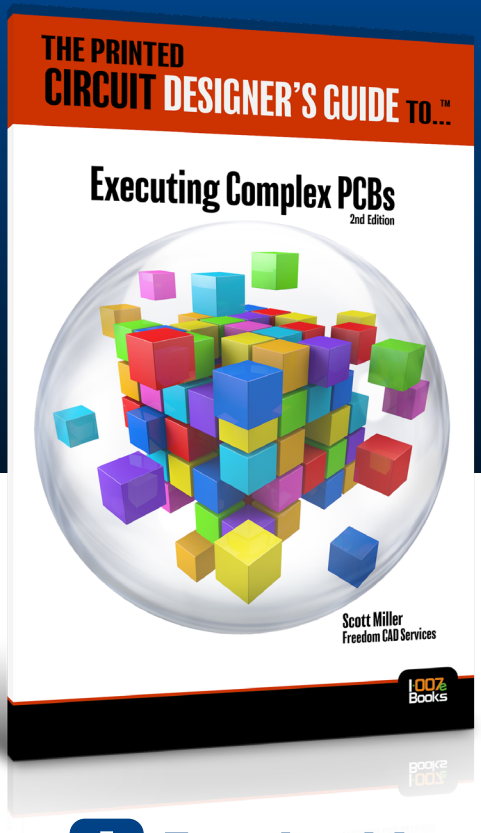
Shaughnessy: What do you find yourself doing to deal with this?

FREEDOM CAD EXPERT PCB DESIGN TIP #2:

DON'T MAKE YOUR HIGH-SPEED TRACES TOO LONG!

Today's PCB designs require more diligence in order to meet the requirements of today's high-speed microprocessors. At higher data rates, insertion loss becomes more critical. This results in signal length limitations due to insertion loss, very tight impedance matching, short stub lengths and the use of higher performance PCB laminates.





Free book!

"Freedom CAD drove the tasks to make the board design come together efficiently and effectively. They became a seamless extension of our team."

Laurie Fraser
Engineering Services Manager
Draeger Medical Systems, Inc.

FREEDOM CAD SERVICES

(714) 490-7600 West Coast
(603) 864-1303 East Coast

freedomcad.com

Beers: To me, it's the buy-in we have from management that the technology is good, and we need to keep up with technology. ISO is a great management tool, as well as a quality system to keep things repeatable. All manufacturers want repeatability, and ISO is a good process for that. How do you have ISO in the engineering department, and how do you make your designs ISO-compatible? That's where you just look at having your procedures in such a way that you can keep up with changing software and development in technology and let the engineers have the free rein to go; at least have the checks and balances for risk-based analysis.

Shaughnessy: I haven't heard anyone put it that way before. What do you think is the answer? Should engineers get more involved with ISO standards?

Beers: From a manufacturing standpoint, you want ISO. To me, it's the best way to produce things—when you have good ISO work instructions, and things are done in a way that's repeatable every time, you ship out great products. In the development cycle, engineers don't do the same thing every time. How do you document something when you're not doing it the same way every time?

Shaughnessy: In some OEMs, the product is almost custom every time.

Beers: Right. To have an ISO procedure to adapt customization is huge. Your procedures need to have the guidelines for where the engineering department goes, but not down to dotting the "Is" and crossing the "Ts" to box in engineers so they can't move ahead with new technology. You need process-based procedures that are flexible and designed to move forward.

Shaughnessy: Is this something that you all discuss a lot at your company?

Beers: We have weekly and bi-annual meetings on ISO quality and how to keep the procedures moving forward, but it takes time and manpower to update procedures. And when you don't have an engineering mindset on your ISO team, how do they understand keeping procedures up to engineering standards?

Shaughnessy: You would have to have a big engineering presence on the ISO team. It's counterintuitive because ISO exists in part to encourage innovation.

Beers: The big thing moving forward is to have engineering buy-in that this is important, but yet vague enough that you can move ahead. I don't know how else to say it.

Shaughnessy: Because if you're too innovative, you run the risk that you might be out of ISO compliance, which is kind of the opposite of what ISO was supposed to do.

Beers: ISO on the production side is great. From an engineering management risk-based analysis, I'm still trying to figure out how to get that incorporated through the whole company rather than just to a production-based environment.

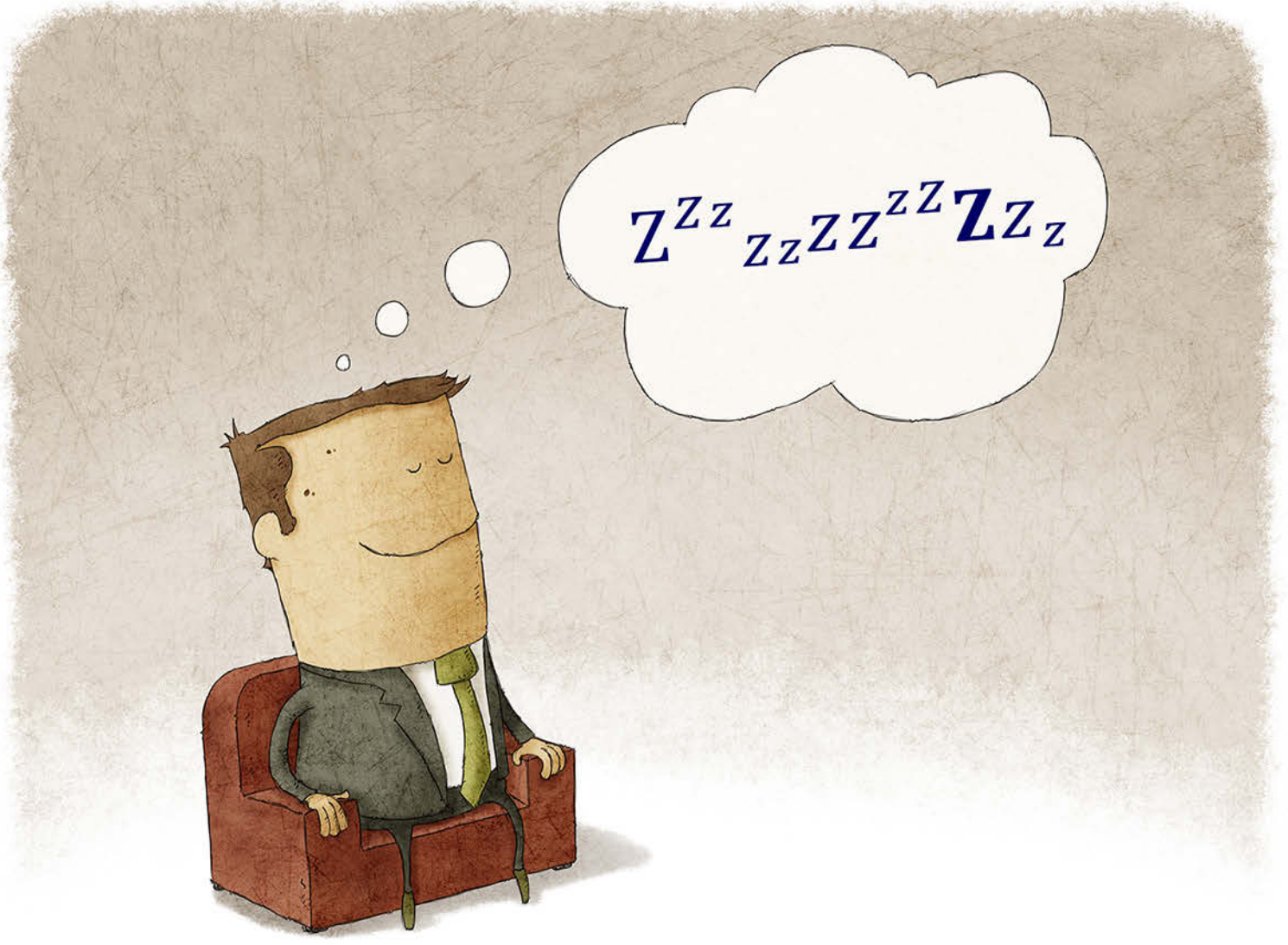
Shaughnessy: That's interesting. I'd like to see what our design engineer readers think about this.

Beers: People should know that it is a concern.

Shaughnessy: Roger, thanks for your time today.

Beers: Thank you, Andy. **DESIGN007**

We DREAM Impedance!



Did you know that two seemingly unrelated concepts are the foundation of a product's performance and reliability?

- Transmission line impedance and
- Power Distribution Network impedance

DISCOVER MORE

iCD software quickly and accurately analyzes impedance so you can sleep at night.

iCD Design Integrity: Intuitive software for high-speed PCB design.

"iCD Design Integrity software features a myriad of functionality specifically developed for PCB designers."
– Barry Olney

iCD
In-Circuit Design Pty Ltd

www.icd.com.au

PCB007 Highlights

The Government Circuit: U.S.-China Tariff War Threatens American Jobs and Investments ▸

Fifty-five percent of companies report they are facing higher costs as a result of higher tariffs, which are affecting, on average, about one-third of the total dollar value of the products they import. Some companies say their costs have increased more than the direct costs of the tariffs due to higher administrative and operational burdens to sort it all out. Chris Mitchell explains

Miniaturization Continues: Day 3 productronica Coverage ▸

The conversations on day 3 of productronica continued, including topics such as flexibility, 5G capabilities, and increased data and intelligence. The theme from test and inspection equipment providers included increased capabilities in sensing, material handling, and visual inspection technologies. And through it all, one common perspective emerged time and time again: PCB manufacturing is now approaching the levels of line, trace, and component density that were once limited to the surface of a silicon wafer.

Future Trends in Flying Probe Testing ▸

Peter Brandt, director of sales for Europe and Japan at atg, sits down with Pete Starkey and Barry Matties, gives his views on market requirements and testing technologies, and explains how flying probe testing is becoming the industry standard at all levels of production—and in many cases, the only practicable solution.

Four Key Developments From TPCA Show ▸

The recent TPCA Show 2019 attracted 31,926 attendees who had the chance to visit 1,432 booths erected by enterprises from 420 countries. In addition, the topics brought forth by the 57 seminars and keynote forums held at the recent IMPACT 2019 have directed the future trends of the PCB industry. The organizer found the following key extended topics.

SCHMID Group Installs PlasmaLine at Hofstetter PCB AG ▸

SCHMID Group installed the first PlasmaLine comprising ICP etch and sputter deposition of its Korean JV SCHMID AVACO at Hofstetter PCB AG in Switzerland.

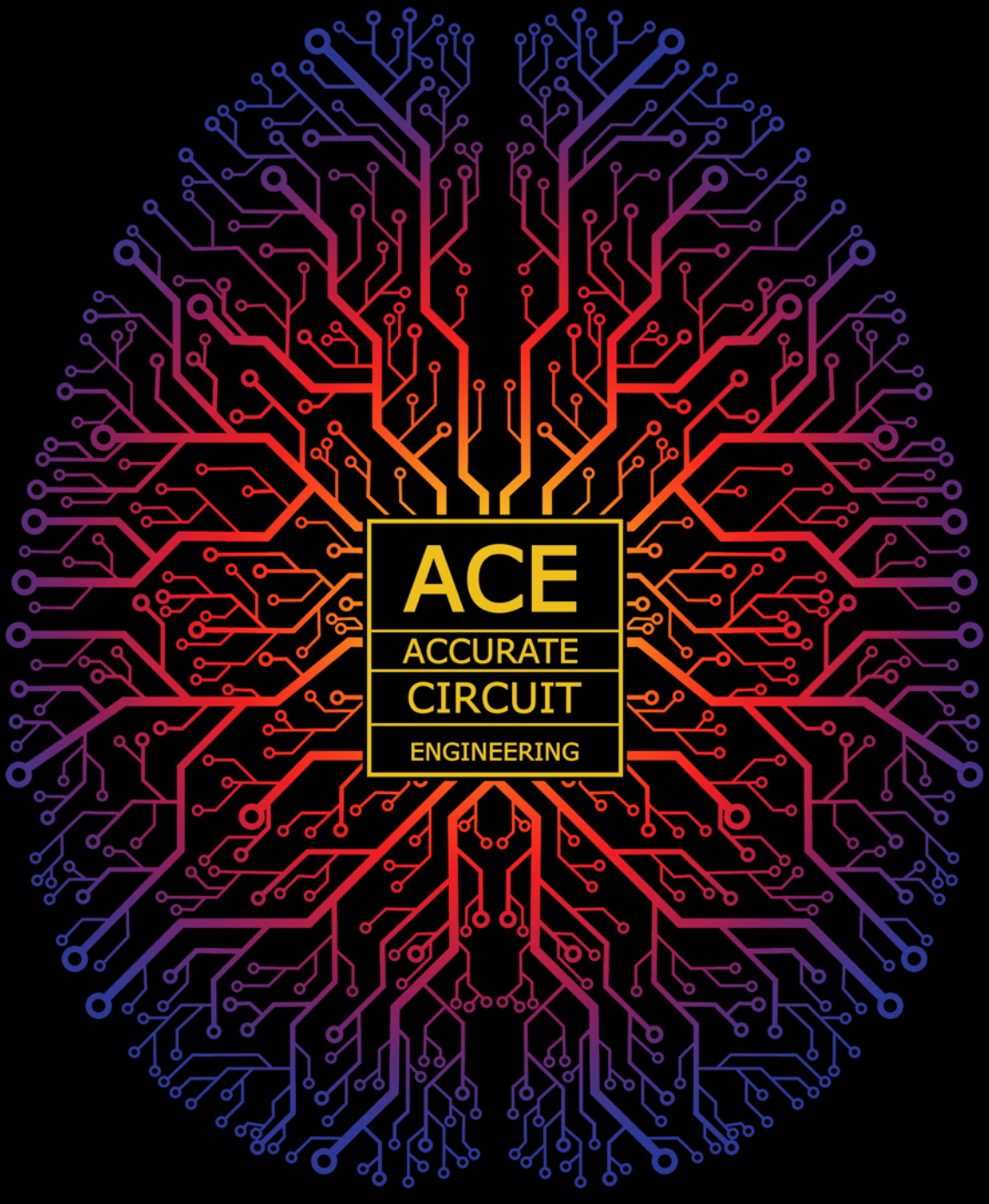
Multilayer Press Technology Using Magnetism to Produce Lamination Heat ▸

A revolutionary concept in multilayer press technology has been developed that uses electromagnetic energy to heat the existing stainless-steel separator plates with a never-before dreamed-of accuracy and precision. The heating and cooling systems—embedded within a robust hydraulic press inside a vacuum chamber design—are controlled using a temperature feedback loop that guarantees perfect fidelity between the press recipe and the press result.

International Electronics Circuit Exhibition Shenzhen Presents Technical Conference ▸

Organized by Hong Kong Printed Circuit Association (HKPCA) and China Printed Circuit Association (CPCA), The 2019 International Electronics Circuit Exhibition (Shenzhen)—formerly the International Printed Circuit & APEX South China Fair—will present the latest market and technology innovations and trends.

Don't Overthink Your Prototype Builds



Go with the team that can make your ideas a reality...

RF • MICROWAVE • HIGH-SPEED • QUICK TURN EXPERTS

ACE
ACCURATE
CIRCUIT
ENGINEERING

Accurate Circuit Engineering
3019 Kilson Dr
Santa Ana, CA 92707

(714) 546-2162 • sales@ace-pcb.com

For technical assistance, call:
James Hofer
(714) 227-9540

www.ace-pcb.com

‘The Want of a Nail’ and the Butterfly Effect

The Bare (Board) Truth

by Mark Thompson, CID+, PROTOTRON CIRCUITS

Last Sunday, I woke up at 4 AM with a song in my head that I had never heard before—at least not that I could remember! The lyrics had me so perplexed that I had to jump out of bed and check it out. Two things came up on my word search when I looked up “the want of a nail:” the first was the song I had just heard in my head, and the second was something about “the butterfly effect” from a number of chaos theory texts.

First, I re-played the song titled “The Want of a Nail” by Todd Rundgren and listened to the lyrics I had heard in my head just 20 minutes before in my sleep.

“For the want of a nail, the shoe was lost,
For the want of a shoe, the horse was lost,
For the want of a horse, the rider was lost,
For the want of a rider, the message was lost,
For the want of a message, the battle was lost,
For the want of a battle, the war was lost,
For the want of a war, the kingdom was lost,
For the want of a nail, the world was lost.”

Then I read the blogs on the butterfly effect. In the example I found, a butterfly flapping its wings can lead to a tornado weeks later, meaning that small events can lead to larger consequences. I believe this wholly. I see that small changes to the initial design or information gleaned at the start of a project lead to much larger issues *all the time.*

After I digested as much as I could about the butterfly effect, I chanced upon a TV show about bridge failures that focused on a device used to check for voids in the concrete-to-steel wire construction. It was very much like a time-domain reflectometer (TDR). Whereas a TDR

Integrated Tools to Process PCB Designs into Physical PCBs

Visualize

Use manufacturing data to generate a 3D facsimile of the finished product.

Verify

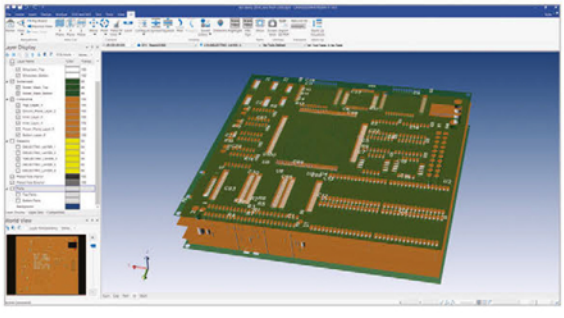
Ensure that manufacturing data is accurate for PCB construciton.

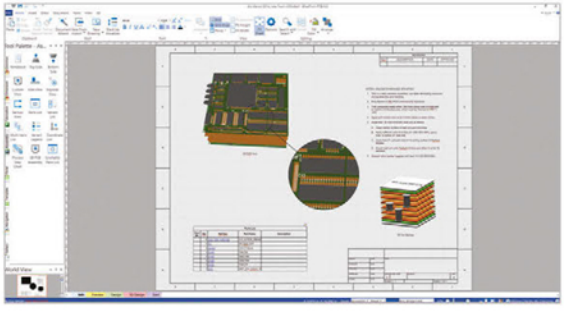
Panelize

Minimize costs in both PCB fabrication and assembly through maxiumum panel usage.

Document

Quickly and easily specify the PCBs form, fit and function.





CAM350®

Verify and Optimize PCB Designs for Successful Manufacturing

BluePrint-PCB®

Create Comprehensive Documentation to Drive Fabrication, Assembly & Inspection

DownStream Technologies, offers an integrated manufacturing data preparation solution for PCB post processing including manufacturing data visualization, stack-up definition, PCB panel visualization and support for 3D PCB documentation. This environment allows PCB designers and engineers to visualize, verify and document a printed circuit board before manufacturing.

DownStream Technologies

800-535-3226 | downstreamtech.com

NOW WITH 3D

is mainly used to determine the characteristics of a given electrical line by observing reflected waveforms, they can also be used to locate discontinuities in a connector. In the case being shown regarding a large braid of steel cable encased in concrete, some of the back-filling of the concrete did not fully encapsulate the wire bundle, leading to areas of moisture entrapment and, therefore, oxidation of the wire strand, causing a failure of the bridge.

So, why the Sunday morning revelation? I began to think about how small changes in design characteristics at the PCB fabrication level can have larger consequences for the final product. Some of these include changes in trace geometry, dielectric, material type, copper weights, etc. Let's go through a basic list of them and discuss each one.

1. Change From "Free Space" to a Coplanar Coupled Trace

Figure 1 shows a free space calculation. Figure 2 shows an added copper pour, creating a coplanar model.

As you can see, with the added coplanar coupling, characteristic impedance went from 49.86 ohms to 43 ohms for a 50-ohm trace due to poured copper close to the affected traces. Let's take that one step further with the next example.

2. Change in the Mask Color From the Original Calculations

Using that last example, let's make things even more complicated. For instance, let's assume that the customer has changed the free

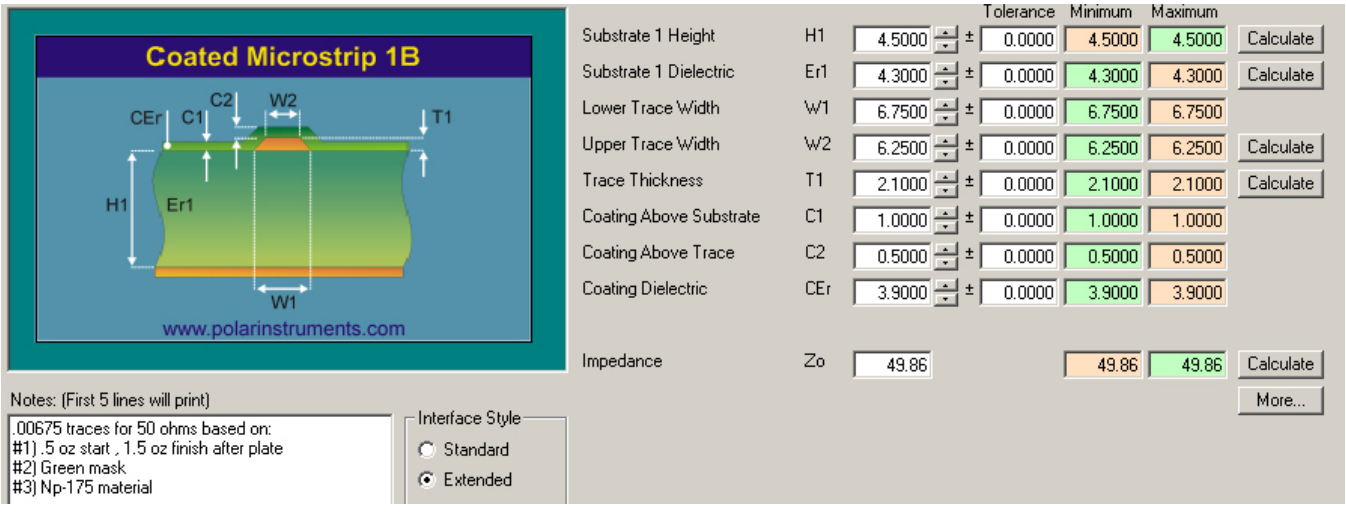


Figure 1: Free space calculation for a .00675" 50-ohm trace.

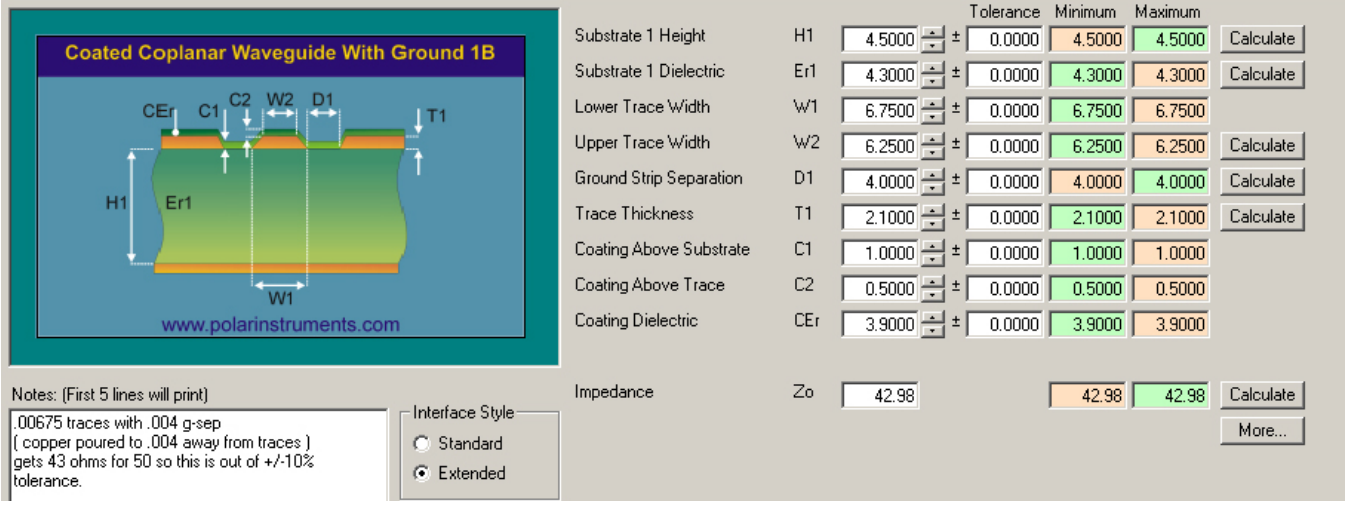


Figure 2: The same trace with added copper pour.

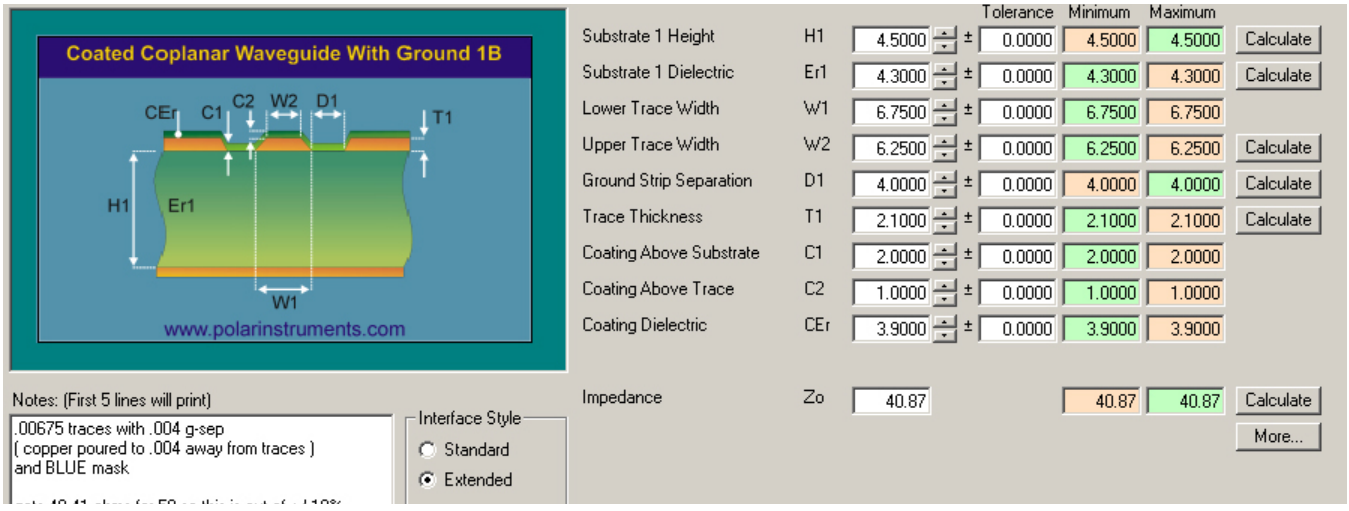


Figure 3: The same trace with a 0.004″ distance to the adjacent copper pour as blue mask.

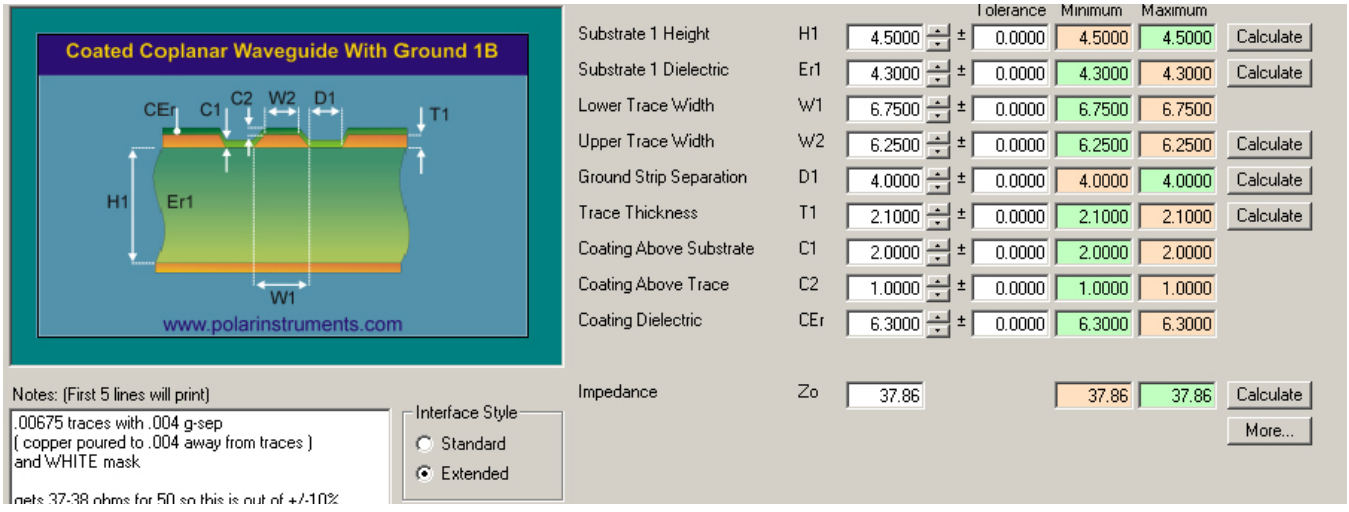


Figure 4: The same 0.00675″ traces with 0.004″ g-sep and white mask.

space trace calculation (approximately 2x to 3x trace width), but let's change from green mask to blue or white mask. Figure 3 shows the original 0.00675" traces with a 0.004" distance to the adjacent copper pour as blue mask.

Figure 4 has the same 0.00675" traces with 0.004" g-sep and white mask.

Let's say the drawing says, "Trace widths can be changed up to ±20% of the original trace width/space to achieve impedances." In the last scenario, with the mask color changed from green to white, we are well out of 20% to get back to 50 ohms. Figure 5 shows the results of changing to 0.00365" traces and 0.0055" g-sep from 0.00675" traces with 0.004" g-sep (linear trace/g-sep changes only), which equals greater than 20% of trace width.

3. Change in Material

Let's change from using Np-175 to P-370HR with the same two plies (1 x 106 and 1 x 1080). As you can see in Figure 6, it is fairly close to the end of the tolerance at 45.97 ohms for a 50-ohm ±10% tolerance, not factoring in any manufacturing variations, such as over-etch, under-etch, higher-than-normal plating, or lower-than-normal plating.

Once again, let's change the material type and the color of the solder mask from green to white. We are out of tolerance again at 40.39 ohms with both a material change and a mask color change (Figure 7).

4. Change in the Starting Copper Weight

Next, let's see what happens when we

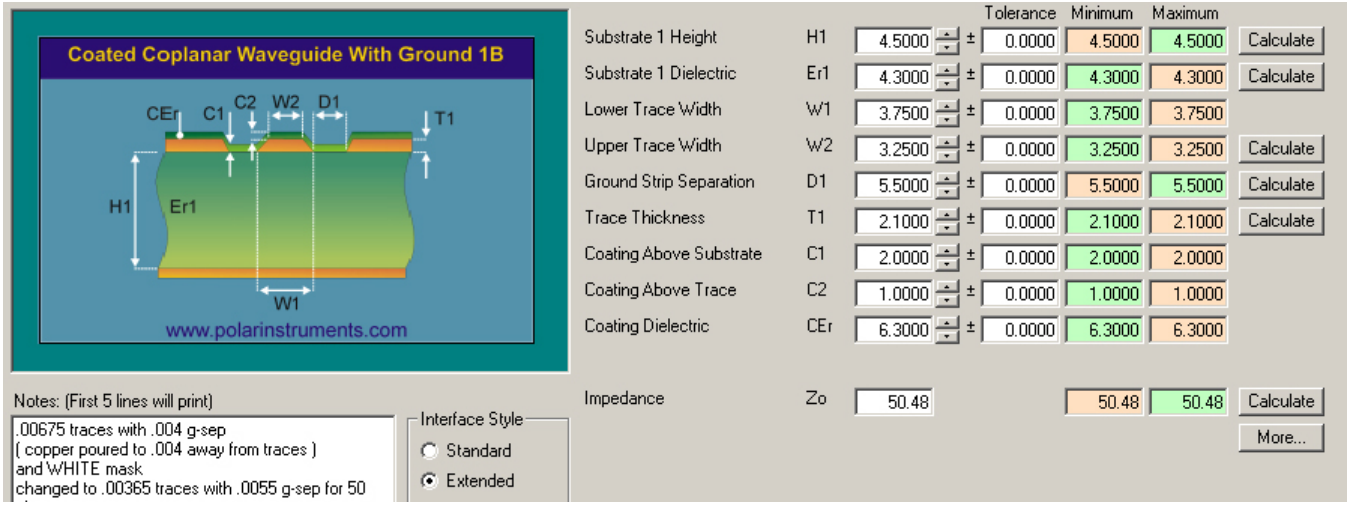


Figure 5: After changing from 0.00675″ traces with 0.004″ g-sep to 0.00365″ traces and 0.0055″ g-sep.

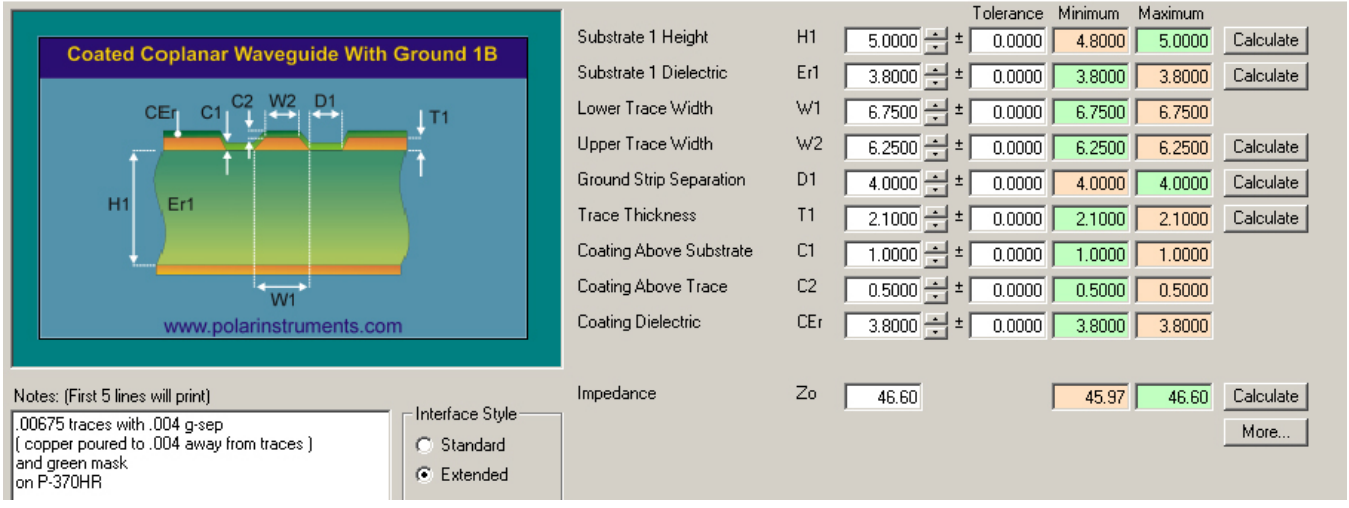


Figure 6: The same trace after changing from Np-175 to P-370HR.

change the starting copper weight. Remember, most U.S. fabricators shoot for an additional ounce of copper for the through-hole plating process to meet the IPC mandate of a minimum of eight-tenths of a mil of plated copper in the barrel of the hole.

Then, let's change from a half-ounce starting copper weight and 1.5-oz. plated finish to a 1-oz. starting copper weight callout with a 2-oz. finish (with the same starting trace width of 0.00675" with 0.004" g-sep achieving 43 ohms for a 50-ohm trace). Now, we are at 41.88 ohms, which is also well below the target of 50 ohms based on the original calculations (Figure 8).

To add insult to injury, this is not possible for many manufacturers, as achieving 0.004" g-sep means we will add an etch compensation for the 2-oz. finish of 0.001" taking the g-sep down to 0.0035". Remember, for a g-sep change, it will be half that of a space change, making the part not possible for many manufacturers without reducing the copper weight.

How to Make Sure This Does Not Happen to You

First, when communicating what your signal integrity needs are to your chosen PCB fabricator, ask them to provide you a calculation based on material type, starting copper weight, color of mask, and whether or not the part will have any close copper pour, creating a coplanar model. Second, if your trace size will be based on component constraints that the fab-

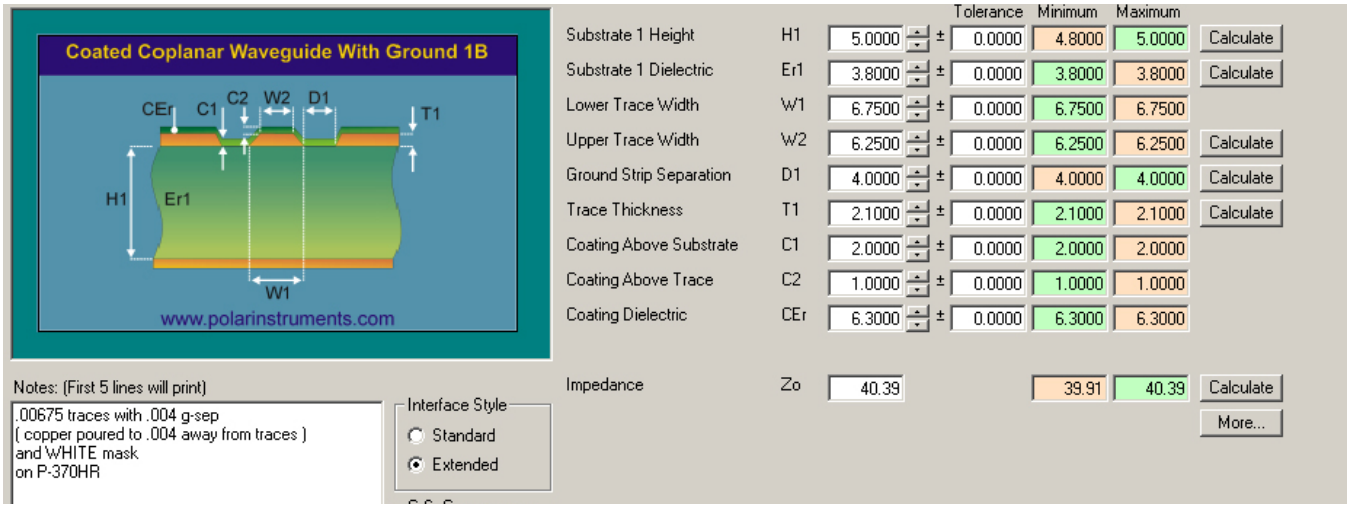


Figure 7: The same trace after changing material and solder mask color.

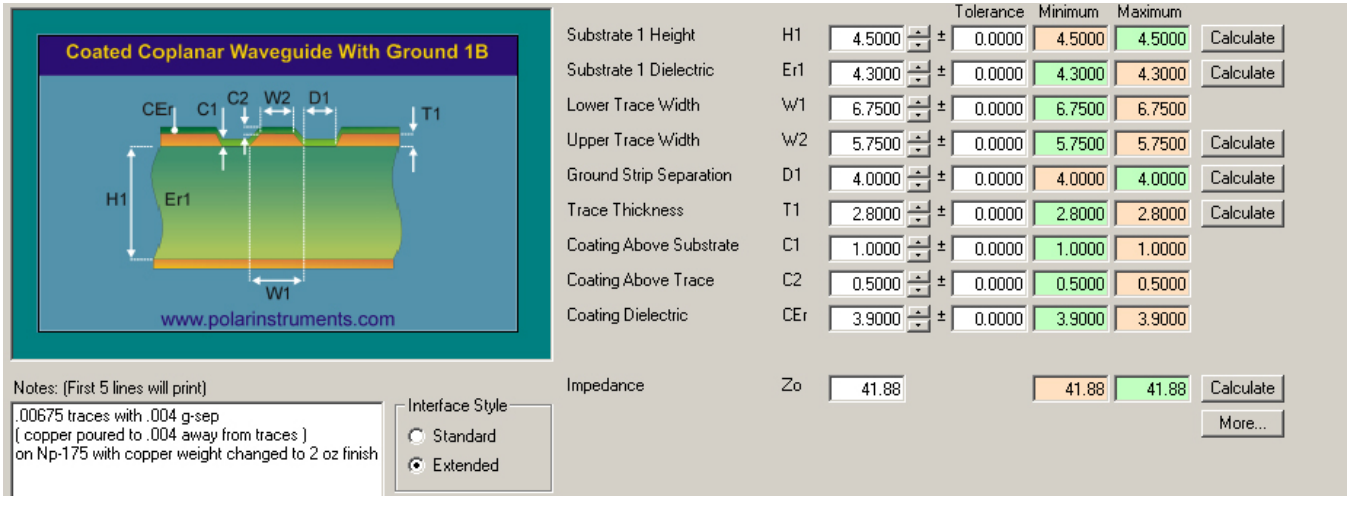


Figure 8: Results after changing starting and finished copper weights.

ricator is not aware of (maximum space to ingress and egress out of a given fine-pitch component), please communicate this information to the fabricator. A typical stackup that works the best for a fabricator does not work well for tight trace and space geometries to meet the desired impedance.

If I give you a trace width of 0.021" to meet 50 ohms based on a standard 0.062" four-layer board, I will hear a lot of coughing at the other end of the phone, as the designer says, "We have no room for a trace that wide." Try to have an idea or a range that a trace width should be before consulting with the fabricator.

Third, as we have learned from the aforementioned examples, do not change things without consulting your chosen fabricator to see what impact they will have on the design. The last thing an engineer or designer wants to hear after they have done the layout is that the part is not possible. For the want of a nail…

Thanks for reading. **DESIGN007**

Mark Thompson, CID+, is in engineering support at Prototron Circuits. To read past columns or contact Thompson, click here. Thompson is also the author of *The Printed Circuit Designer's Guide to… Producing the Perfect Data Package*. Visit I-007eBooks.com to download this book and other free, educational titles.

Top Tips for Conformal Coating Selection

Sensible Design

by Phil Kinner, ELECTROLUBE

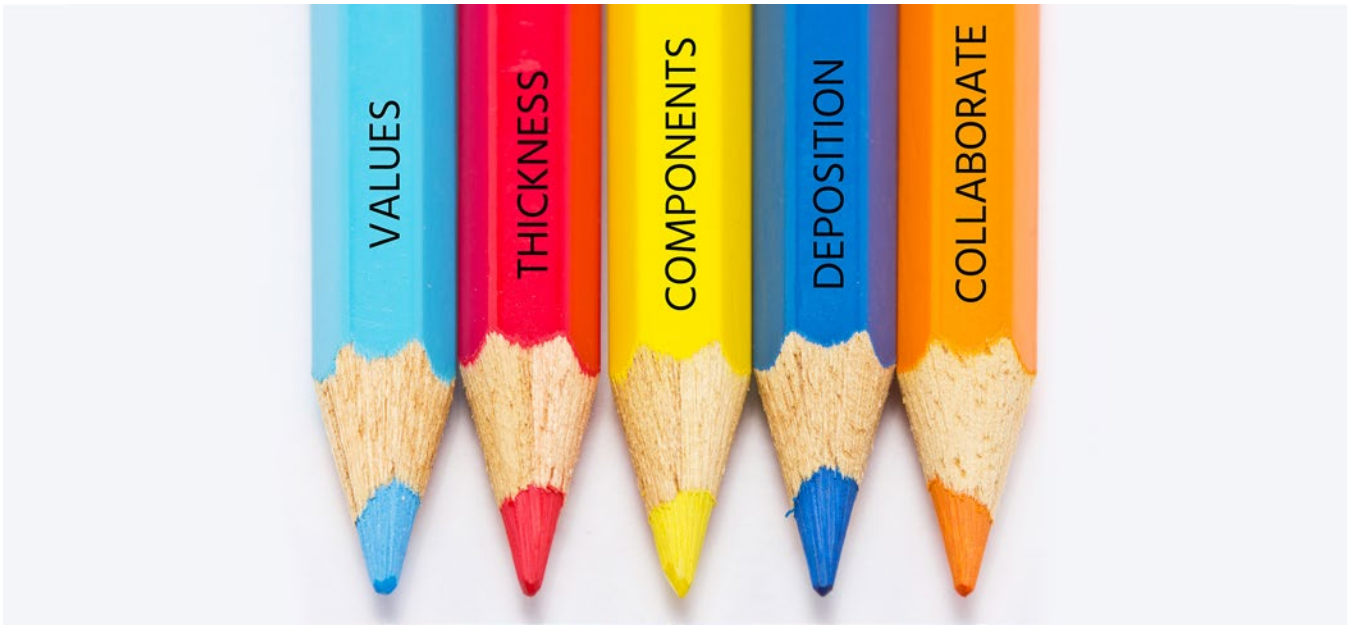
Over the past few months, I have covered the topic of conformal coatings in as much depth as possible. In this column, I'm going to explore some of the essential factors for designers in coating selection. As we have all experienced, sometimes, things are not always as simple or straightforward as we would like them to be, and in any engineering discipline, there is always the slightest chance that something might go wrong. Thankfully, the key to kicking that possibility is to be as fully prepared as possible. Thus, I'm going to concentrate on helping you avoid coating pitfalls in my five-point guide.

Understanding Coating Values

When choosing a conformal coating, consider critical material parameters like CTE, Young's modulus, Tg, and gas permeability, as coatings can vary widely in these respects. Designers should also be aware that the values listed in a vendor datasheet are generally measured at ambient conditions for newly cured coatings. It is important for designers to understand how the behaviour of materials will vary with temperature and time, especially ageing at higher temperatures. The coating vendor may have information regarding how materials will vary with temperature and time; however, it is imperative to bear in mind that materials will have to be tested by the OEM to make certain the material is fit for use on their assemblies. Temperature excursions must be factored in. For instance, if thermal shock or thermal cycling is overlooked, it could lead to the coating cracking, which will severely compromise its protective capabilities.

Getting Coating Thickness Right

Achieving the correct coating thickness is essential; bear in mind that if the coating is too thick, it can lead to entrapment of solvents in



Your circuit boards delivered...

1 DAY LATE

2 DAYS LATE

3 DAYS LATE

WHAT'S THE COST TO YOU WHEN YOUR BOARDS ARE DELIVERED LATE?

With Prototron, you don't have to ask that question. Serving customers for over three decades with a 98% on-time delivery record, we understand that providing you with high-quality PCBs on time, and right the first time, is critical.

Prototron Circuits

America's Board Source

www.prototron.com

areas where the coating does not fully cure. Similarly, it can cause the coating to crack as it cures or as the result of changes in temperature, or due to mechanical shock and vibration. As a rule of thumb, conformal coatings should not be applied in thicknesses greater than necessary or exceed the thickness they were designed to be applied at. Coverage is often more important than thickness, except in the harshest environments; however, if you are in any doubt about how to achieve the best coating thickness, we strongly advise discussing this with your coating vendor.

Knowing your components is central to achieving a successful coating process.

Know Your Components

Knowing your components is central to achieving a successful coating process. Simply by being aware of what kinds of components should be coated or cannot be coated will provide more flexibility to the coater in assembly operations. When a component must not be coated, it becomes a keep-out zone. The engineering drawing should not only identify components that must not be coated but identify the tolerance on that keep-out zone.

It is important to be very specific on the tolerance. The manufacturer needs to know exactly where you want coating and where you don't want coating. It is best practice to specify the areas that must be coated and the areas that must not be coated as well as the "don't care" areas to help the coating process run as smoothly as possible. Avoid vague statements at all costs in an engineering drawing. This is especially true when specifying coating around connectors. It is also worth noting that designers should not rely on a conformal coating to compensate for poor part selection. Some are more moisture-sensitive than others, and if the component fails, the failure may have no relation to the coating. The selection process of the coating, in this instance, will be absolutely critical to achieving a successful outcome.

Vapour-deposited Coating

The designer should be aware, during coating selection, that there is a vast difference between solvent-borne or liquid coatings, and those coatings applied by vacuum deposition or chemical vapour deposition processes. Vapour-deposited coating is an expensive process, and conventional coatings can't be used in this process. The most common vacuum-applied coating is polyparaxylylene (e.g., Parylene). Such processes require different masking, surface preparation, etc. If a designer chooses a vapour-deposited coating, it's important to be aware of the board's design as not all circuits are suitable for vacuum deposition. If in doubt, designers should work with the coating vendor or manufacturer who will do the coating, to identify the special rules that go with such materials.

Communication Is Key

Finally, the biggest gift that a designer can offer to the production team is to invest a few days on the manufacturing floor. I cannot emphasise enough how valuable this time will be in ensuring that the design flows through the coating process smoothly. It provides the opportunity to observe the process and talk with the process professionals who do this every day. Such individuals will be able to give the designer key elements as well as identify bad practices. When design and production work together, the outcome will nearly always be successful.

In my next column, I will look at more coating-related considerations. **DESIGN007**

Phil Kinner is the global business and technical director of conformal coatings at Electrolube. To read past columns or contact Kinner, click here. Kinner is also the author of *The Printed Circuit Assembler's Guide to... Conformal Coatings for Harsh Environments*. Visit I-007eBooks.com to download this and other free, educational titles.

Thursday, February 6, 8:30 am–3:00 pm. **Activities include hands-on, project-based learning, including soldering, PCB assembly and design.**

Help **engage the emerging workforce** at the IPC Education Foundation's **STEM Outreach Event** at **IPC APEX EXPO**. **Sponsorship** and **volunteer opportunities** are still available.

Join our 2020 Sponsors:

I·Connect007 GOOD FOR THE INDUSTRY

Panasonic

Nordson

Weller®

Your Company HERE

Contact CharleneGunter@ipc.org for sponsorship and volunteer opportunities.

MilAero007 Highlights

Defense Speak Interpreted: Executive Agent ▶

After reading one of Dennis Fritz's column, you may have realized that electronics packaging technology development came from the Naval Surface Warfare Center in Crane, Indiana. One of its core responsibilities is the assignment of "executive agent" for PCBs and electronic interconnects. But what is this "executive agent" thing, frequently shortened to EA? Fritz explains.

University Rover Challenge Tests Student Teams ▶

Jose Antelo, Luis Toledo, and Francisco Fernandez, members of the California State University (CSU) Fullerton Rover Challenge team, are working on building a competitive robotic platform to compete with other universities in a competition hosted by The Mars Society known as The University Rover Challenge.

Bosch Sends Sensor System to ISS ▶

With the help of Bosch's SoundSee technology, it is possible to analyze the information contained in emitted noises. In a research partnership, Bosch and Astrobotic are exploring ways to use the technology on the International Space Station to determine whether machines or their individual components need repair or replacement.

How NASA Is Helping Humans Reach the Red Planet, Using GPUs ▶

A group of NASA scientists and engineers is working with colleagues from Old Dominion University and NVIDIA to simulate with unprecedented accuracy the physics needed to land the first manned Mars mission. To do so, they're using the fastest supercomputer in the world, the NVIDIA GPU-powered Summit system at Oak Ridge National Laboratory.

From the Hill: Technology and Reliability Demands Drive Designers and MIL-PRF-31032 Specification ▶

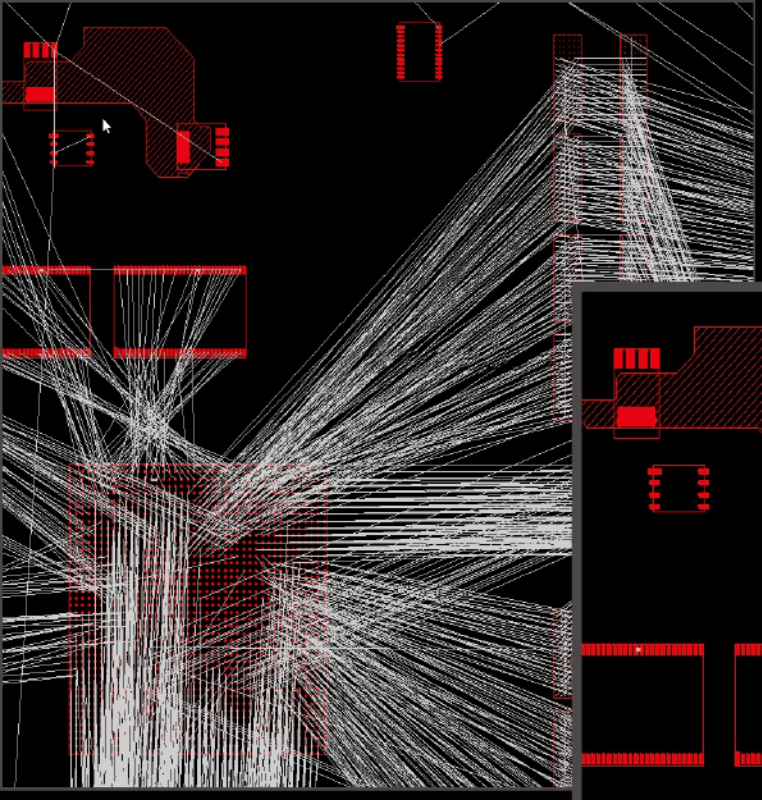
With the future demand for more and more military electronics, certification to the PCB MIL-PRF-31032 specification becomes a business decision for many fabricators. Fluency in the MIL-PRF-31032 language is a key first step to understand the requirements and communicate with the DoD. Mike Hill defines many terms related to this military specification that you should review before informing the DoD of your intent to certify.

Requirements of Being a MIL-certified Shop ▶

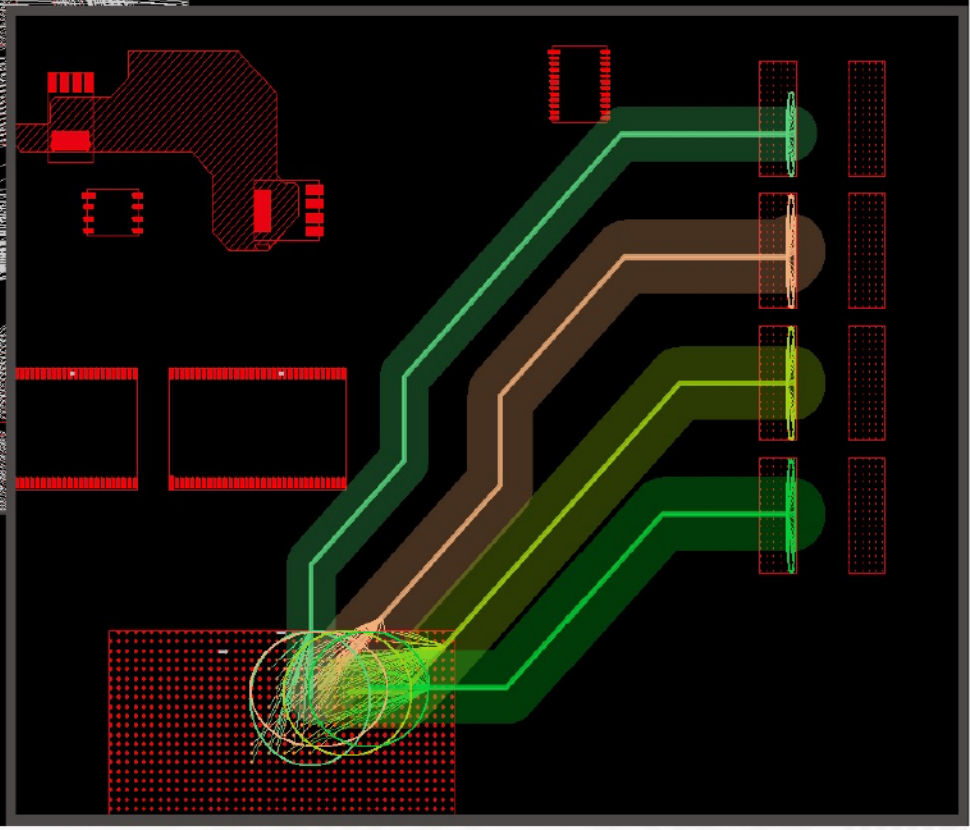
Barry Matties speaks with American Standard Circuits' VP of Business Development David Lackey, who has nearly 40 years of experience producing PCBs for the mil/aero market. David talks about what it's like being a MIL-certified shop and the stringent quality and reporting requirements that it entails.

World's First Electric Race Plane Unveiled at Dubai Airshow ▶

The world's first electric race plane has been unveiled on day one at the Dubai Airshow, showcasing custom modified technology built by one of the racing teams set to take part in the inaugural Air Race E event next year. The racing series will provide a testbed for innovation and accelerate the journey toward electric commercial travel.



Go from a rat's nest to organized in minutes



The industry's first **Bundle Planner**

Introducing the SimplifyPLANR

Bundle planning made easy

PCB Designs

PACKAGE Designs

FLAT PANEL DISPLAY Designs

Simplify Design Automation
simplifyda.com
sales: 925.548.5061
sales@simplifyda.com

SimplifyDA

FLEX007
A SPECIAL DESIGN007 MAGAZINE SECTION

What You Need to Know About Flex

What the Flex?

by Andy Shaughnessy, I-CONNECT007

Sure, rigid PCBs are the industry's workhorses. They brought us this far, and we are grateful for that. The PCB pays our mortgages and buys our boats and guitars. But PCBs are a little two-dimensional.

On the other hand, flex seems to bring out the creativity in a team, turning the most laid-back designers into mad scientists. I think a lot of technologists see flex as the Silly Putty of the PCB world—a medium that can be squeezed, twisted, and changed in an almost infinite number of ways. A rigid board just doesn't offer you those options.

I guarantee that some flex designers spend their lunch break experimenting with random pieces of flex. They form them into seemingly impossible shapes, occasionally duct-taping old PCBs onto the flex here and there to approximate rigid-flex, and then they yell out, "Hey, Bob! Check this out." How many final flex products have come about this way?

Flex circuits' unique characteristics lend them to experimentation. "Since the flex needs to move back and forth right here, what if we tried this? We'll just make it longer and have it twist around right here." And I understand why so many OEMs are moving (or being dragged) into flexible circuits. The more that the engineering managers learn about flex, the more appealing the idea sounds. Why restrict yourself to 2D?

LENTHOR ENGINEERING

MAKING THINGS FLEXIBLE IN A RIGID WORLD®

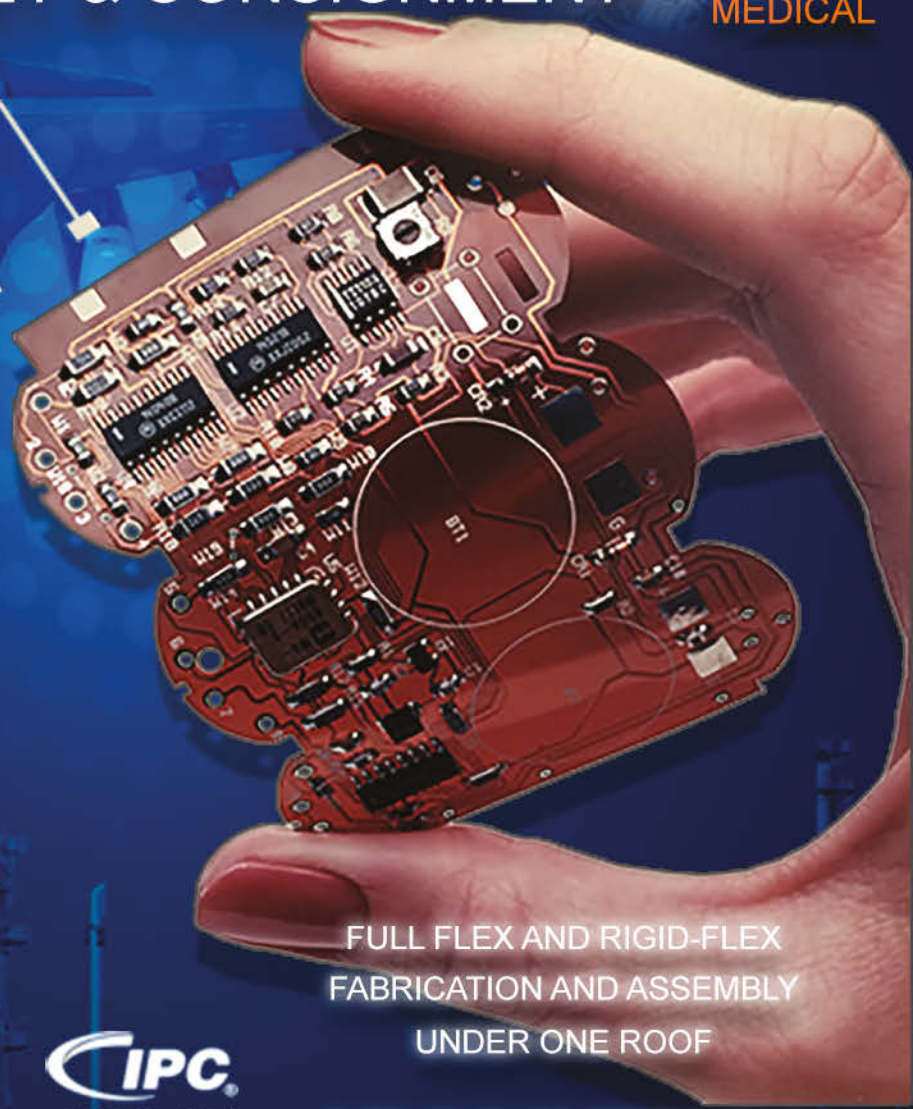
FLEXIBLE CIRCUITS ASSEMBLY

- PROTO-TYPE NPI
- FULL SCALE PRODUCTION
- TURN-KEY & CONSIGNMENT

MEDICAL

MILITARY / DEFENSE

INDUSTRIAL



LENTHOR ENGINEERING
311 Turquoise Street
Milpitas, CA 95035
(p) 408-945-8787
(f) 408-956-1874
(e) sales@lenthor.com
(w) www.lenthor.com

FULL FLEX AND RIGID-FLEX FABRICATION AND ASSEMBLY UNDER ONE ROOF

OVER 32 YEARS SPECIALIZING IN FLEX AND RIGID-FLEX PWB / PWA TECHNOLOGIES

IPC J-STD-001

ITAR

RoHS Compliant

ISO

MADE IN USA

I think that this sort of experimentation is one reason that we're seeing companies doing "super flex," with engineers constantly trying to test the limits of the medium. I thought the old Mars Rover's 35-foot flex assembly was pretty remarkable, but Trackwise has created an 85-foot multilayer flexible circuit. Is there a length limit? With flex, there are no hard, fast rules. If you have an idea, try it out.

There's so much innovation going on in the flex and rigid-flex world that it can be difficult to make sense of it all. In this issue, we provide a snapshot of the flexible circuitry segment. We cut through the noise and tell you what you need to know about flexible circuits.

As Kelly Dack explains in our feature story, he needed to find a fabricator who could attach a piece of metal to a flexible circuit, with the metal acting as a heat sink. He wound up working with Guy Martindale of SEP, a fabricator based in Korea. Guy had the technical knowledge to get the project done, and he was also the first one to visit with Kelly in person. Kelly and Guy share some tips on designing and fabricating advanced flex circuits and discuss why communication is king, especially with complex designs.

Joe Fjelstad of Verdant Electronics shares some of the knowledge gleaned early in his career from technologists of the "Greatest Generation," and he looks into the many disparate skill sets required to create even a simple flex or rigid board. Next, Tara Dunn of Omni PCB discusses the need for technologists to continue their training and education, as well as some of the things to consider before beginning your first flexible circuit design. And Dominique K. Numakura of DKN Research shines a light on a technology that's been around for a few years but is only now beginning to catch on: transparent flexible circuits.

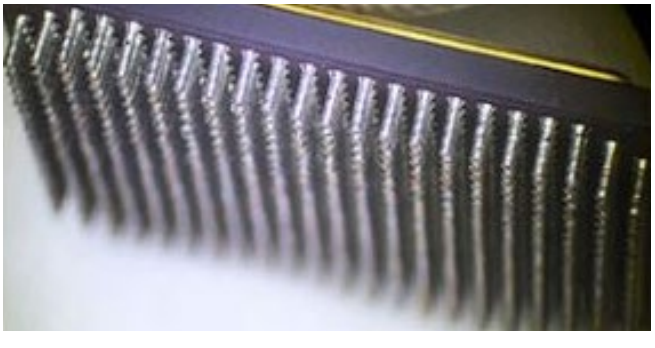
It's hard to believe that New Year's Day is right around the corner. After we clean up the confetti and empty champagne bottles, we'll be heading out to cover DesignCon and IPC APEX EXPO 2020. There's a lot going on in the industry, but we'll keep you informed.

Have a great holiday! **FLEX007**

Andy Shaughnessy is managing editor of *Design007 Magazine*. He has been covering PCB design for 19 years. He can be reached by clicking here.

TopLine and VPT Components Announce Column Attachment Services for RadHard FPGA Devices

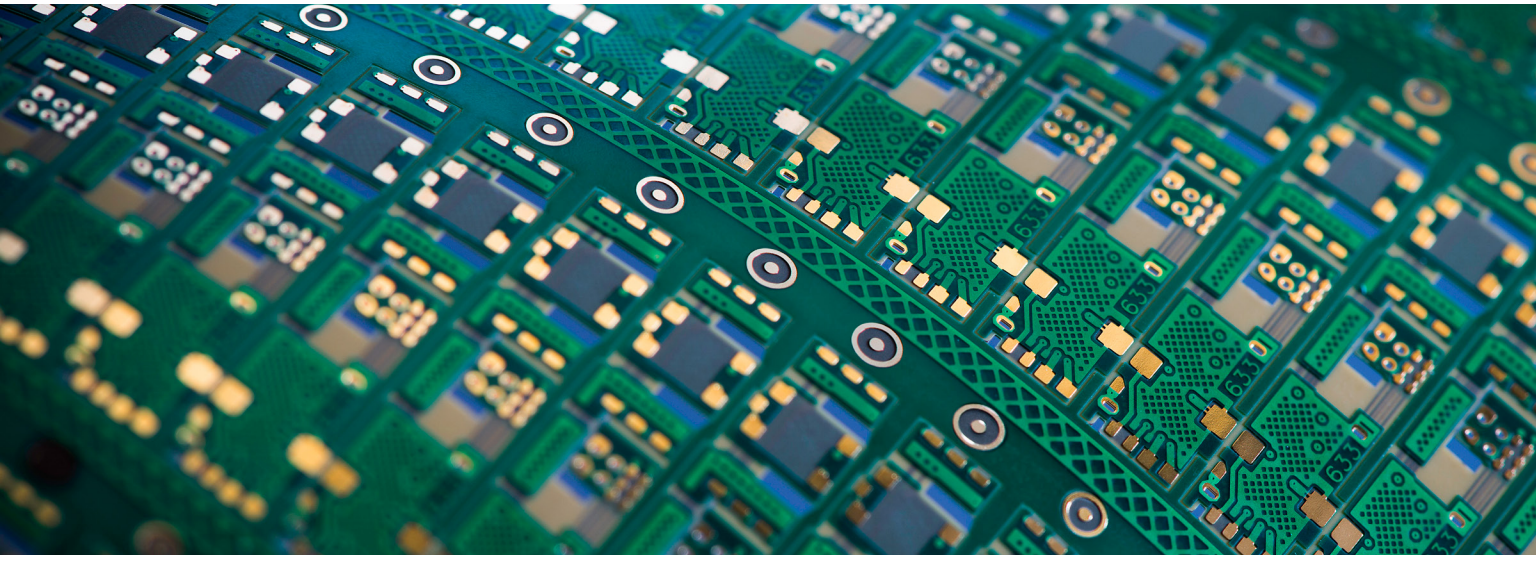
Teresa Farris, TopLine business development manager of radiation hardened technology (RadHard), said the company has expanded collaboration and technical cooperation with VPT Components, a JANS-certified semiconductor manufacturer. VPT Components will provide CCGA column attachment services to the RadHard community, according to Joe Benedetto, CEO of VPT Components, at a recent RHET conference held in Melbourne, Florida.



TopLine is a California-based manufacturer of copper-wrapped Pb80/Sn20 solder columns for ceramic column grid array (CCGA) packages.

Martin Hart, CEO of TopLine Corporation, said, "We are pleased to extend close technical cooperation and support to VPT Components, who serves as an independent contract assembler to provide column attachment services to the greater defense and aerospace industries."

Benedetto added, "There is a need in the radiation hardened IC community for faster turn-times. VPT developed a CCGA process using TopLine columns with excellent attachment and pull-strength test results." (Source: TopLine)



cicor

Cicor is a technology partner in printed circuit boards, printed electronics, hybrid circuits and electronic manufacturing services with 10 production sites worldwide.

cicor.com

What You Need to Know: Advanced Flex Design

Interview by the I-Connect007 Editorial Team

During PCB Carolina, Kelly Dack and Andy Shaughnessy sat down for an interview with SEP's Guy Martindale, who is helping Kelly to develop an advanced metal-bonded flexible circuit. Guy explains why it is so critical for designers and fabricators to begin collaborating early in the design process, particularly with advanced and custom flex applications.

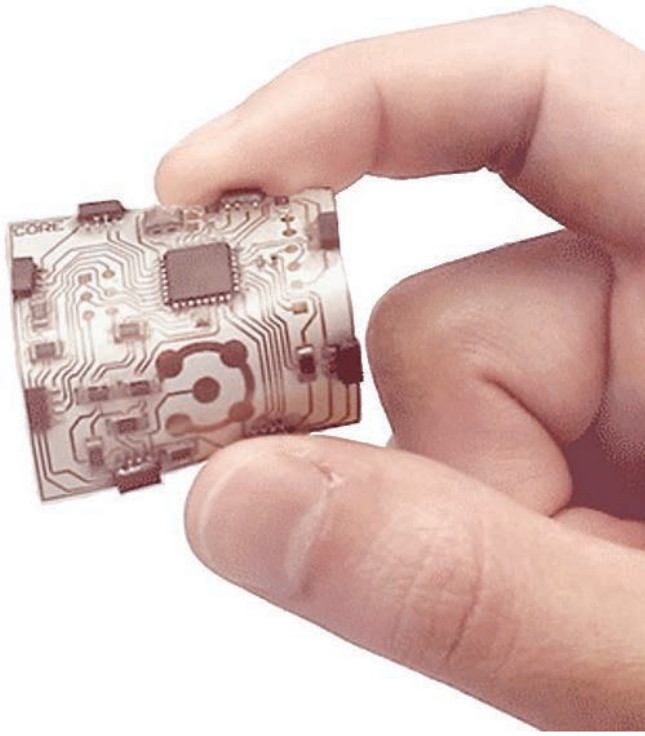
Kelly Dack: Guy, thanks for spending time with us today.

Guy Martindale: Thanks for reaching out.

Dack: Just two months ago, we ran into each other on the show floor at another industry event. I had a specific design need for a flex circuit requiring a thin piece of stainless steel to be bonded to its backside—specifically, a heat sink material attached to the flex. I'd been searching the trade show floors and had interviewed several suppliers, but your capabilities stood out. After setting up our NDA, you presented a detailed stackup proposal that not only addressed the materials, but you followed through with SEP's innovative approach to panelizing this specialized configuration for volume assembly. Guy, tell us about SEP and where your circuits are produced.

Martindale: SEP is based out of Suwon-Si, South Korea. We're a PCB manufacturer that covers all aspects of circuit boards, from rigid to rigid-flex and flex. Our niche is being able to support custom applications like you've come to the table with. Bonding metal to the flex presents a certain challenge, but excising the completed parts from a manufacturing panel array presents an entirely different set of challenges based on the assembly order of operation in the manufacturing environment. So, we're not only a PCB manufacturer, but we support some of the assembly side of the business as well.

DUPONT®



PYRALUX® FLEXIBLE CIRCUIT MATERIALS

PYRALUX® AP
high reliability flex & multilayer circuitry

PYRALUX® AC
for thin, light, high density with chip-on-flex attachment

PYRALUX® HT
for high temp, high performance flex

PYRALUX®TK
Teflon®/Kapton®

PYRALUX® LF
industry standard for high reliability

PYRALUX® FR
flame-retardant for products requiring UL rating

PYRALUX® APR
reliable resistor, temperature tolerant

DuPont offers a comprehensive, industry-leading portfolio of Pyralux® flexible circuit materials and systems designed to meet today's and next generation design challenges.

The Pyralux® portfolio includes a diverse collection of core dielectric materials and customized dimensional constructions that enable the designer and manufacturer of complex circuits to deliver high performance solutions.

DuPont Pyralux® copper clad laminates, bondplys, coverlays, and adhesive systems offer excellent functional performance and high reliability and allow for the fabrication of thin, solderable, high density electrical interconnects for single and double-sided, multilayer flex and rigid flex applications.

PROUDLY DISTRIBUTED BY
INSULECTRO

20362 WINDROW DRIVE • LAKE FOREST, CA 92630 • 949.587.3200 • INSULECTRO.COM

Andy Shaughnessy: You two bonded on the topic of bonding, so to speak.

Martindale: Yes, a lot of the conversations I've had working with Kelly have been centered around how to take this part from a base flex circuit to adding a metal piece onto the circuit for his custom application. How do we best build that and deliver you a panel of circuit boards with the metal-backed application? And how do we take that to the assembly side? We had to get a clear picture of when to singulate the part. Should it be performed before or after the assembly stage? There are pros and cons to each strategy. One may be best for prototyping, but the strategy may need to change when the part goes to volume.

Shaughnessy: What are some of the pros and cons of manufacturing in different parts of Asia?

Martindale: We started out as a South Korean PCB manufacturer and expanded into South China. Now we have circuit board capabilities in both countries due to internal customer demand. About 15 years ago, we had customers who wanted to vertically integrate and have us do the assembly side. So, we built a 30,000 square foot manufacturing facility in South Korea and started doing the assembly side for our current customer base.

We're not a traditional CM; about 45% of our business is bare board today for all different types of circuits. They're very niche applications. Then, we do the assembly side, which we build the circuit boards for as well. It allows us to offer both a low-cost region out of South China and a higher-cost region with South Korea, both based out of Asia, and gives us the flexibility to either build a lot of our complex circuitry out of South Korea, and maybe do the assembly side in South China, or vice versa; we can also do it all in one country. A lot of it comes down to how we price it for customers, what they're looking for strategically in their supply chain, and where they want to build.

Dack: Andy, I have taught EPTAC certification courses for PCB design, and we emphasize how critical it is for designers to reach out to the fabrication stakeholders—the people who are doing the work—to incorporate design for manufacturability into the product. I practice what I preach, and this is exactly what I did. We shook hands, I presented Guy with a challenge, he took the information with him, and within the week, the team at SEP provided us with a solution that helped our design process to point toward DFM/DFT. Guy, tell us about that and why it's so important.

Martindale: Over the years, we've come to realize that a lot of OEM companies have the functions of hardware design and software design, including the fabrication portion of it. I work with clients and look for opportunities to bridge the gap between the two. The design of a circuit, whether it's rigid or flex, can be the best circuit and design that you have out there, but if you can't manufacture it, especially when you are on the production side of it, you have a problem; more often than not, we see this conundrum when working with different customers. We often see tremendous opportunities if we get in early on the design—the R&D and prototyping side of it.

What you brought to the table was instantly a fit because it said that you had a custom application and a problem that you needed to go solve. That's where we say, "How can we take your current design, prototype it, and then scale it to volume without redesigning the whole part? The key is to connect with a customer at the concept stage to consider and define DFM for each phase of the project. That's where I spent a lot of my time with customers on the engineering side. Then, we're looking at the total cost of ownership and not just what the price looks like on a PO.

Dack: And for anyone new to flex design, there are a lot of differences to consider over rigid board PCB design. Standard, rigid FR-4 designs go to standard assembly lines for parts placement and then singulation. But in the world of hybrid flex structures, it's not so easy. In

manufacturing, parts may have to be singulated first or partially assembled then singulated. The order of operation required based on the parts that need to be installed can get really challenging. Again, we came to SEP because we couldn't create individual bare flexes and bond metal to them on our assembly lines. Alignment and other matters make it too complex and time-consuming.

If high-tech peripherals are required to be installed onto this flex board, we must ask ourselves: Are we considered the experts? Do we want to invest and become the experts? In many cases, no. And in this particular case, SEP proposed a solution that appeared to be much more desirable from a time-to-market viewpoint. Oftentimes, PCB flex fabrication and assembly are best done under one roof. They not only manufacture the bare flex, but they do the assembly work as well.

Martindale: Yes, and that's what we look at when we talk to customers. With your specific application, I can deliver you the flex circuit with a metal-bonded piece on it and guarantee that work; however, if we look at building out the entire solution for you, we may be able to help by assembling the parts in a lower-cost region, owning the quality of the parts and saving you more money over trying to do that in the U.S.

Dack: And that just goes back to my point: Sometimes we must admit we need the services of experts because we are not.

Martindale: Right. For our size company, a lot of what we do in the flex circuit world has niche applications. For instance, on the medical side of the industry, there are small flex circuits with mini cameras being mounted on them that are going onto the ends of very small, complex devices. Your application was a perfect fit because when we looked at it, we saw that we could build the flex circuit, get it delivered, and do the full assembly side of it as well as test that solution for you. With our solution proposal, you get a fully tested assembly coming in. You can count on it being right

Guy Martindale shows off some of SEP's flex circuits at the PCB Carolina show in November.

because you have one supplier that's not only doing your circuit board and the assembly side of it, but they're helping you out on the design side as well. What tends to happen when you're buying the circuit from one company and then you're doing the assembly with another company, is that you lose the integrity of the engineering data. The strategies critical to defining which best practices to employ for design for low cost and design for manufacturability can become blurred.

Shaughnessy: What's your background, Guy?

Martindale: I graduated in 1997 with a chemical engineering degree and went to work for a PCB manufacturer called Zycon. Through mergers and acquisitions, Zycon became Hadco, which became Sanmina, and then Sanmina-SCI. I spent about eight years in various process engineering roles within the circuit board industry, transitioned out of that, and went to work for an OEM; there, I mainly worked on the assembly manufacturing side of it, but I spent seven years integrating design and software engineering for DFM and test over to the manufacturing side for that business.

That OEM company eventually got acquired, and I decided it was time to move on, so I went to work for my supplier I'd been working with for six years, who happened to be SEP. I started working directly for them, supporting customers in the United States.

Shaughnessy: Kelly, you said that SEP was the first company to step up with a plan. Tell us about that.

Dack: I was searching for capabilities, and while we did get NDAs with a lot of capable suppliers, SEP responded within a week with a quotation and a solution. On top of that, they jumped on a plane and came to visit us; they talked to our supplier management group and our engineers about their capabilities. They submitted tangible evidence of their abilities to our manufacturing and engineering team representatives. A business card is one thing, but meeting in person with the people who are doing the work is always the best.

A business card is one thing, but meeting in person with the people who are doing the work is always the best.

Martindale: As far as putting a metal plate on the back of a flex unit, there are folks who can do that; it's a matter of putting in the up-front investment in time and energy. This is before you've even thought, "Am I going to get business or not?" It's that engineering support up front to work with designers and people in the supply chain and think strategically about how the partnership can go to market with the product. One aspect of doing that is to be able to technically support the R&D, the production side of it, and work with everyone trying to get there.

Shaughnessy: What was the point of attaching metal to the flex? Did the metal have to flex too?

Dack: The purpose, in this case, was to serve as a rigidized surface that also doubles as a heat sink.

Shaughnessy: So, the flex wouldn't flex?

Martindale: In this flex application, there is a need for high-end heat dissipation, so that's where the metal-backed flex circuitry came in. Usually, when components are to be installed onto a flex part, rigidization using a stiffener made with polyimide or FR-4 is common. But in this case, the rigid metal heat sink could serve as the rigidizer. It works nicely as a two-for-one solution. However, dissimilar materials like stainless steel and polyimide require certain bonding materials that are unique in the industry. Not everyone uses them or is willing to invest the time to R&D them and build them into the prototypes and take it to the production side. We were able to do research in the industry on what standardized materials we can get out in the marketplace. We had to bond this stainless steel plate on the back of the flex, build that into the quote for Kelly in what we're doing here, and then look at the assembly side of it and how we put that all together as well.

Shaughnessy: And you flew right out to visit them.

Martindale: Yes, it worked out well. We're a company in a very competitive industry, and we have a certain niche that works well for us; flex happens to be one. We realize that meeting face to face is where the real work is done from an engineering perspective and developing solutions together. You can work remotely and over email all day long, but meeting in person and understanding who you're working with, why you're with them, and the technology side is where it's at.

Shaughnessy: Thanks for your time.

Dack: Thank you for your expertise.

Martindale: You are both very welcome. **FLEX007**

WHAT DO ALL OF THESE LEADING COMPANIES HAVE IN COMMON?

atg LUTHER & MAELZER

ATOTECH

CHEMCUT
BOUNDLESS INNOVATION | UNBEATABLE PRECISION

CIMS
Creativity in Motion

CYBEROPTICS

DownStream Technologies

ELECTROLUBE
THE SOLUTIONS PEOPLE

esi
Designed for Brilliance. Engineered for Production.

KIC Making Ovens Smarter™

KYZEN

MacDermid Alpha ELECTRONICS SOLUTIONS

Mentor
A Siemens Business

MIRTEC

NANODIMENSION
Electrifying Additive Manufacturing®

orbotech
The Language of Electronics

pluritec

PULSONIX

rehm
THERMAL SYSTEMS

schmoll asia pacific

ventec
INTERNATIONAL GROUP
騰輝電子

VJ ELECTRONIX
Process Control Solutions
CONFIDENCE DELIVERED

YXLON

They all benefit by advertising with us–PCB007 China Magazine.

Should your name be on the leaders' list?

GET STARTED NOW!

I·Connect007
GOOD FOR THE INDUSTRY

pcb007china.com

FlexOO7 News Highlights

Mentor Video: What You Need to Know About Rigid-Flex Design ▸

This on-demand webinar will show you how PADS Professional's rigid-flex technology simplifies the rigid-flex design process, from stackup definition through to manufacturing handoff. The presenters are Technical Marketing Engineering Manager John McMillan and Technical Marketing Engineer Brent Klingforth, both of Mentor, a Siemens business.

Decreasing Bend Radius and Improving Reliability, Part II ▸

Many of the issues that arise when using a flex circuit come from a lack of knowledge about how to properly design one, especially when the circuit is required to bend. Many novices will design a circuit that calls for bending the flex in too tight of a bend radius, which can cause damage to the circuit and lower the reliability of the end product. This series of articles will focus on the seven key aspects to consider when designing for maximum durability and maximum flexibility.

Flex Talk: Additive Electronics Momentum ▸

Tara Dunn has been involved with additive electronics for the past several years and seen the discussion of and demand for sub-75-micron feature sizes slowly grow. Conversations, questions, and research about SAP and mSAP increased significantly when it was announced that the mSAP process was used to create the circuitry in the more recent versions of our smartphones. While this process is available in very high volume in some areas of the world, it is still in the early stages of development in other areas.

EPTE Newsletter: Monocoque Printed Circuits—An Update ▸

Wiring with flexible circuits could be a practical solution. Nowadays, most mobile device manufacturers are consuming huge amounts of thin, flexible circuits to attach on the surface of the housing in limited spaces. However, the cost of flexible circuits and assembling them is another headache for device manufacturers because they are not negligible in the whole cost of the devices.

Nano Dimension's Partner, Accucode, Expands With First 3D Electronics Service Bureau and Second DragonFly System ▸

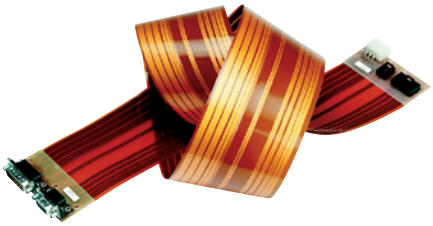
Nano Dimension Ltd., a leading additive electronics provider, announced today that U.S.-based reseller, Accucode, is opening the world's first service bureau for 3D printing of electronics focused on serving the market with end-to-end prototyping and low-volume additive manufacturing services for electronics.

Insulectro Leverages One Source Distribution for Printed Electronic Materials at IDTechEx ▸

Insulectro, the largest distributor of PCB and printed electronics materials, exhibited at IDTECHEX's Printed Electronics World 2019 conference held in Santa Clara, California.

AstroNova Partners With SPF for Flex Tech ▸

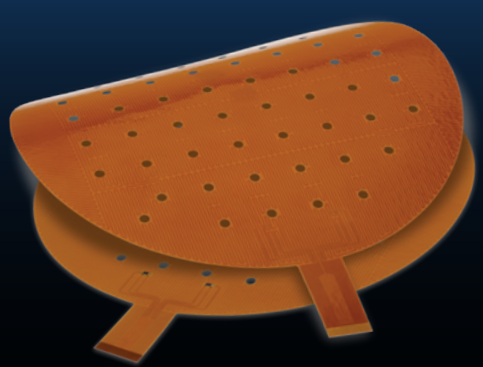
AstroNova has partnered with SPF, a technical consultancy supporting printing and packaging industries to facilitate an easy, cost-effective digital printing method for producing flexible circuits by combining AstroNova's QuickLabel and SPF's metallograph technologies.



all flex

Flexible Circuits • Heaters • Assemblies

- Design application assistance
- Component assembly
- Fast response
- Quick delivery



All Flex Flexible Circuits & Heaters
1705 Cannon Lane
Northfield, MN 55057
(800) 959-0865
(507) 663-7162

allflexinc.com

The Value of Experience

Flexible Thinking

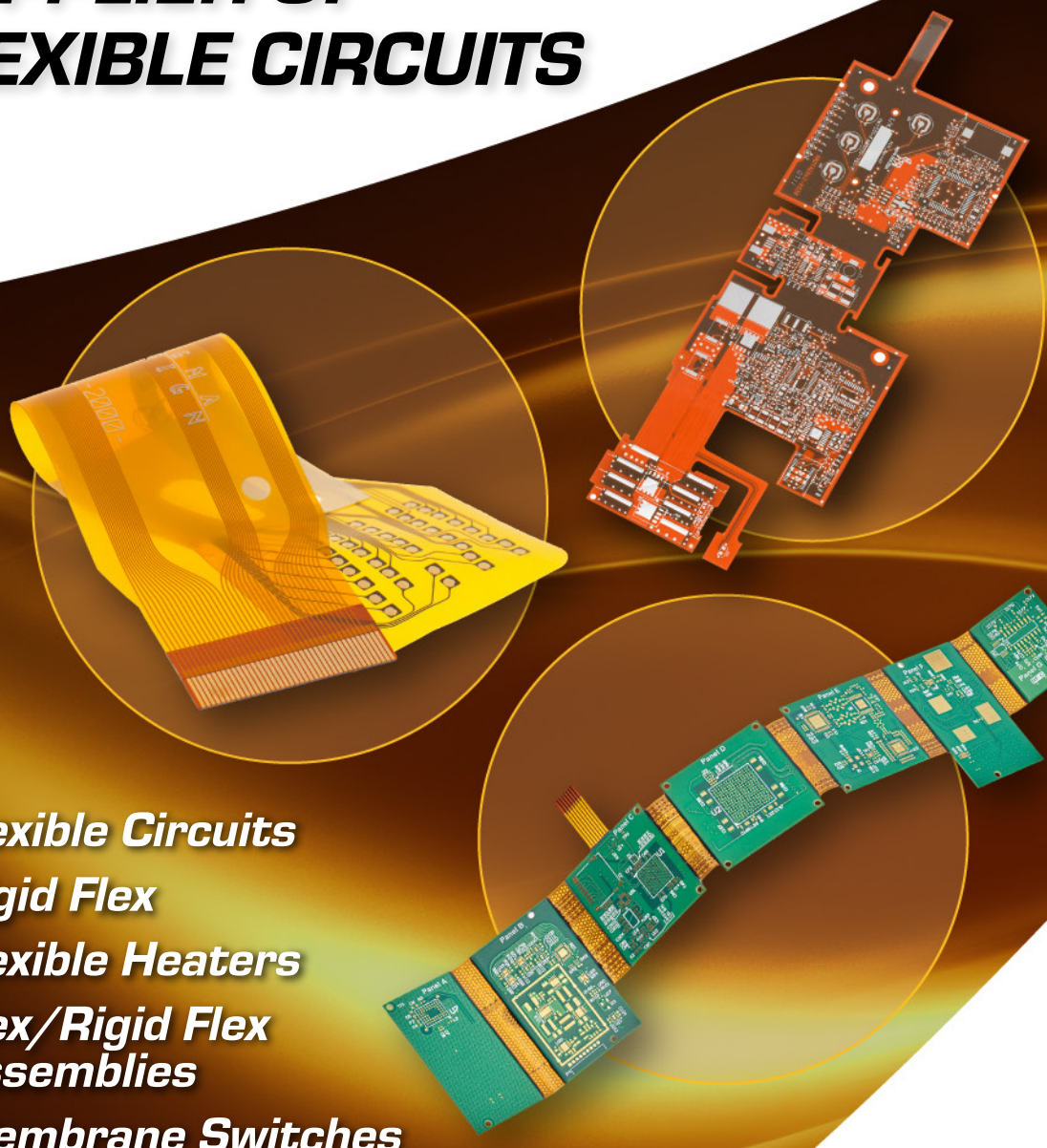
by Joe Fjelstad, VERDANT ELECTRONICS

For many people, December is a month in which to reflect on the experiences and lessons encountered and learned over the past year. As the years pass, I am increasingly thankful for the many experiences that have brought me to this point. In sitting down to collect and share my thoughts, what first came to mind was a timeless story about the value of experience. It goes something like this.

A young engineer asks a mentor a question about a problem that he has been pondering for some time. The older engineer listens attentively and then posits a possible solution to the problem. The young engineer tries the suggestion, and it works the first time. He thanks the mentor for sharing this wisdom, asking, "How did you know that would work?" The older engineer responds, "From my experiences." The young apprentice pressed for more detail, wanting to understand how the mentor had obtained the wisdom to intuit the answer so quickly; the mentor sagely adds, "From my bad experiences."

Looking back at my almost half-century of experiences, I can attest to both the validity and the constancy of this story from both ends of that humorous, but experientially, true tale. I was fortunate to have had several excellent and very experienced engineers as mentors from an early stage in my career. They were largely members of the "Greatest Generation," and many were veterans of World War II and

PREMIER GLOBAL SUPPLIER OF FLEXIBLE CIRCUITS



- Flexible Circuits
- Rigid Flex
- Flexible Heaters
- Flex/Rigid Flex Assemblies
- Membrane Switches
- Product Module to Complete Box Builds

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

FLEXIBLE CIRCUIT® Technologies

WE GO WHERE OTHERS WILL NOT

www.flexiblecircuit.com

the Korean War. I was a veteran as well, but of Vietnam; thus, we had a common bond that allowed for a friendly and interactive relationship.

I was also fortunate to have worked in an engineering "bullpen" in a manufacturing technology group for the manufacture of electronics for Boeing Aerospace. Our shared office space consisted of desks that were three across and eight rows deep, which gave me instant access to an incredible brain trust, who could provide answers to my questions or direct me to a reference where answers could be found (these cherished colleagues were my early version of the internet but with personalities). Printed circuit fabrication was a primary focus of our group, and as those who fully comprehend and appreciate the complexity of PCB manufacturing will attest, it took all manner of engineering and technical skills and disciplines to develop and service the processes required to make the desired products.

Printed circuits are arguably one of the most complex and process-intensive products made today—even more so than semiconductors, one could argue. Consider the following skills required today:

- Electrical/electronic engineers and skilled technicians must define the functions, design, and layout the product
- Packaging engineers and technicians must design the envelope into which the electronics will be placed
- Mechanical and material engineers must determine what properties the PCB substrate needs have to function for the application
- Chemists, chemical engineers, and skilled technicians must operate and maintain the numerous chemical processes involved in PCB manufacturing
- Project and manufacturing engineers must plan the manufacturing flow
- Process engineers and skilled technicians are required at every step to execute the seemingly endless number of process steps and variations that might be required to complete the product
- Metallurgical scientists and engineers must test, evaluate, and bless both the assembly process and the materials used
- Laboratory test engineers and technicians must monitor, test, and evaluate the end-product and its fitness for use

I can think of a few other disciplines not mentioned, but the short list provided should provide ample testament to the complexity.

People of such varied skills were responsible, from the earliest days of the industry, to weave together materials, equipment, and processes, often with origins far from the tasks they have been adapted and are now being used for. Consider the few following industries that have been drafted into service to make today's most advanced printed circuits:

- Lamination industry: Lamination presses for flex and multilayer circuits
- Graphic arts industry: Screen printing and photolithography used for imaging
- Machining industry: NC drilling, routing, punching, and stamping
- Plastics plating industry: Electroless and electrolytic plating of vias
- Aerospace industry: Vacuum lamination and advanced composites
- Shoemaking and paper industries: Steel rule die cutting of flex circuits
- Microelectronics industry: Metal sputtering and plasma cleaning and etching
- Applied physics: NC laser drilling and cutting

Over time, these items have adapted to more specifically address the changing needs of printed circuit manufacturers. Today, the PCB materials, processes, and equipment industries constitute multi-billion dollar market. However, the printed circuit is, unfortunately, too often underappreciated in its continuing role as the virtual foundation of all modern electronics. Without PCBs, semiconductor devices would have no means of interconnecting and performing the designed end function and purpose/mission. They play a vital and irreplaceable role in all electronics manufacturing.

Given global society's increasing reliance and dependence on electronics for getting through the day, we are all, by extension, reliant on PCB technology. Therefore, it is imperative that we not lose sight of the important role printed circuit technology continues to play in our lives. It is also important to keep pace with the advances of semiconductor technology to provide and ensure complete and uninterrupted reliability of the electronics we all increasingly depend on.

In closing, I'd like to share another short story. It comes from an interview given by famed Swedish film director Ingmar Bergman, which I first encountered some 30 years ago: "Old age is like climbing a mountain. You climb from ledge to ledge, and the higher you get, the more tired and breathless you become, but your views become more expansive." Enjoy your climb. FLEX007

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.

MIT: Toward More Efficient Computing, With Magnetic Waves

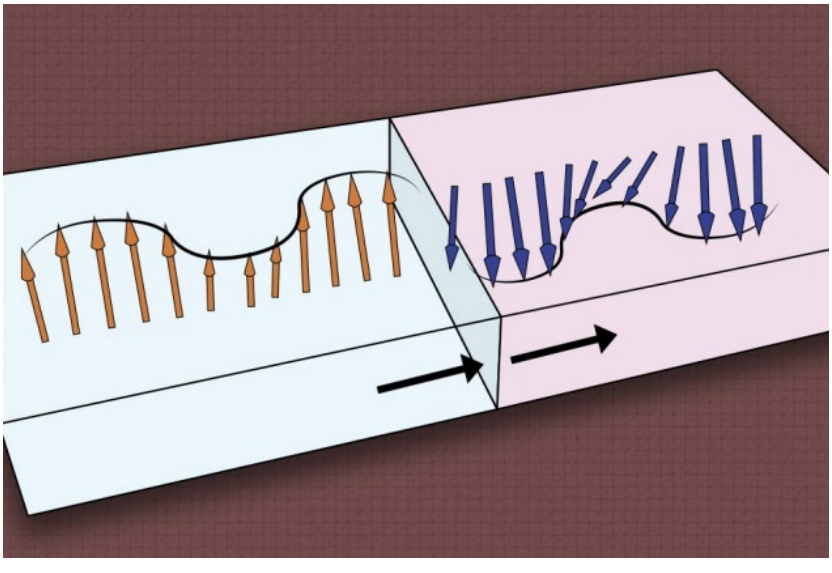
MIT researchers have devised a novel circuit design that enables precise control of computing with magnetic waves with no electricity needed. The advance takes a step toward practical magnetic-based devices, which have the potential to compute far more efficiently than electronics.

Classical computers rely on massive amounts of electricity for computing and data storage and generate a lot of wasted heat. In search of more efficient alternatives, researchers have started designing magnetic-based "spintronic" devices, which use relatively little electricity and generate practically no heat.

Spintronic devices leverage the "spin wave"—a quantum property of electrons—in magnetic materials with a lattice structure. This approach involves modulating the spin wave properties to produce some measurable output that can be correlated to computation.

Spin waves are ripples of energy with small wavelengths. Chunks of the spin wave, which are essentially the collective spin of many electrons, are called magnons. While magnons are not true particles, like individual electrons, they can be measured similarly for computing applications.

By controlling the spin wave, they found they could control the position of the domain wall. This relies on a phenomenon called, "spin-transfer torque," which is when spinning electrons essentially jolt a magnetic material to flip its magnetic orientation.



In the researchers' work, they boosted the power of injected spin waves to induce a certain spin of the magnons. This actually draws the wall toward the boosted wave source. In doing so, the wall gets jammed under the antenna, effectively making it unable to modulate waves and ensuring uniform magnetization in this state.

Such innovations could enable practical wave-based computing for specific tasks, such as the signal-processing technique, called fast Fourier transform. Next, the researchers hope to build a working wave circuit that can execute basic computations. (Source: MIT)

Continually Learning: Why Stop?

Flex Talk

by Tara Dunn, OMNI PCB

On two Friday afternoons in November, the SMTA Upper Midwest Chapter hosted a "Crash Course" at the University of Minnesota. The goal of these sessions was to educate and create interest for immerging engineers about the electronics industry as well as help with soft skills, such as resume building, professional networking, and interview skills.

Two of us from our local chapter were slotted to speak in the first session. We were planning to talk about all the cool, interesting, and fun things you can do with a career focused on PCB design, fabrication, and assembly. There are a lot of stories to choose from. We started the session with a successful search and rescue story, pointing out all the different tools available to the rescue team that all contained flex or rigid-flex: the GPS in the tractor showing the exact layout of the field, the drone, the helicopter, the communication system, and even the cellphone. I probably could have talked for hours! But what quickly became apparent was that although the audience was involved in projects that utilize PCBs, there was very little knowledge about how a PCB was manufactured.

The rest of the presentation was spent going through the fabrication process step by step. What I learned, as we went through the basic process steps required to build a simple custom PCB, was how easy it was to slip into acronyms or process terminology that turned the student's eyes to that glassy, faraway look. We recognized that and worked to dial it back to something more manageable in a short time.

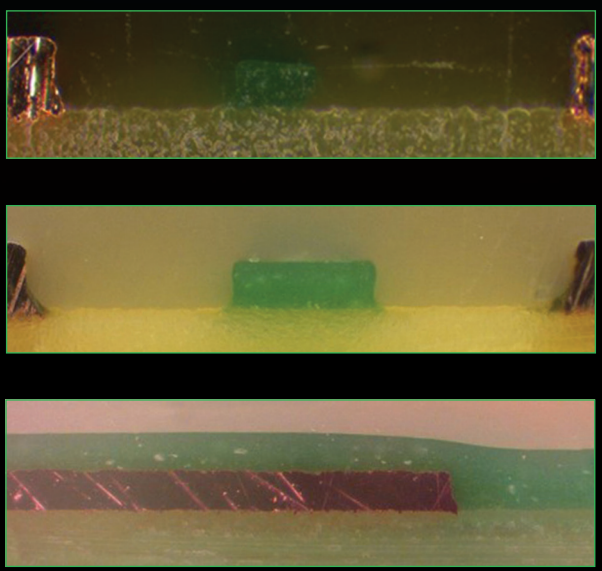
FLEXIBILITY NEVER LOOKED SO GOOD

FLEX SOLDER MASK

BY TAIYO

PSR-9000 FLX03G LDI

- Designed specifically for Orbotech LDI equipment
- Dark green glossy finish
- Fast exposure time
- UL listed VTM-0 on 2 mil polyimide
- Straight sidewalls



TAIYO AMERICA

Your Ultimate Solution

Phone [775] 885-9959 • www.taiyo-america.com • info@taiyo-america.com

What I hope the students learned is that those "little green squares" require custom tooling and complex processing, even for a simple design. I also hope the students are at least a little interested in learning more about careers in this field. During the second session, they were introduced to local companies that are sponsoring job shadowing, internships, etc. I am confident there were some good connections made.

I joked that the student's eyes turned to that glassy, faraway look, but I remember back to a few times I experienced that myself this fall. I had the opportunity to attend several different conferences and a couple of engineering meetings in a field that I am not completely familiar with. A few times, I was quickly scribbling notes and spelling phonetically so I could go back later and get a better understanding of what everyone was talking about. I am sure that I ended up with the blank stare a few times as well!

The PCB industry is moving and advancing so quickly that there is always something new to learn. I encourage everyone to break outside of their comfort zone and search out something new.

The PCB industry is moving and advancing so quickly that there is always something new to learn. I encourage everyone to break outside of their comfort zone and search out something new. It can be uncomfortable to feel like the only person in the room that isn't completely knowledgeable about the topic, but I can almost guarantee there are others.

Where are these opportunities to search out new information? The electronics industry is fortunate to have both large conferences such as IPC APEX EXPO and SMTAi, as well as smaller regional conferences such as the Additive Electronics: PCB Scale to IC Scale Conference and the IPC Electronic Materials Forum. We also have an active network of local chapter meetings through organizations such as SMTA, IEEE, and IPC that meet more frequently and provide a lot of valuable education.

What are some interesting new topics to start exploring? Are you currently working with flex and rigid-flex designs? Although flexible circuits have been used for decades, they are one of the fastest-growing portions of the PCB market, and I regularly speak with people who are just starting to tackle their first designs. One key thing to explore before starting your first flex design is flexible materials. What are the common laminates? What does your supplier regularly stock? Which coverlay type will be best for your application: polyimide or a flexible photoimageable coverlay? Will you need a stiffener? If so, will that be FR-4 or polyimide?

Once you have learned materials, the next things to learn are the tips and tricks to help improve flexibility in the design: teardrops, tiedowns, routing the traces perpendicular to the bend, do not place plated through-holes in the bend area, be sure that coverlay and stiffener endpoints overlap by at least 0.030", and the list goes on!

Another up-and-coming technology is additive PCB processes that are enabling very high-density interconnect technology and achieving trace and space at or below 0.001". SAP, mSAP, and SLP are all new acronyms gaining a lot of attention. Substrate-like PCBs, semi-additive processes, and modified semi-additive processes are new to PCB fabrication, and technologies are available that allow these processes to be integrated with our traditional subtractive etch processes, opening a whole new way of solving next-generation electronics challenges. Imagine the size and weight reduction when able to route with 0.001" trace and space or the added electronics that could be included in the existing footprint with this technology.

There is a lot of work currently being done to help define these benefits and identify creative

ways to implement this technology to solve challenges with complex pinouts, space and weight constraints, and RF performance. This is a topic that I have seen at several industry conferences, and IPC APEX EXPO 2020 will be a very good location to learn more.

As I mentioned, the industry is advancing at an incredibly fast pace, and there is always something new to explore. In the past, there was a time where focused learning and being the expert in one specific process provided a sense of pride and accomplishment. Today's electronics require us to be continually learning well outside the scope of our daily work. New materials are being introduced regularly, and new processes are being developed that break the barriers of traditional process methods and design rules. This is an exciting time to be involved in the electronics field. Don't be afraid to jump in, ask questions, and even spell phonetically as you learn. **FLEX007**

Tara Dunn is the president of Omni PCB, a manufacturer's rep firm specializing in the PCB industry. To read past columns or contact Dunn, click here.

Bell Autonomous Pod Transport 70 Wins Popular Science 'Best of What's New in Aerospace' Award

Popular Science has recognized the Bell Autonomous Pod Transport (APT) 70 with a Best of What's New Award in the Aerospace category for 2019. In 2018, Bell received a Best of What's New Award in the Aerospace category for its Bell V-280 Valor Joint Multi-Role Technology Demonstrator. This continued recognition validates Bell's commitment to technology advancement and innovation in the aerospace industry.

"The Best of What's New is our celebration of the most impactful and exciting innovations of the year," says Popular Science Editor-in-Chief Joe Brown. "This expertly vetted collection lays the groundwork for a healthier, safer, and awe-inspiring future–in our homes, cities, outer space, and everywhere in between. We're proud to bring you the Best of What's New 2019."

APT 70 is part of the eVTOL family of vehicles Bell is developing, capable of rotation and translation in flight to achieve high speeds and long-range flight. APT's diverse mission capabilities allow it to serve many industries while keeping operations simple, efficient and fast. The vehicle's true multi-copter payload capability with fixed wing speed and endurance enables the customer to save time, going faster, longer.

"The team continues to develop innovative technology to meet the transportation needs of the future," said Scott Drennan, vice president, Innovation. "We are honored to be recognized by a leading industry voice like Popular Science, confirming the team's hard work and dedication to advancing technology and expanding the scope of vertical lift."

Bell announced the successful first autonomous flight of the APT 70 in August and recently carried 60lb payload with plans to expand to the maximum payload of 70lbs. The Bell team continues to develop technologies and expand the envelope to prepare the vehicle for entry into market.

(Source: Bell)

Transparent Flexible Circuits

EPTE Newsletter
by Dominique K. Numakura, DKN Research LLC

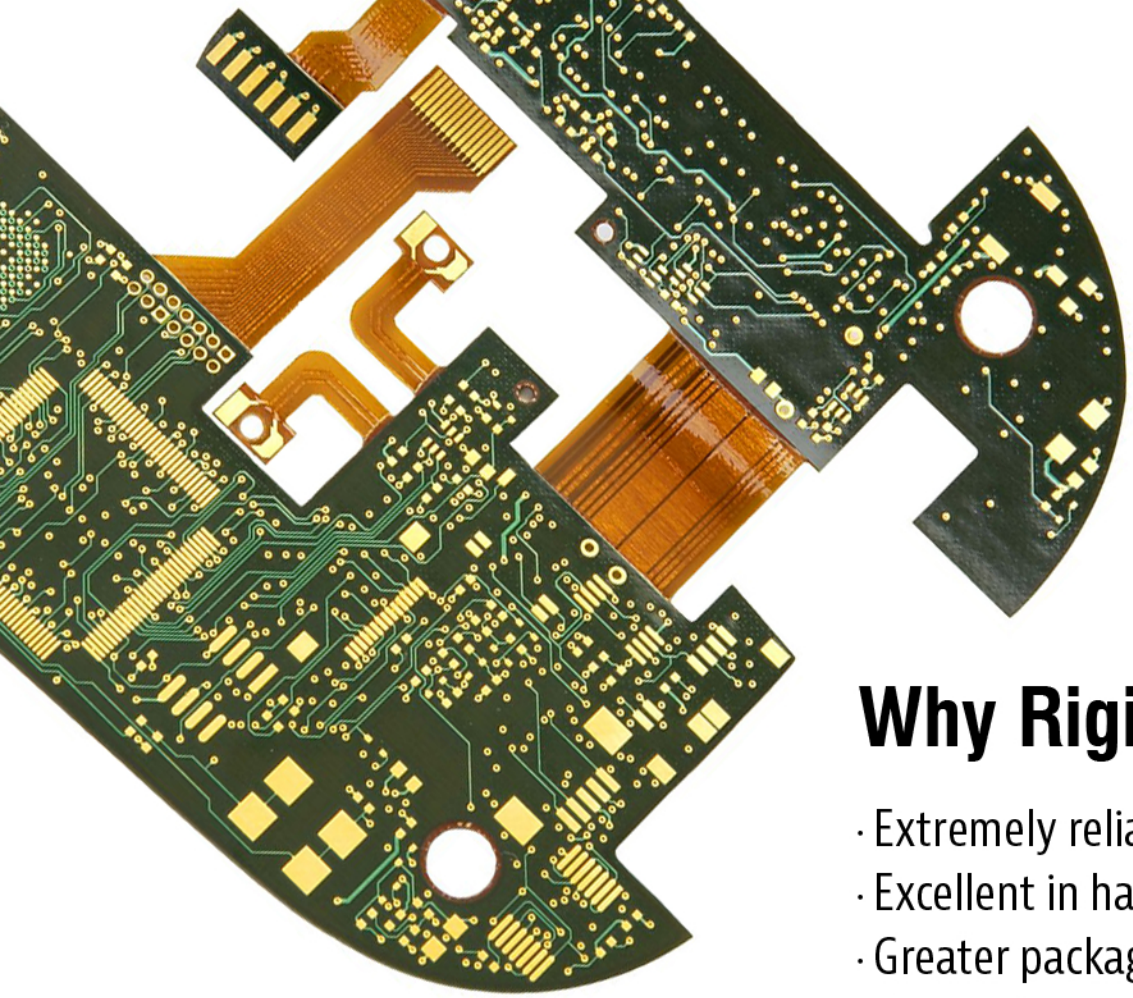
When DuPont developed the heat-resistant polyimide film Kapton in the 1960s, there was no large-volume application. Now, DuPont has developed a new circuit concept for flexible circuits, but the company did not have a business plan for the products. The company developed and commercialized the copper laminate Pyralux® as the raw material of reliable flexible circuits and rigid-flex for aerospace applications. DuPont also developed basic constructions of the circuit with process conditions. When DuPont promoted Pyralux, the company provided the technical information as the design guide and processing manual. The business strategy was successful, and DuPont enjoyed good profits. The basic construction of the flexible circuits was almost fixed.

The industry has grown significantly in the 1980s and 1990s, especially in consumer electronics, such as personal computers and cameras. The flexible circuit segment made another expansion in the 21st century with the growth of cellphones/smartphones. The industry expanded and became larger during the last four decades. However, the basic construction of the circuits has not changed. Kapton® is still a major material as the base film of flexible circuits. The circuit density rose, and via hole size shrank. Most of the flexible circuits retain the Kapton color. Few mobile device manufacturers seem to dislike the Kapton color, and they asked flexible circuit manufacturers to paint the circuits black, but customers don't request changes to the base polyimide films.

On the other hand, new demands for the flexible circuits have appeared in wearable devices and medical electronics, which require abnormal properties that cannot be satisfied by traditional polyimide base flexible circuits; one of these requirements is transparency. Previously, it was not very critical for the applications; both customers and manufacturers of the flexible circuits did not pay attention to performance. However, the transparency of the circuits is critical for optical sensors and lighting devices. Therefore, the requirement about transparent circuit has become very keen (Figure 1).

Figure 1: Transparent/flexible circuit example.

Almost five years ago, a material supplier in Japan commercialized a clear polyimide film as the base material of heat-resistant flexible circuits. However, the etching companies did not



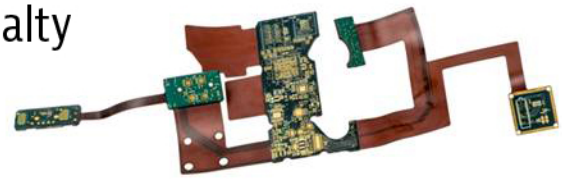
Printed Circuits

Why Rigid-Flex?

- Extremely reliable
- Excellent in harsh environments
- Greater packaging density
- Ultra-thin, ultra-light packaging

Why Choose Us?

- Experienced–rigid-flex our focus since 1997
- Complex, high-reliability rigid-flex our specialty
- State-of-the-art equipment
- We are easy to work with
- Provide design assistance as required
- The most extensive UL qualification available
- Mil spec certified: MIL-PRF-31032, Mil-P-50884
- ISO 9001 2015, ITAR, JCP 4532 certified



12-Layer Rigid-flex Board with Laser Vias

Visit Our Website for Rigid-Flex Design Aids

LinkedIn

Contact Printed Circuits Today! 888.844.8416 **printedcircuits.com**

consume the new polyimide films directly because they could not make the circuits without copper foils. Therefore, laminate manufacturers developed copper laminates using the transparent polyimide film. Unfortunately, the customers were not very happy about the properties of the copper laminates because the glue layer reduced the transparency remarkably, and the glue materials were not very transparent.

Next, it was our turn. We developed a series of flexible copper laminates without a glue layer using plating processes. Several flexible circuit manufacturers could make the circuits with the new laminate with their existing etching process, but they were not very positive about expanding that business. They needed coverlay material to finish the flexible circuits. I asked material manufacturers to develop film base coverlay or screen-printable coverlay ink, and they did.

On the other hand, device manufacturers have been demanding transparent conductors for optical applications. It was a hard requirement for traditional flex circuit manufacturers, so we had to change their minds. We had several choices to reach the goal. The first candidate was an indium tin oxide (ITO) film, but it did not have good flexibility. The second candidate was transparent conductive ink made of an organic molecule, but it was not stable at high-temperature processes, such as soldering. The third candidate was a silver nanowire ink, which was difficult to find appropriate process conditions, and its high cost was another headache. These candidates have larger conductor resistance than 100 ohm-cm. This conductivity restricts applicable areas.

The fourth candidate was a finely etched mesh pattern made of thin copper layer. In this case, the conductivities have a trade-off relationship against transparency. And the transparency depends on the fine pattern etching capabilities of the circuit manufacturers. Currently, we do not have a perfect solution for transparent conductors. We have to make appropriate balances between the materials and performances.

Our new customers sometimes request crazy properties because they do not know standard design rules. Recently, some wearable device manufacturers have asked for elastic, transparent, and heat-resistant flexible circuits, and it's almost impossible to satisfy all of the requirements. However, one of our colleague companies developed silicone rubber base flexible circuits with screen-printable silver inks. The base materials are elastic, transparent, and heat-resistant, and the customer's requirements were 80% satisfied.

There is no limit to customers' requirements, especially the medical device manufacturers, who have been generating ultimate properties for the flexible circuits. Circuit manufacturers have been developing a broad range of materials with various manufacturing processes, and they do not satisfy merely 10% of the requirements, but 80–90%. Additional design ideas will help the requirement satisfaction reach 100%. We will be very happy to share our experiences regarding these new demands.

Headlines

1. Sharp (electronics company in Japan)

Demonstrated the secondary battery system "JH-WBPB150" for home use (capacity: 6.5 kWh; price: 2.6 million yen).

2. JPEA (organization of solar generators in Japan)

Reported that 80% of home solar generator holders utilized self-operation systems during the power outage caused by Typhoon #1915 in September.

3. Toyota (automobile manufacturer in Japan)

Will introduce a flexible organic EL display for the next-generation concept car "LQ."

4. Panasonic (electronics company in Japan)

Co-developed a volume production process to produce chemical chips on a glass substrate with the Institute of Microchemical Technology.

5. Tokyo University (Japan)
Produced wafers of organic semiconductors by using a simple printing process, which could be a valuable step-up to make organic transistors.

6. Sharp (electronics company in Japan)
Co-developed a 30" foldable, flexible OLED display with NHK, which can be put in a 4-cm diameter pipe (weight: 100 grams).

7. SK Hynix (semiconductor manufacturer in Korea)
Opened an R&D center of C-MOS Sensor devices in Hamamatsucho, Tokyo, to catch up with the growing market.

8. AIST (R&D organization in Japan)
Improved the resolution of EDS element analysis in SEM system more than two orders to determine the 3D structure of carbon nanotube molecules.

9. Hokkaido University and Bsan University (Japan and Korea)
Made the chemical reaction process of SrCoOx, the basic material of next-generation memory devices.

10. Touchence (device manufacturer in Japan)
Agreed to co-develop finger type touch sensors using MEMS manufacturing processes with Kagawa University.

Dominique K. Numakura is the managing director of DKN Research LLC. Contact haverhill@dknreseach.com for further information and news.

5G Economy to Generate $13.2T in Sales Enablement by 2035

Qualcomm Technologies, Inc., announced new study findings, forecasting that 5G will generate $13.2 trillion in sales enablement by 2035. The IHS Markit 5GEconomy Study, commissioned by Qualcomm Technologies, shows an increase of $1 trillion in 5G sales enablement by 2035 over the original forecast released in 2017 which estimated $12.3 trillion.

IHS Markit correlated this increase in large part due to the early completion of the initial 5G standard and the subsequent accelerated rollout. As noted at the Qualcomm 5G Summit in October, there are now more than 30 live 5G networks and more than 40 OEMs launching 5G-enabled devices, making it the fastest ever rollout of a cellular technology generation.

Additional key predictions from the study show:

- The global 5G value chain will support 22.3 million jobs in 2035. This is 3.4 times as many jobs as a similar level of economic output supports today
- The global 5G value chain will invest an average of $235B annually to continue to expand and strengthen the 5G technology base. This figure represents nearly 80% of the total US federal, state and local government spending on transportation infrastructure in 2017
- The U.S. is forecasted to lead in cellular R&D and capital expenditures in the 5G mobile value chain with 27% average annual investment, followed closely by China at 26%

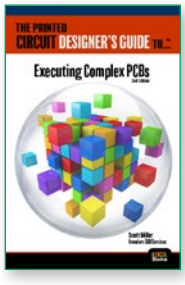
"Like electricity and the steam engine before it, IHS Markit views 5G as a catalyst that will thrust mobile technology into the exclusive realm of General-Purpose Technology," said Bob Flanagan, economics consulting director, IHS Markit.

(Source: Qualcomm)

1 What Does Intelligent Routing Look Like? ▶

Many PCB design projects miss schedule commitments by 70% due to delayed routing and lack of automation. Fortunately, over the past several years, powerful routing capabilities have been added that allow designers to address a number of specific, critical tasks.

2 Book Review: The Printed Circuit Designer's Guide to... Executing Complex PCBs ▶



If you are serious about designing complex PCBs (most designs today are far from elementary) and even more serious about doing it right the first time, then this is the book for you. Loaded with guidelines for designing cutting-edge PCBs, this book is filled with real-world examples and tips, tricks, and techniques by some of Freedom CAD's most experienced designers.

3 Fresh PCB Concepts: The Current Material Situation ▶


Ruben Contreras

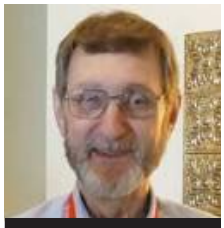
We have all heard about the component crisis in the circuit board industry, and maybe you heard about the CCL shortage, but how many are aware of the bare board material shortage? Ruben Contreras explains the current material situation and tips to address this issue.

4 Communication, Part 5: Internet Impedance Calculators for Modeling ▶


Mark Thompson

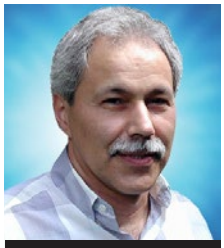
Bob Chandler of CA Design and Mark Thompson of Prototron Circuits address how new engineers use internet impedance calculators for modeling (e.g., formulas versus recipes) in Part 5 of this series. Do you use impedance calculators that you found on the internet? Read on!

5 AltiumLive Frankfurt 2019: Rick Hartley Keynote ▸


Rick Hartley

Introduced by Lawrence Romine, Altium's VP of corporate marketing, as a "low impedance presenter with a passion for his topic," Rick Hartley delivered the opening keynote at the AltiumLive 2019 European PCB Design Summit in Frankfurt, Germany. Pete Starkey provides an overview of Hartley's presentation, entitled "What Your Differential Pairs Wish You Knew."

6 Quiet Power: How Much Signal Do We Lose Due to Reflections? ▸

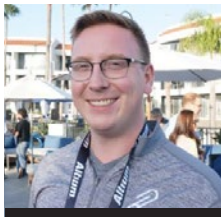

Istvan Novak

We know that in the signal integrity world, reflections are usually bad. In clock networks, reflection glitches may cause multiple and false clock triggering. In medium-speed digital signaling, reflections will reduce noise margin, and in high-speed serializer/deserializer (SerDes) signaling, reflections increase jitter and create vertical eye closure.

7 Dana on Data: The DFM/Data Transfer Process Is Broken ▸

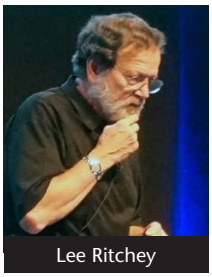
In a world that is showing great strides toward implementing a Factory 4.0 world, why can't a design be passed from a designer to the fabricator without errors every time? Dana Korf emphasizes moving the responsibility up in the food chain, examines key design package error categories, and proposes creating a cultural change.

8 Dugan Karnazes Discusses His New Startup ▸


Dugan Karnazes

Andy Shaughnessy caught up with Dugan Karnazes again this year and discussed his new startup, Velocity Research, which is a one-stop shop for design made up of technical creatives. The Grand Rapids company is already doing design work for a variety of customers, from individuals to multinational companies.

9 Lee Ritchey on the Direction of PCB Design ▸


Lee Ritchey

Andy Shaughnessy and Nolan Johnson recently spoke with Lee Ritchey of Speeding Edge about the direction of PCB design. Lee also discusses some of the changes that he has seen in this industry over the past 40 years and some of the technological drivers that are causing designers to think more like electrical engineers than ever before.

10 Why Does the PCB Industry Still Use Gerber? ▸

Ucamco has over 35 years of experience in developing and supporting cutting-edge software and hardware solutions for the global PCB industry. Our customers include the electronics industry's leading companies, and many of them have been with us for over 30 years. We are dedicated to our industry and excellence in everything we do, which includes our custodianship of the Gerber format.

Career Opportunities

Pssst! Are You Looking for Someone?

Place your notice in our Help Wanted section.

For just $500, your 200 word, full-column—or, for $250, your 100 word, half-column—ad will appear in the Help Wanted section of all three of our monthly magazines, reaching circuit board designers, fabricators, assemblers, OEMs and suppliers.

Potential candidates can click on your ad and submit a résumé directly to the email address you've provided. If you wish to continue beyond the first month, the price is the same per month. No contract required. We even include your logo in the ad, which is great branding!

To get your ad into the next issue, contact:
Barb Hockaday at barb@iconnect007.com or +1.916.608.0660 (-7 GMT)

I-Connect007
GOOD FOR THE INDUSTRY

smt007 pcb007 pcb design007

Career Opportunities

Technical Support Engineer III

The technical support engineer III is responsible for providing leading-edge, high-level technical support to Indium Corporation's customers, potential customers, and sales staff. Due to their senior position and experience, their role also includes conceiving and devising projects, assisting with staff career development, marketing guidance, and more. The technical support engineer III has learned, mastered, and demonstrated unique and specific skills and information throughout their career. They are responsible for at least one sales territory and for leading other engineers. They train and evaluate colleagues on unique and general information. Continuing education/training is critical.

Requirements

- Technical undergraduate degree (B.S. in engineering, chemistry, physics, metallurgy, or materials science)
- 15 years of direct technical experience in applied materials science, electronics assembly techniques, and/or electronics assembly technical service
- Demonstrated technical competency
- Strong interpersonal, communication, and presentation skills
- Ability to work, with ease, with executive-level counterparts
- Strong alignment with the corporate and departmental missions
- Ability to work cooperatively and effectively in a cross-functional team environment
- Ability to travel with limited notice
- Proficient in Word, Excel, and PowerPoint
- Experience with JMP or Minitab preferred
- Special consideration is given to candidates with language skills in Spanish and/or Chinese

apply now

West Software Application Engineer

This position reports directly to the Orbotech West software support manager and works with customers to support Orbotech's pre-production software products. Acts as a focal point for technical issues, manages product implementation projects, provides customer training, and supports the sales process. Advanced knowledge of Frontline PCB products, including InCam, InPlan, InStack, InSight, Genesis, and Genflex. Ability to travel and manage time to maximize results. Requires both written and oral technical communication skills. Skilled in the use of scripting languages, including C-Shell, Perl, or Python. Knowledge of relational databases and HTML/XML highly desirable. Knowledge of PCB manufacturing processes. Familiar with the processes used in front-end engineering departments at PCB fabrication sites. Requires use of project management skills to organize and complete projects that involve the implementation of sophisticated software tools used in printed circuit fabrication facilities.

An expected average of 35%+ travel. College degree or equivalent technical education, in addition to a minimum of five-plus years of related experience. Experience supporting sales and sales activities is a plus. U.S. citizen with the ability to work and travel within the U.S., Canada, and internationally.

apply now

Career Opportunities

ventec
INTERNATIONAL GROUP
騰輝電子

OEM Sales Manager
Chicago/Home-Office-Based

Want to advance your career by joining a globally successful and growing world-class CCL manufacturer and help drive that success? We are seeking to hire an OEM sales manager to grow and manage key customer accounts with OEM's and Tier 1 manufacturers in the USA, focusing on Ventec's core market segments: mil/aero, automotive, and medical, offering a full range of high-reliability materials, including polyimide, IMS, and thermal management products.

Skills and abilities required for the role:
- Non-negotiable: Drive and tenacity!

Required:
- 7 to 10 years' experience in the PCB industry in engineering and/or manufacturing
- Detail-oriented approach to tasks
- Ability to manage tasks and set goals independently as well as part of a team
- Knowledge of MS office products

Full product training will be provided.

This is a fantastic opportunity to become part of a successful brand and leading team with excellent benefits.

Please forward your resume to jpattie@ventec-usa.com and mention "Technical Sales Engineer—Chicago" in the subject line.

apply now

FREEDOM CAD SERVICES

Sr. PCB Designer–Mentor Xpedition

Freedom CAD is a premier PCB design service bureau with a talented team of 30+ dedicated designers providing complex layouts for our enviable list of high-tech customers. Tired of the commute? This is a work-from-home, full-time position with an opportunity for additional compensation for overtime work at time and a half.

Key Qualifications
- EXPERT knowledge of Xpedition VX 2.x
- Passionate about your PCB design career
- Skilled at HDI technology
- Extensive experience with high-speed digital, RF, and flex and rigid-flex designs
- Experienced with signal integrity design constraints encompassing differential pairs, impedance control, high speed, EMI, and ESD
- Excellent team player who can lead projects and mentor others
- Self-motivated with the ability to work from home with minimal supervision
- Strong communication, interpersonal, analytical, and problem-solving skills
- Other design tool knowledge is considered a plus (Altium, Allegro, PADS)

Primary Responsibilities
- Design project leader
- Lead highly complex layouts while ensuring quality, efficiency, and manufacturability
- Handle multiple tasks and provide work leadership to other designers through the distribution, coordination, and management of the assigned workload
- Ability to create from engineering inputs, board mechanical profiles, board fabrication stackups, detailed board fabrication drawings and packages, assembly drawings, assembly notes, etc.

apply now

Career Opportunities

ROGERS CORPORATION

Advanced Connectivity Solutions

Senior Development Engineer

Rogers Corporation is seeking a senior development engineer accountable for the development of more complex products and processes, the establishment of sound technical bases for these developments, and effective interaction with technology, process, and platform innovation; operations; sales and marketing; and process engineering personnel to commercialize these developments.

Essential Functions:

- Design and conduct experiments and interpret the results
- Report on projects in both written and verbal formats at all levels of the organization
- Perform technical troubleshooting of new products and processes; act as new product/concept incubator for new technologies and platforms, identifying opportunities for improvement and incorporation design for manufacturing requirements resulting in a viable, scalable product
- Provide ongoing process and manufacturing support to newly launched products as applicable
- Provide support in terms of analytical equipment maintenance, methods development, material analysis, and documentation of new process or products
- Manage capital projects for the purchase and installation of new process or support equipment; train employees in new processes

Required Education and Experience:
Ph.D., Ch.E., M.E., or material science, or B.S. or higher in a technical discipline with accomplishment in product development and project management.

Rogers Corporation provides equal employment opportunities to minorities, females, veterans, and disabled individuals as well as other protected groups.

apply now

KOH YOUNG AMERICA

Field Service Engineer
(Location Negotiable)

Are you passionate about delivering an exceptional user experience? Come work as a field service engineer at the industry's leading inspection company that offers great benefits with opportunities to advance while learning alongside accomplished business leaders.

The Company: Koh Young is the leading 3D inspection solutions provider in the electronics manufacturing industry. With is new offices in Atlanta and Guadalajara, it helps its customers optimize their printed circuit board assembly process.

The Position: Deliver technical services–including installation, support, and maintenance–to elevate the user experience. Location is flexible, but OH, IN, IL, MA, MI, FL, CA, or Toronto are desired.

The Reasons: An opportunity to apply leading-edge inspection technology to products you know and use every day. A great environment that supports its team and treats everyone like family.

Join the industry's leading provider of
true 3D inspection solutions

Interested? Submit your resume below.

apply now

Career Opportunities

Gardien Is Hiring!

The Gardien Group, a leading solutions provider in the PCB industry, is looking to fill multiple openings in their China, Japan, Taiwan, and United States service centers.

We are looking for electrical engineers, operations managers, machine operators, and sales executives. Prior experience in the PCB industry is beneficial but not essential. Training will be provided along with excellent growth opportunities, a benefits package, and periodic bonuses.

Our global teams are from diverse cultures and work cohesively as a tight-knit unit. With performance and initiative, there are plenty of opportunities for professional growth.

Gardien is an equal opportunity employer. Employment decisions are made without any regard to race, color, religion, national or ethnic origin, gender, sexual orientation, age, disability, or other characteristics.

Interested candidates, please contact us with your resume and a cover letter. Kindly note that only shortlisted candidate will be contacted.

Apply at careers@gardien.com.

apply now

Assistant Department Manager, Operations, Carson City, NV

This is an entry-level professional management trainee position. Upon completion of a 1-2-year apprenticeship, this position will be elevated to facility/operations manager. Primary functions during training: shadow incumbent staff managers to learn and understand the operations and personnel of the operations department. This position will train and learn, develop, implement, and coordinate strategies related directly to the manufacture of Taiyo products. Additionally, this position will be learning all about the facility, environment, and health and safety functions. Eventually, this position will be responsible for the administration, security and maintenance of the facility and warehouse

Required Experience/Education:

- 4-year college degree in industrial engineering or another similar science discipline combined with work experience in ink or coatings manufacturing
- Ability to read, analyze, and interpret common scientific and technical journals, financial reports, and legal documents
- Ability to respond to inquiries or complaints from customers, regulatory agencies, or members of the business community
- Ability to develop and implement goals, objectives, and strategies
- Ability to effectively present information to top management, public groups, and/or boards of directors
- Ability to apply principles of logical or scientific thinking to a wide range of intellectual and practical problems
- Knowledge of governmental safety, environmental, transportation regulations/laws

Preferred Skills/Experience:

- Bilingual (Japanese/English)
- Toyota Production System (TPS)

Working Conditions:

- Occasional weekend or overtime work

See complete job listing for more information.

apply now

Career Opportunities

eptac

TRAIN. WORK SMARTER. SUCCEED.

Become a Certified IPC Master Instructor

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

Qualifications and skills

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

Benefits

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

apply now

INSULECTRO

Technical Account Manager Chicago/Minneapolis

Insulectro, the largest national distributor of printed circuit board materials, is seeking a talented sales superstar for a Technical Account Manager role based out of either our Chicago or Minneapolis office. This role will focus on maintaining the existing customer base and developing new business within the assigned territory in both the printed circuit board and printed electronics industries. We are looking for the perfect fit of education, experience, and attitude that matches our company culture and enhances the service level to our customers.

Qualifications:

- A self-motivated business professional who is driven to succeed with a minimum of 3 years outside sales experience in the PCB or PE industry
- Proven sales/business development record
- Excellent communication and interpersonal skills
- OEM and electronic assembly experience is a plus

We offer:

- Competitive salary and commission plan with a comprehensive benefits package
- A fun, high-energy company with an entrepreneurial spirit
- A great group of people to work with!

apply now

Career Opportunities

Analyst Programmer, Hong Kong

We believe in caring about our people because they are our greatest asset. CML works with multicultural stakeholders daily to achieve more and bring them the best solutions. That's why we continuously invest in optimizing our culture and focus on providing our team with opportunities to develop their skills (e.g., through professional coaching to achieve their highest potential).

The analyst programmer will assist the IT and ERP manager in Hong Kong to support the company's BI systems, ERP systems, and other related IT-landscape applications.

In addition, this post will participate in system development projects and provide support including, but not limited to, user requirement collection and analysis, user training, system documentation, system support and maintenance, enhancement, and programming.

- Develop and enhance related IT systems and applications
- Prepare functional specifications
- Transfer the relevant business and interface processes into IT systems and other applications to get a maximum automation degree and prepare all required business reports
- Conduct function testing and prepare documentation
- Manage help desk/hotline service

CML is a leading provider of printed circuit boards. We develop tailor-made sourcing and manufacturing solutions for our customers worldwide with strong partnerships and reliable connections.

apply now

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

Career Opportunities

Development Chemist
Carson City, NV

Develop new products and modify existing products as identified by the sales staff and company management. Conduct laboratory evaluations and tests of the industry's products and processes. Prepare detailed written reports regarding chemical characteristics. The development chemist will also have supervisory responsibility for R&D technicians.

Essential Duties:

- Prepare design of experiments (DOE) to aid in the development of new products related to the solar energy industry, printed electronics, inkjet technologies, specialty coatings and additives, and nanotechnologies and applications
- Compile feasibility studies for bringing new products and emerging technologies through manufacturing to the marketplace
- Provide product and manufacturing support
- Provide product quality control and support
- Must comply with all OSHA and company workplace safety requirements at all times
- Participate in multifunctional teams

Required Education/Experience:

- Minimum 4-year college degree in engineering or chemistry
- Preferred: 5-10 years of work experience in designing 3D and inkjet materials, radiation cured chemical technologies, and polymer science
- Knowledge of advanced materials and emerging technologies, including nanotechnologies

Working Conditions:

- Chemical laboratory environment
- Occasional weekend or overtime work
- Travel may be required

apply now

Multiple Positions Available

The Indium Corporation believes that materials science changes the world. As leaders in the electronics assembly industry we are seeking thought leaders that are well-qualified to join our dynamic global team.

Indium Corporation offers a diverse range of career opportunities, including:

- Maintenance and skilled trades
- Engineering
- Marketing and sales
- Finance and accounting
- Machine operators and production
- Research and development
- Operations

For full job description and other immediate openings in a number of departments:

www.indium.com/jobs

apply now

Career Opportunities

Manncorp™

SMT Field Technician
Huntingdon Valley, PA

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

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

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

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

apply now

U.S. CIRCUIT

Sales Representatives (Specific Territories)

Escondido-based printed circuit fabricator U.S. Circuit is looking to hire sales representatives in the following territories:

- Florida
- Denver
- Washington
- Los Angeles

Experience:
- Candidates must have previous PCB sales experience.

Compensation:
- 7% commission

Contact Mike Fariba for more information.

mfariba@uscircuit.com

apply now

Career Opportunities

ZENTECH

Zentech Manufacturing: Hiring Multiple Positions

Are you looking to excel in your career and grow professionally in a thriving business? Zentech, established in Baltimore, Maryland, in 1998, has proven to be one of the premier electronics contract manufacturers in the U.S.

Zentech is rapidly growing and seeking to add Manufacturing Engineers, Program Managers, and Sr. Test Technicians. Offering an excellent benefit package including health/dental insurance and an employer-matched 401k program, Zentech holds the ultimate set of certifications relating to the manufacture of mission-critical printed circuit card assemblies, including: ISO:9001, AS9100, DD2345, and ISO 13485.

Zentech is an IPC Trusted Source QML and ITAR registered. U.S. citizens only need apply.

Please email resume below.

apply now

BLACKFOX
Premier Training & Certification

IPC Master Instructor

This position is responsible for IPC and skill-based instruction and certification at the training center as well as training events as assigned by company's sales/operations VP. This position may be part-time, full-time, and/or an independent contractor, depending upon the demand and the individual's situation. Must have the ability to work with little or no supervision and make appropriate and professional decisions. Candidate must have the ability to collaborate with the client managers to continually enhance the training program. Position is responsible for validating the program value and its overall success. Candidate will be trained/certified and recognized by IPC as a Master Instructor. Position requires the input and management of the training records. Will require some travel to client's facilities and other training centers.

For more information, click below.

apply now

Events Calendar

DesignCon 2020 ▶
January 28–30, 2020
Santa Clara, California, USA

IPC APEX EXPO 2020 ▶
February 1–6, 2020
San Diego, California, USA

Medical Design & Manufacturing ▶
February 11–13, 2020
Anaheim, California, USA

Embedded World ▶
February 25–27, 2020
Nuremberg, Germany

CPCA Show 2020 ▶
March 16–18, 2020
Shanghai, China

electronica & productronica China ▶
March 18–20, 2020
Shanghai, China

LOPEC Exhibition and Conference (Driving the Future of Printed Electronics) ▶
March 24–26, 2020
Munich, Germany

KPCA and KIEP Show ▶
April 22–24, 2020
Kintex, Korea

Additional Event Calendars

Coming Soon to *DesignOO7 Magazine*

January 2020: Networking & Education
Our Designers Council issue will also feature previews of IPC APEX EXPO and DesignCon.

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

SALES MANAGER: **BARB HOCKADAY**
(916) 608-0660; barb@iconnect007.com

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

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

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

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

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

ASSOCIATE EDITOR: **KIERSTEN ROHDE**
kiersten@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: **SHELLY STEIN**

COVER IMAGE: **ADOBE STOCK © PETR VACLAVEK**

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

December 2019, Volume 8, Number 12
DESIGN007 MAGAZINE is published monthly,
by BR Publishing, Inc.

ADVERTISER INDEX

I-Connect007

GOOD FOR THE INDUSTRY

FREE SUBSCRIPTION

myiconnect007.com

mediakit.iconnect007.com

EDITORIAL CONTACT

Andy Shaughnessy
andy@iconnect007.com
+1 404.806.0508 GMT-5

SALES CONTACT

Barb Hockaday
barb@iconnect007.com
+1 916 365-1727 GMT-7

www.iconnect007.com