APRIL 2020

I-Connect007

DESIGN007 MAGAZINE

Mitigate the Effects of Tin Whiskers

Expert Phil Kinner explains how conformal coating and cleaning can protect your boards from corrosion and mitigate against the effects of tin whiskers!

LEARN HOW

I·007e Micro Webinars

ELECTROLUBE
THE SOLUTIONS PEOPLE

DESIGN007 MAGAZINE

Design Economics

You may have taken an economics course in school, but you probably won't find many classes on PCB design economics. Not to worry. This month, our experts delve into the economics of PCB design: the total cost of each design, a variety of methods for controlling costs at each stage, metrics that help designers and engineers track costs, when it's time to outsource a design, and much more.

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.

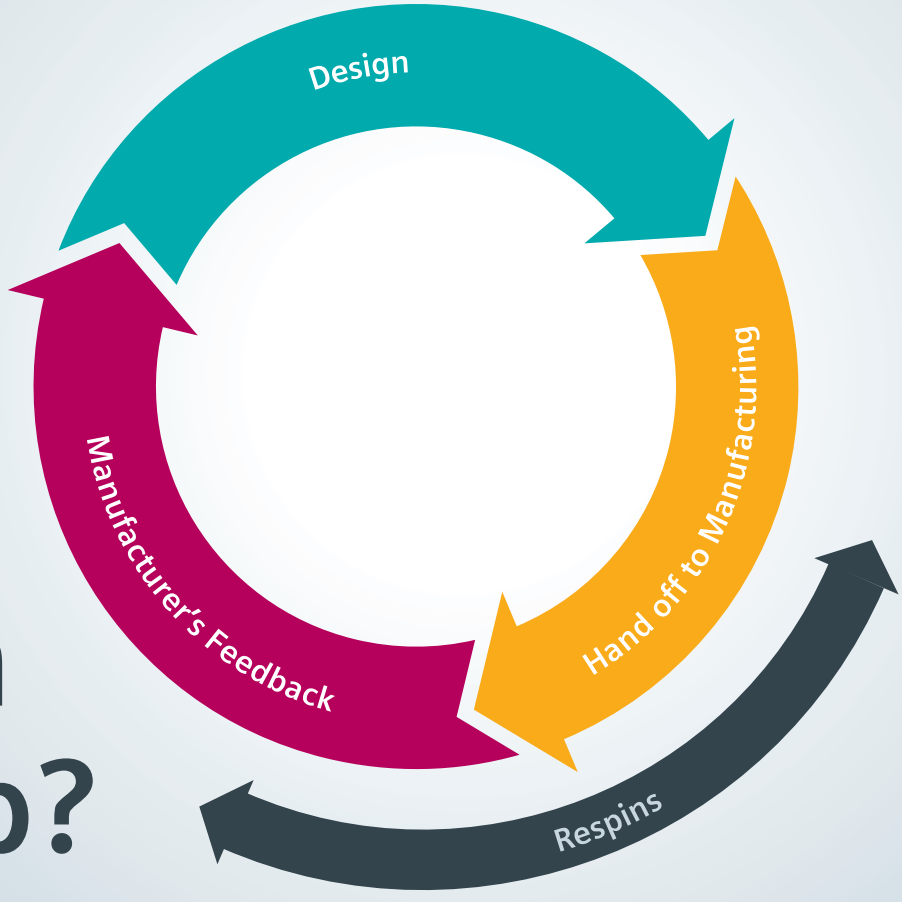
VIEW OUR CAPABILITIES

EAGLE electronics Inc.

www.eagle-elec.com

DESIGN007 MAGAZINE

SIEMENS
Ingenuity for life



Stuck in the loop?

First Time vs. **Next** Time

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

Design → **Analysis with Manufacturer Specifications** → **Hand off to Manufacturing**

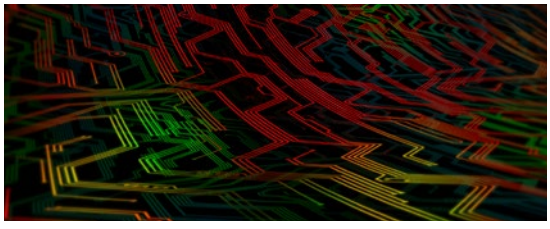
Mentor®
A Siemens Business

Visit website

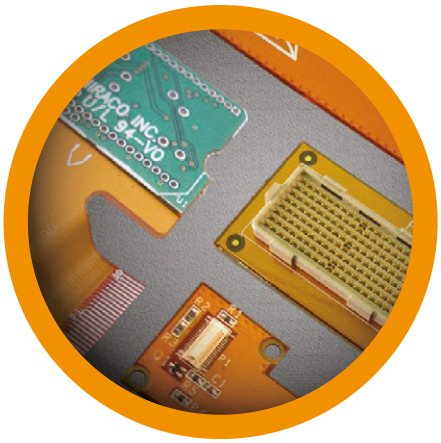
FLEX007

Flex Economics

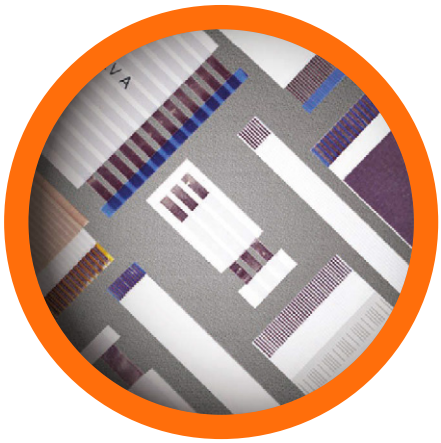
The design economics related to creating flexible and rigid-flex circuits are similar to that of their rigid board counterparts, but the 3D nature of flex can lead to a variety of potential hurdles on the way to cost-aware design. This month, our expert contributors examine the economics of flexible circuit design from a variety of industry viewpoints.



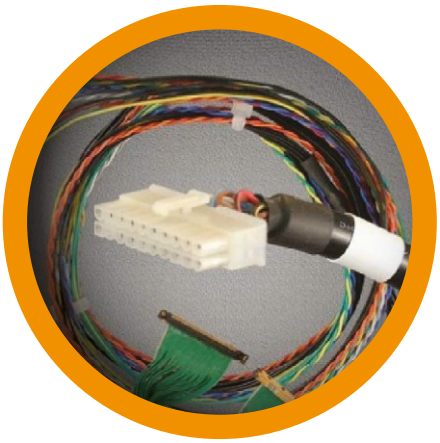
30+ YEARS EXPERIENCE. MORE THAN 30 PATENTS. JUST ONE GOAL.



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

The Economics of Design

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

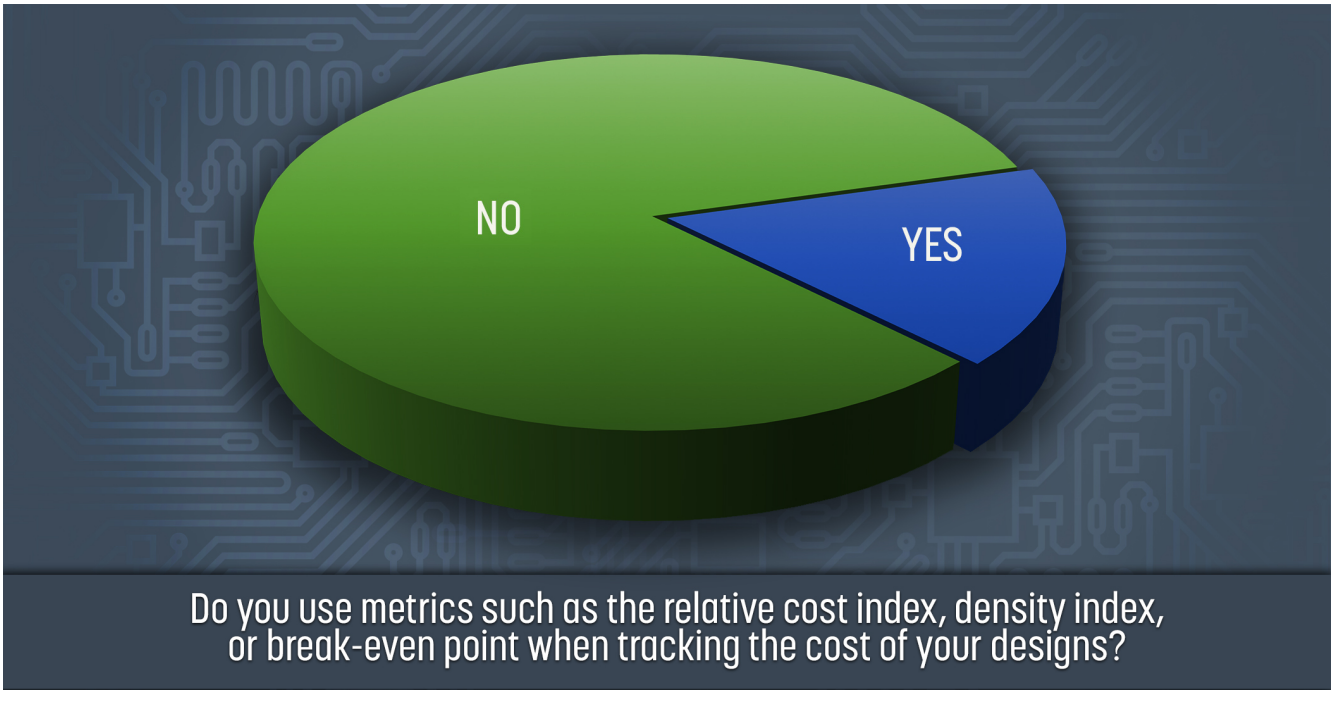
Most colleges teach an economics curriculum. We're not exactly professors, but this month, we're going to whip out our calculators and look into the economics of PCB design.

We first came up with the idea for a design economics issue during our research for the March 2020 issue, which focused on design for profitability (DFP). Everyone can agree that profit is a good thing; however, the main disagreement among PCB designers and design engineers seems to center around the best design practices for more profitable designs.

Along the way, we heard a surprising statistic: Some project managers believe that up to 80% of the cost of the PCB is determined in the design cycle, so I wanted to know a few things. How do designers and design engineers manage and track costs? How do they figure out what to bid on a design job?

In a recent survey, we asked designers if they utilized metrics when tracking the cost of their designs, and 83% said they didn't. Here are a few of the respondents' comments regarding design costs, edited lightly for clarity:

- We currently track BOM cost, part size, and pin count. Total part area vs. available board area gives us a "difficulty" metric that we use to create an estimated cost based on pin count and BOM.
- PCB cost is a small part of the overall product cost. We track this at the product level, not the PCB level.
- We think of it more from a design for manufacturability perspective. We assume (maybe incorrectly?) that if it is readily producible at high reliability and good yields, it will maximize our profit.
- I am a circuit board layout specialist. Others take the lead on a project's cost-related concerns.
- Cost reduction is always on our mind.



We discovered that there are as many different opinions about design economics as there are designers; there are dozens of methods for tracking cost throughout the PCB design cycle. Some designers begin considering cost the second they begin working on the design, and other designers don't track cost at all. Some designers said they wait until they had a working product and try to shave dollars off when specifying components.

What process does your company follow for tracking design costs? Do you have a process?

This month, our experts delve into the economics of PCB design. We look at the total cost of each design, a variety of methods for controlling costs at each stage, some useful metrics that help designers and engineers track design costs, when it's time to outsource a design, and much more.

Chris Young of The Goebel Company explains how he tracks design costs every step of the way, from the schematic through final assembly. Kelly Dack, CID+, discusses why he looks at design economics as a matter of conflict management, and why each decision to manage costs may create DFM or DFA problems later. Rick Hartley shares a story where design costs spiraled out of control, leading to PCBs that cost over $200 apiece. Nick Barbin of Optimum Design Associates explains how design economics ties in with solid DFM skills and why communication between designers and fabricators is paramount for cost-aware design. MK Hicks of the In-FRONT Agency explains why designers can't afford not to adapt to changes in the industry. And Joe Fjelstad posits that cost-sensitive PCB designers should really be looking at design with manufacturing instead of design for manufacturing.

I hope all of you are doing well under these COVID-19 restrictions. Many of you work for companies that are considered essential businesses, and we'll continue to bring you the latest updates from your vendors, suppliers, and partners. Stay healthy, and we'll get through this. I think we're going to come out of this period stronger than ever. **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.

Smart Eye Announces 24 Design Wins With Four Different OEMs

Smart Eye announced 24 new design wins with four different OEMs in one fell swoop. Of the four OEMs, two are new customers, one is an American high-volume car maker, one is a European premium manufacturer, and two are existing European premium customers. The estimated combined life-time value of the four contracts is at least 500 million kronors ($49 million).

Through its leading position for DMS in the automotive industry, Smart Eye has already secured its sales for the coming years. This provides a solid base for the delivery of DMS in the short and medium term.

"This is a solid foundation for the company, and we are determined to continue to lead the automotive DMS industry into the 2020s, as it's becoming as common as seatbelts and airbags," said Martin Krantz, founder and CEO of Smart Eye.

As a precautionary measure with the current market volatility due to the coronavirus, Smart Eye has decided to lower its operating costs in order to conserve cash. The cost reduction program enters into effect immediately. Special emphasis is placed on ensuring that the numerous delivery commitments to the global automotive industry are met. The company retains readiness to quickly scale up if coronavirus-related uncertainties improve. (Source: Smart Eye AB)


Chris Young

A Design Economics Primer

Feature Interview by the I-Connect007 Editorial Team

When you start a new design, do you begin tracking costs right away, or do you wait until you have a functioning product before you start looking at the dollars and cents? Chris Young begins cost-aware design before the design cycle has even begun. Andy Shaughnessy and Nolan Johnson recently interviewed Chris, an engineer with The Goebel Company and founder of Young Engineering Services, and asked him to explain his approach to design economics.

Andy Shaughnessy: Tell us about your position and responsibilities, and we'll go from there.

Chris Young: At The Goebel Company, I'm the chief hardware engineer. My primary responsibilities are product architecture, design, and development, which starts from a customer need and progresses into delivery and sustainability.

Shaughnessy: You have a pretty wide view of things.

Young: I've been lucky to be exposed to quite a bit of stuff. I've worked on product development from the cradle to the grave in the avionics industry, semiconductor and semiconductor test, and developing test systems for space and medical devices, ranging from airborne surveillance systems to electromechanical medical devices. I've had the pleasure of working with many different people in various industries.

Shaughnessy: We have been talking about design for profitability for a couple of years, and we started looking into the whole issue of design economics. In your position, how important is the cost of what you're doing?

Young: Along with the chief technologist, we are primarily responsible for the profitability of the company. The products that I design are directly related to revenue and profit.

Shaughnessy: You're looking at the cost from the very beginning of the design cycle?

Young: You must. In my career, I have developed a systems view of what's happening,

Integrated Tools to Process PCB Designs into Physical PCBs

Visualize

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

Verify

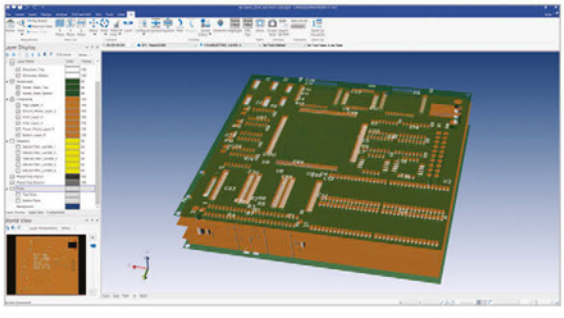
Ensure that manufacturing data is accurate for PCB construciton.

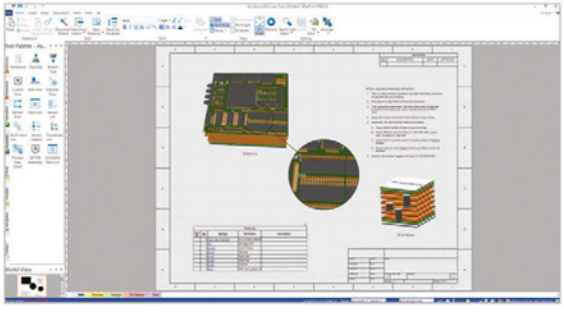
Panelize

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

which has shaped how I approach the issue of costs and profitability. It also depends on the environment that you're in. I always ask myself basic questions, such as, "What are we trying to do? When do we need to do it? How are we going to do it? Which market are we addressing? What do we think our budget is?"

In my career, I have developed a systems view of what's happening, which has shaped how I approach the issue of costs and profitability.

I say "think" because the budget is usually guesswork at the beginning. I think about what it would cost to do something based on metrics and previous design experience. Typically, I take into account planning, design, test, manufacturing, and sustaining costs. I add up the time and materials and then push it into some type of overall estimated budget. As the project/program progresses, the budget is adjusted and/or addressed accordingly.

Shaughnessy: Can you tell us how you approach it from a PCB design perspective as well as an overall project perspective?

Young: Sure. When I'm thinking of board design, the things that initially come to my mind are as follows. What are we trying to do? Is it a test fixture board? Is it a product or an instrument of some sort? Is it something else that goes into a box? Is it a computer? Breaking it down further, what type of interfaces do I have? Do I need analog, digital, power, or RF I/O? What are the number of interfaces on the board? What type of power rails are available from the outside world? What type of connectors do I need to be concerned about?

As I dig deeper, I consider the type of signals I have to work with and what the characteristic impedance of the traces need to be. I create an appropriate board stackup to accommodate these signals and select vias accordingly. Also, what type of finish do I need on the board? What are the copper thicknesses? Can I use a standard laminate, or do I have to use some hybrid laminate or an embedded capacitive substrate? What mechanical items do I need to address, such as specific connector locations, tooling/mounting holes, board outlines, and any special requirements? Do I use the components I have in my library or make new ones? I usually spend a lot of time working on my library of components.

Nolan Johnson: You mentioned you do your own parts in your libraries rather than using any outside services.

Young: Actually, I have a librarian who is responsible for creating the parts, and I'm responsible for reviewing them and making sure they're correct.

Johnson: On average, how much of your overall design hours are devoted to library management?

Young: It depends. I also have my own company and am in the process of integrating with The Goebel Company. I've been working on my current master library for about 12 years. Early on, I spent about $300,000 on setting up my library. Now, approximately 20% of my budget is dedicated to library development. I try to do it upfront. I'm always looking at vendors' websites to see what they have for new components and packaging technology. I have my librarian on an annual retainer, which gives me on-demand library services.

Part of my business model is that I'm asked to do things very quickly. I might do a board that's 5" x 7" and have 100 parts. I can usually do something like that in under 30 hours. Sometimes, I'm lucky, and all the components I need are in my library.

Shaughnessy: That's a good person to have on retainer. Most designers I know hate to spend too much time in the library.

Chris Young crewing a catamaran in St. Martin.

Young: As I need stuff, he does it. I've worked with the same guy now for six years, so he knows how I prefer symbols and footprints to be designed. My philosophy is the front-end work pays off in terms of board design. When I first started, I did not care about the library. Then, an older gentleman, who I respect quite a bit, said, "You need to look at your libraries and what's going on there. We have already found a couple of mistakes on this board that you're working on, but we caught it before it was done and released."

I have a formal process for how I request components for my library. I have broken it down into several stages: parts request, part creation, library delivery, and verification. I'll issue a parts request, which includes a data package containing the component list and relevant datasheets. My component list is in a spreadsheet format that includes the part number, manufacturer's name, description, type of package, type of part, datasheet filename, symbol/footprint notes, and any special design notes. The librarian creates the component to the relevant industry/design specifications and delivers the component to me in a test library. I verify the component accuracy and then issue it to my verified library.

I've seen a lot of times where people will draw a schematic, and then they get to the end and just start thinking about the cost. Then, you have to talk to your manager and say, "You're going to have a loss on this project." It's important to scope things out from the beginning to understand if it is worth it.

Johnson: Back to that old adage, "Fast, cheap, or good? Pick two."

Shaughnessy: Do you use a specific software tool for tracking costs, or do you use a spreadsheet?

Young: It depends on what's needed. The Goebel Company's business manager monitors the financing and uses the financial software to keep track of the budgets, and we bill costs to the budget. As an engineer, I typically use a spreadsheet. I have had to use detailed project/financial management software in the past, in more enterprise-level companies. It's a good rule of thumb to track costs, time, and materials. Doing it on a spreadsheet is significantly better than not doing it at all.

Shaughnessy: Where do you think most designers and EEs make most of their mistakes as far as leaving money on the table?

Young: A lot of people will look at their favorite part of the design and start there. Some of the EEs that I've seen will start drawing schematics. Some PCB designers place components without a perspective on what needs to happen. It's usually some preset idea from past experience or how they like to work.

Shaughnessy: I think it's partly tribal knowledge; that's how they learn it from the person before them. Some designers learned to sprin-

kle hundreds of decaps on a design, whether it needs them or not.

Young: I've seen people do that. I am typically aggressive with decoupling caps because most of my designs end up being in environments where the idea of an interference or signal integrity issue will be very upsetting to the customer. I don't have an excuse because many of my customers require that my designs do not interfere with other equipment. My decision of when and how to use decoupling capacitors is requirements-driven. It may sound cruel, but often, these design habits are a lack of discipline in the process.

My decision of when and how to use decoupling capacitors is requirements-driven.

Shaughnessy: Since you look at the whole process, and you're talking about design through assembly, where are some of the other spots outside of PCB design that you think designers can get more profitability out of the board?

Young: Let me walk you through what I would typically do for a project. First, I ask, "What are we trying to do? What are the scope and purpose? Is this commercial, industrial, avionics, military, or government?" Those are five industries, large entity industries that have different economic factors that affect them. Then, I think, "What's the purpose? Are we trying to design something new or work on product development? Are we working on sustaining, or are we trying to redesign, fix something in the field, or make something more profitable? Are we doing something for test or research?"

A lot of times, research gets left off. Some companies still do it, but I've seen companies blow their entire budget on research and never get out of the door with products. Once you get those factors put into place, then you consider your timeline. When do I need to do this? Do I need this now, or do I have time? Is it one week, one month, or one year? The design process that's going to be followed will sometimes be defined by the timeline. Is it informal or requirements driven? This means are you going to put something together and hope for the best or follow a defined process.

I've developed projects in the avionics industry, and some people in the commercial realm might think that because you have a year and a half to two years that you have all the time in the world. But the reality is that it can be an extremely tight schedule. All the in-process testing and the certification testing that needs to take place can cost as much or more than the design effort. People forget this sometimes, and that's how they blow the budget. As an example, they get to the point where they have a $10 million budget that ignores/underestimates the testing involved in the project. In the end, they realize, "We need $10 million additional worth of test equipment and development." I've observed that happen on that magnitude.

Then, they talk about budget overruns where the project is behind. It was supposed to take a year and a half, but it took three years. What happened? They ignored part of what needs to happen and did not talk about design for test and design for manufacturability. You need to design a board so that it's manufacturable. Attention to industry standards is important. In some industries, there might be standards around design for test, but I think it still needs more work to be solidified into more of a standards approach.

Another point to consider is paying a premium for expediting PCB fabrication and board assembly. A quantity of 20 boards that cost $50 a piece normally for a two-week turn could turn into $150 a board with a 2–3-day turn. That puts you in the area of $3,000 versus $1,000 for PCBs. The assembly of a board with 100 parts (20 ICs, 80 passives) may normally cost around $700–800 with standard assembly times (3–4 weeks) could turn into $3,000 for 2–3-day rush. These costs are real because you're paying for overtime, getting ahead in

Candor
Circuit Boards

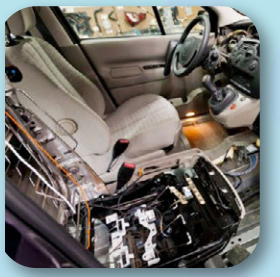
Innovative solutions for complex circuit board manufacturing

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

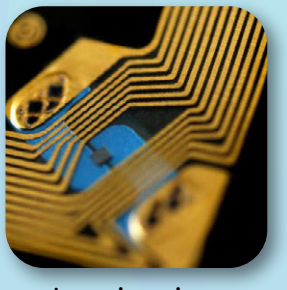
RF/Microwave PCBs

Power Electronics

High-Speed Data



Automotive Electronics



Lamination

Startups

Our technology produces what our customers dream!

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

the assembly queue, expedited shipping, etc. Expediting fabrication and assembly can easily triple your manufacturing costs.

Shaughnessy: It sounds like you look at the cost of each step in the process. This is in contrast with what some of our readers said in a survey about costs. Quite a few designers at OEMs said, "We're at a giant company, and it's almost impossible for me to keep track of the total cost of a design because it happens in so many different offices."

Young: It is possible to do that, but it requires very good planning skills. Also, you have to look at the personality of the team players. Teamwork is important. I'm not talking about groupthink; I'm talking about people who can coordinate and get along with each other. I've worked on a lot of projects in the past, and I've had to hire, fire, and lead people who have been great to work with. I've also failed on projects with a team I considered to be the top of the class, but we didn't communicate well with each other.

Shaughnessy: Who do you think is responsible for the design and keeping track of the cost? Who do you think is the main person responsible for that?

At the end of the day, talking from a corporate perspective, you need good program planning and management.

Young: At the end of the day, talking from a corporate perspective, you need good program planning and management. If your program manager doesn't know what the cost is or is overcome by events, there may be a need for some type of involvement from the executive level. From what I have observed, people are afraid to slow down, and sometimes that is what it takes to get back on track. It is important to accurately assess where you are in a project and where you need to be.

Shaughnessy: What advice would you give to designers who are starting to have to track the dollars and cents in their designs?

Young: Talk to other people in the process. Ask about their problems and ideas. Ask them what they're trying to do, and I'm not talking about in a meeting; I mean to communicate directly one on one. As an example, have the design engineer talk to the layout and manufacturing engineers about issues they see. Talk to the people in shipping/receiving; there may be something in their process/environment that needs design consideration. The problem is that so many engineers think, "I design this circuit, and that's all I do." No, you're trying to design this circuit in a system that other people are working with as well.

Shaughnessy: They look at it as if it's the final product.

Young: Exactly. It sounds corny, but it's worth learning new things. I'm always reading about something, and it's not always engineering. My wife and I have a great library. I have books on microelectronics, power electronics, signals and systems, digital communications, calculus, Active X and OLE programming, FPGA simulation, UNIX systems, mathematical proofs, microwave filters, and engineering. Then, balance that with books on how to write better requirements, talk to people, and understand where they're coming from. Being well-read and well-educated helps a lot, but you have to balance that with talking to people.

Shaughnessy: This has been really helpful. I appreciate your insight. Thanks, Chris.

Young: Thank you, Andy. I enjoyed it. **DESIGN007**

APEX EXPO IPC 2021

Meetings & Courses:
January 23 – 28

Conference & Exhibitions:
January 26 – 28

SAN DIEGO CONVENTION CENTER | CA

TECHNICALLY SPEAKING:

IT'S THE PLACE TO be

Come and collaborate at the one electronics industry event everyone will be talking about.

See you in San Diego at IPC APEX EXPO 2021!

Thank you to all who have contributed to creating and supporting an excellent IPC APEX EXPO 2020 experience.

IPCAPEXEXPO2021.IPC.ORG | #IPCAPEXEXPO

IPC

Predicting and Measuring Impedance

Beyond Design

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

To control the impedance of high-speed signal interconnects, one first needs to predict the impedance of a specific multilayer stackup configuration. A precision field solver is arguably the most accurate way to calculate the single-ended, edge-coupled, and broadside-coupled differential impedance. Once the board is fabricated, the transmission lines need to be physically measured to determine the actual impedance to qualify the board (Figure 1).

The most common method for measuring PCB trace impedance is to use a time-domain reflectometer (TDR). This measures the impedance in the time domain. However, a far more accurate method is to use a vector network analyzer (VNA), which operates in the frequency domain. The VNA sweeps through a range of frequencies determining the impedance and signal losses at particular frequencies.

However, VNAs are expensive, delicate instruments and not as robust as a TDR (Figure 2), which allows unskilled personnel to operate

Figure 2: Zmetrix impedance test systems.

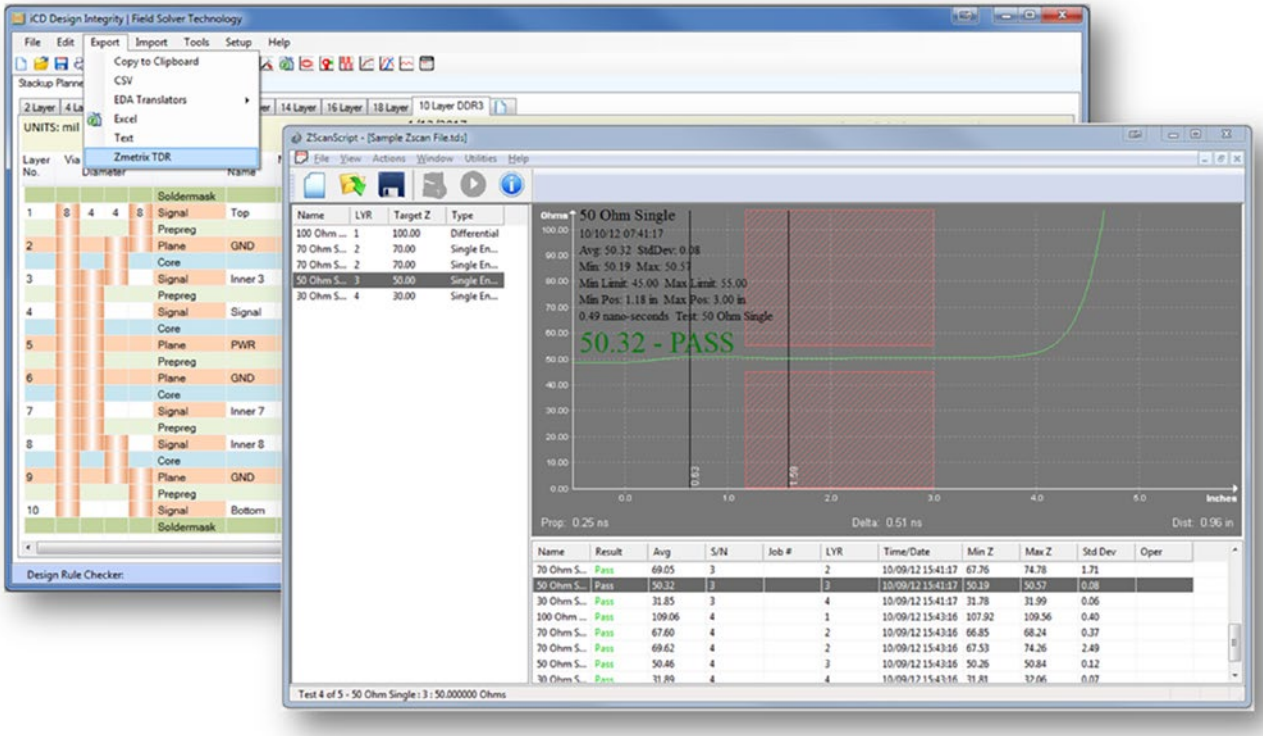
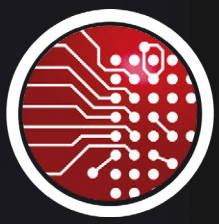


Figure 1: The iCD Stackup Planner coupled to the Zmetrix TDR and Zscan software.



APCT

Passion I Commitment I Trust

Where Advanced Technology, Highest Quality & Fastest Delivery Are Standard

APCT Offers Printed Circuit Board Solutions from NPI to Mass Production

- Standard to Advanced HDI Technology
- Flex and Rigid Flex Technology
- Complete DFM Support
- Cycle Times Faster Than Industry Standards
- Defense & Aerospace Certifications
- Global Offshore Solutions

  Advantages Include:
  - *Ease of Use*
  - *Seamless Recipe Transfer*
  - *Inventory Management*
  - *Risk Mitigation*

Contact APCT Today 408.727.6442 | APCT.com

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

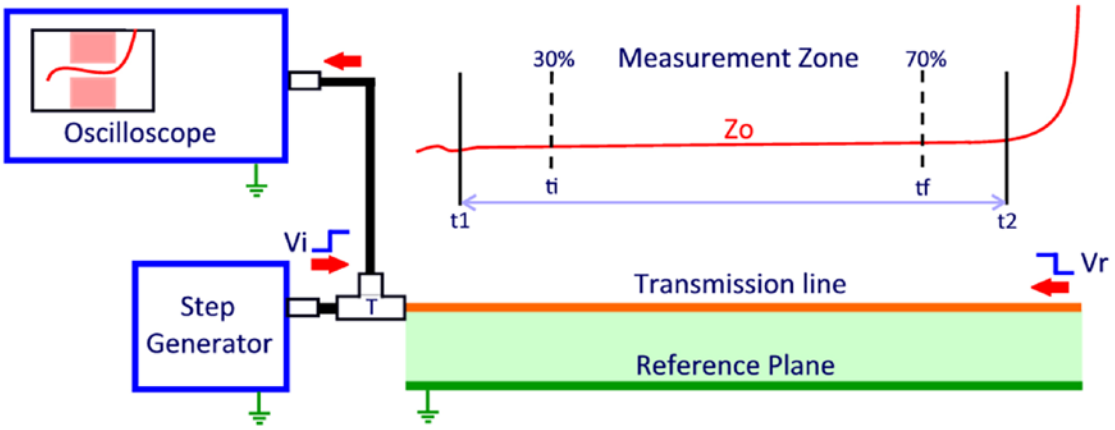


Figure 3: TDR measurement overview.

them in a factory environment yielding repeatable results. Hence, from a cost and practicality perspective, the TDR is the de facto standard impedance measurement instrument in the PCB fabrication industry.

A TDR applies a very fast pulse (< 100 ps) to an impedance test coupon via a controlled impedance cable and matching impedance probe (Figure 3). Whenever there is a change in impedance, part of the signal energy is reflected back to the TDR and is measured by the instrument. The magnitude of the reflected signal is related to the value of the discontinuity. Changes in the cross-sectional area of the trace, distance to the plane(s), and return path or proximity to other pads and traces will also cause a change in the impedance and cause a reflection. From this data, it is possible to graph the impedance and its variation over distance or time.

If a signal propagates from a region with impedance Z1, and enters a region with an impedance Z2, the incident waveform will reflect. The reflection coefficient—defined as the ratio of the reflected to the incident voltage—is related to the two impedances by the reflection coefficient :

$$\rho = \frac{Vreflected}{Vincident} = \frac{Z2 - Z1}{Z2 + Z1} \qquad \text{Equation 1}$$

By measuring the reflected signal and knowing the incident signal and the impedance of the source, the impedance of the transmission line can be extracted. When measured, in the time domain by a TDR, the incident waveform is a fast-rising step edge, and the impedance profile of the trace is measured as:

$$Zo = 50\Omega \frac{1 + \rho}{1 - \rho} \qquad \text{Equation 2}$$

When measured, in the frequency domain by a VNA, the incident waveform is a sine wave, and the reflected amplitude and phase is measured at each frequency value. The reflection coefficient, usually referred to as S-parameter (S_{11}), is related to the total, integrated overall impedance of the transmission line at each frequency by:

$$Zo = 50\Omega \frac{1 + S11}{1 - S11} \qquad \text{Equation 3}$$

The key attribute of a TDR to consider is the pulse rise time, as this determines the measured resolution. The faster the rising edge, the shorter the impedance discontinuity, which can be detected. If you are characterizing a connector or other very short type of interconnect, then a faster rise time will allow you to see anomalies that would be missed with a slower rise time. Most of the energy of the pulse will be in the first harmonic, and it is this frequency that is used to calculate the impedance. Fortunately, impedance does not vary much with frequency although insertion loss does.

Impedance test coupons are generally 150 mm long (see IPC standard IPC-2141A Design Guide for High-Speed Controlled Impedance

Circuit Boards), but by using a very fast rise time of 20 ps (20 GHz), as used in the Zmetrix ST808, traces can be tested down to 20 mm in length, allowing the operator to test actual on-board traces—not just test coupons. This is important as test coupons are normally placed outside the board outline on the panel edge, where the impedance is generally higher than those traces in the center due to an increase in resin and may not accurately represent board trace conditions.

Let's look at a couple of typical examples of impedance prediction and measurement.

1. 50-ohm Single-ended Impedance

The iCD Stackup Planner was used to model a 50-ohm, single-ended impedance for both microstrip (Layer 1) and stripline (Layer 3) on the part stackup in Figure 4. The substrate used Nouya NY2150 dielectric materials.

The data from the iCD Stackup Planner was exported to the Zmetrix Zscan software (.zmx format). The results of the Zmetrix ST600 TDR (75 ps rise time, 7 GHz bandwidth) were then correlated to the iCD Stackup Planner impedance (Figure 5). The measurements of the 50-ohm impedance test coupons passed for all signal layers.

2. 100-ohm Differential Impedance

The iCD Stackup Planner was used to model a 100-ohm differential impedance for microstrip Layers 1 and 6 on the stackup in Figure 6.

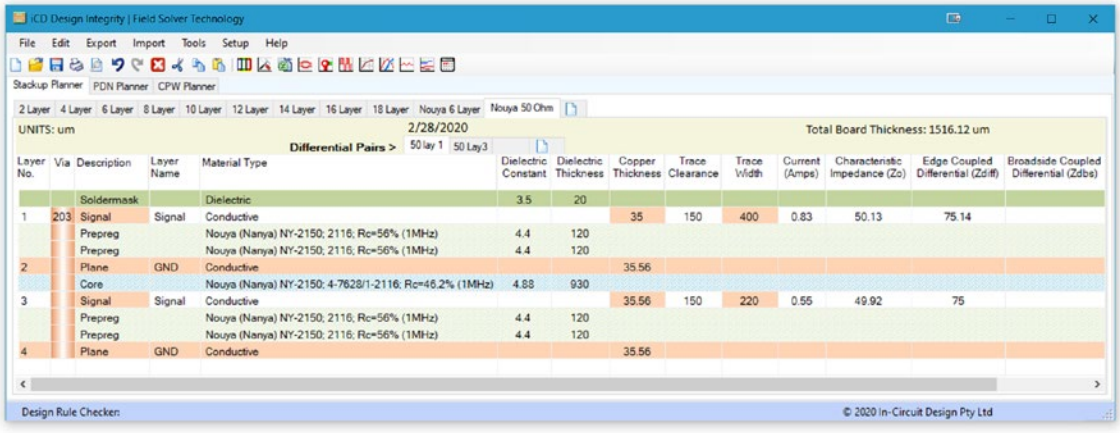


Figure 4: 50-ohm microstrip and stripline impedance calculation (iCD Stackup Planner).

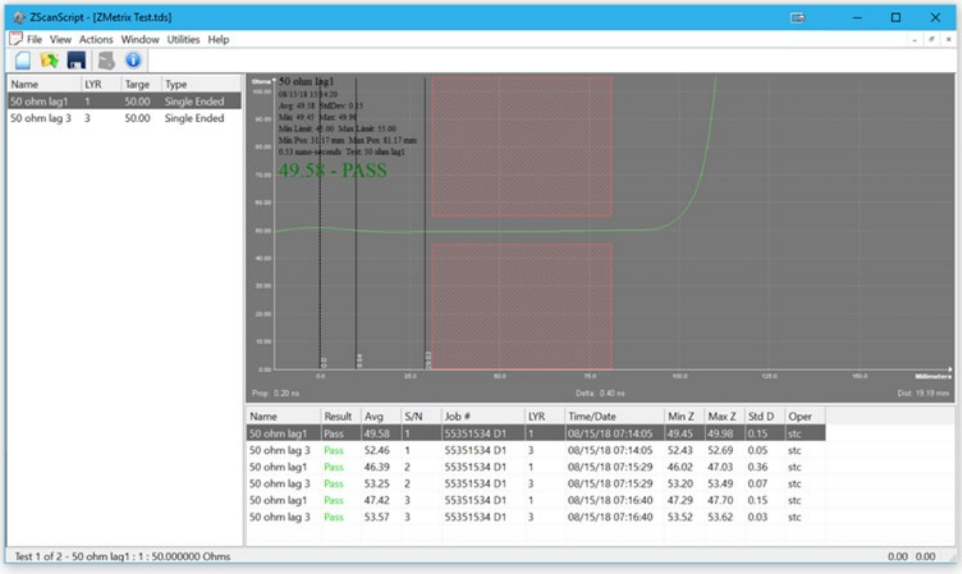


Figure 5: Pass/fail results for the 50-ohm traces (Zmetrix TDR and Zscan software).

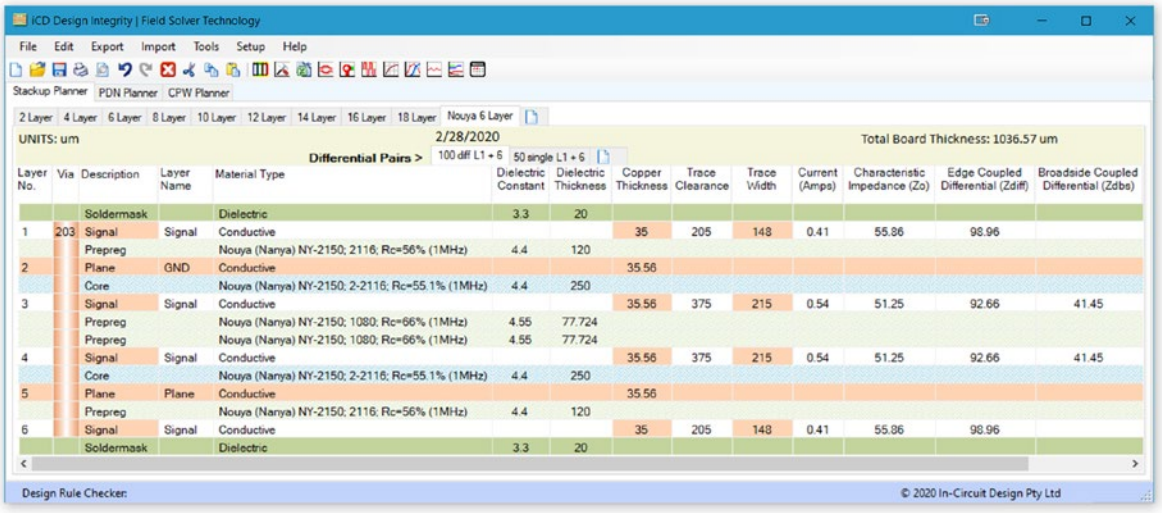


Figure 6: 100-ohm microstrip differential impedance calculation (iCD Stackup Planner).

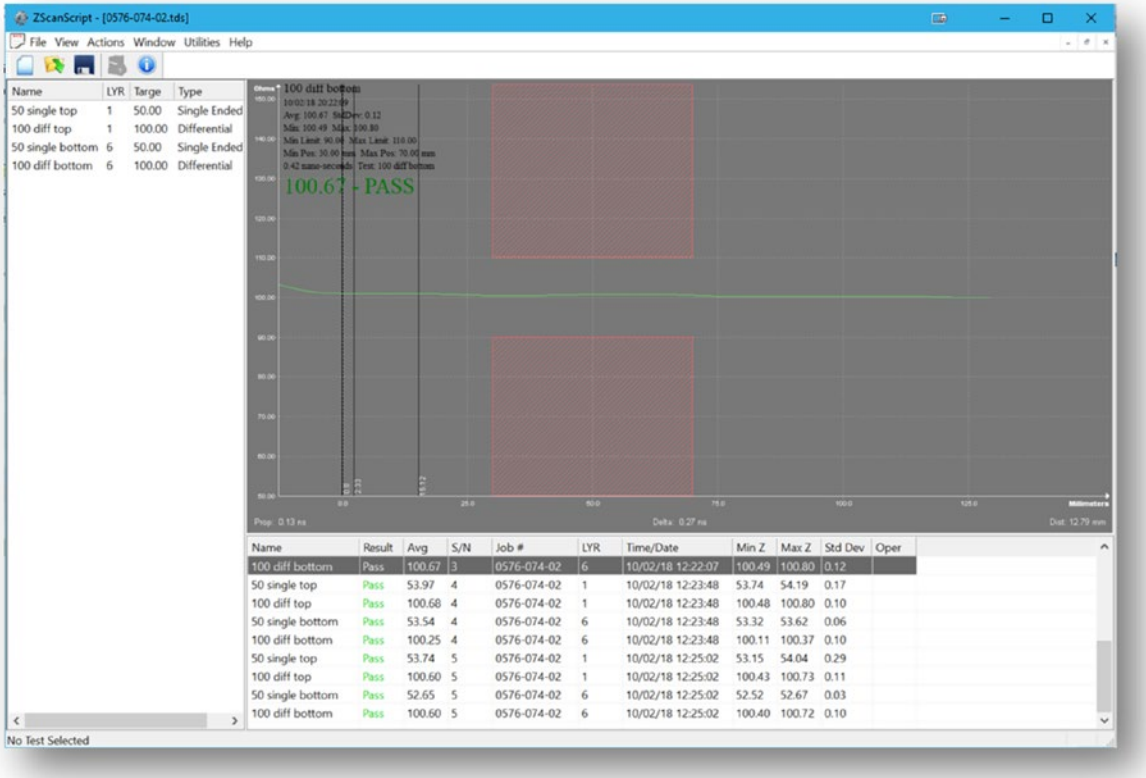


Figure 7: Pass/fail results for the 100-ohm differential traces (Zmetrix TDR and Zscan software).

The TDR measurements of the 100-ohm differential impedance test coupons passed for both top and bottom microstrip signal layers (Figure 7).

Comparing the modeled impedance to the actual measured impedance, in this case, we get a 1.6-ohm variation for microstrip traces and a 2.5-ohm variation for stripline traces. The Zmetrix TDR's accuracy is within 1%, across the entire impedance range from 10 to 200 ohms. Keep in mind that there is also a multitude of fabrication variables to account for—hence a tolerance of ±10%, which is the IPC standard for controlled impedance measurement. And fabrication variables will be different for every process line. This is where an experienced fabricator can improve yields. Some fabrication shops offer a tolerance of

±5%, but that comes at a premium due to reduced yields.

Also, controlled impedance interconnects are intended to achieve target impedance when the transmission lines are loaded with ICs and powered. These conditions do not prevail on a bare board, so these measurements are likely to read higher than that predicted.

Fortunately, providing the impedance is in the ballpark, the most important factor from a PCB design perspective is the constant value of impedance along the transmission line—no discontinuities or reflections. However, impedance becomes more critical as frequency increases and wavelengths become close to trace lengths.

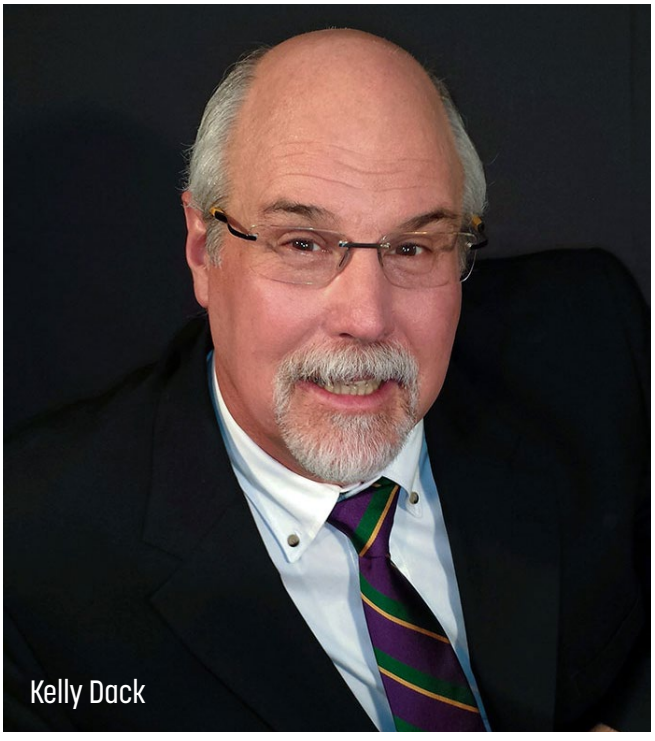
Key Points

- A TDR measures the impedance in the time domain. However, a far more accurate method is to use a VNA, which operates in the frequency domain.
- TDRs are the de facto standard, as VNAs are expensive, delicate instruments and not as robust.
- A TDR applies a very fast pulse to an impedance test coupon via a controlled impedance cable and matching impedance probe and measures the reflected signal and graphs the impedance.
- The key attribute of a TDR to consider is the pulse rise time, as this determines the measured resolution.
- Impedance test coupons are generally 150 mm long, but with a 20-ps pulse, traces can be tested down to 20 mm in length, allowing the operator to test actual on-board traces—not just test coupons.
- Test coupons are placed outside the board outline, which can make the impedance higher due to increased resin on the panel edge.
- There is also a multitude of fabrication variables to account for—hence a tolerance of ±10%, which is the IPC standard for controlled impedance measurement. **DESIGN007**

Further Reading

- B. Olney, "Beyond Design: Controlled Impedance Design," The PCB Design Magazine, May 2015.
- A. Bahl, "How TDR Impedance Measurements Work," November 28, 2017.
- Tektronix, "TDR Impedance Measurements: A Foundation for Signal Integrity," 2008.
- E. Bogatin, "TDR and VNA Techniques for PCB Characterization," IPC APEX EXPO 2002.

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


Kelly Dack

Design Economics: The Buck Stops Where?

Feature Interview by Andy Shaughnessy

I recently spoke with Kelly Dack—CID, CID+, and a PCB designer and instructor who has worked in the design and manufacturing segments over the years. Thanks to his background, Kelly provides an intriguing viewpoint on cost-aware design and the philosophy of design economics in general.

Andy Shaughnessy: Kelly, you've written about cost and profit in PCB design. Give us a basic overview of your thoughts on design economics.

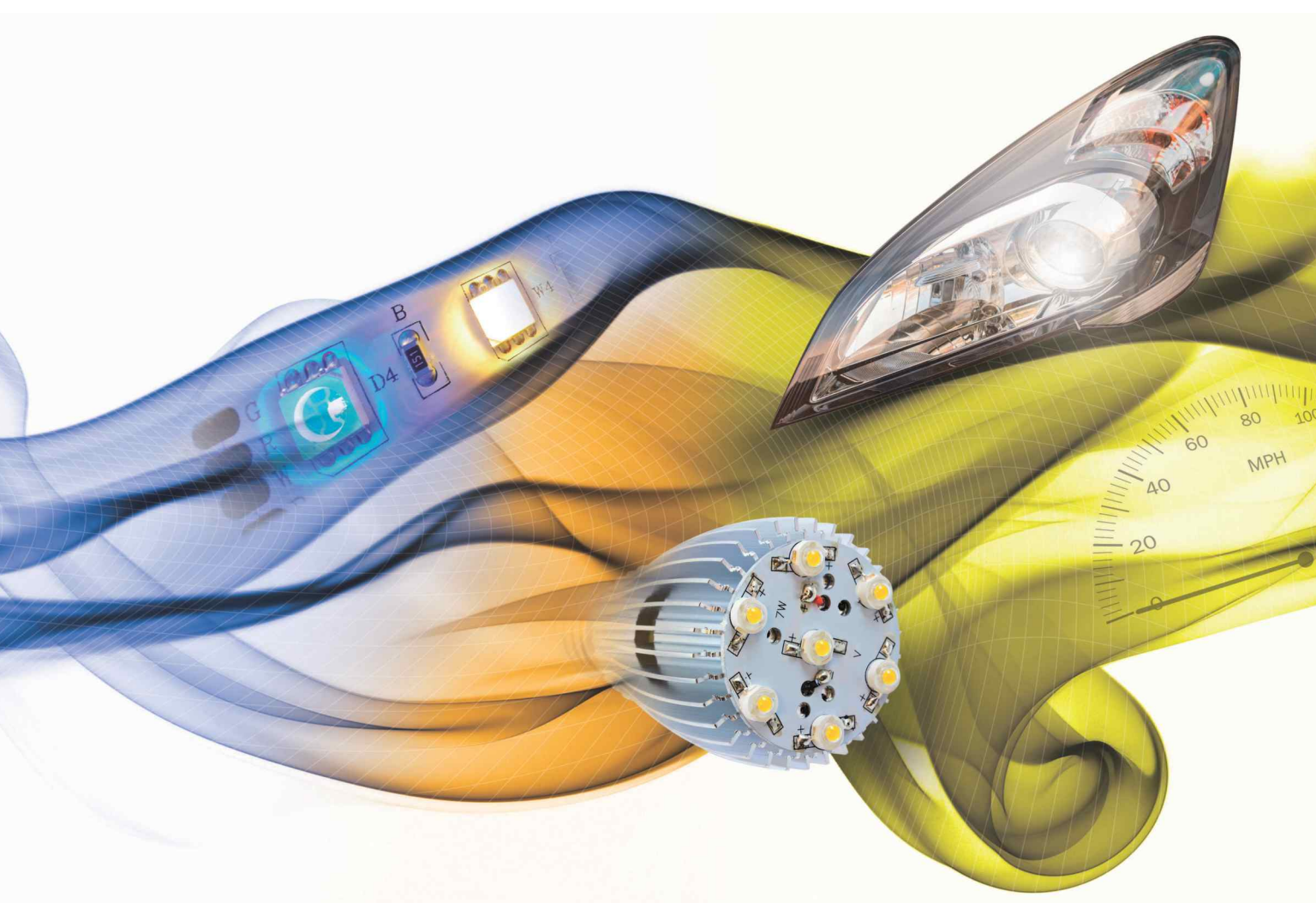
Kelly Dack: Considering cost in design is tricky because there are so many facets of cost. First, it's expressed in product development as "time-to-market." How much is a company losing by not having this product? It depends on priorities. I've always said that the first priority is meeting a performance specification and then meeting a cost specification. But in this tricky industry, if you miss either constraint—cost or performance—you may not have a product.

And rarely do I see a cost constraint on the front end of a PCB design project. In fact, I just had a conversation with an engineer regarding using a new chip and a new sensor to drive down the cost of a product, which has been on the market for years. Here's where it gets tricky: You may not necessarily save money by using a less expensive component. Component packages can have many different form factors, and each may require different design techniques, which could offset any cost savings. A proposed new chip may require a more expensive design process, manufacturing process, materials requirement, or all three. It's never simple. It really requires some good analysis by all of the project stakeholders.

Shaughnessy: Before you begin layout, what are some criteria that you keep in mind for cost-aware design?

Dack: The first thing is to define products by classes. What type of class does the product belong to? Is it a disposable product? Is it a product that has a dedicated use, or is it a product that needs high reliability? These are all different types of product classifications that will play into how much the product should cost.

For instance, a disposable/throwaway calculator that you buy at a dollar store would be presumed to use less costly components, high-



A leading light in engineering improved LED performance

We don't just strive to exceed expectations, we innovate to make the best of industry, perform better.

The LED market is the perfect example of our capabilities in action. LEDs are considerably more efficient than traditional lighting forms, but they do still produce some heat, which without thermal management will have an adverse effect on both the efficiency and operational lifetime of the LED.

From bonding and non-bonding thermal interface materials, to thermally conductive resins and protective conformal coatings, our solutions offer the ultimate level of protection and heat dissipation for your LED application.

With an expansive product range and a strong emphasis on research and collaboration, we provide a complete electro-chemical solution to the world's leading LED manufacturers.

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

Scan the code to discover our full spectrum of 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

er volumes, and techniques to reduce the cost so that a profit would be realized by selling the product in volume. However, a spy satellite or a control mechanism in the cockpit of an airplane will most certainly be more costly due to its limited volume of production and it's higher reliability requirements. The components and parts used in the product have to undergo rigorous testing, which all add cost to the individual components, and that will add cost to the overall product. Then, there are the specifications and regulations that will add to the timeline and cost.

Shaughnessy: Do you start looking at costs at the schematic stage?

Dack: We can and should. A long time ago, when CAD databases began including attributes that could be populated within the schematic, somebody had the great idea to create an attribute for cost. In other words, a schematic symbol represented an off-the-shelf part. The part would correspond to a data sheet and a supplier, and the supplier's costs could be added as an attribute.

However, all one needed to do was try to purchase those parts. You'd find out that depending on the supplier, inventory, or lead times and production quantity, these parts often either were not available or the cost had changed significantly from the time of prototype. The parts would then have to be ruled out because they were perceived to be too costly. On the other hand, if the product were aimed toward volume production in the millions, the costs would go down and be more suitable.

Now, we're seeing schematics being linked directly to online component sources—such as DigiKey, Mouser, and Arrow—who will give up-to-date data, cost, and availability for parts throughout the development cycle. This is a lot more dynamic, and I think it is the go-to methodology for engineers now.

But my perception is that we start first and foremost with a BOM. Any EMS provider will require a BOM to initiate raw materials analysis. Time and process analysis will come after the BOM is presented. A BOM will contain all the electronic parts, mechanical hardware, processes, process materials, and mechanical hardware. The BOM will most likely be reviewed by a quoting team, which will break down all of the part classifications.

And then there's a manufacturing engineering group, which will analyze all of the parts, including the raw PCB to access all of the manufacturing processes that will be associated with the parts to calculate their manufacturing time, which is measured down to seconds per step. Any seconds they can shave off of the manufacturing process has the potential to result in huge cost savings in volume production.

Shaughnessy: A few weeks ago, you mentioned that cost-aware design techniques are often just a matter of conflict management. You said it was tough to squeeze profit out of a design without messing with time to market, DFM, DFA, etc.

Dack: It is conflict management or the convergence of all of these disciplines. Often, we are put in the position of robbing Peter to pay Paul, with Peter being the electronics engineer, and Paul being the manufacturing engineer. Here's an example that's so common it is almost cliché. Many times, an EE will require that SMT decoupling or bypass capacitors be placed as close to a through-hole pin as possible for reasons of performance. If the board design is two-sided, the EE will want those chip caps on the opposite side of the part, snuggled right up on the pin.

Here's the rub: Design for performance says that the capacitor must be placed as close to the pin as possible, but DFM says that that surface mount capacitor will interfere with the solderability of that pin if it's not placed a hundred thousandths away from the pin. The designer is put in the unique position of having to deal with those two opposing forces. It must be our specialty and expertise to present both viewpoints to the engineering stakeholders and bring them to a solution agreeable to each of them, respectively. A cost attribute must be placed upon each so that an agreeable cost can

be determined and described in terms of performance or manufacturability.

If not addressed in a DFM review on the front end, this type of scenario may be brought up as a cost reduction idea down the line. This can make the manufacturer look good and come across as a proactive stakeholder even though anything corrected down the line in production increases costs by a large factor. Design teams must do a proactive job of engaging and soliciting a review from the manufacturer, even during prototyping. If you change something early in the concept stage, there's no cost adder. If you correct it in prototype, it's a 2X cost adder; but if you have to correct that in production, it can be an 8X or 10X cost factor, for instance.

Shaughnessy: You're managing a series of trade-offs.

Dack: Yes. Another example is automated manufacturing. Is it better to have it hand-processed or processed by automation? Automation will save lots of costs if it's done right. However, if it's done wrong, it will cause more problems because it will have to be done by hand. And cost savings are established by conserving line time.

Shaughnessy: Do you typically know the total cost of a design? Can you look at it and say, "Here's how much that design costs?"

Dack: I think good business says you count the costs upfront. The cost must be counted before you start any project. A wise builder counts the cost. I can't imagine that a company would have a good business plan without at least estimating the costs for a build. And often, for EMS companies, that's provided as a quotation to the customer, and then the customer can use those numbers to compare with their own estimations to determine value.

When you move through the prototype phase and the initial checkout testing phases, product testing, consumer testing, agency testing, the assembly will eventually move on to an EMS provider to transfer it onto production. It is at that point we typically see product building materials loaded into the data management systems of the EMS provider who has far greater purchasing power. In addition to the single customer's engineering design, the EMS provider is building scores of other customers' products and buying parts at a rate that their customers' organizations cannot.

Now, the costing game takes on a different, more aggressive form of costing. To win the business, the EMS provider must not only demonstrate that they can supply the lowest cost, but also provide the highest quality; in other words, the ratio of cost to quality equals value. While sometimes the supplier with the lowest cost wins, some would argue that both customer and supplier win when purchasing value.

While sometimes the supplier with the lowest cost wins, some would argue that both customer and supplier win when purchasing value.

Shaughnessy: We did a survey recently about design economics, and some designers replied that profitability is primarily a production concept, but not their problem.

Dack: It's true. On the front end, it's all about performance and meeting the performance specification. I think it's the job of manufacturing, engineering, and management to realize a sellable product. "We made it work. Now, build it and get the cost down."

Shaughnessy: How about when the board goes from prototype to production?

Dack: Here's a story I tell my CID classes about the availability of materials and prototypes that meet performance with a total disregard

to cost, availability, and production. At a company I worked for, we had mechanical engineers come to us with a performance requirement—a board had to flex or bend into an arc to be installed. They knew that flex materials were expensive. They were considering cost, but they were trying to come up with a mechanical solution that involved normal FR-4 materials. They came to us with a requirement that a surface-mount board had to bend a certain amount to be installed, and its radius typically would exceed the amount that we like to with rigid-flex boards, which is by nothing, as it induces stress on the SMT components.

They were considering cost, but they were trying to come up with a mechanical solution that involved normal FR-4 materials.

We reluctantly took on the challenge. We sent the board out to our prototype supplier to be prototyped with very thin laminates. It came back to us in panel form. And this board was assembled with the LEDs, and then excised from the panel and bent to install into the machinery. We were looking at this and thinking, "You're going to crack the LEDs." But they installed the boards and plugged it in, and guess what? The board worked. All the LEDs came on.

We're scratching our heads, thinking, "Give it a few weeks. The stresses are going to transfer into the LEDs, and they're finally going to crack in this board, and it's going to fail." Weeks went by, and it seemed that everything worked well. How do they do that? We appeared to have mud on our face from a PCB design standpoint.

We released the design, and the board was released for volume production in China. The boards came back, and they looked the same. But weeks later, we started getting phone calls from the final test and inspection department. There were failures, and the lights weren't lighting up. There was a mushroom cloud over our facility!

What happened? I called our CAM guy at the board shop to find out what he did to provide the solution for us. And he said, "We used really 'juicy' prepregs on this design to solve your problem." That made sense because I could take this board and wiggle it. This FR-4 board would flop around. It was so juicy, meaning low glass content with lots of resin on it.

It was basically a custom material, custom-laminated board, using custom dielectrics that were not specified, and even if they had been, the offshore suppliers didn't have access to these materials. They built it out of normal FR-4 laminates, which are a lot more rigid, as we knew. The new, offshore-supplied boards had stress induced when they were bent into shape—a lot of stress. And it did pop a lot of the LEDs. This is what happens when the availability of the materials used during prototypes is ignored in production.

Shaughnessy: One hand doesn't know what the other is doing sometimes.

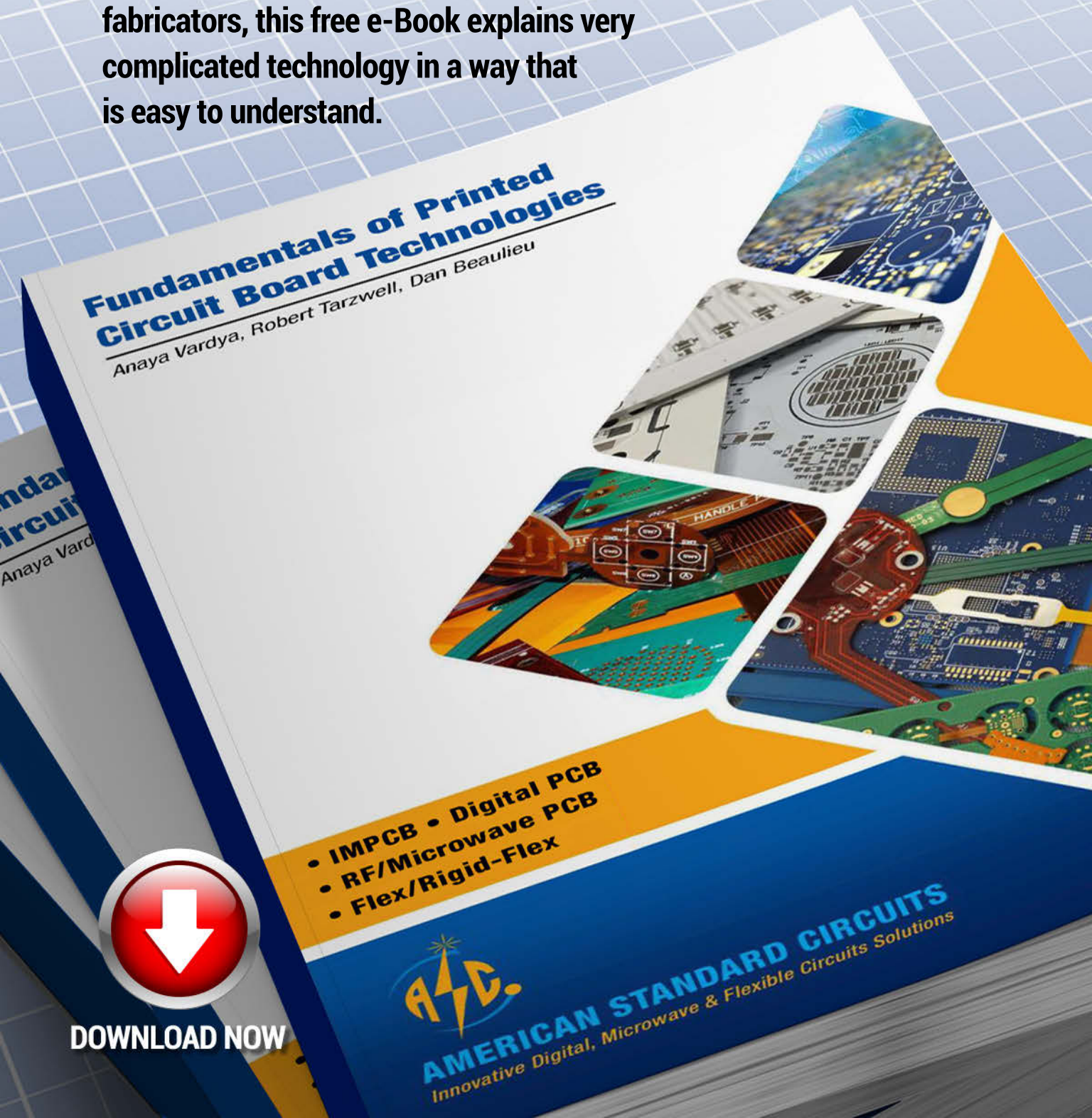
Dack: We use those types of metaphors quite often. We rob Peter to pay Paul when our left hand doesn't know what our right hand is doing. Exactly. Cost-aware design involves a lot more than proper layout techniques. It also means looking ahead and connecting with all of the engineering, manufacturing, and supply chain stakeholders to try to foresee and keep problems from happening at volume production. As your survey respondents said, most of the cost savings are realized in manufacturing. You're managing the design, and it's up to you to look out for these things before they can occur.

Shaughnessy: Good stuff. Thanks for your time, Kelly.

Dack: No problem. Thank you. DESIGN007

Everyone in the industry should get their hands on this book.

A "must-have" for PCB designers, OEMs and fabricators, this free e-Book explains very complicated technology in a way that is easy to understand.



DOWNLOAD NOW

Increased Focus on Health and Wellness Transforms the PCB Industry

Connect the Dots

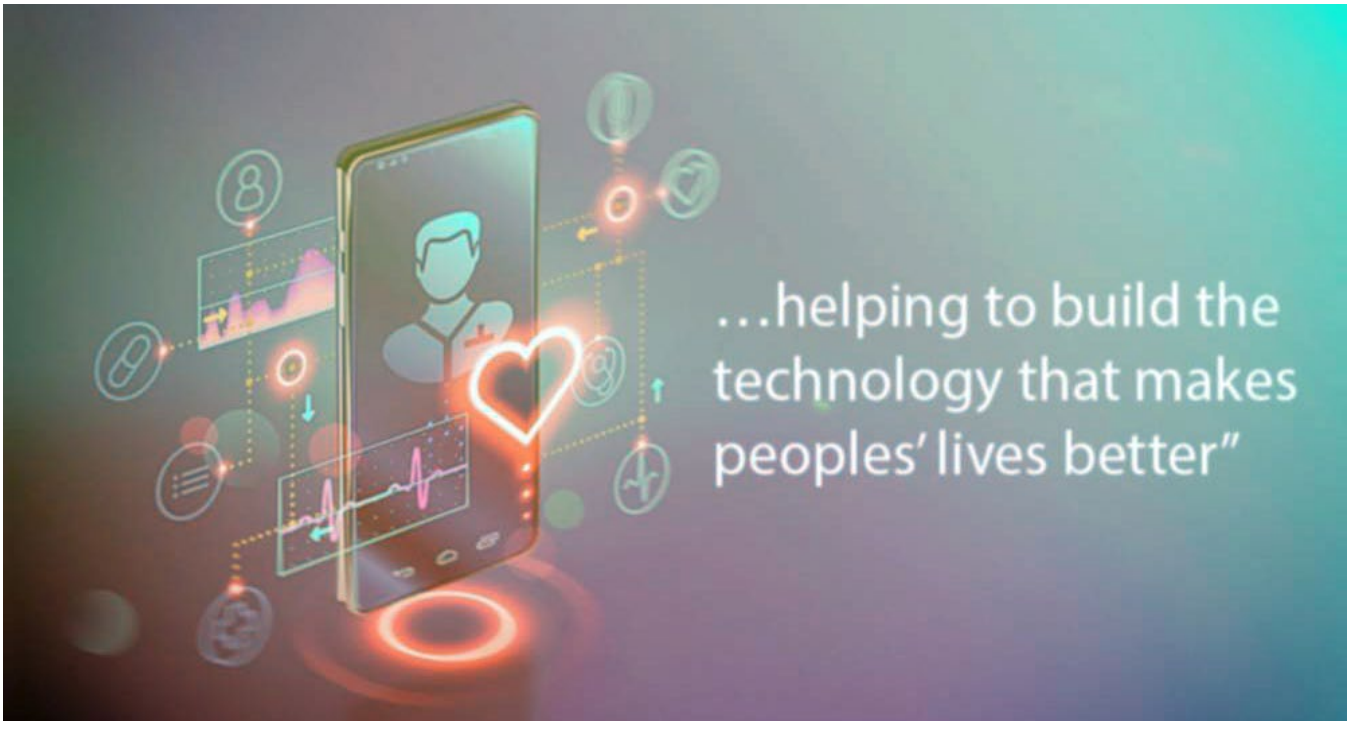
by Bob Tise and Matt Stevenson, SUNSTONE CIRCUITS

Can you imagine a time when people went jogging with nothing more than a self-winding wristwatch and a street map? How many of the gizmos in your doctor's office were there five years ago? What would your boss from 10 years ago say if you asked them to install a napping pod at the office?

Things have changed a lot. Our population is aging, living longer, and seeking to maintain a healthy quality of life in the process. In a job market that's been at full employment for years, employers are feeling the squeeze from rapidly evolving expectations surrounding workplace wellness and rising healthcare costs. These drivers are converging to create rapid change that is impacting the PCB industry in many ways.

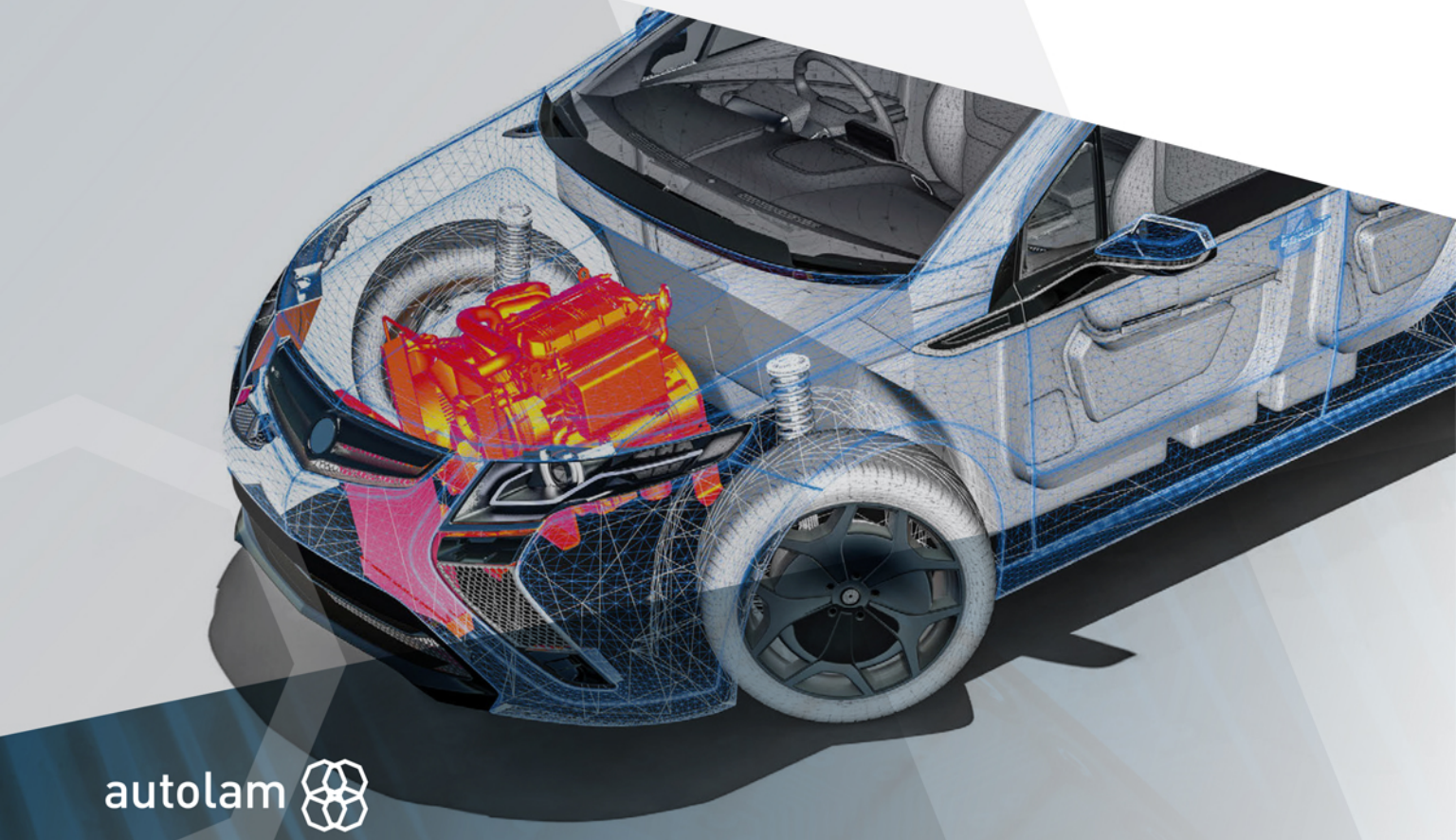
Our increased focus on health and wellness drives technology advancement for personal devices and those used in the delivery of healthcare. This trend also drives both PCB production innovation and a long-overdue update of the employer/employee relationship.

Wearable technology was barely a thing five years ago. When we go for a run now, we expect detailed information about where we went and how our body performed. This health and



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

wellness information must be reliable and easy to use during our workout. It also needs to be available to review and analyze later to see if we are reaching our goals.

From the safety lights on the bicycle you ride to GPS tracking, sophisticated fitness monitors, and small-rugged video cameras that record fitness adventures, electronic devices are capturing more and more information even as they become smaller and lighter, as well as more durable and affordable.

PCB manufacturers are responding to the demand for increased board complexity by upping our game. Thanks to new equipment and enhanced process controls, we now have the capability to miniaturize circuits with smaller footprints and reduced weight. Using laser direct imaging, we can produce boards with trace and space narrower than a human hair. Advanced drills put holes in panels visible to the human eye only with help from a loupe magnifier.

In addition to deploying more advanced equipment, we are improving production efficiency to reduce the number of steps needed for a production run. Continuous process improvement like this is a primary focus in our industry because wellness technology is about more than consumer wearables.

Medical technology also represents an emerging opportunity for the PCB industry. Small, seemingly simple devices capable of complex functionality such as real-time health monitors, smart prosthetics, and intelligent surgical equipment are all part of this innovation trend. Manufacturers serving this vertical need not only International Organization for Standardization (ISO) 9001 certification, but also ISO 13485—a medical device-specific quality management system (QMS) certification that sets a high standard for production documentation and traceability.

Investment in QMS and ISO certification can run into the six figures for PCB manufacturers, but we believe the return on investment is there. Improved quality, increased production capability, and reduced waste are just some of the benefits of implementing QMS and maintaining ISO certifications. Doing so also sends a message to the marketplace that we are committed to delivering a quality product and top-notch service.

New equipment and better processes can't deliver quality by themselves. Quality products are made by skilled personnel who take pride in their work. Quality of life depends on our health, so we do our best to inspire the

team at Sunstone to live healthy lifestyles both at both home and work. For seven years, Sunstone Circuits was named one of the Healthiest Employers of Oregon.

In addition to incentives for getting annual physicals, seeing the dentist, and participating in company-wide challenges, we provide an onsite walking trail, fitness room, and company-organized running and walking events. We have received a lot of positive feedback from employees who are healthier as a result of a lifestyle change we helped promote.

From health tech to wellness programs, our industry is uniquely impacted by this new paradigm. We not only participate in the trend toward healthier lifestyles; we're also helping to build the technology that makes peoples' lives better. Whether it's a treadmill that measures distance covered and calories burned, a wearable device that measures your VO max, or an AI-powered insulin pump, the future of wellness is here, and we are thrilled to be part of it. **DESIGN007**

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

Bob Tise

Matt Stevenson

RoboSense Smart LiDAR Sensor Wins the 2020 Edison Awards

RoboSense MEMS-based LiDAR M1 uses 905nm lasers, boasting low cost, automotive grade, and compact size. Parts have been reduced from hundreds to dozens in comparison to traditional mechanical LiDARs, greatly reducing the cost and shortening production time–achieving a breakthrough in manufacturability. The coin-sized optical module processes the optical-mechanical system results to meet autonomous driving performance and mass production requirements.

In December, RoboSense's LiDAR production line, including RS-LiDAR-M1 has obtained the IATF 16949 certification in the automotive field, which now fully qualifies it to supply to automotive customers. As the winner of the CES Innovation Award in 2019 and 2020, RoboSense RS-LiDAR-M1 also demonstrated that it is the world's first and only Smart LiDAR capable of real road tests on open roads at CES 2020.

RoboSense, the world's leading autonomous driving LiDAR perception solution provider, has been honored as a winner in the transportation and logistics category for 2020 Edison Awards. The Edison Awards is one of the world's prestigious awards that symbolize the persistence and excellence in innovation and honors the most innovative products and business leaders in the world.

"We are very proud that our Smart LiDAR Sensor has been recognized by 2020 Edison Awards. It stands for the global industry experts' recognition of RoboSense product design, development, and production processes and also indicates that RoboSense has achieved a new milestone of complete readiness for mass production of automotive LiDARs. We will continuously improve the product performance for the safety and mass production development of autonomous vehicles," said Dr. Leilei Shinohara, RoboSense co-partner and vice president.

(Source: RoboSense)

A Design Economics Horror Story

Feature by Rick Hartley
R HARTLEY ENTERPRISES

Editor's note: During a recent conversation with Rick Hartley, he shared one of his favorite PCB design horror stories. This is a cautionary tale about what can happen when design teams place too much faith in app notes and do not follow cost-aware design techniques. Enjoy!

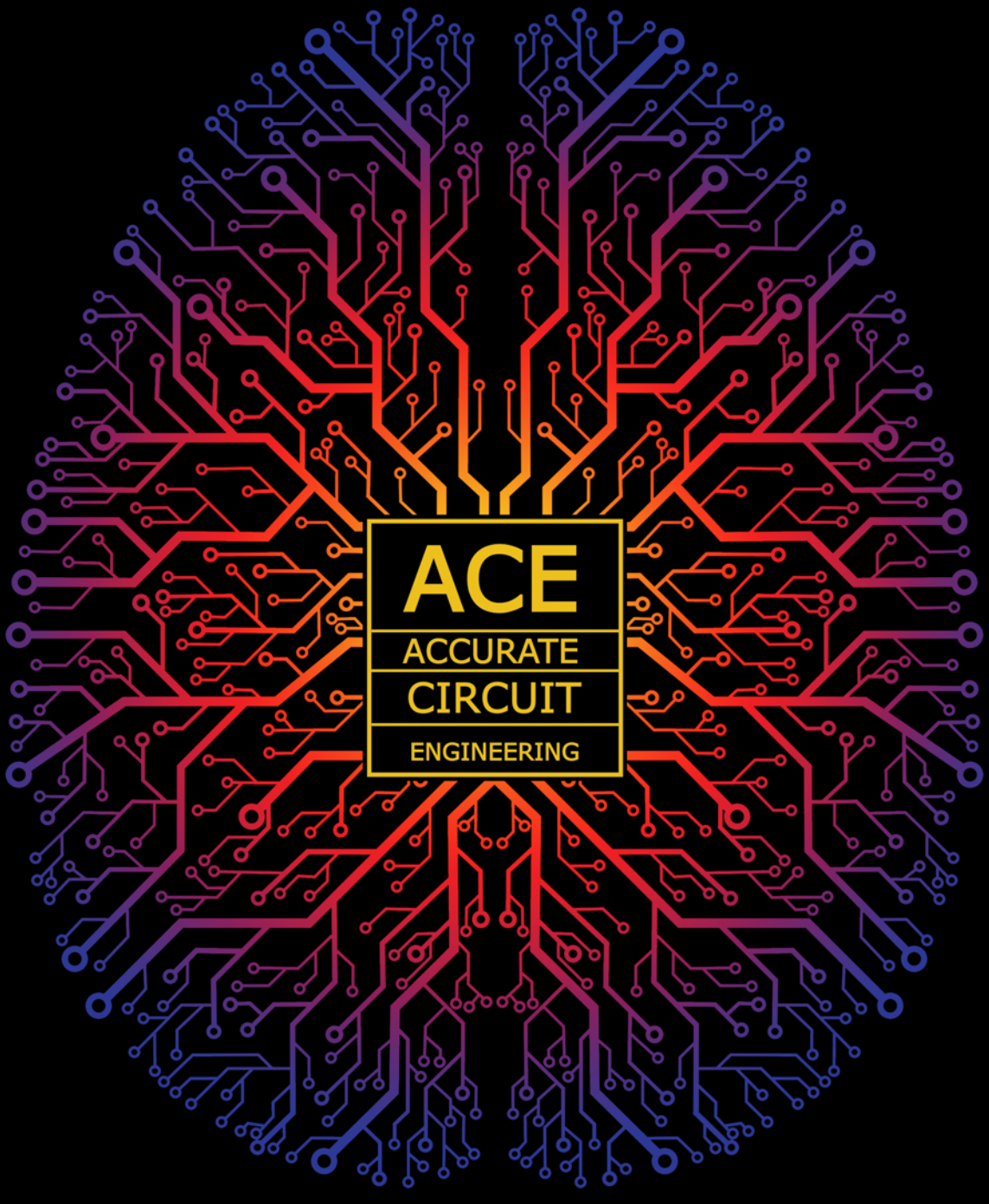
When I first went to work at a company in 2003, I was asked to look at a circuit board that was not working properly; the few they did get to work had major EMI problems. I discovered they had placed the parts badly so that routing the memory lines to be even close to a similar length was going to be nearly impossible.

The engineer who did the schematic looked at the app note for the particular memory that was used with this processor, which stated, "Route all the memory lines within the same length of one another at ±50 mils." The engineer thought, "50 is good, and 25 is even better," so he put a note on this schematic that the memory lines should all be the same length at ±25 mils. What's really interesting is that engineers often don't understand that inner and outer layers propagate at different rates. If you make all the lines the same length, they don't have the same propagation time, and they don't match anyway.

Then, they farmed out the layout. And the people who did the layout—because of the poor positioning of components—couldn't get all the lines routed anywhere close to the same length without a ton of tromboning or serpentining of the memory lines. They ended up with 14 routing layers, and the only way they could get it to the thickness required was to make it an 18-layer board. They had an 18-layer board with only four plane layers and 14 routing layers. Anybody who knows anything about design is already thinking, "How could this possibly work?"

It all happened because of this overabundance of length-matching. To make matters worse, they had to have 65 ohms impedance on some of the lines because there was a PCI bus on the board. In order to get the dielectric constant—with these very thin dielectrics—low enough to hit the target impedance, they had to go to Rogers 4000 series material (at 6 to 8 times the cost of FR-4). The bare boards were seven by nine inches, and they were go-

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

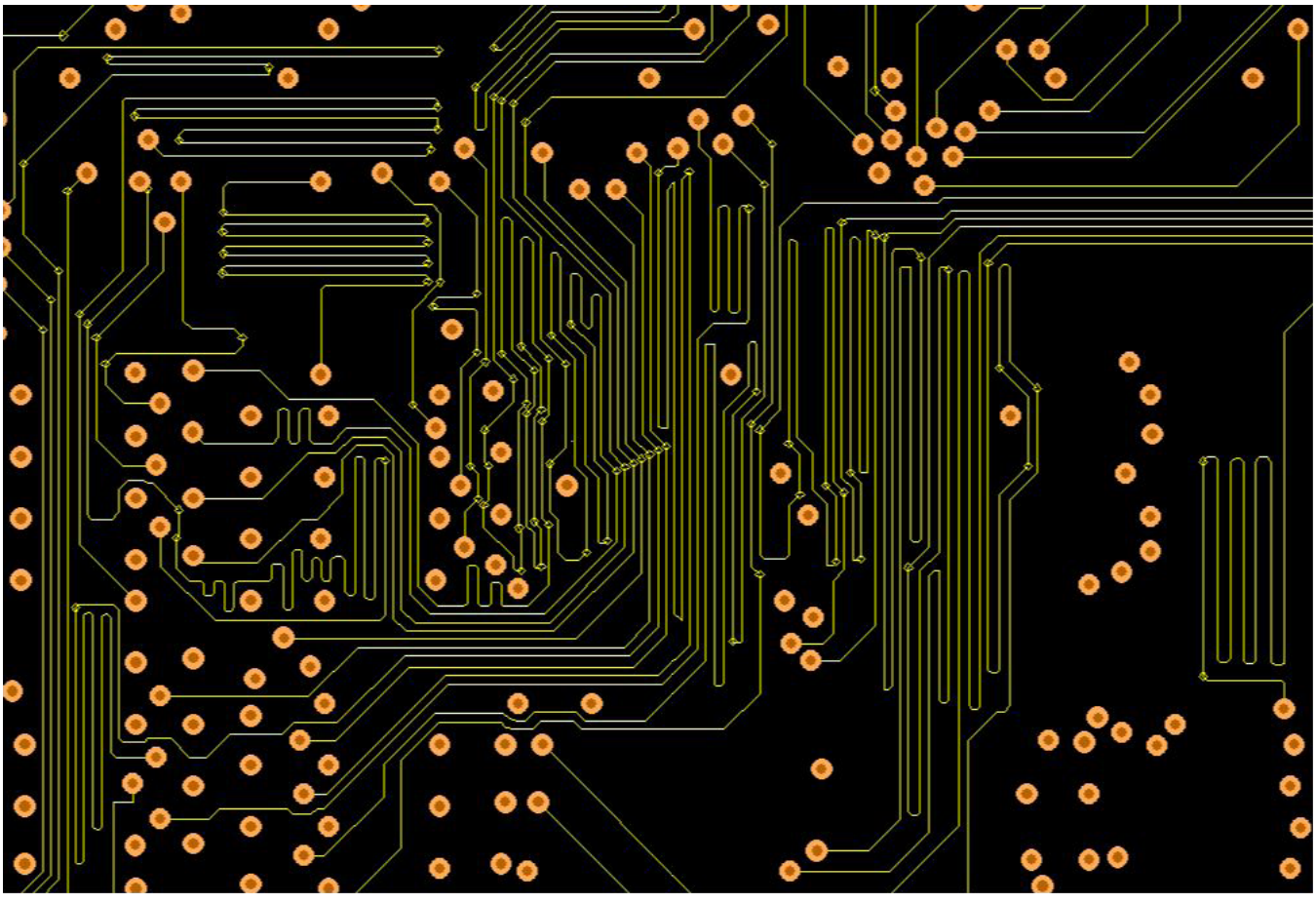


Figure 1: Close-up of the original 18-layer board showing an abundance of tromboning.

ing to cost an estimated $235 each in quantities of 1,000 at a time. Again, most of the boards didn't work, and the few that did work failed EMI testing and were expensive.

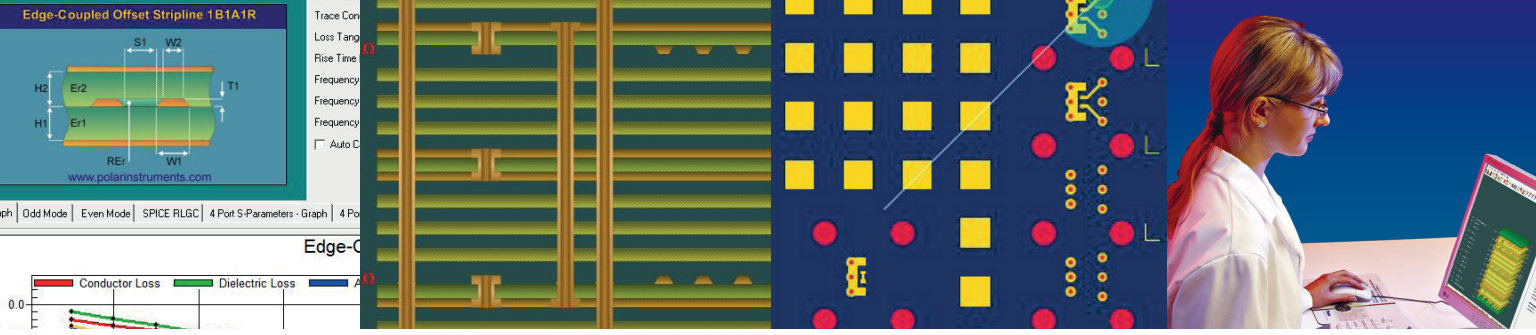
I looked at the board and realized what the problem was. I asked the engineer why he put that note on the schematic. And he said, "It was in the app note. I figured they knew what they were talking about." I asked, "Did you do a timing analysis?" He replied, "You know what management around here is like. We never have time for things like that." I said, "Do you have time now to do a timing analysis?"

The engineer came back to me a day later and said, "By my calculation, we have about ±200–300 picoseconds of available skew." I asked him, "Do you know how much that is in terms of length? It's somewhere between two and three inches. Let's even call it an inch. That means you could have made all these memory lines the same length ± 1inch, and they would have all worked."

He exclaimed, "Bull. I don't believe it." I said, "Well, we are going to prove it to you!" We stripped away the routes, repositioned the components to make things route properly, routed the board with no line matching at all, and checked them when we were done, and they were all within an inch of the same length. We did no serpentining at all. We made a 10-layer board out of it instead of an 18-layer board with six routing layers and four plane layers.

It worked perfectly. It passed EMI testing, and the price dropped to $34 per board from $235 per board. That's what happens when people blindly follow app notes. **DESIGN007**

Rick Hartley is the principal engineer at R Hartley Enterprises.

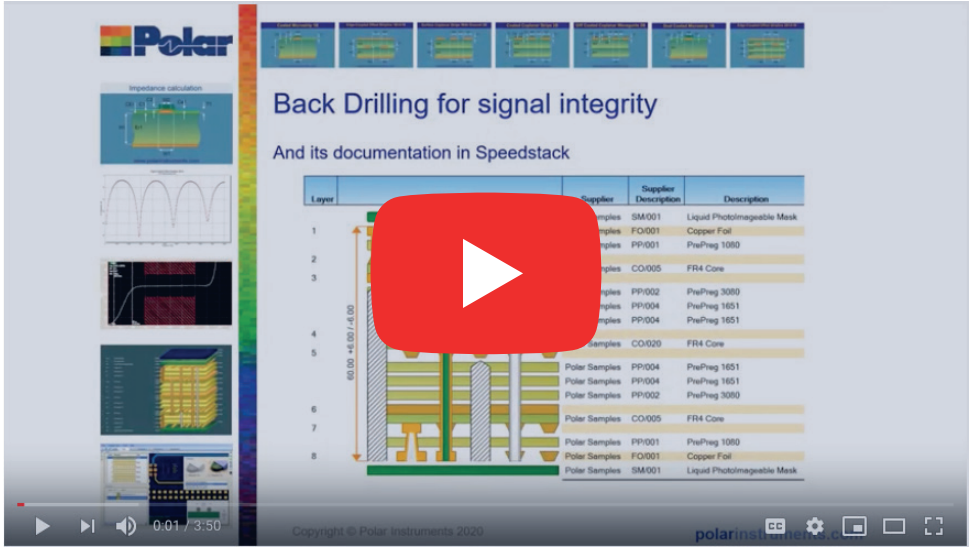


SUBSCRIBE

Subscribe to the Polarinstruments YouTube channel for helpful impedance and stackup videos

PCB Signal integrity tools for design & fabrication

- Impedance & insertion loss modeling with Si9000e
- PCB stackup design & documentation
- Test systems for controlled impedance & insertion loss
- Application notes on a wide range of Si topics



polarinstruments.com

Onward!

The Digital Layout

by Kelly Dack, CIT, CID+, PCEA

Introduction

In my debut as the new author of this column, I provide updates on the direction and leadership of the PCEA, and Stephen Chavez shares his first "message from the chairman." In addition, Bob McCreight, president of the Silicon Valley Chapter, discusses their group's most recent "lunch and learn" event held in February. Lastly, we share our most updated list of professional development and event opportunities, although some may be affected by the COVID-19 outbreak. Stay tuned for more updates.

PCEA Updates

Over the past few months, our writer for "The Digital Layout" column, Stephen Chavez, was unanimously elected as chairman by our newly formed executive board at the PCEA. Steph graciously accepted, and the PCEA is delighted to be part of a vibrant, highly skilled team of PCB industry folks now led by an experienced, young, and newly-minted chairman. Steph has demonstrated a real zeal for helping the electronics industry and is surely capable of leading us successfully into the next decade.

Source for All Quality PCB Manufacturing Needs!

PCBCart is one of the very first Chinese suppliers who take care of worldwide customers' PCB manufacturing demands. Till now, we have produced PCBs for over 10,000 customers from 80+ countries, our overal customer satisfaction rate is over 99%!

20 first order **% OFF** up to $300

Quote Now ▸

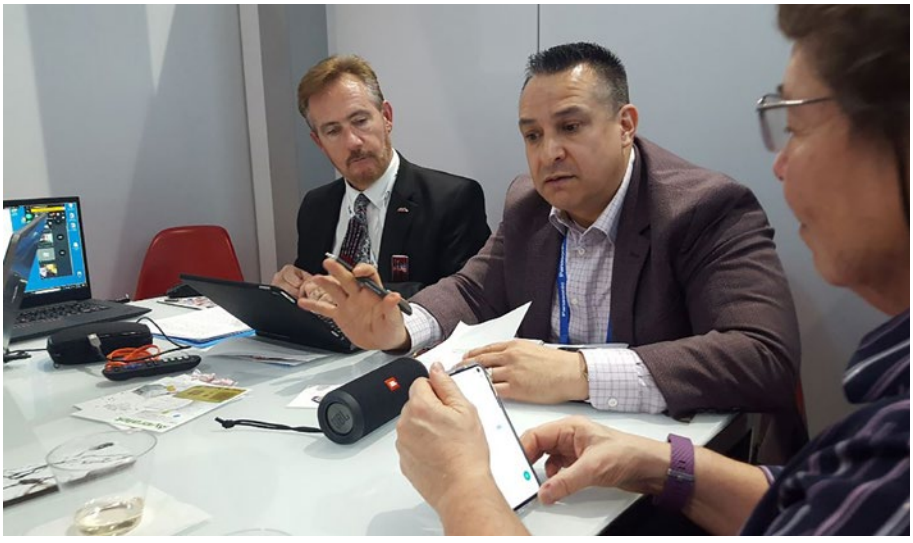
Why Choose PCBCart:

- 14-years' experience in all kinds of PCB fabrication
- Advanced manufacturing capabilities
- Rigorous quality control
- Full-scale testing and inspection
- Multiple value-added options
- Online quoting & ordering system
- Best balance on price-quality-service
- Knowledgeable support team

Pcbcart

WWW.PCBCART.COM

SALES@PCBCART.COM



Mike Creeden, Stephen Chavez, and Cherie Litson.

Steph has written before that his main objective in writing was to promote continued globalization of knowledge-sharing for any and all involved in the design, fabrication, assembly, and test of printed circuit boards. "The Digital Layout" has been Steph's labor of love for the electronics industry. He has put in many hours of his own time to lay the foundation, solicit interesting content, and craft this column into a must-read for people who want to engage.

All of us in the PCEA wish to thank Steph for setting such a high standard for this column and for inspiring excellence in all of us as we move forward. In closing out his column last month, Steph mentioned a process of evolving from a writer to a leader. As chairman, he has already proven himself a leader by becoming an effective delegator. Within a short time, the PCEA team was led to move quickly to nominate and elect a replacement communication officer so that this column would not miss a beat.

Well, here I am, and I want to thank Steph for his kind words and for the confidence shown by the executive team in nominating and electing me as PCEA's communication officer. I won't take this service lightly, and I will do my best each month to bring you coverage of this unique association of professionals who are on fire for promoting collaboration, inspiration, and education within the PCB engineering industry.

Okay, let's try this out. Tap, tap… Is this mic on?

Who Serves the PCEA?

Over the past several months, a group of people active in the PCB design and electronics industries transitioned to form the PCEA. In order to move forward with the preliminary ideal and goals that the group had set for the long term, important organizational roles needed to be created and filled immediately. Leveraging time together while meeting at several industry trade shows and employing online meeting apps, the group was eager to begin nominations and elections to fill critical roles.

I am very proud to announce our progress in electing the first servant leaders to the PCEA executive board.

- Chairman: Stephen Chavez
- Vice-Chair: Mike Creeden
- Chairman Emeritus: Gary Ferrari
- Administration: Cherie Litson
- Treasurer: Mike Creeden
- Communications: Kelly Dack and Stephen Chavez
- Media/Social Media: Judy Warner and Tara Dunn
- Sponsorship/Endorsement: Mike Creeden
- Education: Susy Webb, Gary Ferrari, and Rick Hartley
- Chapter Liaisons: Scott McCurdy and Terri Kleekamp

New ideas for the organization are being identified at just about every meeting. These are exciting times for the PCEA. In the months that follow, I will not only be providing coverage of the activities of the executive staff and their duties serving the PCEA, but I will also be working with Stephen Chavez to feature expanding chapters and report on how they will represent and grow the benefits and values of the PCEA in their own local areas and beyond.

From left to right: Mike Creeden, Stephen Chavez, Randy Kumagai (president of the Phoenix PCEA Chapter), Cherie Litson, Judy Warner, and Gary Ferrari.

Message From the Chairman

by Stephen Chavez, MIT, CID+

CHAIRMAN OF THE PCEA

As chairman of PCEA, let me start off this first-ever "Message From the Chairman" by saying that it is an honor and privilege to be selected to lead such a large global group of talented and passionate veteran industry leaders. As we move forward and establish a path for PCEA, collaboration, inspiration, and education are at the core of what the organization is all about.

Looking back earlier in my career, when I made the decision (more like a huge nudge from one of my early mentors) to get involved and become an industry contributor, it made such an enormous positive difference in my career and within my personal life as well! Not only have I had continued success as a printed circuit engineer, but I have had the great fortune to meet many great industry colleagues who have become dear friends over time. I was lucky enough to get guidance back then, but only after lots and lots of struggling on my own.

As the newly elected chairman of PCEA, serving our global chapters and members, collaboration, inspiration, and education are truly at the core of my passion as well. I intend to give my all to make a positive difference in our ever-evolving industry and to help others succeed in their profession, like those who have helped me along the way.

I highly encourage you all to get involved! Join the PCEA by visiting our website (www.pce-a.org) and registering as a member to become part of the PCEA collective. You can al-

ways reach out to me (stephen.chavez.pcea@gmail.com) or Kelly Dack (kelly.dack.pcea@gmail.com) to get more information, as well.

Our evolution as a professional association within the industry is progressing as planned. We continue to solidify the foundation of PCEA now that the executive board leadership has been voted on and is in place. Our website is a work in progress with a new and improved version ready this spring. Our incorporation status of 501(c)(6) non-profit is coming together as planned. We should be getting our employer identification number (EIN) within a few weeks.

Things are moving very fast and coming together very nicely due to the outstanding efforts, positive attitude, and passion that each of the PCEA executive board members brings to the table. This is truly the right group of industry leaders to take on such a task and make it successful! I feel very humbled and blessed to be a part of such a team.

PCEA Chapter Spotlight

by Bob McCreight

PRESIDENT OF THE SILICON VALLEY CHAPTER

On February 13, the Silicon Valley Chapter enjoyed a sponsored "lunch and learn" event and some very interesting presentations. After lunch, the technical director for design education at Insulectro, Mike Creeden, began the presentation portion by filling us in on the formation of the PCEA.

As the electronics industry is becoming aware, the PCEA is a new association that has been formed to inspire and support collaboration and education between all facets of the PCB engineering, manufacturing, and test sectors. There was a great amount of interest from the audience with regard to the new organization, and several followed up with various queries.

Following a round of introductions from everyone in attendance, Faisal Ahmed, an application engineer at Cadence, provided a short demonstration of some new enhancements to Allegro, such as DesignTrue DFM Technology. This fresh take on design settings leverages technology files that are programmed to match the manufacturing capability from participating manufacturing partners. Designers now appear to have a direct link to DFM settings for some suppliers. Faisal also explained Cadence's 3D Canvas—a high-quality 3D visualization engine—and the Symphony Team Design Option, which allows multiple designers or teams to perform concurrent engineering using a shared canvass without having to set up a partitioned project.

Faisal Ahman

Atar Mittal

Following the Cadence presentation, the meeting progressed to the feature presentation by Atar Mittal, general manager of Sierra Circuits' Design and Assembly Division. Atar presented his paper "Signal Degradation on PCB Transmission Lines: Causes and Remedies" and shared some vital points—including signal reflections, crosstalk, EMI, and PCB materials—comprising a very insightful talk that held audience attention and resulted in a detailed Q&A session at the end.

All in all, the meeting went quite well, with 30 PCB engineering professionals in attendance. Our thanks go to Mike Creeden, Cadence, and Sierra Circuits for providing their resources and expertise, which contributed to the success of this event.

Our next PCEA meeting will be on April 23, hosted by Amazon and sponsored by Altium. Chris Carlson, an FAE with Altium, will be the main speaker. I hope to see you then in Sunnyvale, California. Detailed announcements will go out later this month.

The PCEA Executive Board

Stephen Chavez
- *Chairman*
- *Communications*

Mike Creeden
- *Vice-Chairman*
- *Treasurer/Sponsorship/Endorsement*

Gary Ferrari
- *Chairman Emeritus*
- *Education*

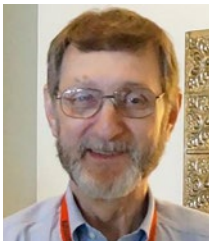
Cherie Litson
- *Administration*

Kelly Dack
- *Communications*

Judy Warner
- *Media/Social Media*

Tara Dunn
- *Media/Social Media*

Susy Webb
- *Education*



Rick Hartley
- *Education*

Scott McCurdy
- *Chapter Liaison*

Terri Kleekamp
- *Chapter Liaison*

Professional Development and Events

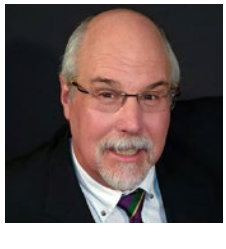
It has been our custom to highlight all up-and-coming industry events to look out for in 2020. We will continue this; however, with the challenges brought upon our industry by the COVID-19 outbreak, we can only remain hopeful that these events will not be affected. If you have an interest in any of these events, please search and contact the event coordinators directly for the latest event status.

- May 6-7: Del Mar Electronics & Manufacturing Show (San Diego, California) *Postponed*
- May 12-14: IPC High-Reliability Conference (Baltimore, Maryland) *Postponed*
- June 9–10: PCB2Day—SMT Assembly Boot Camp (Austin, Texas)
- June 11–12: PCB2Day—Design Essentials for PCB Engineers (Austin, Texas)
- June 14-20: IPC SummerCom 2020 (Raleigh, North Carolina)
- June 22–25: Realize LIVE 2020 (Las Vegas, Nevada)
- October 7–9: AltiumLive 2020 (San Diego, California)
- September 8–11: PCB West (Santa Clara, California)
- November 11: PCB Carolina (Raleigh, North Carolina)

We also want to spread the word. If you have a significant electronics industry event that you would like to announce, please send me the details by clicking here, and we will consider adding it to the list.

Conclusion

No matter what your part is in this industry, let's continue to move forward together by finding solutions for the unknown and having hope for the future. **DESIGN007**



Kelly Dack, CIT, CID+, is the communication officer for the Printed Circuit Engineering Association (PCEA). To read past columns or contact Dack, click here.


Nick Barbin

Design Economics With Optimum Design Associates

Feature Interview by the I-Connect007 Editorial Team

We recently interviewed Nick Barbin, CEO and co-founder of Optimum Design Associates. Originally a design bureau, ODA has expanded over the years and now includes prototype and mid-volume assembly, test, and inspection. We asked Nick to break down the economics of PCB design for us.

Andy Shaughnessy: Thanks for joining us, Nick. We're discussing the economics of PCB design. When you look at a design, do you track things like the total cost of the design?

Nick Barbin: Absolutely. I look at it on a per-project and per-designer basis.

Barry Matties: Are you looking at it from billable hours or the design economics? For example, do you look at the impact of a design on the finished product for a supply line cost?

Barbin: Being a service provider, we are not privy to the overall cost of a particular product in regard to development. With that said, PCB layout would be a very small percentage of the overall cost. Our interest is in providing our customers with a very good estimate of hours, and then billing for logged hours, not an hour more or an hour less. Some of our customers require fixed bid, and we can accommodate that as well.

Matties: When someone comes to you for a project, are they coming in with a budget in mind?

Barbin: In 30 years, I've never had anyone tell me what their budget was.

Matties: Is there some sticker shock in some cases when they realize what the design might cost?

Barbin: Yes, but we do a very good job of breaking down the various milestones to provide the details of hours. They can get a less expensive price from somebody working from home but will come to Optimum for consistency amongst our 20+ member design team and because of


We are dedicated
to excellence through
innovation,
technology
and most importantly,
service.
COMMERCIAL • MILITARY • MEDICAL • BAREBOARD
U.S.C.
U.S. CIRCUIT
www.uscircuit.com
ISO
9001 CERTIFIED
ITAR

our high degree of competence with today's SI and manufacturing technologies.

Matties: When you're putting a project together, there's no budget. How do you manage the filler materials and the supply chain?

Barbin: Generally, when a customer comes to us with a layout project, they are providing the schematic, BOM, and mechanical specification, so we don't generally provide much into the input package We do, however, collaborate and make suggestions where we have experience with a specific component or technology.

Dan Feinberg: One of the discussions that we've had lately, particularly at IPC APEX EXPO this year and last year, was about reliability. Have you seen any changes in the last few years regarding specifications and the need for higher reliability with the finished device?

Barbin: Reliability is always a major concern for our milaero and automotive customers. In addition to understanding and implementing Class 3 and our specific customer workmanship standards into a design, our designers will collaborate closely with the engineering and manufacturing team to help ensure we don't miss anything. It really takes a team effort. Conversely, for the customers where cost is a big issue, being a manufacturer, our designers have a strong awareness of manufacturing cost drivers built into our DNA. Optimum designers are keenly adept at switching modes between reliability and cost objectives.

Optimum designers are keenly adept at switching modes between reliability and cost objectives.

We also incorporate Valor DFM analysis for all design projects at the various major milestones. The DFM is run by one of our DFM specialists who puts together a report for the designer of any and all issues. Between our strong understanding of the fabrication and assembly processes and the nuances between Class 2 and Class 3, our designers are able to put together a design package that can flow seamlessly through the manufacturing chain.

Shaughnessy: Do many designs arrive over-constrained, maybe using expensive technology when they don't have to?

Barbin: To ensure proper SI, PI, and complex enclosures, very little comes in over-constrained today. In some cases, we may have seen a specific technology or topology that we have incorporated on a design that has worked great on another design; we are then able to share that when a different constraint might not be achievable. To ensure the various circuits function correctly, it's very important to have all aspects of the design constrained within the tool to ensure all requirements have been met.

Shaughnessy: Do you ever tell a customer that you have found a way that they can save some money on their design? Is that a fairly common thing?

Barbin: Sure, that is very common. This is especially important when it comes to the raw PCB, as it is normally one of the highest cost items on a BOM. Our designers work very closely with our customers and the manufacturing engineers of the various fabrication shops to construct a stackup that will meet their electrical and cost requirements; this is paramount to making a successful design. Choosing the right material has a big impact on cost. Although our designers don't specify the material, they regularly collaborate with the engineer and may occasionally suggest a material that might be less expensive than what the engineer originally specified.

I believe Optimum's experience with incorporating HDI technology into a design is a major

differentiator between us and our competition. Implemented wrongly or inefficiently, it can result in a design being more costly than needed or not manufacturable at all. On a daily basis at Optimum, we incorporate some type of HDI strategy into many of our customers' designs. It is essential that we're able to collaborate as a team to find a strategy that achieves the lowest cost possible. A couple of years ago, one of our designers wrote a case study article about the strategy that was used for one of our customers in which they needed to reduce the form factor space in half. It's available on our website.

Shaughnessy: I know you all use Valor DFM, which lets you track costs.

Barbin: Yes, you can plug a lot of that into Valor. I've seen some EMS companies do a better job than others, where they put in a cost element by incorporating feedback from a DFM review about what that cost factor was.

Shaughnessy: Do your customers come to you primarily for technology reasons, or for bandwidth or cost?

Barbin: Just about everything. We work for anybody who will write us a purchase order. But we do very well for large OEMs and milaero companies that require consistency and scalability. To ensure all our designers are on the same page, we have documented our processes with flow-charts, design guidelines, and have incorporated checklists throughout the various design phases. Additionally, we are ISO certified and have cybersecurity measures in place to ensure our customers' data is protected. From a technology standpoint, we design approximately 250 boards a year from a very diverse customer set, so we have a very broad capability range.

Matties: Do you see an increase in activity for orders right now, or is there a slowdown?

Barbin: We have been growing an average of about 16% year over year for the last six years, and we're trending that way again this year.

Matties: What technology trends do you currently see?

Barbin: Higher frequency and data rates. For SerDes, we see data rate ranges from 16 to 128 Gb/s, and for RF products, we see 12–16 GHz. We also see a lot of boards utilizing RF system-on-a-chip throughout many industries, including milaero, automotive, and IoT. For all these markets, we see more requirements for rigid-flex designs.

Matties: When keeping your team staffed, one of the things we hear about is the shortage of PCB designers. Have you experienced that?

We find designers wherever they live, and we set them up to be successful in working for us.

Barbin: Yes. We find designers wherever they live, and we set them up to be successful in working for us. Once we have a possible candidate, we will test them by giving them a small board to design. The idea is to see how they interact with our design manager. If we like them, we'll make them an offer and bring them on. For the first year, our design manager will perform peer reviews and provide observations at the various milestones. Our intention is to learn from each other and to continuously improve our processes and increase our knowledge as a team. As we scale our design team, it is vitally important that we do so in a way that ensures we are able to maintain our reputation for quality and our culture of pride of workmanship. Our designers are the rock stars at Optimum.

Feinberg: What are your biggest design challenges?

Barbin: The biggest challenge is for designers to keep up with the increased role that they

play in understanding the circuitry without being an engineer. Designers need to understand so much more than when I got into this business—back when there was more of a drafting element to it. Designers are the bridge between electronics and manufacturing, so they need to help bring together power integrity, signal integrity, cost, and reliability aspects of the design.

They will also need to be able to see boards together in a 3D environment and be able to take measurements, so they're not just constraining based on one board but a system. Being a manufacturer, design for robotics is going to be important in the near future as robotic arms are being used more frequently.

Feinberg: Have you seen an increased level of communication between the design teams and the suppliers who are supplying things like the laminates and the photoresist, solder masks, etc., as well as the fabricators?

Barbin: We are communicating with the various fabricators on a daily basis, as they're a great source for us in helping put together stackups as well as understanding all the different materials that are available.

Matties: When you hire somebody, what do you expect from a designer? What sort of skill sets do they have to come in with now?

Barbin: Going forward, I think that some college-level of electronics is going to be required, if not an EE background. You can see the EDA tools are heading in that direction by providing intuitive interfaces, simulation, and library services. When I see resumes that discuss different projects—especially the types of technologies that we're working on here—that's a positive from my standpoint.

Shaughnessy: How do you manage the customer's design?

Barbin: We do this as a partnership between the lead designer and design project manager. The lead designer is the captain of the ship, responsible for the technical and on-time delivery while the project manager communicates all schedule and business-related items to our customers. We do that on an ongoing basis. We don't want any surprises as we get toward the end of the design schedule. We have taken a very proactive approach, which has worked very well for us.

Every Monday afternoon, all our designers do a video call, and they must have their video-enabled. Each designer will have two minutes to discuss their respective designs. This meeting helps increase the collaboration and communication between our designers. We're a team, and as we see different issues coming up, we can share them with each other.

Matties: That's great, and I imagine that you're looking to hire.

Barbin: Yes. We don't like to run more than at 75–80% of capacity; if we do that, then we can't take on new business. If we tell a customer, "We can't take on your work because we're too busy," we may never see that customer again. But I'm always on the hunt for passionate designers who love what they do. If you support them, they're going to dazzle our customers.

Matties: It sounds like you're doing a good job running your business; it's systems-based and data-driven. What's the best piece of advice you would give a young designer?

Barbin: Dive into the books. Scott Nance is director of PCB layout, and he and Brendon Parise—one of our technology managers—wrote a book called *A Practical Guide to RF and Mixed Technology Printed Circuit Board Layout*. I give this book away to our customers, but it is also available for sale through Amazon. We also put out many articles on the resource portal on our website regarding subjects that we're passionate about, whether it's a case study on HDI or how to implement DDR the correct way. You also have to continue to learn the EDA tools. Tools are always changing, and designers generally get complacent in doing things in a particular way. I would advise them to take a fresh look at the functionalities within the tools to see if there is a better way of doing something.

Matties: Where are you based, Nick?

Barbin: We're located in the Bay area in a small town 20 miles east of San Francisco and Oakland called Pleasanton.

Matties: We appreciate your time, Nick. This has been great.

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

BYD, Toyota Launch BYD Toyota EV Technology Joint Venture to Conduct Battery Electric Vehicle R&D

BYD Company Ltd. (BYD) and Toyota Motor Corporation (Toyota) announced that preparations have proceeded since they signed an agreement for the establishment of a joint venture company to conduct research and development of battery electric vehicles (BEVs) on November 7, 2019, and registration of the new company has been completed. Operations are scheduled to commence in May 2020. The name of the new company is BYD TOYOTA EV TECHNOLOGY CO. LTD. (BTET). Hirohisa Kishi from Toyota will serve as chairman, and Zhao Binggen from BYD will be CEO.

New Chairman Hirohisa Kishi said, "With the engineers from BYD and Toyota working together under the same roof, we aim to develop BEVs that are superior in performance and meet the needs of customers in China by merging the two companies' strengths and also through friendly rivalry."

Binggen added, "This joint venture company will focus on the research and development of battery electric vehicles with technology and know-how from both China and Japan. The company is committed to promoting and populating high-quality technologies that make battery electric vehicles more environmentally friendly, safe, comfortable, and intelligent. Our vision is to create a future customer-first mobility style and harmonious society for humans and nature."

BYD and Toyota will work together to meet the diverse needs of customers by researching and developing BEVs that appeal to customers and promoting their widespread adoption and also hope to contribute to improving the environment in China. (Source: JCN Newswire)

Understanding Dk Data Key to Cost-aware Design

Lightning Speed Laminates
Feature Column by John Coonrod, ROGERS CORPORATION

In the development stages of a circuit for a new PCB application, there are usually several iterations to the circuit, including testing, redesigning, building new circuits, etc. These many changes can be costly, and it is not uncommon for a project to have 4–8 changes before it can be released to the market. One item that can substantially reduce the number of changes and the associated costs is the use of a good circuit simulation software.

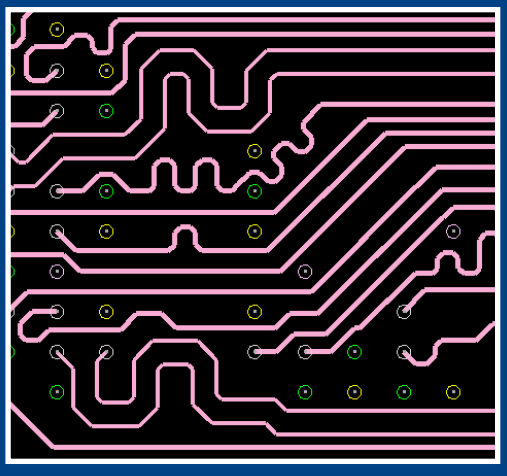
There are many very good software programs on the market that will allow a circuit designer to predict the electrical performance of a circuit. The predicted electrical performance simulation is done on a model of the circuit and is often related to impedance and insertion loss. There are many other circuit attributes that can be simulated, but there are usually some differences between the actual circuit performance and the predicted performance of the simulated model. Sometimes, these differences are small; other times, the differences are very significant.

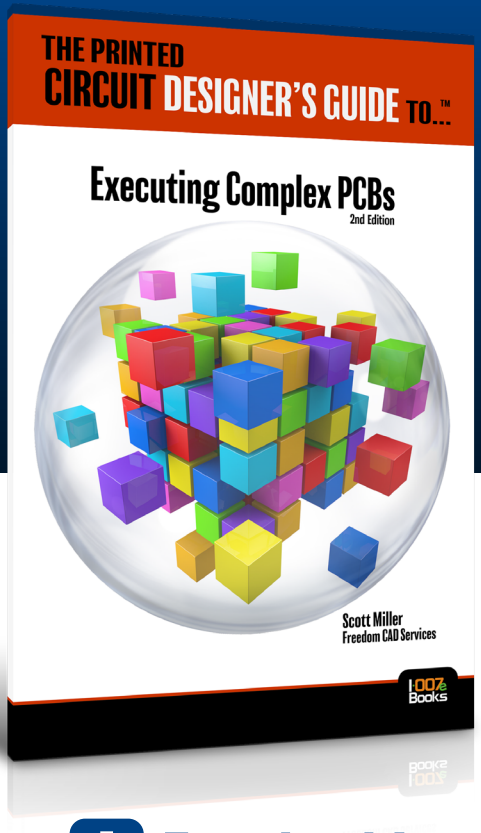
Before the designer inputs the data into the simulation software, they need to make sure they know the details behind how the data was generated. Since all simulation models are not the same due to the unique desired performance of the circuit, the data being input may not be appropriate for a specific model. The inaccurate information could be caused by the input from the user, who is defining the model. Sometimes, it could be an oversight from the user, or it could be inaccuracies from the information which the user assumes to be accurate. One area of potential inaccuracy for these software programs is the dielectric constant (Dk) value for a circuit material. Although, even when the Dk value is accurate, the user may use it inappropriately due to a misunderstanding of how the Dk value was obtained and what it represents.

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

The Dk of any dielectric material is frequency-dependent. In other words, when testing the same piece of material and using the same test method, it should be expected to get slightly different Dk values when testing at different frequencies. There is a range of frequencies where the Dk changes more with a change in frequency, and that is typically from a few megahertz up to about 5 GHz to 10 GHz. After about 10 GHz, and out to about 250 GHz, the Dk frequency curve will have a slight negative slope for most low loss circuit materials. Considering this range of frequencies, and depending on the degree of polarity of the circuit material, the slight decrease in Dk with an increase of frequency is usually in the range of 2% or less. For more accurate circuit modeling, it is important to use material data that was produced at the same frequency as the circuit being modeled.

For more accurate circuit modeling, it is important to use material data that was produced at the same frequency as the circuit being modeled.

Another issue, which is typically a little more problematic for Dk values used in circuit simulation, is anisotropy. Most circuit materials are anisotropic, which means the Dk is not the same on the three axes of the material. It is pretty common for most circuit materials to have a Z-axis (thickness axis) Dk, which is different from the x-y plane of the material. The X-axis and Y-axis are typically similar for Dk values, but the Z-axis is often quite different. Also, for the testing to determine the Dk of a material, there are common test methods that evaluate the Z-axis of the material only and other test methods that evaluate the X-Y plane only.

If a designer uses the Dk information in their model, which was produced by testing the X-Y plane (instead of the Z-axis), it may or may not be appropriate for their particular model. It is good for the designer to be aware of what type of test method was used to determine the Dk, as well as the frequency at which the Dk value was obtained.

In the case of most high-frequency circuit materials, which have a Dk around 4 or less, the anisotropy is typically not that significant. In most of these cases, the difference between the Z-axis and the X-Y plane Dk values is 3% or less. However, for non-filled, glass-reinforced circuit materials, these Dk differences can be much higher.

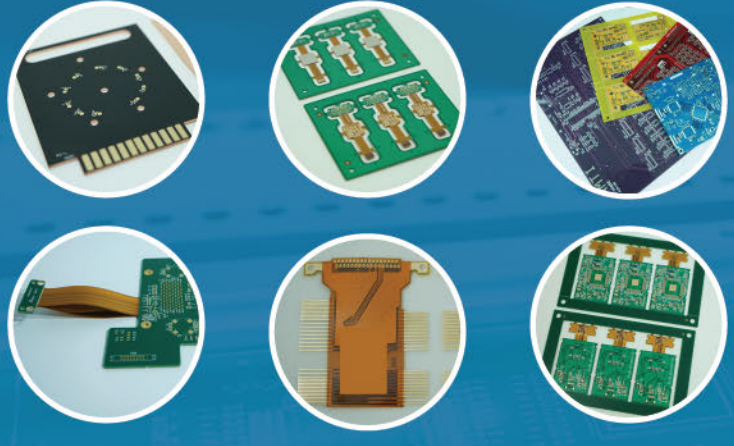
In the case of higher Dk materials, such as materials with Dk values in the range of 6 or higher, there can be much larger differences in the Dk values of the Z-axis when compared to the X-Y plane values. For these materials, it is not uncommon to see Dk differences due to anisotropy of 5–15%, depending on the material. There are some exceptions, where some high Dk materials are formulated to have minimal anisotropy; however, the designer should consider anisotropy when using materials with higher Dk values.

Lastly, the designer should use Dk values from a test method that is most similar to their circuit design and at the same frequency as their application. Since there are a limited number of test methods and a very diverse number of circuit applications, it can be difficult for the designer to find a good match between the test method and their model. Regardless, the designer should investigate and try to use a Dk value that was generated in a manner that is as close to their model as possible. Further, engage with the material supplier to see if they have other Dk information that may be more appropriate for their design. **DESIGN007**

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

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

PCB007 Highlights

2020 EIPC Winter Conference, Day 1 ▶

Pete Starkey recaps how the 2020 EIPC Winter Conference, held in mid-February, attracted around 90 delegates from a dozen European countries—as well as a few from North America—to an outstanding learning and networking experience for members of the PCB community. The theme of this year's event was: "The Needs for Next-Generation Electronic Devices and Changes in Fabrication Solutions for PCBs, PCBAs, Materials, and Technologies."

IPC Chief Economist Dr. Shawn DuBravac Shares Industry's Financial Outlook ▶

On March 23, IPC Chief Economist Shawn DuBravac, Ph.D., CFA, spoke with I-Connect007 publisher Barry Matties in an exclusive phone interview immediately following DuBravac's online briefing to IPC member attendees.

Institute of Circuit Technology Spring Seminar 2020 ▶

Back to Meriden, the nominal centre of England where the daffodils were blooming. A good crowd made it to the spring seminar that followed the Annual General Meeting of the Institute of Circuit Technology (ICT), with five specialist presentations and excellent opportunities to network with their peers in the industry. Pete Starkey shares his overview of the event.

I-Connect007 Researchers Survey Industry on COVID-19 Outbreak Effect ▶

I-Connect007 surveyed our readers regarding the coronavirus (COVID-19) outbreak and its effect on our industry. After reviewing the first 100 survey responses primarily from North America and Europe, one-third of the respondents reported that the outbreak was having at least a small effect on their orders. Two-thirds of our readers have yet to see any effect on their orders.

Audio Interview With IPC President and CEO Dr. John Mitchell: COVID-19 Global Industry Update ▶

On March 20, Dr. John Mitchell, IPC president and CEO, spoke with I-Connect007 publisher Barry Matties in an exclusive phone interview with updates on COVID-19-related current events in the manufacturing industry. In this information-packed 14-minute audio interview, Dr. Mitchell shared key takeaways from Friday's IPC Executive Forum conference call.

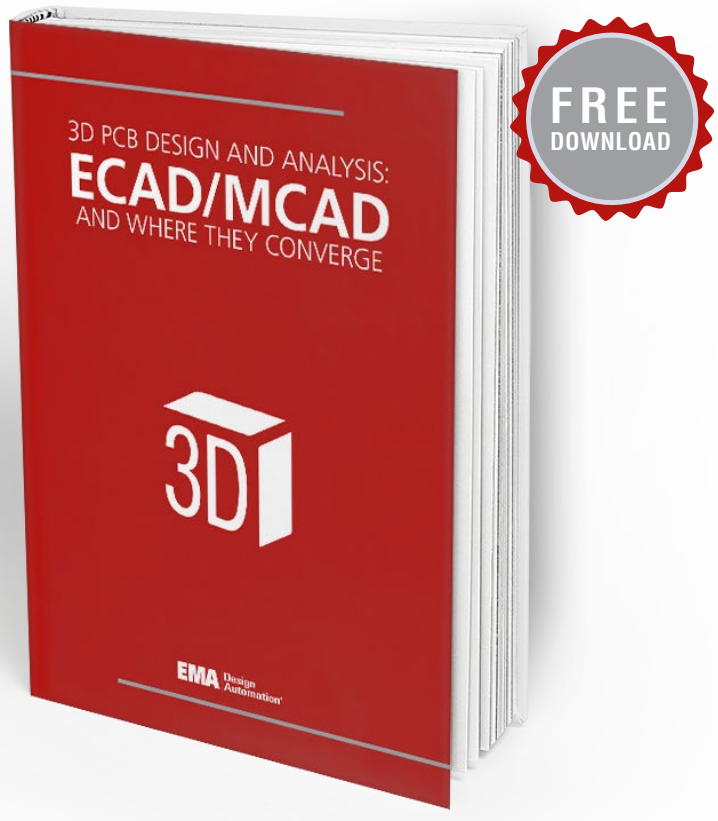
NCAB: A 'Hub' That Works With Designers From Start to Finish ▶

In this video interview from the show, Kelly Dack meets with Ruben Contreras, field applications engineer at NCAB Group, and Shaun Salas, key account manager. They discuss a variety of ways to overcome challenges with tariffs and broker relationships, as well as emphasizing upstream expertise and how the company operates as a "hub." Part of NCAB's objective is to work with designers and design engineers to make the board more manufacturable.

Standard of Excellence: Successful R&D With Your PCB Partner ▶

Now, more than ever, we have to rely on our PCB partners to help us with new product development. A great deal of trust and confidence in our PCB vendors is required to create and fulfill this type of partnership fruitfully. Anaya Vardya shares eight things that must be in place to have a successful R&D relationship with your PCB partner.

UPGRADE YOUR KNOWLEDGE



Learn about ECAD/MCAD INTEGRATION in our latest e-book and view our growing library of resources created by the PCB design experts at EMA Design Automation.

Get the E-book Today ›

EMA Design Automation®

Thermal Management: Five Tips for Application Success

Sensible Design

by Jade Bridges, ELECTROLUBE

In Electrolube's recent columns, my colleagues Phil Kinner and Alistair Little have outlined various problems and solutions regarding conformal coatings and encapsulation resins, highlighting their role in circuit protection across an array of applications. This month, it's my turn to share five tips to improve your thermal management process based on some typical questions that our Technical Support Team receives.

With so much to consider when choosing a thermal management material, it's important to do your calculations, and consider the equipment's operational and environmental conditions and experiments. Underestimating these could compromise the reliability of an electronic assembly and shorten its life expectancy. I'd like to concentrate on how to apply a thermal interface material correctly, take a look at screen-printing methods, and address the issues you are likely to face if you don't employ the appropriate TIM.

1. Is there a specific method for screen printing?

Screen printing can be a manual or an automated process, and screens can be plain or produced to the desired shape for the accurate application of thermal management material to your device. An automated process requires specialist equipment; however, a manual process just requires a screen and a squeegee.

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

These are low-cost items that are readily available; all that you need to do is match the screen mesh diameter to the product you are using and the required thickness you wish to apply. Your TIM supplier should be able to assist with this. How you apply a thermal interface material, or thermal paste, is important for the effectiveness and reliability of the product. For the best results, the entire surface must be covered using a thin, uniform layer.

For the manual application of a thermal paste using a screen, we recommend starting with a screen thread count of around 70 threads per cm. The thread count of the screen controls the thickness of paste applied and may need to be adjusted to suit your particular application.

Lay the cleaned heat sink or substrate, face-up on a flat work surface. Place the screen evenly over the surface and apply a thick line of thermal paste at the far end of the screen using a spatula. Using a squeegee that is wider than the substrate, slowly and carefully drag the paste over the surface, ensuring all areas are completely covered.

Apply high pressure to the squeegee when in use and only pass over the surface once. Gently lift the screen, slowly applying pressure so that you can see the screen separating from the surface. Do not lift the screen off quickly or unevenly as this may affect the distribution of the paste left on the substrate.

If any errors do occur, clean the product from the screen and substrate by wiping off the excess with a dry cloth and then clean. Once removed, you should be left with a uniform layer of thermal paste on the substrate. Assemble the parts together immediately to avoid any contamination migrating onto the applied paste.

2. Why is less often more when it comes to the application of thermal management products?

Thermal management products are designed to remove air from between a heat-conducting component or device and its heat sink. This could be a standard heat sink, or it could be the device housing designed to offer both protection and heat dissipation. As such, the air gaps could be very small (in the case of thermal interface applications) or up to a few mm in the case of a gap between component and casing.

In all cases, air is the worst conductor of heat, and the heat sink or casing will be the best heat conductor. To ensure the most efficient heat transfer, all air must be displaced without adding extra thermal management material to widen the gaps further; therefore, keep the quantity of thermal management material to a minimum.

3. Why does a thicker layer of thermal paste not enhance the performance of the paste?

The thermal management product is designed to be flexible in terms of its application and should, to some degree, conform to the contours of the interface in a way that a solid metal material could not. However, a solid metal heat sink will have a much higher bulk thermal conductivity than a thermal management material, such as a paste or phase change product, for example.

Therefore, if you apply too much of the thermal management material, you are not only displacing air, you are also increasing the bond line thickness between the heat-generating component and the heat sink. By pushing the heat sink further away from the component and putting a lesser thermally conductive material at the interface, the overall thermal resistance will increase, consequently reducing the efficiency of thermal transfer.

4. What are the potential issues to be faced by not employing correct thermal management?

If the temperature of a device or component is not controlled, it could lead to inconsistency of performance, reduced efficiency, or—ultimately—failure. The actual results will depend on each individual design requirements, hence the need to correctly assess which type of thermal management is most suited to your application.

5. How do I know if my thermal management is a success?

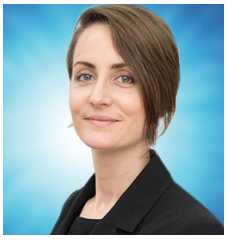
You will know your thermal management process has been successful when you see improved efficiency of heat transfer, reduced thermal resistance, and a lower temperature observed around the heat-generating component/device. Effectively, the chosen thermal dissipation media will operate within the temperature limits defined for the device, whilst maintaining performance during changeable conditions.

Conclusion

Efficient thermal management is an essential part of both modern and future electronics design, particularly given the ongoing trend for product miniaturisation coupled with higher-powered devices.

With so many different types of TIM at your disposal, it can be overwhelming to determine the correct thermal interface material for your application. But by careful consideration and identifying the correct test regimes, it is possible to differentiate between products and select the most suitable material for your application.

At Electrolube, we regularly assist customers with thermal interface questions to help them maximise their heat transfer efficiency. If you are in any doubt, contact your material vendor or us. I hope this month's top tips for TIMs have been informative. Please watch for my next column where I will examine thermal management issues further. **DESIGN007**



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

Bosch Develops Rapid Test for COVID-19

The coronavirus SARS-CoV-2 is posing major challenges for healthcare systems and medical institutions worldwide. An ability to rapidly diagnose the virus is of invaluable help in curbing its exponential spread in many countries. Bosch's new, fully automated rapid test for COVID-19 can help medical facilities such as doctors' offices, hospitals, laboratories, and health centers make fast diagnoses. The rapid molecular diagnostic test runs on the Vivalytic analysis device from Bosch Healthcare Solutions.

Developed in just six weeks, the rapid test can detect a SARS-CoV-2 coronavirus infection in patients in under two and a half hours–measured from the time the sample is taken to the time the result arrives. Another advantage of the rapid test is that it can be performed directly at the point of care. It also means patients quickly gain certainty about their state of health while allowing infected individuals to be identified and isolated immediately. With the tests currently in use, patients must usually wait one to two days for a result.

A Bosch Vivalytic analyzer can perform up to ten tests in the space of 24 hours. This means it takes just 100 devices to evaluate up to 1,000 tests per day. Given the dynamic spread of the coronavirus SARS-CoV-2, laboratories are already working beyond capacity. The Bosch Vivalytic will thus help to increase available testing capacities.

(Source: Bosch)

Counseling Startups From Design Through NPI

Interview by Andy Shaughnessy and Happy Holden
I-CONNECT007

Andy Shaughnessy and Happy Holden sat down for an interview with Richard Marshall, CEO of Xitex Limited, a company based in the U.K. that provides consultancy to startups launching wired and wirelessly connected products. Richard explains some of the most common factors companies overlook when introducing a new product. He also discusses one startup's new product—a Fitbit-type device worn by horses—and why he advises startups to "Make your mistakes close to home."

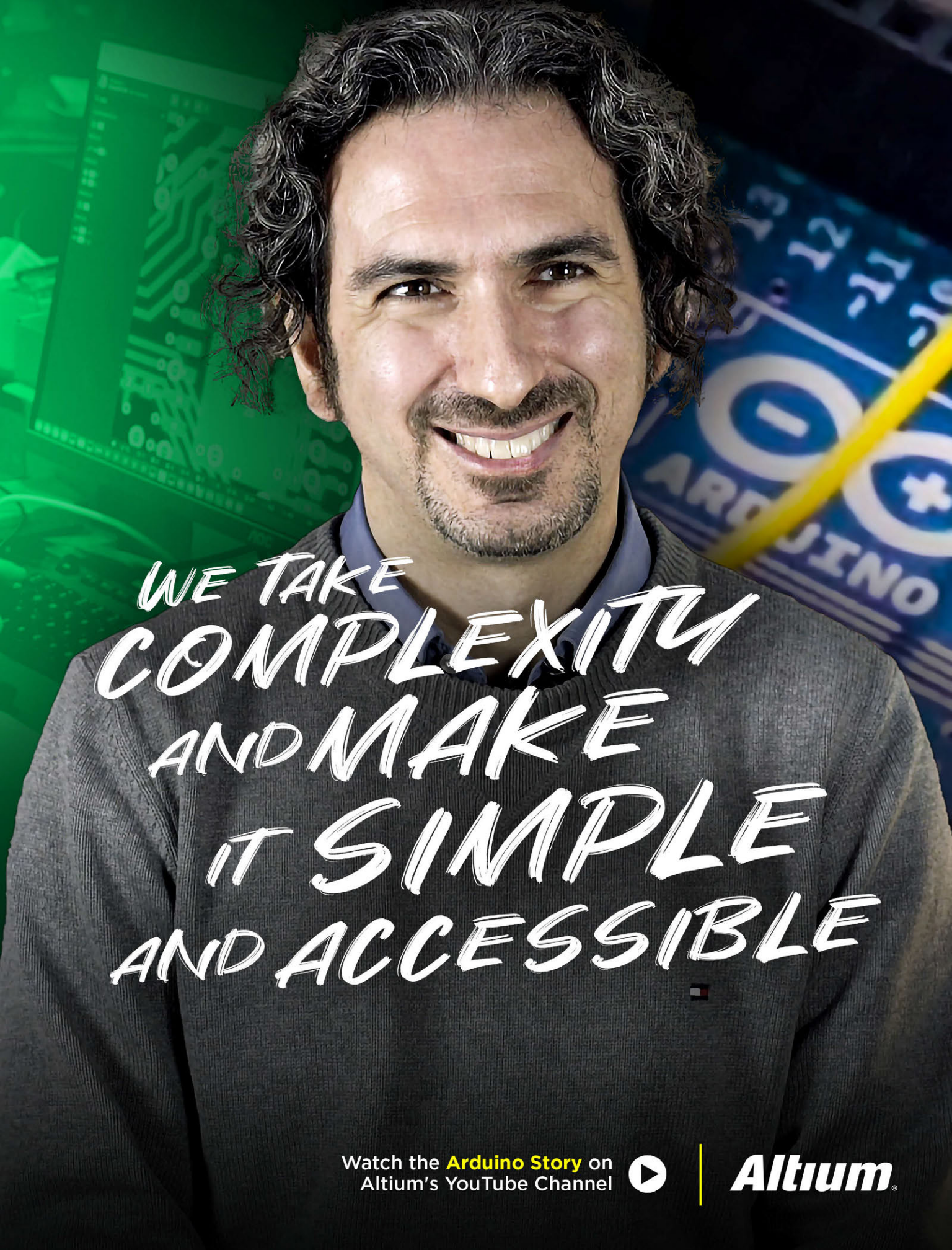
Andy Shaughnessy: How did you get involved in electronics?

Richard Marshall: As a kid, I grew up having various bits of electronics around. You don't realize until later in life how much you learn from your own father. I grew up with that stuff, soldering and knowing resistor color codes, long before I studied physics.

Shaughnessy: As CEO, give us some background on Xitex.

Marshall: Xitex has been going for about 25 years. It's my consulting company that I've been in and out of for a chunk of time. My last major startup was Ubiquisys, which was bought by Cisco in 2013. They were a small cell company, taking a 3G base station and miniaturizing it into a consumer product. We did one of the world's first femtocells, which we started working on back in 2006. Then, it went for the usual acquisition phase with Cisco. I came out of Cisco in the backend of 2014, and since then, I've been running my design consultancy.

Partly, I've always enjoyed startups because you have to be able to turn your hand to do a lot of things. You have to get things done. As I often say to founders, execution is king. If you want to convince investors to fund you and back you, then execution is king. The moment you don't start fulfilling your promises, then your business is likely to stop. There's also a feeling of wanting to give


WE TAKE
COMPLEXITY
AND MAKE
IT SIMPLE
AND ACCESSIBLE
Watch the Arduino Story on
Altium's YouTube Channel
Altium


Richard Marshall

something back in life, mentoring and helping other startups.

A couple of years ago, I was involved in a startup event in the U.K. and was talking to a couple of the founders, who were at Hive, which was bought by British Gas. One of the founders mentioned how hard it was to find people who had been through NPI in OEM manufacturing. And of course, for those of us who have worked for OEMs with our own production facilities, we tend to forget the fact that the world has changed so dramatically since 2000–2005, when so much manufacturing was first outsourced. Obviously, the majority of electronics have tended to be outsourced to Southeast Asia.

For the designers coming up in the last 10 years, it's quite a challenge. When the manufacturing is on the other side of the world—particularly for anybody who's going for a bigger scale—it's that much more of a challenge to learn. Of course, we see one or two startups in startup incubators in London, where they have spent months and months in Southeast Asia to get a product up. A lot of that is not through any lack of enthusiasm or effort; it's because they're going up a very steep learning curve in a very different culture.

Shaughnessy: It sounds like helping startups is a real vocation for you.

Marshall: Absolutely, and one of the things is the fact that there has been this idea that you can't manufacture things in Western Europe. I think there is something of a parallel to that in the U.S. Yes, you can still get things manufactured in the U.S.; however, I often say to people—particularly if you're going into volume manufacturing—you must have a plan to go to a low-cost area of manufacturing, whether that's Central or Western Europe or Southeast Asia. It's a lot easier to make your mistakes closer to home and be able to see where things are going wrong.

Shaughnessy: Because you're going to make mistakes if you're a startup, it's better to fail nearby.

Marshall: Right. Is it all going to go perfectly? That is part of the challenge, but you can mitigate so much risk by being able to do those things locally. We talked about this at the AltiumLive 2019 conference, often on the PCB side. Have you been to the fabrication plant? Have you talked to the fabricator? When you come to build the products, it's exactly the same thing, if you can't stand on a production line and see the product going together. There is a bunch of things that no amount of emails and conference calls will ever tell you that you will see within a minute of standing on a production line.

Happy Holden: That's an old paradigm. A smart factory in North America manufactures, at half the price of China, the perfect PCB in two days, for any quantity, from one to 100,000. You don't have to go to Southeast Asia to get the best quality and the lowest price at any volume.

Marshall: It's a really good point. If you look at it, it's not a one-sided thing; it's almost like a seesaw. We put robotics in factories. Then, due to a whole bunch of reasons, manufacturers started to migrate out to China. Typical-

you were on very manual lines without much automation. Back then, Foxconn announced that they were going to have to put automation in because labor costs had come up. It's not going to be about who can automate; it's going to be about who can start making those inroads into some of the other parts of the process.

Holden: Do you know what's funny about your quote? I was the CTO of Foxconn from 2010–2013 who said that (laughs).

Marshall: Forgive me! I didn't realize that (laughs).

Holden: My quote was, "Foxconn is now going to have to automate because of the rising costs." I worked as the number three man below Terry Gou, the chairman of Foxconn.

Shaughnessy: As you're counseling these startup companies, that's one of the things that you would impress upon them.

Marshall: Absolutely. The other area we get quite heavily involved in, and where I have another one of my ex-Sony colleagues involved, is the regulatory side. Particularly in the startup world, we've all seen the Kickstarter project that says, "We're going to ship in three months. All we have to do is do regulatory." Then, you see the project log go on for nine months or a year, and there's a big long tale of woe. One of the things we always say to people is, "You need to think through the regulatory work upfront because it will save you so much time later."

I've been working with a U.K. startup, Trackener, that is doing the equivalent of a horse Fitbit. It has thrown up some interesting challenges, starting with learning about horses. It is a small startup. They have done an enormous amount on a very constrained set of resources and a good proof of concept, which they designed in CircuitStudio. Then, we took the product and started to redesign it in certain places.

At the "proof of concept" stage, it started out with a lithium battery, and our take was that a lithium battery in a product you might strap to a very expensive racehorse might not be a great plan. There are a few things that could go wrong there, so we have helped them migrate to a nickel-metal hydride one. It isn't sexy, but it's inert. Obviously, there are issues where you have a chunk of metal close to a set of antennas. We had a form factor we had to meet, but the first thing to do was to get the antennas in and work out how the rest of the electronics and case were going to fit around the antennas and be able to perform efficiently.

The device goes in a harness between the forelegs of the horse in its chest region. There is also a strap, which goes around the girth, with a heart rate sensor and motion tracking as well. Now, all of the core technology data processing the startup developed themselves in-house. They have done an astonishing amount of development, with a very small amount of resources, but we came in and supported them on the hardware side.

Holden: How will this benefit the user, the horse, or the jockey?

Marshall: Sadly, I'm not the best person to ask, but some of the veterinary colleges, along with researchers and leisure owners, are using the technology. In the horse-racing world, there's a lot of interest similar to human athletic performance. Through the device, you can understand rest heart rate, peak heart rate, motion, etc. Overall, there is a lot of work going on in the veterinary world that is way beyond my comprehension. In a way, it's a great case study because it brings together battery safety and how to put a metal battery in close proximity to a GPS and GSM antenna.

Holden: Why do you have to start with the antenna first?

Marshall: It is about arranging the antenna in the right parts of the design because it's a patch antenna on the board. We looked at off-the-shelf antennas, but the problem was there were so many tradeoffs, as well as the diffi-

culty of matching them as opposed to doing an F-type patch PCB antenna, in this case. We can tune that for the exact case and battery arrangement. On the assembly side, the other issue is to get good isolation between the cellular module and the GPS module. We have also designed another patch antenna for the GPS transceiver.

Holden: Does this have applications for humans?

Marshall: Analog Devices has been doing a lot of work in this area, and Apple is way ahead when it comes to the Apple Watch.

Holden: A little known fact is that the Australians and New Zealanders developed tracking electronics that animals swallow and stays in their guts. They are one of the biggest electronics manufacturers in the world in terms of volume, but these allow them to track all of their animals, such as sheep. I was flabbergasted because I kind of never heard of that before. These things are sitting in these animals' stomachs, but it allows them to electronically monitor health.

My wife is a horse fancier, and the one thing that interests me is watching her rub down the horse. Horses sweat enormously, so I'm sure that needs to be taken into consideration to make this thing functional.

Marshall: I have not been involved with the device's harness design, but from some brief conversations, I've gathered that getting the harnesses right takes a fair amount of effort.

Shaughnessy: The horse might not like you putting the device on, and it has to be higher reliability and robust to deal with shock and vibration.

Marshall: It is not in full production yet, and the outlier is finalizing the mechanical design. I've learned that there's a huge tolerance with horses. You could be talking about anything from 250 kilos to 750 kilos in weight. I have a few pictures of very early cases that came back from trials where it was obvious that it did not quite go to plan, and we ended up with some cracked cases.

Again, it is an engineering story, which is, "Do you understand the problem?" One of the challenges my mechanical colleague has had is that, because you are putting it on an animal, you are trying to get a compromise between something that is robust enough but also has a certain degree of give. At the moment, we're running on an HP material, and some of the printing is done in Germany. It is obvious that you can use many different materials, such as glass filled or Kevlar, but the problem is if you make it too rigid, you run the risk that whatever it is could be too brittle. Then, you have a similar problem if it suddenly starts breaking up with nasty shards on it.

Shaughnessy: How long have you been working with this group on that?

Marshall: About 18 months.

Shaughnessy: And they reached you through one of these incubators?

Marshall: It was through one of the startup meetups in London. We're probably the majority of the way through the journey, but there's been some interesting stuff on the way, which is why—in the end—you have that "throwaway" conversation. Listening to some of the conversations led to some points that will hopefully help other designers, particularly those who are not so familiar with radio. I've spent a large portion of my career, not as a radio engineer, but usually on the baseband engineer butting up to go to the RF side.

Holden: Are there particular radio frequencies that you're restricted to for an application like this?

Marshall: This one is classic licensed spectrum cellular radio. The current product is 2G/3G, but it would not be that difficult then to take it onto 5G frequencies. If you look at them, they're all pretty much the same. The big distinction for 5G is you come down to 700 MHz instead of 850 MHz. You're operating from 700 MHz to over 900 MHz, and then you work from about 1,750 MHz to over 2,100 MHz, so there is some antenna tweaking to do.

Shaughnessy: Is that UHF or VHF?

Marshall: It's all UHF. We have been involved with things like Bluetooth and some of the high-frequency stuff like DSRC up at 5.9 GHz. One of the other things we'll be talking about is picking a PCB laminate. Don't tell me what Dk is at 100 megahertz; I want to know what it is all the way across the whole frequency of operation.

Holden: As you brought up earlier, you are going to visit your fabricator and bring them in early in the project.

Marshall: In this particular case, that was absolutely essential, we had a number of long conversations with the fabricator, such as what sort of stackup? Then, we have my antenna colleague, saying, "We are putting the antenna on the PCB, so we don't want too thin of a substrate because we don't want to create something that's too lossy or capacitive." You need to understand the materials you are using, and you need to understand the people who are using those materials and how that it is all going to come together.

You need to understand the materials you are using, and you need to understand the people who are using those materials and how that it is all going to come together.

Shaughnessy: And it's a regular FR-4 board?

Marshall: That is why we have had this conversation already. One of the questions often posed is, "What's your board designed with?" If somebody says to me FR-4, I say, "That's nice;" there are thousands of them, so you have to be quite specific. Again, when looking at materials, if there is particular local support from the laminate vendor, it can help you so much. They know their material, and they can give you lots of helpful advice, which can save you a lot of pain later.

Holden: Will this be completed by next year at this time?

Marshall: I hope so.

Shaughnessy: It has been great talking with you, Richard. Thank you.

Marshall: Thank you, Andy and Happy. DESIGN007

When Your Fabricator Is Late

Interview by the I-Connect007 Team

The I-Connect007 Editorial Team recently had a wide-ranging discussion with John Watson, CID, of Legrand. Questions covered include, "What happens when your fabricator is late, whether it's a prototype or volume production?" and, "What are the costs and ramifications up and down the chain?"


John Watson

Andy Shaughnessy: You and I have spoken before about time to market, and how we're all battling time. Let's talk about what happens when one cog in the system is late. What are the costs?

John Watson: Time is the big issue that I hear about. The VPs at my company say, "We need to decrease our time to market," because they see our competitors. The lead dog always gets that biggest piece of the pie if they can get out there with their product first.

Barry Matties: To that point, when you're working with your fabricator, you place the order, and they give you a delivery date that then goes one, two, or three days late. What happens if they miss it by one day?

Watson: That has a significant impact.

Matties: From a designer's point of view, what does that do to you?

Watson: It puts us in a place where you can't put a number on it for how much market share you've lost. We have a lot of external things involved in our design process that throw monkey wrenches into it, such as tariffs because we do work with China. One of our places is in China, so we have some outside influences on our designs, but we try to keep it to a consistent schedule. We identify what we call "blockers" in what we're trying to accomplish, meaning things that are blocking us both internally and externally. We need to identify those blockers and get them out of our way because those can constantly be problems. I would talk to the fabrication house about potential issues.

Happy Holden: The first thing you learn as a young engineer at Hewlett-Packard is break-even time. It's like ROI, but for designers. It came about because if design managers hit obstacles and they're going to be late, they go to management and say, "I need more resources," and managers say no. Lo and behold, they are late. Hewlett-Packard made money off of being the first to market. Then, Stanford MBAs used Professor William Ireson's idea of break-even time, which the financial guys understood. Break-even time is when the profits have paid for all the R&D development money, so it's not a return on investment. After the break-even time, you're truly making a profit.

Matties: You had your schedule.

Holden: And if they were too late, they would pay us more.

Matties: Do they agree to that in advance?

Holden: Yes, before they ever took the orders. If they didn't, they didn't get any orders.

To read this entire interview, which appeared in the April 2020 issue of *SMT007 Magazine*, click here.

Create Comprehensive PCB Documentation to Drive Fabrication, Assembly and Inspection

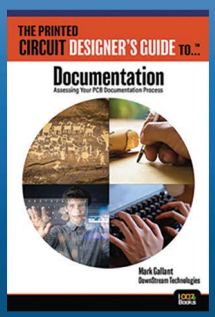
BluePrint-PCB®

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

MilAero007 Highlights

Burt Rutan's Keynote: SpaceShipOne ▸

In this video clip from his presentation, Burt Rutan discusses some of his aircraft's revolutionary flights that drew the most public attention, including SpaceShipOne, which flew three of the five manned space flights launched by man in 2004. He also explains the benefits of working with Microsoft's Paul Allen, who agreed to fund SpaceShipOne based on a simple handshake.

Defense Speak Interpreted: Be Prepared for CMMC ▸

If you are a current or future Defense Department contractor or subcontractor, you need to be prepared for the next cybersecurity requirements coming online during 2020. This is the Cybersecurity Maturity Model Certification, or CMMC, in Defense speak. Dennis Fritz explains how there will be five levels of cybersecurity requirements for various amounts of Controlled Unclassified Information (CUI) you handle, with increasing requirements from one (least) to five (most).

FLIR Inks $23M USAF Order for Centaur Unmanned Ground Vehicles ▸

FLIR Systems announced that the United States Air Force ordered more than 180 of the company's Centaur unmanned ground vehicles (UGV), plus spares. The $23 million contract is sourced through the Dept. of Defense Man Transportable Robotic System Increment II (MTRS Inc II) program.

Boeing Reveals its U.S. Army Future Attack Reconnaissance Aircraft Design ▸

Boeing is offering the U.S. Army an agile, fully integrated, purpose-built system for the Future Attack Reconnaissance Aircraft (FARA) prototype competition.

What It Takes to Be a Milaero Supplier, Part 2 ▸

The decision to pursue military and aerospace (milaero) certification impacts every facet of the organization, and not every shop is prepared to make this transformation. In Part 2, Anaya Vardya focuses on what it takes to be a milaero supplier in the areas of engineering and CAM.

Lockheed Martin's HELIOS Laser Weapon System Takes Step Toward Ship Integration ▸

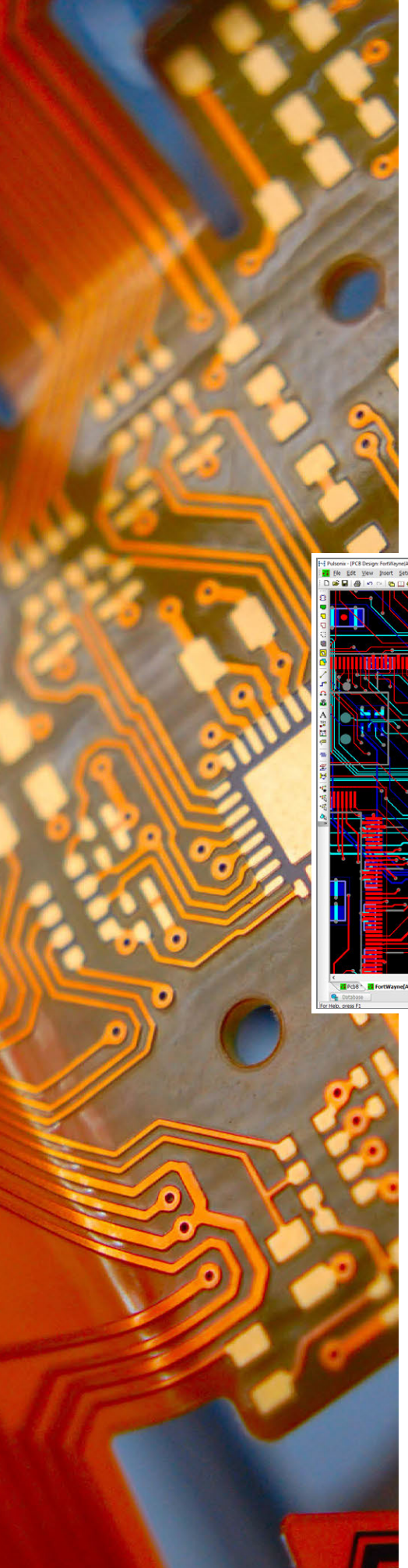
Lockheed Martin and the U.S. Navy moved one step closer to integrating a laser weapon system onto an Arleigh Burke destroyer after successfully conducting a Critical Design Review (CDR) for the High Energy Laser with Integrated Optical-dazzler and Surveillance (HELIOS) system.

Catching up With Midstate Electronics ▸

Dan Beaulieu recently sat down with Susan Matteo, Joan Allen, and Gerri Wooten from Midstate Electronics' sales team to discuss their company, how long they have been in business, the key ingredients to their longevity and success, and how they see the market today in general. Midstate offers everything from multilayer PCBs to flex and rigid-flex circuits and PCB assembly.

BAE Systems Secures $188 Million Contract for U.S. Navy's AEGIS Combat System ▸

BAE Systems Inc. was awarded a five-year $188.2 million contract to provide the U.S. Navy's AEGIS Technical Representative (AEGIS TECHREP) organization with critical large-scale system engineering, integration, and testing expertise for the AEGIS Weapons and Combat Systems aboard U.S. Navy surface combatant ships.

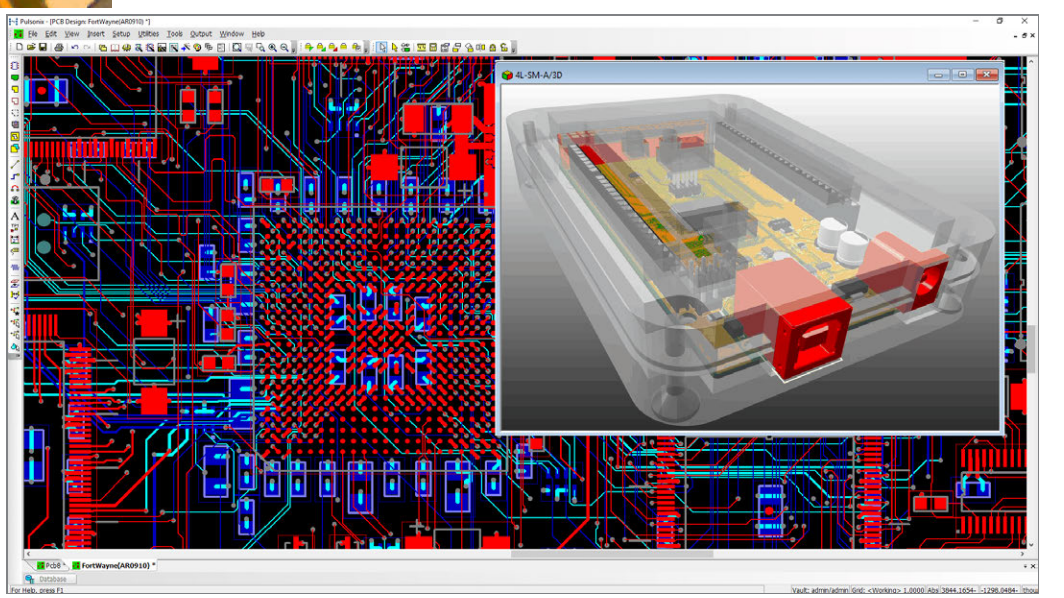


Pulsonix PCB Design Software

PCB design done right

Maximum Productivity in Minimal Time

With a modern, easy-to-use interface and advanced capabilities to automate your layout process, Pulsonix can be the key to a critical reduction in your design time. That's PCB design done right!



Advanced PCB Design Features

- Intuitive placement and routing
- Dynamic copper pour
- Full rules engine for automatic DRC
- High-speed design rules
- Heads-up display length matching
- Rigid-Flex, embedded components, and chip-on-board
- Revision control and PLM integration

Try Pulsonix yourself at pulsonix.com/trial

PULSONIX

FLEX007
A SPECIAL DESIGN007 MAGAZINE SECTION

Mina: Enabling Soldering to Aluminum

Flex Talk

by Tara Dunn, OMNI PCB

Averatek recently launched Mina™, a chemistry that offers exciting benefits over traditional methods of soldering to aluminum. I had the opportunity to speak with Divyakant Kadiwala, director of manufacturing at Averatek, to learn more.

Tara Dunn: Divyakant, I am excited to learn more about Mina™. Before we start, could you introduce Averatek and your role there?

Divyakant Kadiwala: Averatek is a high-tech company based in Santa Clara, California. It was founded by SRI International and private investors. It has two primary products: LMI™, a catalytic ink that enables the fabrication of very high-density circuits with the patented A-SAP™ process, and Mina™, a surface treatment that enables soldering to aluminum. I am the director of manufacturing, and my role includes overseeing process engineering, quality control, facilities management, and business development. Lately, I have been concentrating on promoting Mina™.

Dunn: It sounds like you are busy. I am sure it is exciting to be working with new product development. Can you please explain Mina™ for us?

Divyakant Kadiwala

Kadiwala: Absolutely. Mina™ is a unique surface treatment that enables soldering to aluminum just as is done to copper and even at low temperatures. The reason this is unique is that, normally, there is an oxide that is present on all aluminum surfaces that inhibits solders from bonding with the core metal. To tackle this oxide, processes exist that involve a series of etchants and treatments that remove this oxide. This is commonly called the zincate process. Once the oxide is removed, the surface is plated with a noble metal finish like ENIG or ENEPIG to cover the surface and protect it from forming a new oxide layer.

Mina™ eliminates the need for all the wet chemistry and surface finishes. It is printed onto the pads using a stencil printer and cured at low temperatures. Cured Mina™ is non-conductive, so this leaves the pads with a uniform coat of Mina™. This helps with easy print registration, should the trace and space of a component's pads be too tight. After this, the boards

THINK SMALL

DEFENSE **MEDICAL** **AEROSPACE**

Flex circuits for those who value low TCO

The right flex circuits make a huge difference in your design, on the assembly line and in the field. Whether it's a basic circuit or something more advanced, Minco engineers can help you minimize costs while improving overall efficiency.

Some of our best designs integrate flex circuits, heaters and sensors. When you engage Minco engineers early in the design process, they often find efficiencies that minimize costs, shorten assembly time and save space. Every day, Minco engineers work in high-reliability industries to help our customers achieve the most efficient and reliable solutions.

Email us today for more information at info@minco.com

MINCO

7300 Commerce Lane NE | Minneapolis, MN 55432

minco.com

can be stored to prepare for assembly because cured Mina™ remains stable, or the boards can go straight through the component assembly process wherein solder will be printed onto the pads, followed by loading of components in the pick-and-place and then reflow. Mina™ activates during the reflow cycle and ensures a good solder joint to the core aluminum.

The unique thing is that the solder selection depends on the substrate. A common use of aluminum is associated with aluminum on PET substrates or Al-PET substrate. PET cannot withstand high temperatures for an extended time, so low temperatures solders become the only option.

Mina™ activates during the reflow cycle and ensures a good solder joint to the core aluminum.

Dunn: This appears to significantly simplify the current standard process. Are there other benefits of Mina™ that we should be aware of?

Kadiwala: Yes, there are several benefits to using Mina™. It is green because it eliminates several wet chemistry process steps that would otherwise be needed to solder to an aluminum pad. It also widens the scope for the use of Al-PET substrates. These are often limited to inexpensive applications like RFIDs that are assembled using silver epoxy. With the ability to solder components, Al-PET substrates can now replace single layer copper-based circuits for applications like smart tags, LEDs, and other applications. Overall, aluminum is about one-third of the cost of copper, and when combined with the cost-benefit of PET film over polyimide, it is one-eighth the cost of copper on a usage basis. This is a significant cost savings. From a performance perspective, soldered joints are electrically superior and more reliable than those made using conductive epoxies.

Dunn: What industry segments do you see as the early adopters of this technology?

Kadiwala: From our conversations, I expect that the early adopters will be those industries that are already using Al-PET substrates. Again, these include smart tags, LEDs, automotive, RFIDs, and those using aluminum for EMI shielding and similar applications. Once these adopt, we will start targeting other conventional copper-based circuits that will be driven to aluminum for its cost advantages.

Dunn: Is Mina™ commercially available at this time?

Kadiwala: Yes, it is available now. It has been qualified for use by some customers and is under testing at others.

Dunn: What type of testing and development has been completed?

Kadiwala: The testing and development are always specific to the end application of the product. Mina™ has passed the surface insulation resistance (SIR) test. This was done at an independent lab per IPC specifications. We routinely do shear tests to confirm adhesion while qualifying new solders. We get shear values that are greater than 15 N/mm2, and typically, failure mode is between the aluminum and PET film. We also examine the joints using X-ray to examine for voids. The voids are consistently within expected norms of less than 30%. It has been tested and qualified for use in soldering to aluminum for an EMI shielding application. Further, it is undergoing testing for LEDs, smart tags, and automotive applications.

Dunn: Will there be any specialized equipment, or storage or handling procedures that will need to be addressed before bringing this into a manufacturing environment?

Kadiwala: Mina™ is very simple to use and requires no special handling other than common PPE per best practices when handling a

chemical-based product. These would include gloves, eye protection, aprons, etc. We recommend that it be stored at room temperature, away from direct sunlight. It does not require refrigeration. Thus, it is cost-effective to store when compared to silver epoxies that need freezers for storage.

Mina™ can be printed using conventional stencil printers and cured in reflow ovens that already exist in SMT assembly shops. Therefore, there is no need for any specialized equipment, storage, or handling constraints around bringing Mina™ into a manufacturing environment. This makes it very easy to adopt.

Dunn: How can people learn more?

Kadiwala: People can visit our website or email me at divyakant@averatek.com.

Dunn: Divyakant, thank you so much for talking with me and educating us about Mina™. The ability to solder directly to aluminum is certainly an exciting development with many benefits.

Kadiwala: Thank you. 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.

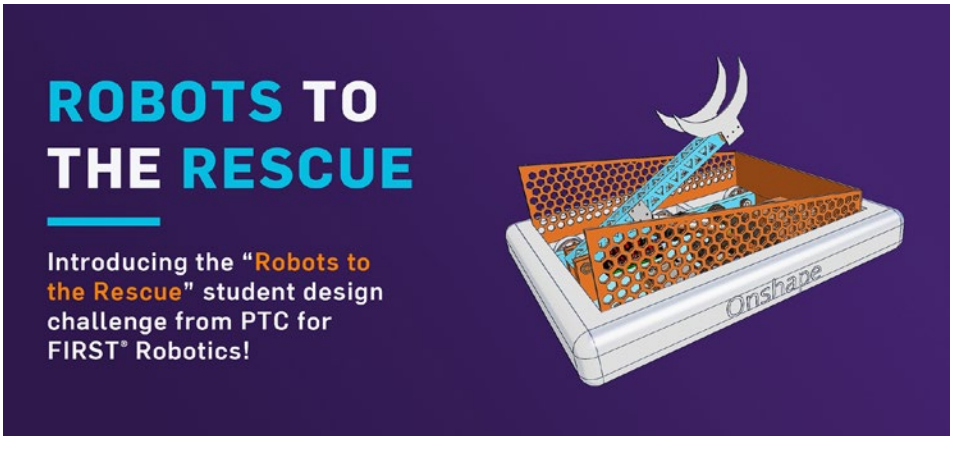
PTC to Sponsor Global Design Competition, "Robots to the Rescue"

In response to the suspended FIRST season caused by COVID-19, PTC is inviting FIRST teams around the world to compete virtually in its design competition, "Robots to the Rescue," in which teams will be challenged to design a robot that can solve a current real-world problem.

FIRST is a robotics community that prepares young people for the future. Each year, the global non-profit organization offers a suite of robotics competitions for students in grades PreK-12. In mid-March, amid school closures and expanded travel restrictions due to the COVID-19 outbreak, FIRST announced the suspension of its current FIRST season.

Students are invited to design and compete virtually in Robots to the Rescue using Onshape–the world's only pure Software-as-a-Service (SaaS) product development platform. Because Onshape is web-based, FIRST students can collaborate remotely while continuing to hone their engineering skills. As part of the six-week competition, FIRST teams–specifically FIRST Tech Challenge and FIRST Robotics Competition teams–will design a robot that can help solve a real-world problem, such as providing food and supplies to people in remote places, or removing pollution from the atmosphere.

The competition launches on Friday, April 3, 2020, with entries accepted through May 15. Students from around the world can register for the competition online here. For a complete list of rules and guidelines, please click here. Both the competition and the use of Onshape are free to students.



Rockwell Automation–also a proud sponsor of FIRST, and with which PTC has a robust strategic alliance–is also supporting the competition.

(Source: Business Wire)

Flex007 Highlights

With DIS, Accurate Registration is Everything ▸

In this video interview from IPC APEX EXPO, Pete Starkey and Jesse Ziomek, VP of sales for DIS, discuss how the company achieves ultimate accuracy in layer-to-layer registration, not just in rigid multilayers, but also in flex and rigid-flex builds. Jesse also comments on keeping technology exciting enough to attract young engineers into the industry.

Multi-board Etching: Managing Rigid-flex Designs and Conductivity ▸

Good troubleshooting techniques involve considering a system as individual parts rather than as a whole. The same techniques apply to your work with multi-board PCB designs. Each board consists of a single unit that has its own lifecycle. Some product designs may use a single PCB design for multiple functions or for multiple devices. Others may interconnect multiple PCB designs.

Lenthor Engineering Remains in Full Operational Mode During COVID-19 Advisories ▸

Lenthor Engineering Inc.—a California-based designer, manufacturer, and assembler of rigid-flex and flex PCBs—announced that the company is fully open for business and is ready and willing to support your requirements. This action ensures that Lenthor is able to continue to move products and continue to deliver against the vital needs of our customers.

Royal Flex Circuits Supports the Production of Ventilators for Medtronic ▸

Royal Flex Circuits in Santa Fe Springs, California, is uniquely positioned to help out in the crisis. Royal Flex Circuits manufactures rigid, rigid-flex, and flexible printed circuits. The company has an inventory of copper-clad cores and prepreg, allowing it to start production as soon as the design files arrived from ventilator manufacturers.

Advantages of Using ZIF Connectors as a Termination Method ▸

There are many types of connectors and termination methods available when designing a flexible circuit. One of the most common is the zero insertion force (ZIF) connector. The reason why the ZIF connector is so popular is that they eliminate the requirement for an added connector. They create a direct connection from the circuit to the mating connector reducing overall weight and cost. Here are a few general facts about ZIF connectors.

Flexible Circuit Technologies (FCT) Appoints Edward "TJ" May to Director of Business Development ▸

Carey Burkett, vice president of business development for Flexible Circuit Technologies, announced the appointment of industry veteran TJ May to the position of director of business development.

APCT Now Offers Quick-turn Capabilities for Their Rigid-flex Product ▸

After a year-long expansion that has allowed our company to innovate our process and invest in new manufacturing equipment, along with the ongoing search and acquisition of high-end personnel, APCT Anaheim is now proud to announce that they are capable of offering quick-turn production for its rigid-flex product.

LENTHOR ENGINEERING

MAKING THINGS FLEXIBLE IN A RIGID WORLD®

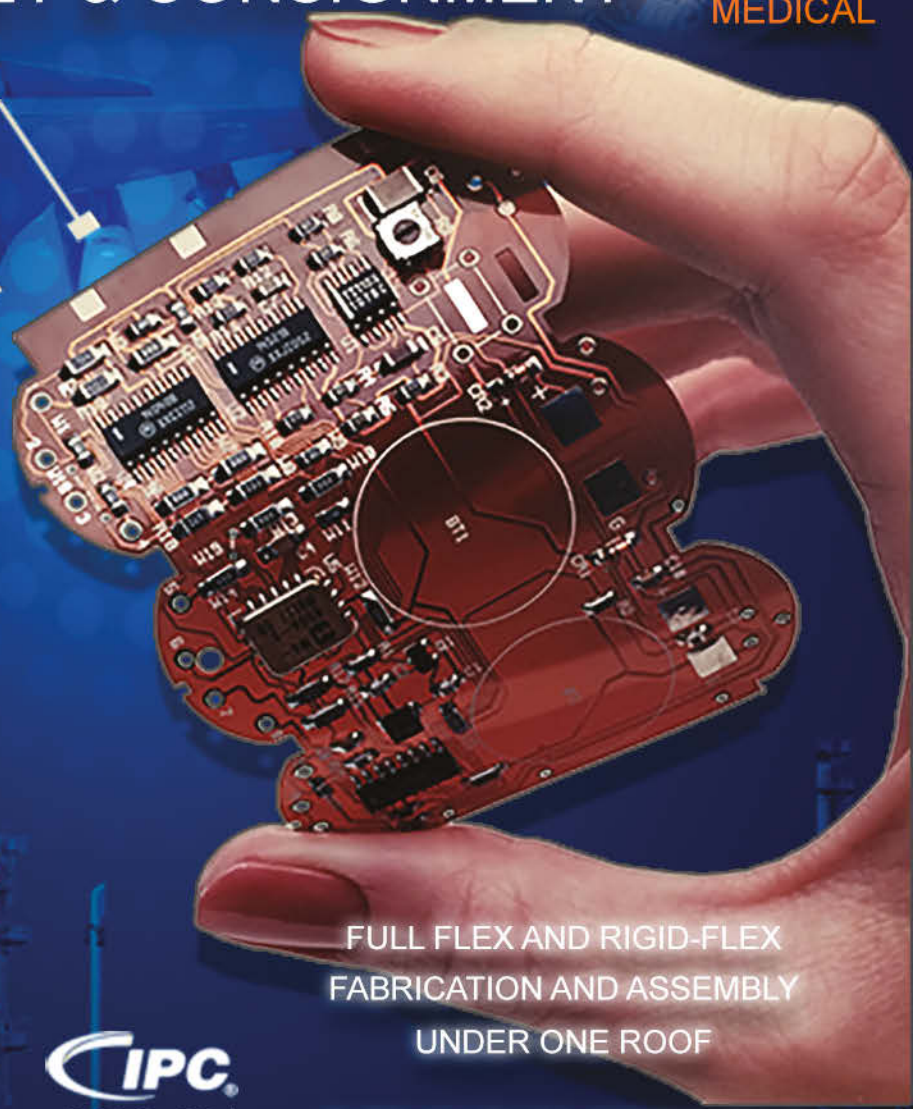
FLEXIBLE CIRCUITS ASSEMBLY

- 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

DFM or Design With Manufacturing?

Flexible Thinking

by Joe Fjelstad, VERDANT ELECTRONICS

The great Irish author, playwright, and humorist Oscar Wilde once defined a cynic as an individual who knows the price of everything and the value of almost nothing. Unfortunately, over the decades, that same analysis could often be applied to procurement agents in electronic product companies around the globe. The reward for a purchasing agent is too often derived not from getting the best solution for their company but the best price. It is a simple fact that if the cost of a decision is not tracked through the entire manufacturing system, the dollar saved at incoming could cost the company much more at the end through process adjustments, rejects, rework, repair and, much worse, through product failure in the field and returns.

In earlier times, many companies were run by accountants (AKA bean counters) who simply ran the numbers with expectations that profit could be assured by controlling the cost of raw materials in the door. If the sales exceeded the cost of the raw materials, labor, and overhead, there was profit, and everything was good. However, in a globally competitive business environment, one can quickly lose position and market share to those competitors more attentive to inefficiencies in their operations and skilled at "cutting away the fat." One of the important ways they have accomplished this is by giving manufacturing technicians and engineers a greater voice in manufacturing management. Those closest to the problem are typically those who understand it best. No one truly wants to make inferior products or contribute to waste.

However, manufacturing is limited in terms of what it can do in many situations. Design is the drum major of any manufacturing march-

SPECIALTY CHEMISTRY DESIGNED FOR SPECIAL MATERIAL NEEDS

Technology. Service. Innovation.

We have a full range of solutions for your flex circuitry needs.

SHADOW DIRECT METALLIZATION

Superior copper to copper bonding on a wide range of materials with reduced process waste

VIA DEP 4550 LOW STRESS ELECTROLESS COPPER

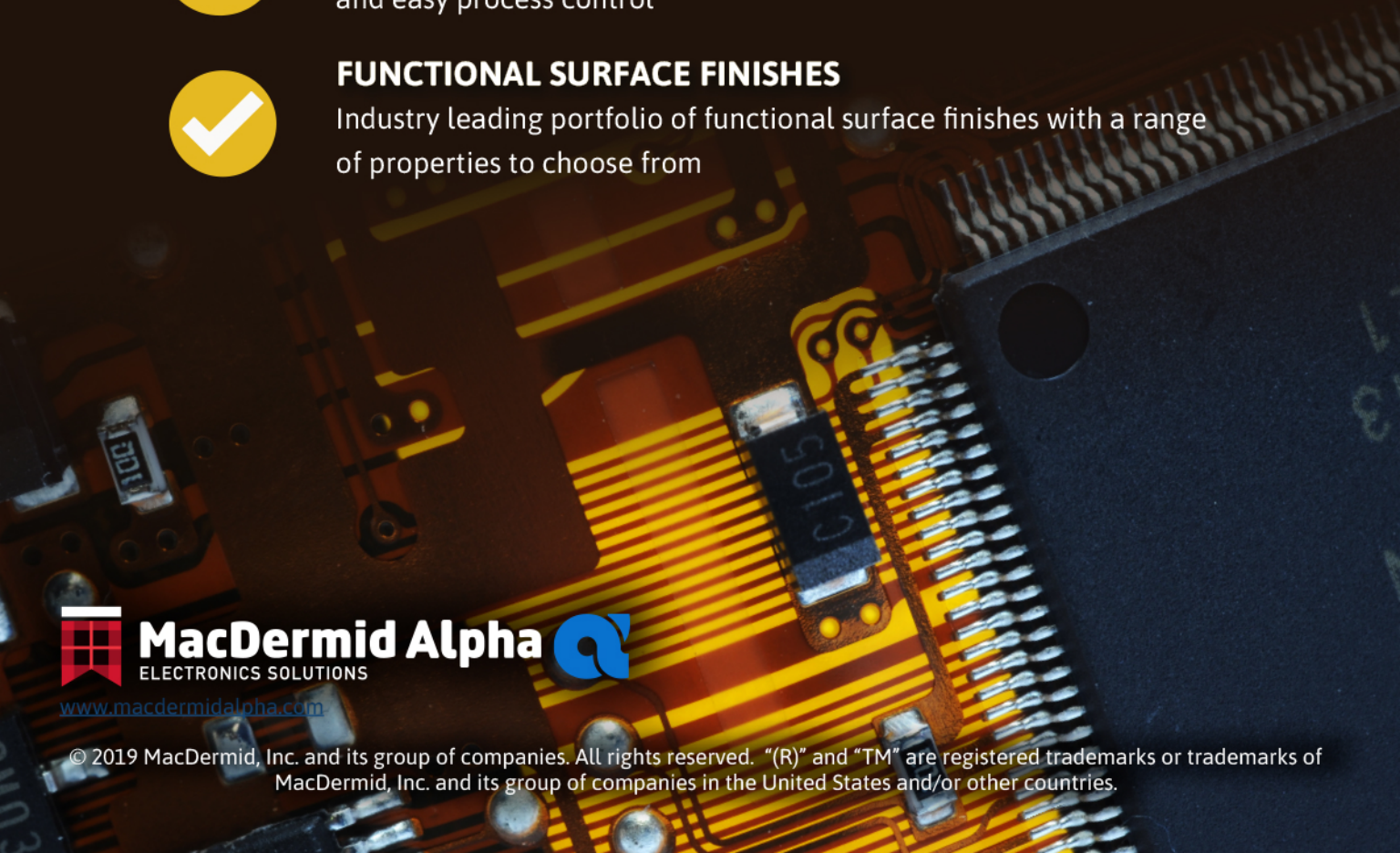
For applications that require exceptional adhesion and bend performance

MACUSPEC ELECTROLYTIC COPPER

Portfolio of copper plating options with excellent physical properties and easy process control

FUNCTIONAL SURFACE FINISHES

Industry leading portfolio of functional surface finishes with a range of properties to choose from



MacDermid Alpha ELECTRONICS SOLUTIONS

www.macdermidalpha.com

ing band regardless of the end-product. The decisions made by designers are critical to success. The phrase "garbage in, garbage out" was coined by computer programmers, but it can be applied to design as well. It is simply not possible to make a 3D model of an Escher drawing (though it is entertaining to think about).

Thus, one important prescription for increasing profitability must include the education of designers to the realities of manufacturing, including assembly. In this regard, truly skilled designers are increasingly designing *with* manufacturing rather than designing *for* manufacturing (DFM).

In the past, major OEMs were vertically integrated. In such environments, designers typically had a chance to interface internally with manufacturing engineers to check in advance and see if what they wanted to do was possible. On many occasions, what the designer wanted wasn't really possible at the moment, but by simply asking the question, they caused manufacturing engineers to reconsider the challenge and often alter their thinking as to what was possible. In that regard, the interaction between designer and manufacturing engineer created a springboard for process innovation and improvement. Not every attempt to integrate into a new process was a success, but each one was a learning event. To quote another author and seminal thinker of the last century, T.S. Elliot, "Only those who will risk going too far can possibly find out how far one can go."

Going too far is not a bad thing. Stretching out our limits is key to making progress. I have been challenging others in the electronics manufacturing industry for over a quarter-century to think in new ways through my various startup activities. In 1990, at Extended Length Flex (ELF) Technologies, we created a roll-to-roll machine and process to direct print catalytic toner on a flexible material web and plate it with copper.

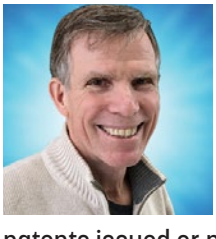
At Tessera in the mid-'90s, we developed the micro-BGA and fan-out wafer-level packaging concepts; in the process, we helped to usher in CSP technology. At Silicon Pipe in 2001, we re-envisioned electronic interconnection in the 3D dimension with over the top (OTT) interconnection of high-speed signals, novel 3D interconnections, and backplane structures that allowed us to demonstrate 40-Gbps data rates through two copper wires over a distance of a meter through two connectors. That was in October of 2003. All of those concepts have found or are now finding their way into products.

The Occam process, which I rolled out for consideration by the industry in 2007, is still waiting to get some consideration by designers and manufacturers who find themselves stymied by the limitations of current technology that relies on solder to make interconnections between components and circuits. Such an approach could prospectively save a substantial amount of money by reducing both the X-Y dimensions and circuit layer counts of the assemblies while integrating thermal management and increasing reliability. Fortunately, it appears that some fertile ground has finally been found where these ideas may be able to take root in the U.S. and in Europe. I look forward to keeping those with interest informed from time to time through my future shared thoughts and observations.

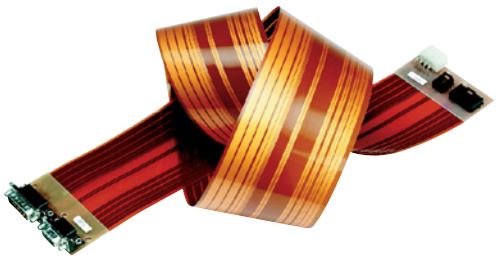
On a personal closing note, I would like to share an observation a dear and wise friend once shared with me: "One really only needs three things to enjoy a full life: something to do, something to look forward to, and friends." I am fortunate enough to have many in each category, and the only thing I can add to that list is good health, especially at this moment in history.

Stay safe and stay well! **FLEX007**

Editor's note: Click here to read a related interview titled "Joe Fjelstad Breaks Down His Occam Process."



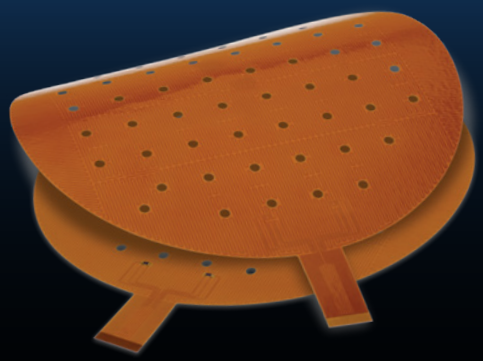
Joe Fjelstad is founder and CEO of Verdant Electronics and an international authority and innovator in the field of electronic interconnection and packaging technologies with more than 185 patents issued or pending. To read past columns or contact Fjelstad, click here.



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

DFM for Advanced Technology Rigid-flex

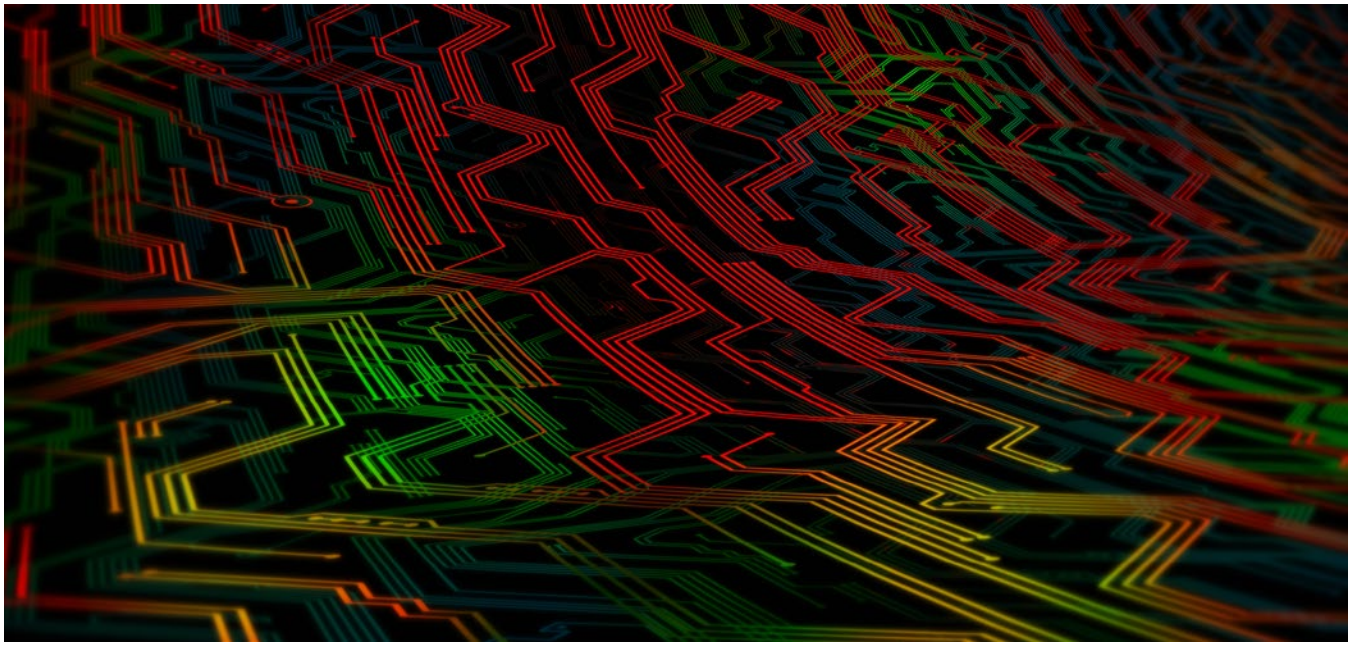
Flex007 Feature by MK Hicks
INFRONT AGENCY

Business as usual is gone. Exponential innovations are the norm. Why are industry leaders aggressively pursuing innovation at a record pace? It's simple—they must. All elements of the technology development chain are advancing at the same time to provide advanced chips, sensors, manufacturing, ever-advancing material and material production methods, and communication networks. How can design companies stay competitive in innovation advancement and gain market share? Collaboration is becoming the essential element that spurs continued and repeated success—especially in groundbreaking, new technologies.

Start With Advanced DFM

The competitive nature of technology—especially hardware—is requiring shorter design-to-commercialization cycles. If the initial concept can be designed with manufacturing efficiencies at its core, not only does this bode well for confirming the buildability of a design, but it also shortens the time to commercialization.

Designs are becoming all too sophisticated. PCB designers need to embrace working with advanced DFM engineers from the initial moment of the design concept. These are leading experts at the foundation of hardware design whose job it is to show up every day, stay informed of the latest materials and new processes, and be constantly challenged to deliver new capabilities. Expert DFM engineers thrive on applying advanced technologies and are driven to design them for the highest yield in the most essential and profitable way. DFM experts in rigid-flex are few, due to the unique experience required with upfront engineering and manufacturing capabilities such as details to put a build plan in place, perform test runs and proof-of-build. Relying on these experts is



THE NEXT BIG THING IS HERE

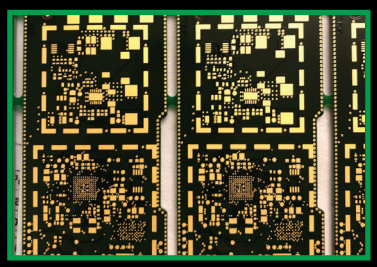
TOTAL ADDITIVE PROCESS

INKJET SOLDER MASK

SOLDER MASK

IJSR-4000 JM SERIES

- JMO2DG, JMO3G
  - Excellent Processing
  - Available in Multiple Colors
  - Available for Rigid and Flex



IJSR-4000 JM03G

LEGEND INK

IJR-4000 SERIES

- IJR-4000 FW100 / FW200
  - Flexible White Legend Ink
- IJR-4000 LW100
  - Rigid White Legend Ink Cured by LED
- IJR-4000 MW300 / MW301
  - Rigid White Legend Ink

IJR-4000 MW300

IJR-4000 FW100

TAIYO AMERICA

UNMATCHED INNOVATION

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

the key to continued collaborative success as technology races forward.

Who's Leading This Dance?

Today's cutting-edge technology is demanding 20+ layer rigid-flex, flex with stiffeners, rigid-flex HDI, and rigid-flex platforms by nearly all industries—from defense and aerospace to medical devices and commercial. The needs of these different industries once-segmented manufacturers and the technologies that each offered. But now, manufacturers who are becoming dominant solutions providers excel in collaboration as they ramp up to offer the same high level of technology, capabilities, reliability, quick-turn applications, testing, and appropriate certification across all industries.

Perhaps the delivery expectations that we have grown accustomed to in our personal lives are also influencing expectations in our business lives. However it started, shorter lead times and quick turn are becoming the norm, not the exception. Where once 40-, 50-, or 60-day lead times were typical for the defense and aerospace industries, for instance, the demand for decreased lead time continues down to 15 days or less. To meet speed demands, manufacturers are being challenged to create more efficient processes and to increase discipline by constantly monitoring key production KPI to meet demands without sacrificing quality or economics. They don't want shorter lead times or quick-turn to live in isolation but to be a companion with advanced technology solutions, reliability, and capacity.

Views From the Industry

We asked three technology leaders to provide their views of today's manufacturing challenges and changes.

Sherry Barone
CEO/Co-founder of ATTP

"The most challenging thing for the defense industry is the manufacturing process."

It takes a long time to develop the most sophisticated designs. The most critical determinant of development-to-deployment time is manufacturing. As an example, it can take up to three to six months to develop a design, which then may take a year or two to test, integrate, and potentially field test. Another challenge in manufacturing is for designs to become lighter and stronger, functionally advanced with capabilities that are modular. You want to have software that is adaptable for the smaller modules so that you can add on features as you continually advance the software applications.

The defense industry is highly focused on flex designs for mobile applications with 5G. Mobile technology is going to expand to enable more software applications and more data exchange at higher speed with signal integrity and enabling advanced cybersecurity. 5G is an energy alternative application that uses significantly less energy. The new 5G technology and current applications are projected to be a $35 billion industry over the next couple of years.

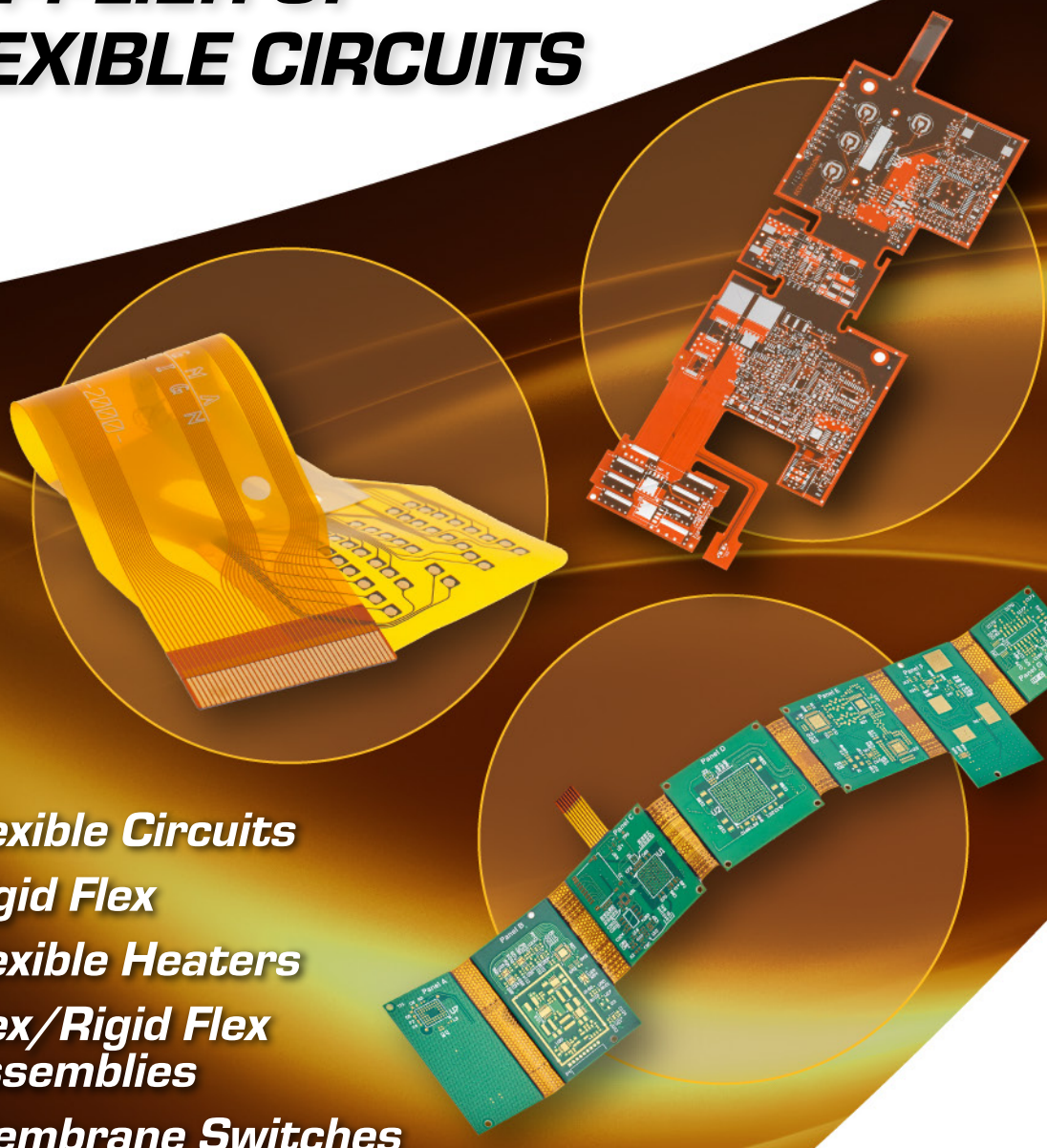
Flexible PCBs are going to be in high demand for the development of defense industry applications. Innovation in the PCB industry is going to need to get faster to keep up with a growing demand for product speeds. It is a unique opportunity for designers and manufacturers to come up with capabilities that offer high speeds, quicker and cheaper.

Steve Robinson
CEO/president of APCT

"We are committed to continually investing in the future."

In the last 18 months, we have been in conversations with our defense, aerospace, and medical industries customers, to name a few, listening to what they value the most from a North American supplier. All of our customers are driving their technologies to new levels, and in support, this has allowed us the flexibility to launch QTA and quick-turn, reduced cycle time offerings in the flex, rigid-flex with stiffeners, and rigid-flex platforms.

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

PCB manufacturers are needed to implement solutions with a combined suite of technology options—including rigid, rigid-flex, advanced materials, stiffeners, HDI, complex stack-ups, panelization, trace and space, HDI-specific laser microvias, stacked microvias, multilayer processing, sequential LAM processing, coverlays, overlays, and mylar—all while producing a more reliable product and mitigate costs.

In essence, we are making a huge shift in changing the legacy DNA of the PCB industry—such as the technology, service, engineering support, broad range of technologies, ability to be flexible and nimble, many manufacturing capability options, cycle time offerings, and delivery capability. All industries are feeling the same demands to catapult technology.

Andra Keay
Managing director of Silicon Valley Robotics

"Embracing new opportunities for radical collective change."

Many companies and manufacturers had too much invested in current processes to want to change things. But now, there's too much that they have invested in their businesses, and they can't afford not to change. That's the real difference right now. Until now, the cost of failure was too high. Today, the threat of failure for companies is so high that they need to become more innovative. I think that means that those companies that are providing suites of services to a variety of companies will do quite well.

The next trend is to have networks in place that can provide benefits for small operations that are immediately located in a whole range of different areas. There is a strong pull in technology to follow the example of retail and move toward microfulfillment, which comes in many different offerings. To stay competitive, companies know that they have to reshape manufacturing, change their business model to engage in innovative logistics, and support collective networks to engage in smaller operations—especially for emerging hardware. **FLEX007**

MK Hicks is the co-founder of inFRONT Agency. She is a hardware engineer turned international marketer, layered with finance.

EyeOn LifeLine Deployed to Hospital ICUs to Combat COVID-19

EyeTech Digital Systems has launched FDA-registered EyeOn LifeLine—a potentially life-saving eye-tracking communication tablet for hospital ERs and ICUs—so that doctors, nurses, and caregivers can easily and quickly communicate with their critical COVID-19 patients who are unable to speak while confined to their hospital beds.

The stand-mounted tablet faces the patient who simply looks at word squares to express pain level or a request, and the tablet voices it through speakers for nurses and doctors.

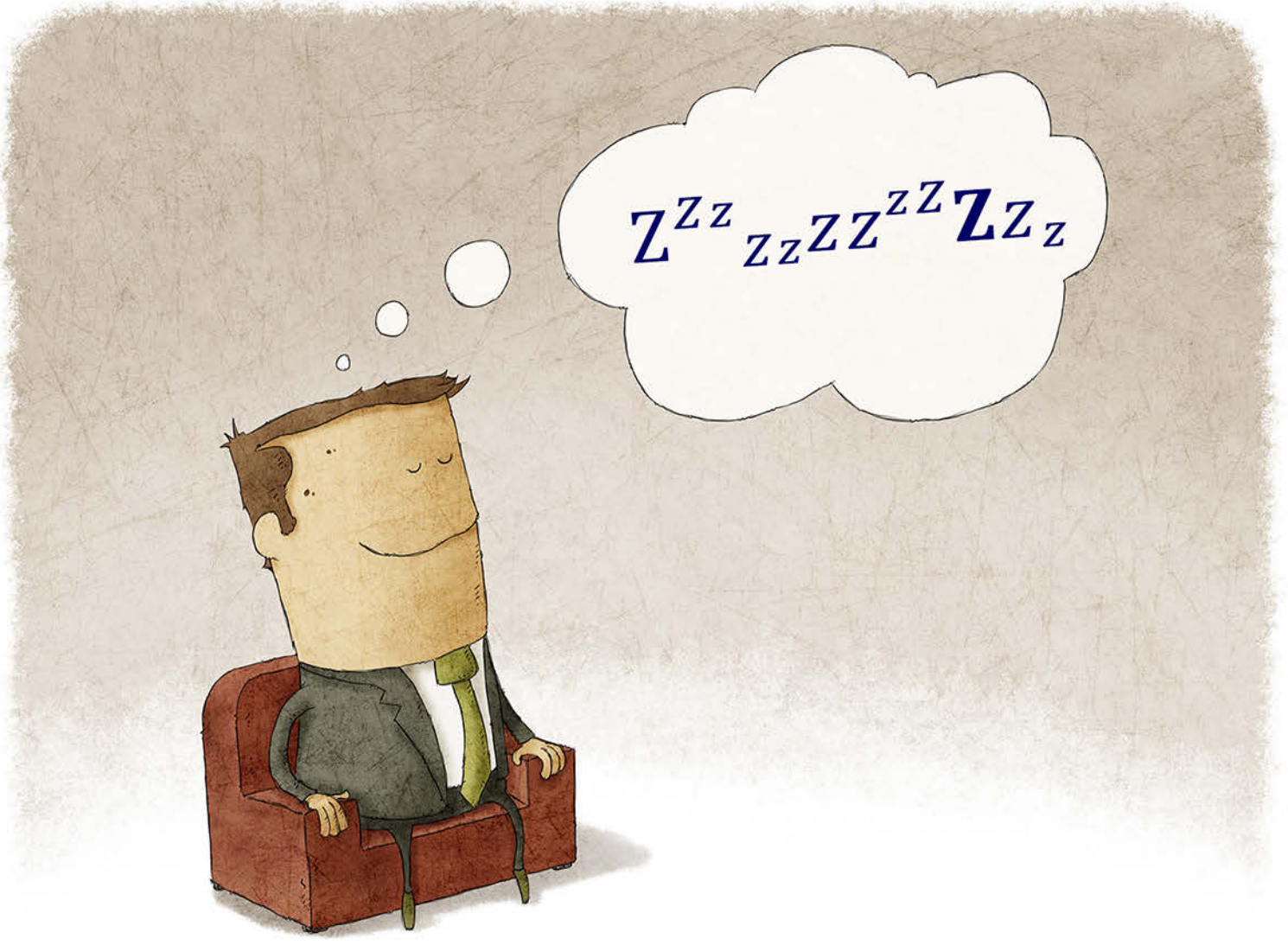
EyeOn LifeLine is ultra-portable, so it can be utilized with multiple patients during this pandemic to improve patient outcomes. EyeTech launched the EyeOn self-calibrating augmented and alternative communication (AAC) device earlier this year for people with ALS, cerebral palsy, muscular dystrophy, spinal cord injuries, Rett Syndrome, traumatic brain injuries or stroke, as well as those on the autism spectrum.

Image credit: EyeTech Digital Systems

"As a long-time eye-gaze user with hand disabilities, I was devastated to see doctors and nurses not being able to communicate with ICU patients under a ventilator," EyeTech CEO Robert Chappell said. "If we can save one life, it will be a huge contribution. I just wish we had enough units to serve all the ICUs around the country."

(Source: EyeTech Digital Systems)

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.

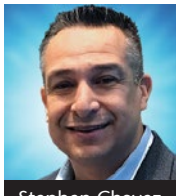
iCD Design Integrity: Intuitive software for high-speed PCB design.

"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

1 The Digital Layout: The New Printed Circuit Engineering Association ▶

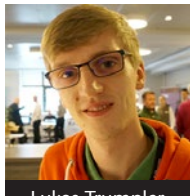

Stephen Chavez

Stephen Chavez takes a first look at the new Printed Circuit Engineering Association (PCEA), including an overview of members and membership and why we established the organization. He also looks at the mission of the PCEA and how it will unfold to the industry.

2 Dana on Data: Automating DFX Transfer and Analysis Using IPC-2581C ▶

We are inching closer to a world where a complete intelligent PCB data transfer is realized. The IPC 2-16 Digital Product Model Exchange (DPMX) Subcommittee has just sent revision C out for IPC-2581 Consortium review with final industry approval targeted for this June. Dana Korf discusses the significant additions and their impact.

3 A Young Designer's Journey After Grad School ▶


Lukas Trumpler

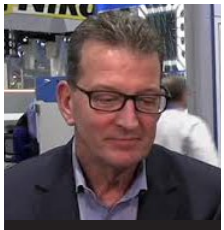
Lukas Trumpler is a young PCB designer from Germany who designs boards for sensors with ITK Engineering GmbH. Andy Shaughnessy met with Lukas and spoke with him about his job, what he's learned in this industry so far, and his impressions of his first AltiumLive event.

4 Tim's Takeaways: Working From Home–5 Tips for Newbies ▶


Tim Haag

Due to the COVID-19 outbreak, many people who have worked in an office environment for their entire career have suddenly found themselves shifted to working remotely. Tim Haag, who has worked from home for over 17 years, shares five tips for making the most of this situation and working successfully from home.

5 DownStream: Smoothing out the Post-Processing Bumps ▶


Joe Clark

Joe Clark, co-founder of DownStream Technologies, gives Kelly Dack an overview of the company and their innovative product line, which serves to smooth the bumps that can occur between source design output and manufacturing line input. As Joe explains, 2019 was a great year for the company, and he expects that trend to hold through 2020.

6 IPC APEX EXPO 2020 Attendees Speak: Caleb Buck ▶


Caleb Buck

"I am a PCB designer and an electrical engineer," said Caleb Buck of EaglePicher Technologies. "I'm not involved in the manufacturing, but I tell manufacturers what to do via my drawings. I had an interest in learning more about the IPC standards, my manufacturer's capabilities, and some of the tools they use."

7 Ultra Librarian Announces UltraBOM for Digi-Key ▶

Ultra Librarian, the world's largest free cloud-based CAD library provider, announced UltaBOM, enabling design engineers to search and research Digi-Key Electronics parts within OrCAD® Capture, and then submit BOM parts directly to Digi-Key for purchase.

8 Elementary, Mr. Watson: Are We There Yet? ▶

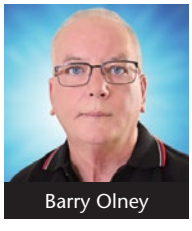
Anyone who has taken a road trip with children knows the question, "Are we there yet?" very well. This question also applies to PCB design. If you are not careful, your PCB project could easily go off track, and you could lose sight of what you are doing (objective), why (motivation), how (process), and when (schedule). John Watson emphasizes the importance of these fundamental questions.

9 Publish Your Work, Become an Industry Leader ▶


Bert Simonovich

Andy Shaughnessy met with Bert Simonovich, founder of Lamsim Enterprises, to discuss a new mentorship program and panel discussion at DesignCon, where veteran authors of technical papers mentor new authors through the process. Bert explained the benefits of publishing technical papers and shared Wayne Gretzky's advice for any engineers or technologists who are considering publishing their technical work.

10 Beyond Design: Interconnect Impedance ▶


Barry Olney

We know that transmission line drivers must be matched to the impedance of the line for the perfect transfer of energy. Energy is never lost but rather transforms into other forms of energy. Barry Olney looks at why interconnect impedance is so important to the correct performance of the system.

Career Opportunities

Looking for the purrrrfect applicant?

Find industry-experienced candidates at I-Connect007.

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

In addition, your ad will be featured in at least one of our newsletters, and your posting will appear on our **jobConnect007.com** board, which is also promoted in every newsletter.

Potential candidates can click on your ad and submit a resume directly to the email address you provide. 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!

Don't delay, contact us right *meow!*

To get your ad into the next issue, contact:
Barb Hockaday at barb@iconnect007.com or +1.916.608.0660 (-8 GMT)

I-Connect007
GOOD FOR THE INDUSTRY

smt007 pcb007 pcb design007

Career Opportunities

Employment Opportunities

National Technology, Inc., a manufacturer of high-quality printed circuit boards, is currently looking for candidates for the following positions in our Rolling Meadows Illinois Facility:

Quality Control Manager

- Manage QMS in accordance with the ISO 9001:2015 system.
- Manage inspection departments, including final inspection, pre-mask inspection, AOI inspection and all associated quality inspections.
- Maintain continuous improvement initiatives.
- Generate and maintain monthly quality reporting.
- Manage internal and external corrective and preventive action.
- Responsible for maintaining the ISO status, including audits, training, procedures, etc.
- Maintenance and scheduling of calibrations.
- Be a liaison to our facility in India regarding customer related issues.
- Customer contact with RMA and corrective action.

Process/Quality Engineer

- Develop and document new processes and technologies.
- Review existing processes for improvement opportunities.
- Assist in identifying and addressing manufacturing issues.
- ISO internal auditing and process related audits.
- Set-up and monitor process controls through manufacturing.
- Maintain regulator compliances.

Candidates for these positions should have a solid background in printed circuit board fabrication. An in-depth knowledge of applicable IPC standards as well as ISO 9001 standard will be required.

apply now

Talent for Hire

Have a position to fill? D.B.Management has the people you're looking for. Candidates currently available include:

- **Director of Operations, PCB and CEM**
  Grew previous operations to over $100 Million.
- **Technical Field Applications Engineer**
  Over 15 years of experience working in the U.S. with offshore companies.
- **Two Highly Experienced Front End Engineers**
  Over 25 years of experience. Will work remotely.
- **Four Experienced Sales Professionals for Both PCB Fabrication and Assembly**
  All four exhibit successful growth records.

Let D.B. Management help you with your staffing needs. Contact us now by calling Dan Beaulieu at:

207-649-0879

or emailing him at:

danbbeaulieu@aol.com

more details

Career Opportunities

FREEDOM CAD SERVICES

Sr. PCB Designer–Allegro

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 overtime at time and a half.

Key Qualifications

- EXPERT knowledge of Allegro 16.6/17.2
- 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
- Experience using SKILL script automation such as dalTools
- Excellent team player that can lead projects and mentor others
- Self-motivated, with 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, PADS, Xpedition)

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 work load
- Ability to create from engineering inputs: board mechanical profiles, board fabrication stack-ups, detailed board fabrication drawings and packages, assembly drawings, assembly notes, etc.

apply now

EAGLE electronics Inc.

CAM Engineer

Eagle Electronics is seeking a CAM engineer specific to the printed circuit board manufacturing industry. The candidate should have a minimum of five years of CAM experience and a minimum of two years of experience in Frontline InCAM software. The candidate should also be fluent in PCB and CAM language pertaining to customer and IPC requirements. The ideal candidate has experience with scripting Frontline InCAM software.

This is a first-shift position at our Schaumburg, Illinois, facility; this is not a remote/off-site position. Any offer would include relocation costs to the Schaumburg, Illinois, area along with competitive salary and benefits.

If interested, please submit your resume to HR@eagle-elec.com and include "CAM Engineer" in the subject line.

About Eagle—Since 1979, Eagle Electronics has provided customers with the highest quality printed circuit boards at fair and competitive prices. From providing customers with short standard lead times to very low premiums on quick turns, Eagle strives to provide the best total value in high technology rapid turn-around PCBs in the industry.

apply now

Career Opportunities

orbotech®
A KLA Company

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

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

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

GO Gardien

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

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

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

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

Career Opportunities

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

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

Career Opportunities

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

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

Career Opportunities

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

YOUR JOB AD HERE

For information, please contact:
BARB HOCKADAY
barb@iconnect007.com
+1 916.365.1727 (PACFIC)

I·Connect007
GOOD FOR THE INDUSTRY

I-007e EDUCATIONAL RESOURCE CENTER

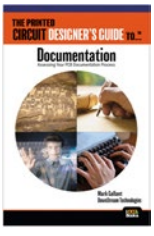
I-007e Micro Webinars

Learn from the Experts in Our On-demand Webinar Series

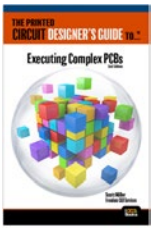
NOW AVAILABLE:
COATINGS UNCOATED from Electrolube, a 12-part webinar series.

I-007e Books

The Printed Circuit Designer's Guide to...

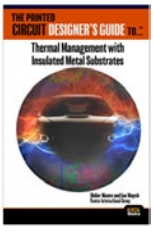


Documentation, by Mark Gallant, Downstream Technologies
When the PCB layout is finished, the designer is still not quite done. The designer's intent must still be communicated to the fabricator through accurate PCB documentation.



Executing Complex PCBs, by Scott Miller, Freedom CAD Services
Designing a complex circuit board today can be a daunting task. Never before have PCB designers on the cutting edge faced more formidable challenges, both electrical and mechanical.

Producing the Perfect Data Package, by Mark Thompson, Prototron Circuits
For PCB designers, producing a comprehensive data package is crucial. If even one important file is missing or output incorrectly, it can cause major delays and potentially ruin the experience for every stakeholder.



Thermal Management with Insulated Metal Substrates, by Didier Mauve and Ian Mayoh, Ventec International Group
Considering thermal issues in the earliest stages of the design process is critical. This book highlights the need to dissipate heat from electronic devices.

Fundamentals of RF/Microwave PCBs, by John Bushie and Anaya Vardya, American Standard Circuits
Today's designers are challenged more than ever with the task of finding the optimal balance between cost and performance when designing radio frequency/microwave PCBs.

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

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

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

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**

DESIGN007 MAGAZINE

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

April 2020, Volume 9, Number 4
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