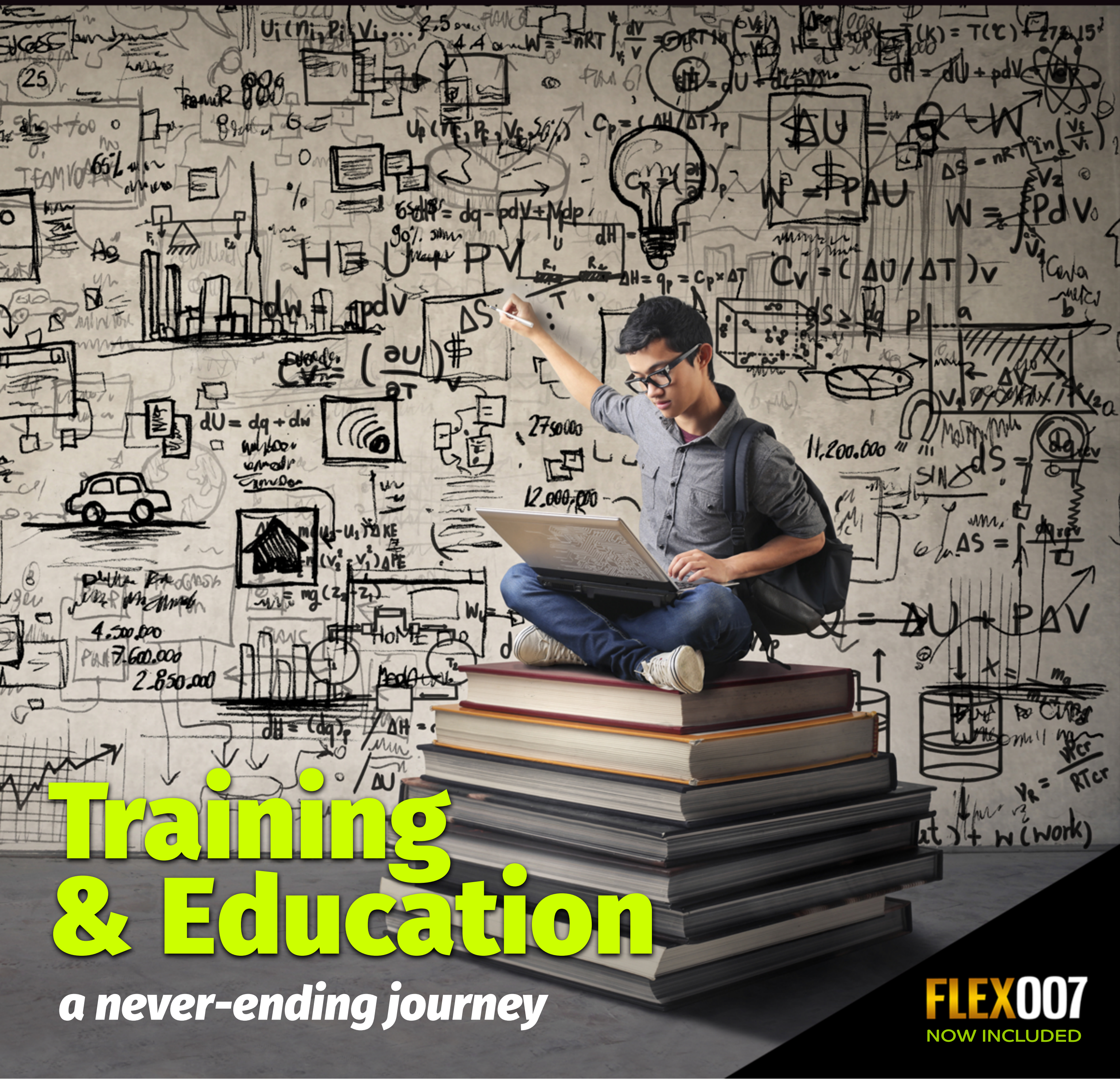
I-Connect007

AUGUST 2019

DESIGN007 MAGAZINE



Training & Education

a never-ending journey

FLEX007 NOW INCLUDED

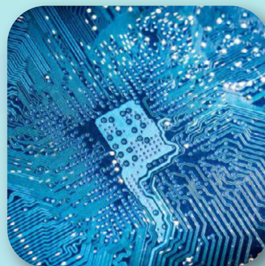
Candor
Circuit Boards

Innovative solutions for complex circuit board manufacturing

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

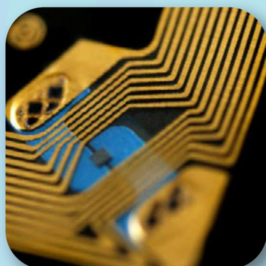
RF/Microwave PCBs

Power Electronics



High-Speed Data

Automotive Electronics



Lamination

Startups

Our technology produces what our customers dream!

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

DESIGN007 MAGAZINE

Training and Education

Not too long ago, if you couldn't attend a trade show or technical conference, your chances of continuing your PCB design education were slim to none. But the times are a-changin'. This month, as the kids head back to school, we look into the growing opportunities for PCB design instruction.

FEATURES:

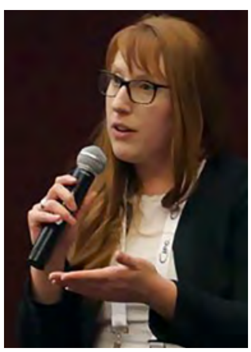
10 **Bill Brooks on Teaching PCB Design at Palomar College**
by the I-Connect007 Team

24 **PCB Design Training: More Critical Than Ever**
Interview with Gary Ferrari

28 **Leo Lambert on Training the Next Generation of Technologists**
Interview by Andy Shaughnessy

38 **Pulsonix Is Bullish on Next-gen Designers**
Interview with Bob Williams

44 **Teaching Young Girls About STEM Careers**
Interview with Brooke Campbell



FEATURE COLUMNS:

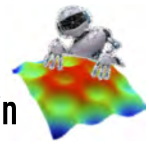
18 **Teaching the Next Generation: An Overview of Today's University Courses**
by Mark Thompson

34 **CA Design's Bob Chandler on Training PCB Designers**
by Dan Beaulieu

DESIGN007 MAGAZINE

SHORTS:

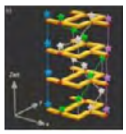
27 Making Robots Sweat With Smart Surfaces That Act as Artificial Skin



32 Eye-controlled Soft Lens Paves Way to Soft Human-machine Interfaces

42 Physicists Make Graphene Discovery That Could Help Develop Superconductors

47 Novel Process for Structuring Quantum Materials



53 Cooling for Quantum Electronics

57 Global Commercial Drone Market to Reach 36% CAGR Over 2018-2022

65 NASA's Mars 2020 Rover Does Biceps Curls

69 Researchers Demonstrate Low Voltage LEDs

88 New Nanoantennas to Improve Ultra-fast Wireless Connections

HIGHLIGHTS:

43 PCB007

83 MilAero007

112 Top 10 PCBDesign007

DEPARTMENTS:

115 Career Opportunities

126 Events Calendar

127 Advertiser Index & Masthead

ARTICLES:

70 Decoupling Capacitors' Impact on Power and Signal Integrity
by Chang Fei Yee

76 Design Rule Checks Cut Down on Board Respins
by Rebecca Lord and John McMillan

84 SimplifyDA's Floorplanning Tool Optimizes Autorouting
by the I-Connect007 Editorial Team

COLUMNS:

8 PCB Design Education Never Ends
by Andy Shaughnessy

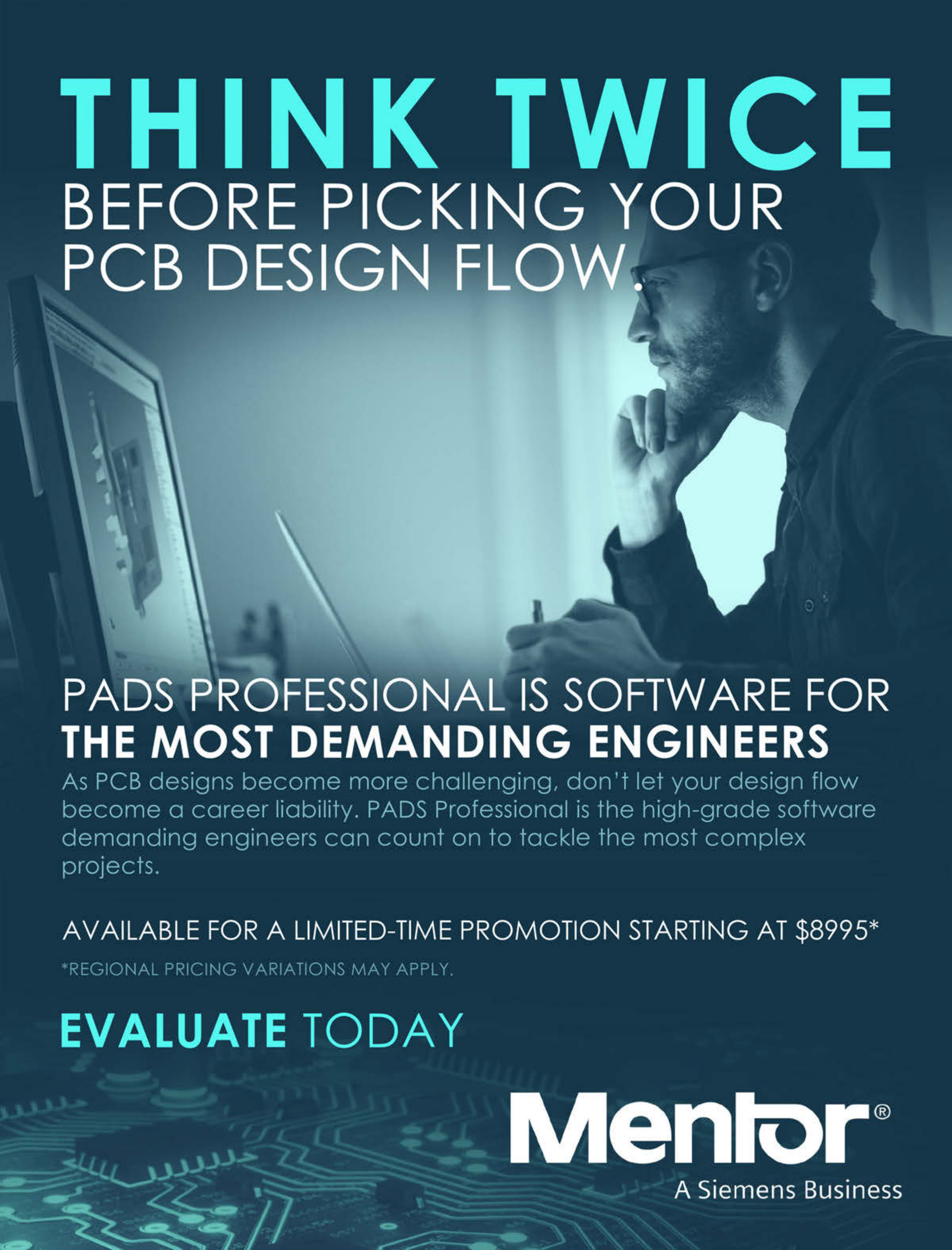
48 The Curse of the Golden Board
by Barry Olney

54 A Bright Future: The Sonora Chapter
by Stephen V. Chavez

58 High-frequency Material Technical Resources
by John Coonrod

62 Managing Global Supply Chain Uncertainty
by Bob Tise

66 How to Avoid Pump-out and Achieve Efficient Heat Transfer
by Jade Bridges


THINK TWICE
BEFORE PICKING YOUR PCB DESIGN FLOW.
PADS PROFESSIONAL IS SOFTWARE FOR
THE MOST DEMANDING ENGINEERS
As PCB designs become more challenging, don't let your design flow become a career liability. PADS Professional is the high-grade software demanding engineers can count on to tackle the most complex projects.
AVAILABLE FOR A LIMITED-TIME PROMOTION STARTING AT $8995*
*REGIONAL PRICING VARIATIONS MAY APPLY.
EVALUATE TODAY
Mentor®
A Siemens Business

FLEX007

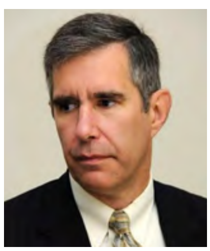
Special Section

It's a simple fact: There aren't many flex designers around. If your company moves into flexible circuits, rigid board designers are going to have to get up to speed in a hurry. This month, we explore a variety of avenues available for flex and rigid-flex design curriculum.

FLEX007 ARTICLES:

92 **Industry Experts Talk Flex Design**
Mike Creeden Interviews Chris Hunrath and Steve Bowles

98 **John Michael Pierobon Discusses His Flex Design Class**
Interview by Andy Shaughnessy



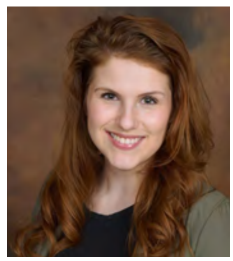
HIGHLIGHTS:

106 **Flex007.com News**

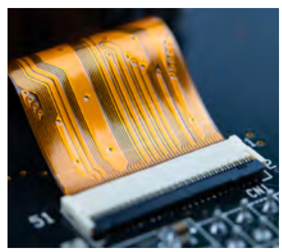
FLEX007 COLUMNS:

91 **Flex Design Education Opportunities Growing—Slowly**
by Andy Shaughnessy

102 **SMTA Pilot Program for Emerging Engineers**
Tara Dunn Interviews Tamara Shephard



108 **Flexible Circuits Go to College**
by Joe Fjelstad



FLEX007 SHORTS:

101 **A Wearable Device So Thin and Soft You Won't Notice It**

105 **Soft Wearable Health Monitor Uses Stretchable Electronics**

PCB Design Education Never Ends

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

I think it must be a requirement for every commencement speech to include the words, "You may be a graduate now, but your lifetime of learning has only begun." It's an overused line, but it's truer than ever for PCB designers and design engineers today.

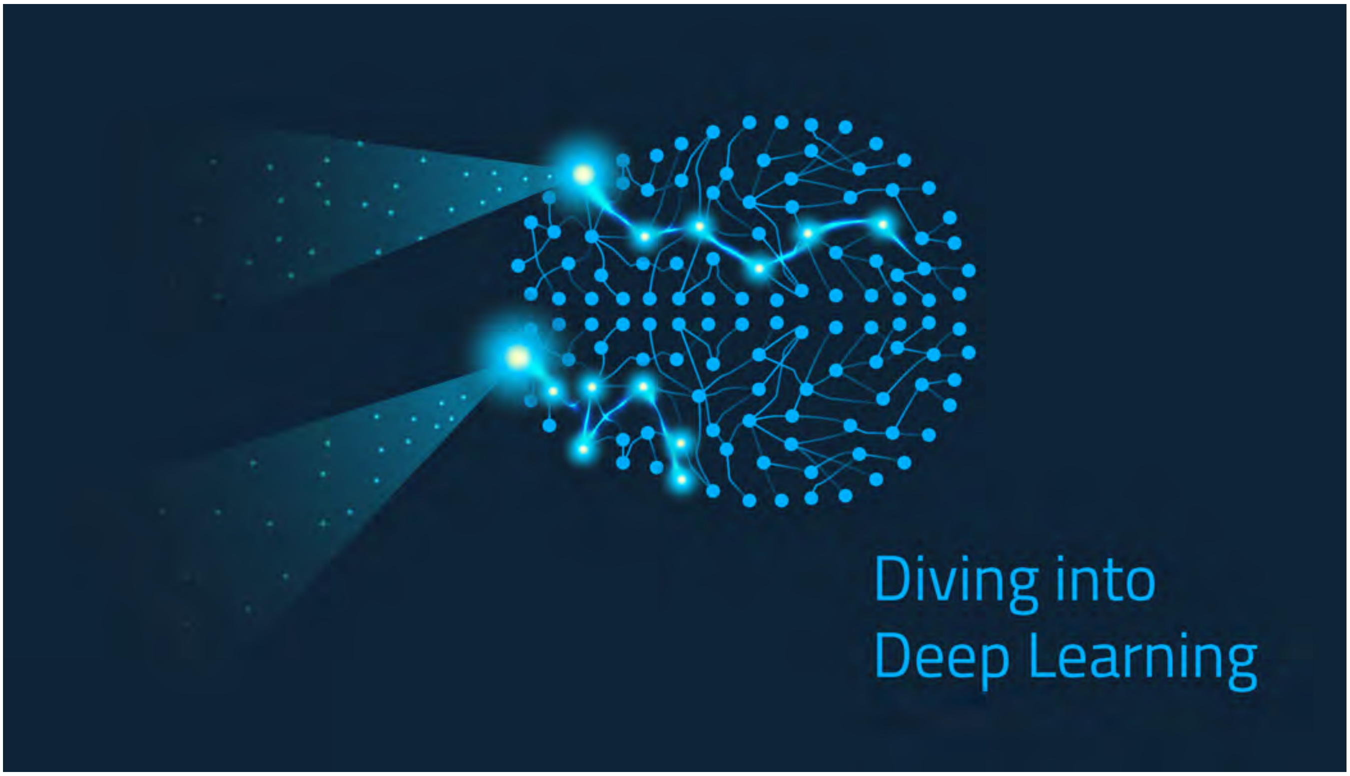
As longtime instructor Mary Sugden once said, "The designer is the hub of the wagon wheel." You could argue that PCB designers have to know far more about what happens downstream than anyone downstream needs to understand about the design process. Designers need to understand every fabrication and assembly process while keeping current on materials, components, and upcoming technologies like 5G. You have to keep educating yourself, or you're going to fall behind. It's a career packed with constant change, which is what keeps it exciting, even for some of you who have been doing this since before Apollo 11.

Once upon a time, if you couldn't travel to a conference or trade show, you were out of luck. You could take an EDA vendor class, but that basically teaches you how to use that particular tool. There weren't many options for designers who wanted to continue their education.

Fortunately, there are more ways than ever to continue your design education. PCB design training and education have come a long way in the past few decades. Now, even designers at companies that have no travel budget can keep their skill set current. Dozens of vendor-neutral PCB design classes are available online, many free of charge. YouTube is full of design courses and tutorials, though you'll have to separate the wheat from the chaff, so to speak. The IPC Designers Council is a great source of information; their chapter meetings always feature an expert guest speaker. Plus, magazines like this offer a solid source of technical information that you need to stay ahead of the game.

But there's still one place you won't typically find much PCB design instruction: at the college level. PCB design is still not available as an accredited four-year degree, and it's rarely available to college students unless it's part of an electrical engineering degree.

Bill Brooks of Nordson Asymtek has had more luck than most at getting PCB design curriculum into the college system. In this month's first feature interview, Bill explains how he created and taught a PCB design course at Palomar College in Carlsbad, Califor-



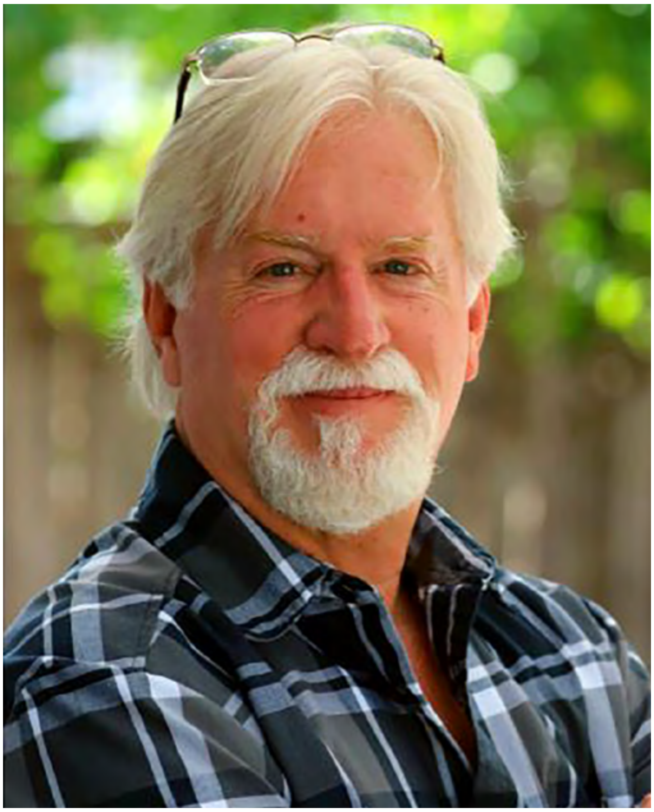
nia, and details some of the hurdles he faced. Next, columnist Mark Thompson of Prototron Circuits outlines the top 19 electronics courses available at the University of Washington that he would recommend to anyone starting out in the industry today. We also have an interview with Gary Ferrari of FTG Circuits who explains why designers must continue their education, even if their company doesn't support their efforts.

Leo Lambert of EPTAC Corporation discusses the company's move to open training centers around the country to cut down on attendees' travel expenses and why training methods are evolving to meet the needs of the next generation of PCB technologists. From the design bureau viewpoint, columnist Dan Beaulieu and CA Design owner Bob Chandler discuss the "gospel of Bob," which is based on the idea that all PCB designers need to be properly trained, and the need for companies to train their designers. Bob Williams, managing director of Pulsonix, explains how this EDA company offers real design training that is not specific to Pulsonix tools as well as their efforts to advocate for PCB design careers by meeting with high school and college students. Finally, Brooke Campbell of BTG Labs describes how she works to draw more young women into STEM careers and why girls shouldn't avoid hard science classes, and she shares some highlights from her presentation to the Girl Scouts.

We have a variety of columns from regular contributors Barry Olney of iCD, Stephen Chavez with the IPC Designers Council, John Coonrod of Rogers Corporation, Bob Tise of Sunstone Circuits, and Jade Bridges of Electrolube. We also have some great articles by CF Yee of Keysight Technologies, Rebecca Lord and John McMillan of Mentor, a Siemens company, and an interview with Zen Liao of SimplifyDA.

We're quickly approaching the show season with PCB West, SMTA International, AltiumLive, and productronica just around the corner. If you can't make it to these events, don't worry: We have you covered! **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.



Bill Brooks on Teaching PCB Design at Palomar College

Feature Interview by the I-Connect007 Editorial Team

Bill Brooks of Nordson ASYMTEK is a longtime PCB designer and one of the first people to teach PCB design courses in a college setting. He recently spoke with the I-Connect007 editorial team about his history in design and his time as a PCB design instructor, the curriculum he developed and taught, and various techniques that might be enacted today to better educate the designers of tomorrow.

Andy Shaughnessy: Bill, can you start by giving us a little background about yourself and how you got into PCB design?

Bill Brooks: My dad was a technician in the aerospace industry. He started a PCB shop out of his garage when I was in high school. I spent some time learning the process with him and ended up working in his shop for about five years. I ended up getting my first PCB design job with Sub Sea Systems in Escondido. Then, I did job shopping for various companies around the San Diego Area for many years.

Barry Matties: And how did you learn printed circuit design?

Brooks: My father taught me from the beginning. He bought the materials and a design book from Bishop Graphics. I learned the whole manufacturing process before doing design, such as drilling, plating, etching, routing, and putting in eyelets in boards. He got me started, and I learned a lot more as I worked for other companies throughout my career and attended seminars and workshops at design-related events.

Matties: What year was that?

Brooks: It was around 1970.

Matties: Fast forward to when you started your design classes. What inspired you to do that?

Brooks: When I competed in PCB Top Gun at PCB West in 2000, Rick Hartley interviewed us during the competition. After I told him my story, he asked, “Have you ever thought of mentoring?” I had not, but that seed he plant-

ed inspired me to find some way to give back to the designer community and made me find the IPC Designers Council booth at the show. I joined the IPC Designers Council and later became acquainted with Gary Ferrari, Andy Kowalewski, Glenn Wells, Dieter Bergman, Susy Webb, Cherie Litson, Paul Fleming, Jack Olson, Bill Gebhardt, Leslie Gomez, Professor Rainer Thüringer, Happy Holden, Tom Hausherr, Doug Brooks, Jeff Condit, Ben Jordon, Jean Stout, and a large community of professional PCB designers. I accepted an invitation to join the Designers Council Education Committee and Executive Committee as a volunteer to help lead the design community and represent them to IPC.

I became certified to teach the Designers Council CID workshops and had many discussions with the designers and professionals along the way before I took on the idea of teaching a class in the California Community College System. I joined the leadership in the local IPC chapter in San Diego and was elected to office there to help grow the chapter. I later was introduced to Ann Reese, the department chair at Palomar College who was running her own PCB design classes. I spoke with her about adding curriculum based on the Designers Council, and she wanted to have me teach the class as she was planning to retire. They put me through the process of getting approved as an adjunct professor/instructor based on my 40+ years of design experience since I didn't have a teaching degree.


Bill began sculpting in 2011, and he's a painter and photographer as well.

Once I had the approval, they hired me as the instructor for the beginning and advanced PCB design courses. I taught semester-based classes for about 10 years at Palomar College. The most important thing is that I found a way over some hurdles to bring quality education to the students in the local college system, which aligned with my desire to help more designers get into this industry. This occupation was otherwise devoid of any real channels through which designers could get a decent introduction.

Matties: Tell me about the students. Were they newbies or people who had already had some design experience?

Brooks: We had a diverse cross-section of people. There were newer students that were coming up out of the high school system, some mid-level students that already had some mechanical CAD classes, and a few that qualified as my peers who were only really interested in the CAD tool that was available in the classroom. I had to split my time between teaching the tool to veteran designers and helping newer students that were challenged with the basic concepts.

Shaughnessy: When you got into Palomar, you said that they already had a class going there, but it was mainly AutoCAD and PADS and that you came in and made some changes. Did IPC help you with this or did you have anybody giving you pointers on how to set this up?

Brooks: Yes, and no. I did get some pointers from Glenn Wells and help from some others. We brought up the question of how to set it up in the Designers Council Executive Committee. Glenn had been working on the same thing in Texas. He wrote a guide based on his experiences with the university system. I used that guide to give me some ideas for the Community College System. Gary Ferrari negotiated a deal with IPC to make some of the PCB-design-related standards available to the college students for a discount because most of the students didn't have much money for books. It was the equivalent of the discount that members could get in the Designers Council. Some of our students were approved for federal assistance to take the courses and had to qualify for the books and papers that were associated with the classes. There is a lot of review and scrutiny in the public education system and lots of red tape.

We were going to put them in the college bookstore, but the bookstore marked them up by double, so there went the savings we were hoping to pass on. Then, I negotiated another way that the students could go directly to IPC and order the design standards, cutting out the bookstore. All they had to do was tell IPC that they were students in my class and have me confirm it. That worked out well.

We wanted to get as much Designers' Council information into the hands of the students as we could. I even had a Designers Council meeting in class one semester. About 20–25 students and designers showed up. It was a full room, which was a cool experience for the students and designers. The students asked questions about the industry, and the designers encouraged the students.

Shaughnessy: Did they want to learn more about the tools?

Brooks: Of course! The CAD tools are expensive and not readily available to the average person. That was one of my personal pet peeves about the classes that were generally available. I remember taking the PADS class at Palomar with my son years before. It was really a CAD-tool-based class. They walked you through the CAD training but didn't dig into the "why" of how things were done. Basically, they said, "You push this button to get it to do this and that." It's tough to get a job as a designer when you can't answer basic questions about a board design much less about DFM, auto insertion, pick-and-place, clearances, creepage, plating thickness, tolerances, etc.

I wanted to try to change that. I wanted the education focus to be on design-related skills and not make it so much about how to run a specific CAD tool. Mentor offered us their Xpedition tool for free, but that turned out to be very difficult for the average students. I later chose Altium as the platform for the classroom, and they gave us 25 seats for a generous discount. Nobody in the class really needed a powerful autorouter to lay out the boards we did in my class; we needed good manual routing, snap grids, DRC, and a design rules-driven environment that would alert you to issues but not stop or crash if you made a mistake. Altium suited us pretty well throughout my time at Palomar College.

Matties: Talk about the curriculum a little bit. Were you starting with best practices? For a new designer, where should their starting point be?

Brooks: I started with the basics. A prerequisite to the class was Electronics Drafting 101. Familiarity with schematic design, symbols, reference designators, notation, formats, bills of material or parts lists, fabrication drawings, dimensioning units of measure—all the types of documentation required to manufacture a board assembly. The first PCB class was a beginning class, covering the use of the tool, making library parts for schematics, PCB components, and the parameters involved in doing that correctly.

Matties: It was an elementary beginning.

Brooks: It has to start there. Later in the semester, I introduced information as it related to their projects: IPC-2221 and IPC-2222 for rigid boards, IPC-4101 for board materials, IPC-6011 and IPC-6012, IPC-2615 for dimensions, IPC-D-325 for documentation requirements, IPC-T-50 for terms and definitions, IPC-7351 for land pattern development, IPC-J-STD-001 for requirements of soldering components, and IPC-A-600 and IPC-A-610 for assembly. It wasn't a requirement for the students to get the specifications, but it served to familiarize those who really wanted to get into the industry to have personal access to them.

I showed them examples of boards and let them solder some through-hole components to some donated boards we had in the classroom. I introduced them to Ohm's law, and how to calculate the current in a conductor and use that to safely size it so as not to damage the board under load.

We also worked with local board and assembly shops and arranged tours so that the students could see the other side of the process: Gerber generation, etching, drilling, plating, solder mask, paste mask, silkscreen legends, and the steps that the manufactures had to go through to make a board. Thanks to Hallmark Circuits, Crown Circuits, Hughes Circuits, Circuit Logic, U.S. Circuits, and Generation Circuits, our students had the opportunity to see boards being manufactured and assembled.

I ended up with two semester-long classes, including beginning and advanced PCB design. We met twice a week for three hours a night. That's about 96 hours of class time per semester and about 3x that in non-paid time preparing for classes, grading papers, etc., all while holding down a 40-hour-a-week day job.

Matties: What's the status of the class now?

Brooks: Currently, they're looking for a new instructor. They approached me again recently, but I told them I'm just too busy. I can't do it right now. I'm a full-time designer at Nordson ASYMTEK, but I have other activities too. I teach art once a week, and I sculpt and paint. I am also an active photographer.

Shaughnessy: Plus, I imagine that colleges have their own hoops for you to jump through, right?

Brooks: Sure. They used to have a lot more vocational classes at the college, but the department chair cut most of them; they went back to almost pure academics. I think they kept the woodshop because it was fully funded from outside of the college. You had to have at least 20–21 students for a 25-seat class to keep it open because of the way funding comes from

Bill brings a sculpture to life.

the state. They are funded by how many students are occupying classroom seats. You must have them full, or they cancel the class for that semester.

Matties: That's the problem. I just did an interview with a company here in the U.K. who's looking to bring in designers, and they're having to hire people and set up their own curriculum in-house to train them because there's not a place where you can hire exciting designers.

Brooks: Yes. And it's tough for the students or prospective employees to find a comprehensive book on PCB design and have them say, "This is going to teach you everything you need to know about PCB design, so get going." It really doesn't exist. And even if it did, it's certainly not going to give them any common practical design sense. I hate to admit it, but when I walked into this place in 2012, they were paying exorbitant prices for their boards with very poor yields because they were assigning the EEs to make their own PC boards, and they didn't know much about board design or DFM or assembly. They said, "Well, it says on the vendor website they can do this, so that's what I'm going to use." They were having all kinds of PCB-related problems designing to the minimums, or the impossible. Nowadays, we typically get it right the first time, every time.

Dan Feinberg: You have a lot of kids now who want to get into high-tech and electronics, but they're saying, "I don't know if I want to go into

debt like that from going to college for four years or five years." What I'm seeing with some of my family members is it took them five years to get through college because the classes were full that they needed to take for the degrees they wanted. They could monitor the class, but they couldn't take it. Isn't that strange? I think it's a trick for the colleges to get another year's tuition out of them.

One of Bill's sculptures, "Passion and Heart," resembles the Western works of Frederic Remington.

Brooks: I think the majority of the designers I've known are non-degreed people. They've come from different backgrounds, the opportunity presented itself, and they had the skills to be able to absorb what needed to be done and to go after it. It's something that most learned at the shoulder of another designer in the beginning. The college system can be expensive. I opted to teach in the Community College System here because it had been so vocational in content and affordable compared with the university system. Private schools, such as Mary Sugden's Copper Connection back in the early 2000s, were an answer to some of the companies that could afford to send their employees to her facility. In Europe, Premier EDA taught a lot of people as well as the FED in Germany, PIEK in the Netherlands, and Collin County Community College in Texas. It's not easy to find a place to learn PCB design in any community.

Shaughnessy: What do you think about online training? Do you think with webinars and YouTube that there's some value to that? Everybody seems to be doing it.

Brooks: I'm really challenged to try to come up with an online curriculum. We're used to going to seminars and having somebody talk to us with a PowerPoint presentation running through a concept. But I think hands-on training with somebody sitting in front of the CAD tool and working with it is essential. Regarding visiting a shop and seeing how it runs, you might be able to do that in a video way and give them a sense for it, but it's not quite the same as walking through it yourself. Somebody could create that content and put it out there with YouTube and make it a part of the curriculum. There needs to be a hands-on feel and they need to be able to use a CAD tool.

It's going to take an investment of resources to put something together and run it. Maybe one of the big CAD companies would back it and create the online content. "If you want to learn how to become a designer, come to us. We'll teach you how to do it, we'll have the curriculum right here, and you can walk through it step by step. At the end, we'll give you a certificate." You pay something up front to get involved in it, or they make it available in a cost-effective way.

Matties: Even beyond the certificate, companies will hire them to train people.

Brooks: Sure. That certainly can be an option. There are people who I'm sure would be willing to do it, but I'll bet that a lot of them have the same problem that we all do. They're not sitting with a book in their hand, saying, "I need to walk through this material and teach somebody how to do this." No, they must develop it from ground zero, and that makes it a large task. I did start putting something together, but I've been too busy to dedicate my life to it. It would take a team of people to create a PCB 101 book that could teach someone what they needed to know online.

Shaughnessy: Thanks for your time, Bill.

Brooks: Glad to share what I can. Thanks. **DESIGN007**

Teaching the Next Generation: An Overview of Today's University Courses

The Bare (Board) Truth

Feature Column by Mark Thompson, CID+, PROTOTRON CIRCUITS

In this column, I will focus on our "home team," if you will—the University of Washington (UW). I counted approximately 163 programs in their catalog of electronic courses. I have pulled out what I think are the most valuable for emerging electronic engineers. This does not mean the other courses are not valuable as well; in fact, I am sure I am missing some. But these are the top 19 courses I would choose if I were to start my electronics career over again.

1. Introduction to Signal Conditioning

This course requires some math courses and a basic physics course. It introduces analog circuits interfacing sensors to digital systems, including connection, attenuation, amplification, sampling, filtering termination strategies, controls and Kirchhoff's laws, op-amps, resistors, capacitors, and inductors. This course is not intended for EE majors but would be a great help to anyone going into the tech sector.

2. Fundamentals of Electrical Engineering

As an introduction to electrical engineering, this course covers basic circuit and systems concepts, mathematical models of components, Kirchhoff's laws, resistors, sources, capacitors, inductors, and operational amplifiers as well as the solution of first- and second-order linear differential equations associated with basic circuit forms. Prerequisites are math courses and physics.

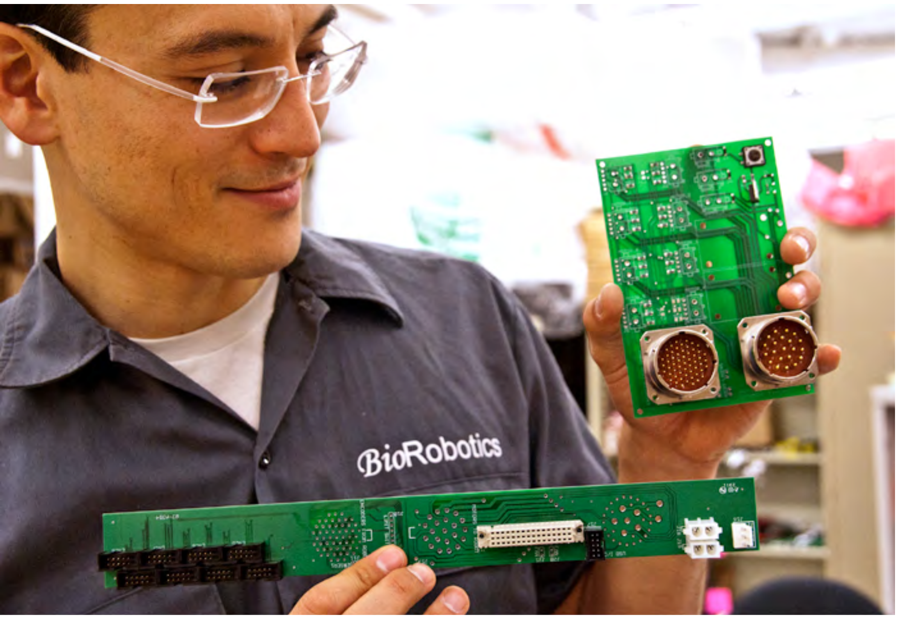
3. Circuit Theory

This course addresses electric circuit theory, analysis of circuits with sinusoidal signals, phasors, system functions, complex frequency, frequency response, computer analysis of electrical circuits, power and energy, and two-port network theory. The laboratory covers basic electrical engineering topics.

University of Washington, Seattle, electrical engineering center. (Source: Wonderlane, Flickr)

4. Digital Circuits and Systems

This course provides an overview of digital computer systems, such as logic, Boolean algebra, combinational and sequential circuits, logic design, programmable logic devices, and the design and operation of digital computers, including ALU, memory, and I/O. This course also



UW electrical engineering doctoral student Hawkeye King holds the circuit boards and connectors for the Raven II robot. [Source: UW News]

has weekly labs that are invaluable for understanding the reality of electronics.

5. Devices and Circuits

Topics covered in this course include characteristics of bipolar transistors, large- and small-signal models for bipolar and field-effect transistors, and linear circuit applications—such as low- and high-frequency analysis of differential amplifiers, current sources, gain stages and output stages, internal circuitry of op-amps, op-amp configurations, and op-amp stability and compensation.

6. Applied Electromagnetics

This course aims to teach students about introductory electromagnetic field theory and Maxwell's equations in integral and differential forms, uniform plane waves in linear media, boundary conditions and reflection and transmission of waves, guided waves, transmission lines and Smith charts, and electrostatics.

7. Design of Digital Circuits and Systems

This course provides a theoretical background in—and practical experience with—tools and techniques for modeling complex digital systems with the Verilog hardware description language as well as maintaining signal integrity, managing power consumption, and ensuring robust intra- and inter-system communication.

8. Analog Circuit Design

Topics introduced in this course include the design of analog circuits and systems applying modern integrated circuit technology, operational amplifiers, differential amplifiers, active filters, and voltage references and regulators.

9. Power Electronics Design

This course helps electrical engineers with electronic conversion and control of electrical power, including semiconductor switching devices, power converter circuits, the design of magnetics, and the control of power converters. Other topics include AC/AC, AC/DC, and DC/DC power converters and circuit simulation with extensive laboratory work and a four-week power converter design project.

10. Computer-aided Design in Power Systems

In this design-oriented course in power system engineering, students are assigned a project concerning system operation and planning, steady-state and dynamic behaviors of power systems, or distribution systems. Each project involves formulating design criteria, developing an approach, and applying existing software.

11. Microwave Electronic Design

This course addresses microwave engineering, the design of microwave circuits using S-parameter techniques, measurement techniques, CAD of microwave systems, and the design, fabrication, and evaluation of a microwave amplifier.

12. Electromagnetics I: Microwave Engineering

This course examines microwave transmission line models and their applications, electromagnetic waves in layered media, mode structures in metallic and dielectric waveguides, resonators and cavities, and Green's functions (something near and dear to my heart).

13. Microfabrication

This course introduces principles and techniques for the fabrication of microelectronics devices and integrated circuits and includes cleanroom laboratory practices and chemical safety, photolithography, wet and dry etching, oxidation and diffusion, metallization and dielectric deposition, compressed gas systems, vacuum systems, thermal processing systems, plasma systems, and metrology.

14. High-frequency Circuits and Antennas

Planar microstrip structures are high-frequency circuits and antennas used in communication, aerospace, and computer industries. This course addresses the computation of fields and waves in such structures and how to calculate circuit parameters and radiation characteristics. Structures studied include microstrip lines, coupled lines, antennas, resonators, and discontinuities.

15. Electromagnetics II: Microwave Engineering

This class covers radiation from apertures and beam waves, periodic structures and coupled-mode theory, dispersion and anisotropic media, antennas, and scattering of waves by conducting and dielectric objects.

16. Antennas: Analysis and Design

This course covers the fundamentals of antennas, analysis, synthesis, and computer-aided design; applications in communications, remote sensing, and radars; radiation pattern; directivity; impedance; wire antennas; arrays; numerical methods for analysis; horn antennas; microstrip antennas; and reflector antennas.

These last three are excellent final courses to take before starting an electronics career.

17. Introduction to Professional Issues

Topics of interest in this class are for students planning their educational and professional path, including salaries, the value of advanced degrees, societal expectations of engineering professionals, the corporate enterprise, ethical dilemmas, patents and trade secrets, outsourcing, and the global market.

18. Master's Thesis

19. Doctoral Dissertation

As I said before, I am sure there are many that I am missing out of the 160+ courses that the University of Washington offers, but these are some of the courses I would recommend emerging electrical engineers to consider taking.

University Groups

At Prototron Circuits, I deal with a number of groups at the University of Washington. In the UW Applied Physics Laboratory (APL), we build a fair amount of projects for this group. One of the unique things they did was create robotic autonomous ocean robots (Seagliders) that survey the Antarctic ice shelf for how waves, ice, and coastal erosion are affecting the coast. This

UW Bothell electrical engineering master's degree students. [Source: UWJ]

Mark enjoyed his time working with the team at UWashington Formula Motorsports. (Source: UW Formula Motorsports)

was a collaborative effort between the UW APL and the UW School of Oceanography.

Another group that used Prototron was the UW CubeSAT Team, which designs and manufactures small satellites. This is a nationwide program that vies for load space on sub-orbital and orbital missions. In our business, there are often two distinct times a year where fiscal budgets break loose, and we get a lot of business: January and July–August. Launch dates do not necessarily coincide with those two times of the year, which creates a lot of work for us year-round.

UWashington Formula Motorsports is my favorite group at UW. The competition directive is for students to design and build a Formula-style racing car, and then compete against similar race cars built by other students from all over the world. Today, over 500 teams compete worldwide. They have two big races every year: one for all of the American universities in Lincoln, Nebraska, and one at the Hockenheim F1 racetrack in Germany.

A typical Formula SAE car weighs from 300–600 lbs. with horsepower figures ranging from 40–100 hp. Most cars can boast a 0–60 mph in 3–4 seconds with lateral acceleration of up to 2.5 g and a 60 to 0 mph braking distance of about 115 ft.

During one particular visit, I asked the students if they watched F1 racing. I was surprised to hear many of them did not, yet they were building a Formula-style car. So, I asked about things like kinetic recovery systems (KeRS), which is basically a generator that stores energy for a short burst of additional power. They understood this and gleefully showed me their current KeRS. There is a specific weight for the cars that they cannot exceed, so when I brought up drag reduction systems (DRS), they were particularly concerned with adding weight.

Current DRS systems have moving back-wing slats that open on the straights and slam shut when they reach a corner to effectively plant the rear of the car for better cornering and adhesion to the track. I explained that before these systems, there was a rudimentary way of doing a similar thing that did not incur additional weight on the car by simply drilling a series of small holes on the underside of the back wing and creating a venturi-type tube for the driver to put their thumb over; this effectively does the same thing as the modern DRS systems with moving wings and planting the rear of the car without additional weight. They became very excited and immediately began drilling small holes in the underside of the back wing. Fun!

I hope this has given you a better idea of what courses are available for the advancement of their electronics career.

As always, feel free to contact me if you have any questions about this column or any of my previous columns. Thanks for reading! **DESIGN007**

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

PCB Design Training: More Critical Than Ever


Gary Ferrari

Feature Interview by Andy Shaughnessy
I-CONNECT007

I interviewed Gary Ferrari, director of FTG Circuits, at the IPC High-Reliability Forum and Microvia Summit in Baltimore. Gary is a co-founder of the IPC Designers Council and a longtime advocate for PCB design and PCB designers. We discussed the crucial role that PCB designers play in the entire electronics development process, and how IPC and the Designers Council are helping to educate and inform the next generation of designers.

Andy Shaughnessy: Gary, how are you doing? It's good to see you.

Gary Ferrari: I'm doing well. The industry is moving ahead, and I just keep on plugging away.

Shaughnessy: It's a good time to be in this industry. It seems as if more people, especially at IPC, are starting to get the idea that design is a pretty important part of the process, and, in some ways, that everything starts with design.

Ferrari: I recently had a conversation with Dr. John Mitchell, president and CEO of IPC. We were discussing things, and he told me, "Gary, you're going to like what I have to say. I got my staff together and asked them, 'What's the most important thing that we should be focusing on?'" Surprisingly, he said that design was mentioned because everything starts with design. That's a message I have been sending for many years. The designer is responsible for understanding electronic engineering—in some cases, even more information about engineering than maybe the engineers—mechanical engineering, board fabrication, materials, signal integrity, and it goes on and on.

There is much more that PCB designers need to know now. First, they must fully understand the fabrication processes used for various types of materials. What materials will work for a specific design, both mechanically and electrically (signal integrity)? They need to know the assembly processes, which include the types of soldering processes used for technologies, such as through-hole, SMT, buried components, etc. Then, how to create a layout that addresses the optimum component orientation, so that everything is soldered properly. They need to know when they are pushing the envelope, which affects performance and cost. Finally, they also need to know something

about field service. Is the product serviceable in the field, or back at a repair depot? The field service engineers don't necessarily have the same tools that we used in the factory.

If that's the case, then the designer has to design in such a way that it can be repaired, if that's the mission out in the field, rather than going back to the factory or repair depot. As you can see, the designer is responsible for a lot. If they don't do it right, it's going to cost you more time and dollars. Without a good understanding, you may see more failures because of designs that were not given the proper attention to details.

The IPC Designers Council was created by designers for designers to exchange ideas, talk about technology, and to learn between themselves; it's not just what their one company does. The council has been in existence for so many years now, and a lot of people have learned that message and applied it successfully. Now is the time for the designer to be recognized for what they do, or don't do.

Now is the time for the designer to be recognized for what they do, or don't do.

Shaughnessy: Until fairly recently, many designers didn't have a lot of avenues for training, unless they could get to a trade show or conference.

Ferrari: It's unfortunate that designers basically have to learn by osmosis. Many are not given an education budget. And if they are allowed to partake in an educational event, they still have to work on their design in the evenings.

We're at this reliability conference, and a lot of the topics that NASA was talking about, and some of the other speakers, highlight some of the problems we have with reliability. Within each one of those, I can see where the designer has some control of that. I have always said that the designer doesn't have to know every single thing, but a good designer is someone who will ask a question. Did somebody do thermal analysis, etc.? Once you bring it up to a group, then you can figure out if that needs to be looked at and addressed.

I don't advocate that the designer has to know everything or every standard, but what they do have to know is what the standards contain that may be able to help them. Then, they can go look it up. I don't expect them to memorize everything. I sure don't. I wake up in the morning and can barely remember my name (laughs).

Shaughnessy: I look at the designer as kind of the maestro. They have to speak to the EEs and know something about that language, and they have to deal with the CAM people too. They have to understand fabrication and some assembly. They're the ring leader of the circus.

Ferrari: I like to say that the designer takes an engineer's design and the symbolic representation of functionality and converts it to the language of the physical world—the language of the board fabricator, assembler, and field service. They convert the symbolic representation. That's where it starts. Engineers have come to me and asked, "How come they can't do this thing?" When you look at it, it's not buildable, reliable, etc. It's the message we've been sending for years and years. Hopefully, by 2020, we'll be in better shape than we were in past years. Technology is getting very complex nowadays.

Again, it's encouraging that John Mitchell understands the importance of PCB design. He attended our executive board meeting at IPC APEX EXPO 2019. He said that he could only spare about 15 minutes in his schedule after the conference, and he spent almost two hours with us. There's a lot of input that we can provide throughout the product development cycle.

Shaughnessy: How do we expose the next generation of young people to PCB design?

Ferrari: Years ago, several of the chapters used to hold open forums for local high schools; it has to start there. I went around to several high schools, speaking to their student guidance counselors. Remember, high school students have to decide in their junior year if they want to go to college, and if so, which one and what kind of field they're interested in pursuing. I asked them, "What opportunities do you have on your list for printed wiring board design?" because those occupations do not exist on their list. How do you encourage a student to go into printed wiring board design if the profession doesn't exist?

Shaughnessy: Most high school guidance counselors have no clue about this job. What can we do to break through?

Ferrari: Some Designers Council chapters brought in a CAD supplier with computers. The students loved the challenge of routing those conductors like a Pac-Man game from years ago. We brought in assemblers with assembly videos and samples, and we did the same for fabricators. These events were very successful. Now, we are in a generation where technology is going crazy. Currently, the industry does not have replacements for older designers reaching retirement age. Seeking replacements at the college level is worth a try, but I think it's too late by then. You have to reach them at the high school level so they can select appropriately when looking at further education.

We have several Designers Council chapters now that are looking into being able to provide that, and I think it goes along with what we are talking about with the technology. It all starts with design and the younger generation. Who's flying all of our drones over in Asia and the U.S.? They keep plucking them out of the schools.

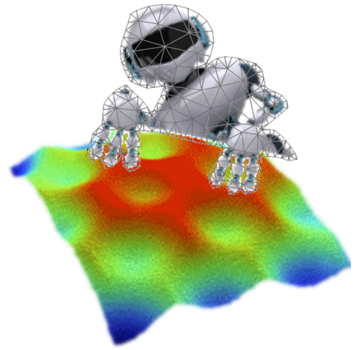
Shaughnessy: Well, I'm glad to see design and designers are getting a little bit more respect.

Ferrari: Let's see if it keeps up.

Shaughnessy: It has been great talking to you, Gary. Thanks.

Ferrari: Thank you. DESIGN007

Making Robots Sweat With Smart Surfaces That Act as Artificial Skin



The Dutch Research Council (NWO) has awarded three million euros to seven early-stage researchers in physics and chemistry through the START-UP programme. Among them, Danqing Liu, assistant professor at the Department of Chemical Engineering and Chemistry of TU/e. Liu receives nearly 411,000 euros, which will be used to develop smart surfaces that can secrete fluids and absorb them from their environment, in response to light or to electric fields. These surfaces will be used to study friction during motion, for self-cleaning systems, and for robotic and health care applications.

Inspired by the skins of living creatures, Liu develops smart surfaces that can repeatedly release and reabsorb substances under environmental stimuli, such as light and electricity.

In robots, especially in humanoid robots placing high torque demands on their motors, the generated heat represents a major constraint on their performance. Currently, engineers solve this problem by using fans or bulky radiators, which take up space and add mass. In the future, the smart surfaces developed by Liu might be used to develop artificial skins which could "make robots sweat, cool down and perform better."

Liu attempts to fill the gap between molecular sciences and material science. "I develop new materials like silicones, hydrogels and liquid crystal polymers, at submicrometer length scales," she explains. These materials are "responsive," meaning that they can sense external stimuli and adapt to those via built-in sensory systems.

(Source: Eindhoven University of Technology)

Leo Lambert on Training the Next Generation of Technologists

Feature Interview by Andy Shaughnessy
I-CONNECT007

I recently sat down for an interview with Leo Lambert, VP/technical director for EPTAC Corporation, during the IPC Summer Meetings in Raleigh, North Carolina. We discussed the company's growth, including plans to have training centers across different regions of the country to help cut down on students' travel times, and why training methods must constantly evolve to remain effective.

Andy Shaughnessy: Leo, what's new at EPTAC?

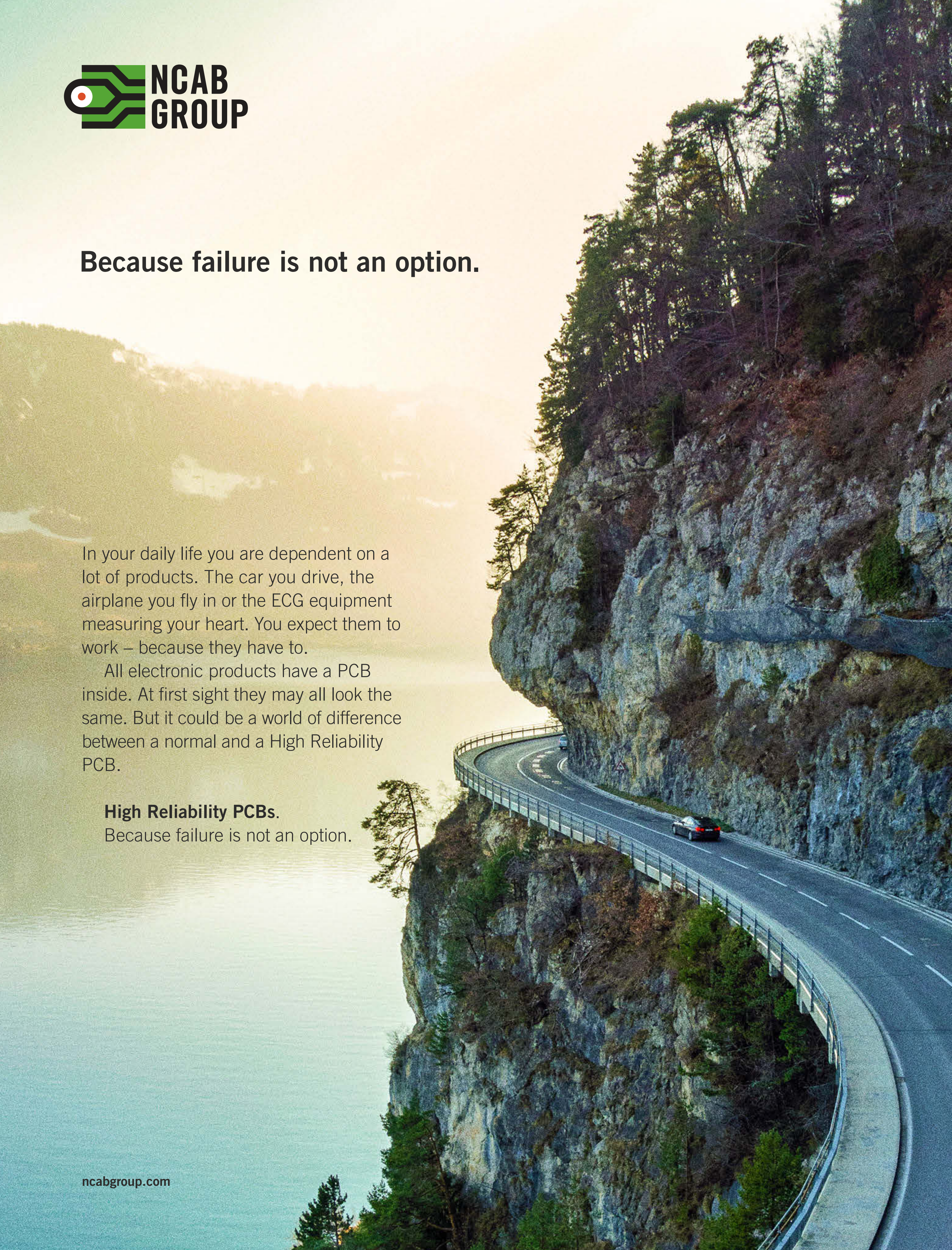
Leo Lambert: We're very busy. The issue happening now is to get instructors located around the country because it costs us a lot to travel. By having instructors in various parts of the country, we eliminate a lot of travel, so it is a big deal for our customers and us.

Shaughnessy: Because it is a service industry, more or less.

Lambert: Correct. New state government rulings on taxation for services rendered are going impact us, and we need to be aware of that impact relative to the total cost of the programs. Secondly, from a training perspective, one of the things that happened at the IPC meetings is the new IPC policies and procedures. At EPTAC, we try to follow those to the letter, but there are so many changes that we are questioning the validity of those changes; for example, they are now going to allow people to change the visual PowerPoint presentations. Individuals were not allowed to change them previously as they were all copyrighted; allowing this to happen automatically changes all of the presentations, so the commonality disappears within the same program, which will impact the industry.

So, my function is to make sure the instructors don't have four different versions for the same specification. Because if they have modified their presentations, we don't want to hear, "I heard this from instructor A. Now, I have instructor B, but instructor A said this, and instructor B said that." It becomes a big effort to standardize the presentation and knowledge being disseminated to the students.

Another issue is IPC has a philosophical bent while creating the programs to do three things:


NCAB GROUP
Because failure is not an option.
In your daily life you are dependent on a lot of products. The car you drive, the airplane you fly in or the ECG equipment measuring your heart. You expect them to work – because they have to.
All electronic products have a PCB inside. At first sight they may all look the same. But it could be a world of difference between a normal and a High Reliability PCB.
High Reliability PCBs.
Because failure is not an option.
ncabgroup.com

"Let me tell you what I'm going to tell you, let me tell you, and then let me tell you what I told you." To present the material in this fashion produces many slides. So, I question the need to have so many repetitive slides. If the instructor goes through every slide, it ends up being a long day, so again, my question is do we really need all that material to certify the individual?

Shaughnessy: And you're talking about 200–300 slides.

Lambert: Yes, and if it could be reduced by a number of slides, it would eliminate most of the duplications being experienced. Because of this repetitive process, the instructors have to balance the class during the day to compensate for the total time of the presentation. This also involves taking into consideration where the students are coming from, as we get students traveling on Sunday. Therefore, by having students and instructors around the country, we can help save the students' travel time. The other thing we're looking at is online training; we've done some, but it hasn't caught on in our industry.

The other thing we're looking at is online training; we've done some, but it hasn't caught on in our industry.

Shaughnessy: What do you see as the value in online learning? I don't think it will ever replace face-to-face training.

Lambert: There's definitely some value in online training. Anything picked up and learned is going to add value to the programs. What they, the students, want is to keep the training program interesting by having video snippets in the PowerPoint slides. With the lecture, you could also have a 30-second video to show you how it's done. We've mentioned it to the developers that they need to have physical demonstrations to provide some perspective of the application as it changes the concept of understanding.

Do you remember the days when we used overhead projectors? Then, we went to the 35-millimeter slides and PowerPoint? Some of the comments we hear today with the existing training media include "death by PowerPoint." So, we're focused on the methodology of changing the programs and the needs of the students who are going to be younger and have learned differently than we did years ago.

Shaughnessy: Are you seeing an influx of younger people in your classes?

Lambert: Yes. Unemployment is so low today that the new people coming in have no knowledge of this technology; they're learning a new dictionary with new words. Many of these people used to work at retailers, restaurants, etc., so we have to learn how to be patient and present the information in a new and interesting manner. The technique of teaching is going to change.

Shaughnessy: How involved is IPC with you as far as the actual techniques of teaching?

Lambert: One of the steps that happened back in the day was each student who was going to be an instructor had to give a platform demonstration. They had to take a section of the book and explain it by creating a beginning, a middle, and an end to the short presentation. But if you have 10 students at six minutes apiece, that's an extra hour. IPC started cutting the section as it became time-consuming, and there was no metric to measure its effectiveness.

There are programs on how to train the trainer, which are important and should be considered for new instructors. From our experience, many military people appear to have gone through some of those programs, but not as many commercial people have. This training has to happen, and IPC hasn't mentioned

creating those programs at this time. They've talked about having a "train-the-trainer" program, but they haven't talked about incorporating them as a part of any of the program or packaging them separately.

Therefore, if you're a new instructor, depending on the time as to when you're going to teach your first class, this becomes a critical issue. The instructor has to prepare, but a lot of people don't want to or don't know how to. They come to the class and say, "How do I do this and that?" IPC has changed the programs with the introduction, creation of the training programs, and distribution of their certificates and all their information is on a better software package in their office. Not only do you have to learn the subject matter with the class, but you also have to learn all about the software to become a better instructor.

For a training center, we have people doing that work, but if you're all alone working for a separate company, the instructors have to set up their own class; if they don't do it right away, they might forget. We spend a lot of time helping people to set up their programs. IPC wants to create videos to show how to do those things, but they change so often; it is difficult for them to stay current, so this is the piece in the training that's changing; new students have to learn new methodologies. We have some students say, "I don't want to do that." Well, if you don't want to do that, you're going to disappear because it has become mandatory.

Shaughnessy: I've noticed that a lot of instructors and training companies seem to be staying very busy. Is it because of subject-matter experts leaving the industry during the two downturns combined with the older technologists retiring?

Lambert: Those are two of the reasons. One reason is what you just mentioned—tribal knowledge is disappearing, and the material is newer, so you have to get the material out. The other one is the contracts, the implication being you get a contract to build something, and the customer is going to make sure the people who are going to build their products are certified and qualified. How are the contractors going to prove the validity of their certifications? How are the sub-contractors going to do their due diligence? This creates a lot of new business. Think back to 1995 when Secretary of Defense William Perry canceled all of the military documents/specifications without replacement. At the time, the knowledgeable people who could do training were still around, but that was 24 years ago, so we need to add more new instructors and get them trained in.

Shaughnessy: But you're having fun, right?

Lambert: I am. It's fun. I interview so many people, talk about different things, and find out that we all have the same problem. We work with different people. Designers understand the functionality of the product and everything else. Manufacturing folks are putting components on boards and soldering them. It makes no difference whether it goes in a radio or a rocket ship; you want to get the solder right, have it function well, and be sure the product is doing what it's supposed to do. The third thing that's going to happen is the evolution of the automotive industry. If they start going to self-driving cars, there are rumors that those companies want to go up to Class 3.

Shaughnessy: And each autonomous car will have 300 million lines of code or something like that.

Lambert: And when you're driving those cars, they want you to have your hands near the steering wheel just in case because if a mistake is made, somebody could die.

Shaughnessy: And 5G and smart manufacturing are coming too.

Lambert: This will include another technology—fiberoptics. Some people are currently working on fiberoptic specifications, but the question remains: How are we going to teach that piece? One will be able to teach the theory, but for the practicality of it, you have to talk about venting systems, etc., and many companies don't have the capability.

Shaughnessy: Do you still do a lot of traveling?

Lambert: I have one more training meeting going on tomorrow. I still teach some classes and travel to visit customers. From a training perspective, visiting customers is how you find out what they need; these are technical exchanges, such as, "They're shrinking our components down." Then, I can ask what size, they can show me, and we can determine if we need to develop training for it. You get an understanding of where the customer is and where our service is. I get to find out what they need and how to provide it. It's fun, and it's a learning process.

Shaughnessy: Some of your instructors joke about how they're going to keep teaching until they drop dead. They don't want to stop because there's no one to take their place yet.

Lambert: My wife asked, "Are you going to retire? You're not going to stay home every day of the week. You have to find something to do!" I'll do it until it's no longer fun. You share information and learn from talking to people. The bottom line is I still want to learn something every day.

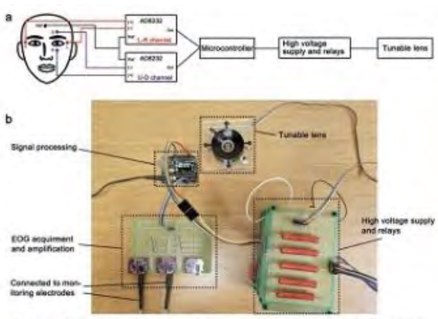
Shaughnessy: It's always great talking to you, Leo. Thank you.

Lambert: Any time, Andy. **DESIGN007**

Eye-Controlled Soft Lens Paves Way to Soft Human-Machine Interfaces

A research team led by the University of California San Diego has developed a soft robotic lens whose movements are controlled by the eyes—blink twice and the lens zooms in and out; look left, right, up or down and the lens will follow.

The lens is the first example of an interface between humans and soft machines. "The human-machine interface, as we know it, features classical machines: computers, wheelchairs, and rigid robotics, for example. The innovation here is the interface with soft robotics. This can really open up new opportunities in the field," said Shengqiang Cai, a professor of mechanical and aerospace engineering at UC San Diego who led the research.



Schematic and photo of the soft lens system. Images courtesy of Cai lab/Advanced Functional Materials

The prototype system responds to the electric signals generated around the eyes during movement, called electrooculographic signals. Patches of electrodes placed on the skin around the eyes measure these signals and transmit them through a signal processor to the lens.

The system is designed to mimic how the human eye works. The lens itself is made up of saltwater encased within two electroactive elastomer films that act like muscles. They can expand, contract, or change their structure when an electrical potential is applied. This enables the lens to look in four directions and change its focal point. Because the lens is made of soft materials, it can change its focal length by as much as 32%.

(Source: University of California San Diego)


NCAB GROUP
Because failure is not an option.
In your daily life you are dependent on a lot of products. The car you drive, the airplane you fly in or the ECG equipment measuring your heart. You expect them to work – because they have to.
All electronic products have a PCB inside. At first sight they may all look the same. But it could be a world of difference between a normal and a High Reliability PCB.
High Reliability PCBs.
Because failure is not an option.
ncabgroup.com

CA Design's Bob Chandler on Training PCB Designers

It's Only Common Sense

Feature Column by Dan Beaulieu, DB MANAGEMENT GROUP

CA Design of Santa Rosa, California, is one of the country's leading—and the most interesting—independent design service bureaus that I know. Owners Bob Chandler and his partner Robin Reynolds have worked together for many years, preaching the "gospel of Bob," which is based on the theory that all PCB designers need to be properly trained. Part of that training has to include a complete understanding of the process of creating PCBs. I caught up with CTO Bob Chandler about design, training, and what we have to do to improve the designer/fabricator relationship.

Dan Beaulieu: Thanks for spending the time with me today. First, can you tell me a little about yourself and your background?

Bob Chandler: My pleasure. I have an MBA in operations management, and I started in 1980 with a contract manufacturing company. I became interested in mechanical design and went from there to design, layout, and PCBs, of course.

Beaulieu: Why PCBs and design in particular?

Chandler: I was working in the drafting room of a large military contractor, which exposed to me to PCB design. Someone I know suggested that I start doing design full time and join him and his company, so that's what I did.


Bob Chandler

Beaulieu: After working in that for a while, you started your own company—CA Design. What drove you to go into business for yourself?

Chandler: My Dad died at a relatively young age, and that was an eye-opener for me. It made me reflect on my own life and I realized that it was better to call my own shots than to work for someone else. So, I hung my shingle, which is the nice thing about being a designer; it's very easy to start your own business—especially today—because you don't need all of the equipment we used to need to do designs and layout.

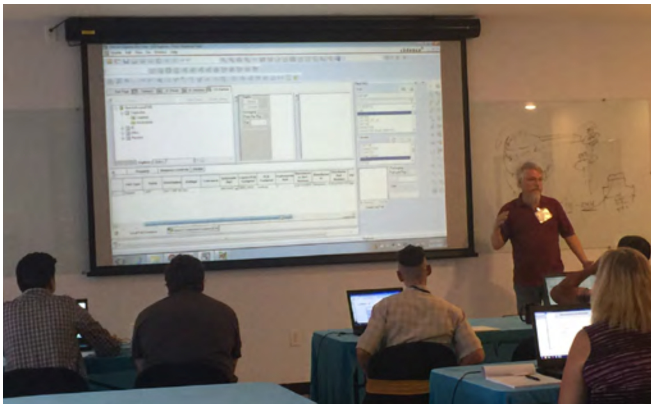
Beaulieu: Back in the day, you needed an IBM mainframe to do designs; now, you just need a good laptop.

Chandler: A couple of large monitors help, but that's pretty much right.

Beaulieu: Where were you located when you started the company?

Chandler: I was in Silicon Valley, and luckily, so much of the PC business was there. A few years ago, we moved to Santa Rosa in the wine country north of San Francisco. With the internet and screen-sharing software, I can and have worked with clients all over the world from India


NCAB GROUP
Because failure is not an option.
In your daily life you are dependent on a lot of products. The car you drive, the airplane you fly in or the ECG equipment measuring your heart. You expect them to work – because they have to.
All electronic products have a PCB inside. At first sight they may all look the same. But it could be a world of difference between a normal and a High Reliability PCB.
High Reliability PCBs.
Because failure is not an option.
ncabgroup.com



to Italy, Australia, Texas, New York, and even across the street. It doesn't matter where you are anymore as long as you are online.

Beaulieu: You do a lot of training. Has that always been the case?

Chandler: Yes, I enjoy it and know it's important. Both a proficiency in the software tool and also basic knowledge in design principles are critical if you're trying to get to market in a timely manner and come up with the most cost-efficient, reliable product. Companies are just not as focused on training their designers as they once were, and you can see it in the designs. Many times, younger designers have no clue what they are designing and how it all needs to work.

Beaulieu: I know exactly what you mean. I work with a lot of board houses, and the design data packages they are getting are worse than ever. One industry expert made the observation last year that less than 10% of the data packages that shops receive are flawless.

Chandler: I completely agree. Now, some companies want to get their engineers to start doing the layout and not even using trained designers to try to save money and short-cut the process as much as possible. In the end, that's short-changing the client.

Beaulieu: In your training experience, what is the most unusual thing that has happened?

Chandler: One time, I was doing a one-day training for a large corporation with a class of 15 designers. After the morning session, everyone left for lunch. But when I came back for the afternoon session, only one designer was sitting there. When I asked him what happened, he told me that they had all been laid off during the lunch break. I asked him why he was still there. He said that he had been laid off too but wanted to complete the course so he could put it on his updated resume he would be writing.

Beaulieu: I also knew a lot of times like that. Stick around long enough you'll see it all. What platforms do you design on these days?

Chandler: We work with OrCAD and Allegro primarily for both design and training services.

Beaulieu: What kind of companies do you design for?

Chandler: We design for a wide range of companies from the biggest corporations to high-tech startups and single consulting engineers. Our clients have ranged from Apple to Xerox, Intel, IBM, Teledyne, Raytheon, HP, Lockheed, etc. The "elevator speech" Robin Reynolds, our director of operations, gives introducing me to classes is, "With over 35 years in business, we've designed boards that are on other planets, under the ocean, and in your phones, homes, and cars. We've had both Bill Gates and Steve Jobs stand on stage to introduce devices we've laid out boards for, and we've designed for exciting new hi-tech wearables and mundane testing equipment."

Beaulieu: And you train designers at these companies as well?

Chandler: Yes, we train primarily in the Cadence Allegro software tool geared toward preparing the student to hit the ground running.

Beaulieu: One of the things I'd like to come back to is the relationship between the board houses and designers. How do you rate that relationship in terms of importance?

Chandler: It's of the utmost importance. I tell my classes that they need to get into board houses, take tours, find out how they build their boards, and ask them about laminates to use, especially the special thermal and high-temp materials. They need to find out everything they can about how a board is built and then design accordingly. Many companies today are having their engineers and designers work in a vacuum. They completely ignore the people who are the experts in building the products they are designing. It makes no sense to me. This I why in my classes, I urge company collaboration between the PCB designer and fabricator. I go even further and urge them to include the engineers as well.

Beaulieu: Based on this philosophy, you are also initiating a collaboration with a PCB fabrication shop, right?

Chandler: Yes. We are planning to start producing columns that will be co-authored by Mark Thompson of Prototron Circuits in Redmond. I will represent the designers, and Mark will represent the fabricators, and we will create a series where we give examples of the importance of designers and fabricators working together. Mark and I even plan to make a video of the both of us at Prototron doing a comprehensive tour of the facility, showing how a board is built and what the takeaways should be when a designer tours a board shop—as all designers should. It should be fun.

Beaulieu: And more importantly, very helpful for not only designers and shops but for the entire industry as well. How do you see your design world today and going forward?

Chandler: One of the things that has helped us has been the ability to work with companies all over the world through the Internet. But it has also hurt those of us in this country some because it has increased the global competition. And the other thing is that some companies are trying to get their engineers to do board layout themselves, eliminating the trained designer, which is often hard on everyone. It's the same as if you would expect the architect to not only design the building but build it as well. The engineer has to reinvent the wheel in every phase of the layout using a tool they're generally unfamiliar with. A seasoned designer knows a thing or two because they've seen a thing or two. Especially in a service bureau, the designer can leverage all they've learned from the thousands of different types of designs they've done through the years to expediently deliver an accurate DFM layout.

Beaulieu: And this is all coming at a time when the designs are getting smaller and denser with many more chip-on-board features, which is not a good time to be pulling the designers from the process.

Chandler: Exactly. We have to coordinate better with the board houses and end users, and we have to train these designers today. Most of the new designers today are learning online, even through YouTube. They are not getting face-to-face, hands-on training that it takes to be a great designer. I am trying everything I can to reach new designers. I would love to see a good school program based on PCB design. I don't know of one, do you?

Beaulieu: I don't either, but maybe we ought to start one. What do you say, Bob?

Chandler: I'm game if you are!

Beaulieu: Right. Well, thank you for your time today.

Chandler: You too. **DESIGN007**

Dan Beaulieu is president of D.B. Management Group. To read columns by Beaulieu or to contact him, click here.

Pulsonix Is Bullish on Next-gen Designers

Feature Interview by Barry Matties
I-CONNECT007

During a recent trip to the U.K., Barry Matties spoke with Bob Williams, the managing director of Pulsonix, about training the new generation of designers. He explained how the company reaches out to local high schools, colleges, universities, and user groups to advocate for careers in PCB design and manufacturing.

Barry Matties: One topic we've been following recently is training for PCB designers. And as we see the generational shift taking place, we're looking at how new designers are coming in and where they're being trained. It looks like there's a real deficiency out there.

Bob Williams: There is.

Matties: How can we address that?

Williams: A lot of people look at YouTube for training. But the problem with YouTube is that it shows you the feature and gives you the steps to follow, but it often doesn't explain why things are done one way versus another. From the years of training that we've done, we've learned where our customers struggle with the user interface, and we've made improvements to make it easier. As a result, we've built a very intuitive user interface allowing us to provide training that focuses more on using our toolset for good PCB design rather than just how to use our tools.

Matties: Where are designers getting that training?

Williams: We're seeing universities pick this up. They're looking at courses for electronics design and they're trying to run very small modules. Some of the new university courses in the U.K. are running courses for how to design

PCBs, including everything from what vias are to why you do or don't use microvias. And we run training courses on PCB design not specific to Pulsonix. The training covers what the board is made up of, the facets of the board if you like, and once they have an electronic concept, how they design the board itself through to fabrication.

Matties: Are you seeing a lot of interest from young people to move into this as a career?

Williams: No, and that's the problem. Today, younger people we're seeing want glory jobs, not what might be considered traditional engineering jobs, such as electronics. And they tend to be single-disciplined as well. A graduate might be a capable electronics designer theoretically, but they lack the real practicalities of core PCB design and knowledge of the whole process.

A graduate might be a capable electronics designer theoretically, but they lack the real practicalities of core PCB design and knowledge of the whole process.

Matties: So, the only thing that's going to drive that is the wage for designers going up so somebody who's salary-driven will look at that, right?

Williams: Potentially, yes. However, if they can see over the initial salary and look beyond this, the future is very good for them in the electronics industry. It opens up a lot of very lucrative avenues.

Matties: Because if there's not a strong appeal, what's left?

Williams: I think it's the same in the U.S.; you also have a shortage of good PCB designers. The only place that I haven't seen shortages is in China and India, but then they are people-rich.

Matties: Well, there are 3 billion people between those two (laughs).

Williams: There's a problem with all sorts of skills in Europe and the U.S. It's almost like a modern spin on the traditional trades and skills of plumbing, carpentry, bricklaying, etc., that were around 60–70 years ago. Now, the new skills are electronic design, mechanical design, CAD, and these types of disciplines.

Matties: We were starting to talk about training and bringing new people into the industry. And you're right that we have the same issues in America. We see IPC and others reaching not just into high schools but even into elementary schools to start introducing them into this area. The fastest way to bring kids into this industry is to create a video game that they start playing as a kid. I mean, look at where your RC cars took you.

Williams: That's true.

Matties: We're watching kids build cities, roller coaster parks, etc., through video games.

Williams: And they say, "I'd like to be an architect." In the U.K., senior school education starts at 11, so it's basically 11- through 18- or 19-year-olds. They teach basic electronics/circuits with a piece of card where they draw the circuit and punch the holes in it. Sometimes, they prototype them, but not very often. Even my daughter understands our business because of the use of the electronics CAD package (DesignSpark) they used at school. They taught them the basics of how to build the board, and then they built it as one of the educational elements of a module in their schooling.

Matties: It's a life skill at this point. As you look at the way the world is evolving to Industry 4.0, on average, people have five or six sensors or internet connection points right now. In a couple of years, they're going to have hundreds, and there's opportunity surrounding all of that for the youth.

Williams: And IoT is a big thing because kids have been brought up with it. Even though they don't know it as IoT, they still have everyday technology that just happens.

Matties: And we have our electronic "dog tags" that we carry around with us. We are so connected. And for you and me, we're from of a generation where it wasn't always that way.

Williams: Definitely not; it's a new thing to which we've adapted.

Matties: And some kicking and screaming because we understand the illusion of privacy now where kids today are growing up online and wide open. It's a completely different attitude.

Williams: For our generation, we've had house phones and public payphones through to the generation of PCs and smartphones. Now, we have smart gadgets in our hands all the time. For example, I always travel with my iPad; it connects me everywhere.

Matties: Why would you leave it behind?

Williams: That's right. It gives me everything I need. I can see who's at my door because I have a camera on the door for security. You can also see if your children are okay, such as if they're infants and lying in their crib. It's phenomenal.

Matties: And every one of those needs a designed circuit board. It's a huge opportunity, but it's not viewed as a glamorous position.

Williams: Right. Wearable technology today is also phenomenal because of what we can do with wearables to make everyday life more bearable.

Matties: You're a tool provider. Are you taking part in the local schools and going in and educating them?

Williams: We do it at high school and university levels.

Matties: How does that work? What's your typical approach?

Williams: As a local company, we would give a talk on something that the students may find interesting, challenging, scientific, and technological. We pitch it as something that's a possibility to explore, whether it's electronics, the design, or something mechanical.

Matties: Is the strategy to teach the career counselor about what this is?

Williams: That could be an idea. We haven't done that yet, but we tend to go to them.

Matties: Because if the career counselor understands the opportunities in a stronger way, then they could be a champion voice in the school. If you have 100 schools and you reach each of their career counselors, you'll be able to reach 100,000 kids.

Williams: And we're seeing more women coming into it as well, which is great.

Matties: Because not every kid is going to go to university or find a glamorous job. But with a job like you're talking about with an eight-month to a one-year training program, they're going to walk out making 70, 80, or 90 thousand dollars, for example, each year.

Williams: That's right. We have a continual process of recruiting, and what we're looking for is not somebody with a degree necessarily, especially for coders. We're looking for somebody with an interest in using software. Have they designed projects outside of college? We have an interesting set of challenges that we pose to them. And we do live code tests, too, to see how they process the problems that we're looking for and their level of interest.

Matties: I was at a local design house last week, asking them how they were finding designers, and they said they're not. They're bringing people in; they've created their own training curriculum, and they have to train their own designers.

Williams: We're definitely seeing that because others send those people to us. We train them on PCB design first, and then we train them on Pulsonix.

Matties: We talked about training and tools. What advice would you give a designer in this market space?

Williams: Do your research. Ensure you understand the full process and learn the tool before you start, especially for new designers. Also, understand what you're designing in terms of how it's going to be physically made, and then try and understand some of the manufacturing constraints and rules. Talking to your manufacturer earlier is important too; sooner is better than later.

Matties: And that's great advice. But sometimes, they may not know who the fabricator will be when they're designing the board. What advice would you give for that situation?

Williams: Talk to a couple of people and take a consensus because you'll generally find that a low-cost manufacturing process will have certain rules, and I expect those rules are applied across multiple manufacturers. If you're doing tracking gap for 8–10 mils, that's going to be translated across multiple vendors for making boards, regardless of whether they're a prototype or not. And whether you're doing a two- or four-layer, you must understand the ramifications of doing a four-layer rather than a six-layer or a two-layer instead of a four-layer due to the costs and the technologies involved, what drilling sizes you need, etc.

Matties: Bob, thanks for speaking with me today.

Williams: Thank you very much for coming in. I appreciate you taking the time. **DESIGN007**

Physicists Make Graphene Discovery That Could Help Develop Superconductors

When two mesh screens are overlaid, beautiful patterns appear when one screen is offset. These "moiré patterns" have long intrigued artists, scientists and mathematicians and have found applications in printing, fashion and banknotes.

Now, a Rutgers-led team has paved the way to solving one of the most enduring mysteries in materials physics by discovering that in the presence of a moiré pattern in graphene, electrons organize themselves into stripes, like soldiers in formation.

"Our findings provide an essential clue to the mystery connecting a form of graphene, called twisted bilayer graphene, to superconductors that could work at room temperature," said senior author Eva Y. Andrei, Board of Governors professor in Rutgers' Department of Physics and Astronomy in the School of Arts and Sciences at Rutgers University-New Brunswick.

Graphene—an atomically thin layer of the graphite used in pencils—is a mesh made of carbon atoms that looks like a honeycomb. It's a great conductor of electricity and much stronger than steel.

Using a technique invented by Andrei's group to study twisted bilayer graphene, the team discovered a state where the electrons organize themselves into stripes that are robust and difficult to break.

"Our team found a close resemblance between this feature and similar observations in high-temperature superconductors, providing new evidence of the deep link underlying these systems and opening the way to unraveling their enduring mystery," Andrei said. (Source: Rutgers University)

PCB007 Highlights

The PCB Norseman: From Wooden Huts to Homemade GO-karts—It All Srarts With Design! ▶

Whether building the coolest go-kart or the most sophisticated electronic hardware, the story is the same: It starts with design. And for designers and manufacturers, early involvement and commitment between all the involved parties in a product development process diminish the risk for mistakes and misunderstandings.

Dissecting the IPC Regional Survey on PCB Technology Trends ▶

Sharon Starr, Denny Fritz, and Mike Carano talk about the global 2018 IPC Technology Trends Report released early this year—the size of the survey, how it was conducted, the general findings, and regional differences. They also shared their takeaways and regional insights, and the industry outlook over the next five to 10 years.

An Examination of Glass-fiber and Epoxy Interface Degradation in PCBs ▶

Multilayer organic laminates, which make up over 90% of the interconnecting substrates in electronics (standard FR-4 represents 85% of the substrates used for laminates), can develop a loss of electrical insulation resistance between two biased conductors due to conductive filament formation.

Microvias: Links of Faith are Not Created Equally ▶

Microvias connect adjacent copper layers to complete electrical paths. There are copper-filled microvias, which can be stacked to form connections beyond adjacent copper layers, and staggered microvias, which stitch adjacent copper layers with paths that meander on the layers between the microvias. This article discusses the various laser-drilled microvias and presents SEM photographs to begin the search for the root cause of weak copper interface.

EPA, Industry Come Together in Visit to TTM Facility ▶

IPC member TTM Technologies is proud to show off the new wastewater-treatment system at its Sterling, Virginia plant, which is helping to enhance the company's pollution prevention and resource recovery performance.

Avoiding CAF Failures at the IPC High-reliability Forum ▶

Foresite CEO Terry Munson recently spoke with Andy Shaughnessy during the IPC High-Reliability Forum and Microvia Summit in Baltimore about his presentation on the causes of conductive anodic filament (CAF), the dangers of resin starvation, and what advice he'd give to PCB designers to avoid those types of failures.

Global PCB Market Analysis and Outlook ▶

The global PCB market is expected to reach an estimated $89.7 billion by 2024 with a CAGR of 4.3% from 2019 to 2024.

It's Only Common Sense: It's Time for Summer Reading ▶

Some people go to the beach excited about their summer reading. When Dan Beaulieu goes to the beach (yeah, right!), his summer reading involves much more useful, and for him, fun (believe it or not) business books on customer service, business strategy, and my personal favorites, advertising, and marketing. Here are some of the best!

Teaching Young Girls About STEM Careers

Feature Interview by Andy Shaughnessy
I-CONNECT007

Andy Shaughnessy recently spoke with Brooke Campbell, R&D chemist from BTG Labs, during the IPC Summer Meetings in Raleigh, North Carolina. Brooke discussed her efforts to get more young girls involved in STEM careers through the Girl Scouts and other learning opportunities, and she offered advice for technologists who are interested in drawing young people into these rewarding careers.

Andy Shaughnessy: I'm here at the IPC summer meetings with Brooke Campbell from BTG Labs. Nice to meet you, Brooke.

Brooke Campbell: Nice to meet you too.

Shaughnessy: I enjoyed your story about how you spoke to the Girl Scouts of USA about STEM and trying to get girls into science and math careers. How did you get involved with that?

Campbell: My local Midwest Chapter reached out. They do an annual leadership conference where they get groups of several hundred girls in the Greater Tri-State area together and take over a hotel. They have different sessions on gardening, skill-building, etc. They were looking for, and struggling to find, women to support the STEM area, to speak to younger ladies about pursuing a career in STEM and what that entails as well as some of the challenges that we face being in a STEM field. It was mentioned earlier today that there are six women here out of a conference with 300 people. That is one of the challenges associated with being in the industry and being a woman.

The girls there made comments such as, "It's too hard," or, "I've been told that's a hard subject matter." Sciences can have challenges, but we spoke to that and how it doesn't have to be as hard as they think it is. Some of the mathematics and fundamentals can take some time to learn, but saying to someone who is younger, "This is hard, so don't pursue it," is such a huge disservice. We talked a little bit about the scientific method and observation, and how that can be applied to the medical industry where doctors do small experiments all the time to see what works to help someone if they have a health issue. We also covered experimentation in the industry or on a big project and everything that goes into that.

It was great to hear the feedback from the girls. They enjoyed it. Three years ago was the first year that they were able to find someone in STEM. We're trying to find other local females who are in engineering or a hard science field to share with these girls what it means to be a scientist or an engineer and to encourage them. Becoming a scientist/engineer is something that's obtainable. It's not just geared towards men. You have the skills and brainpower to do this.

They also liked the hands-on aspect. I did an acidic solution project. We used lemon juice on paper and made an invisible ink. Then, if you apply heat, the acid from the lemon oxidizes at a different rate than the paper. You can bring about a secret message. They requested a more messy and explosive experiment for next year. It's just those little things that they can get their hands on that can be impactful. You can demonstrate a fundamental scientific concept to someone who is young; hopefully, that piques their interest. My main goal was to get them interested.

Shaughnessy: Right. Now, you're a chemist. How did you get interested in this field?

Campbell: I was always studious. My first bachelor's degree is in nursing and I took several semesters of anatomy and physiology. I had a professor who was very difficult and bright. After three semesters of those classes, our final was to take a chocolate chip cookie and put it through the body. You had to name all of the structures of the body that the cookie encountered and name the biological processes and enzymes. There was a lot of biochemistry, and I loved it. It was a huge essay, but I thought it was a good way to summarize what you learned over a year and a half in anatomy and physiology.

She always encouraged me, saying, "Brooke, you have a great brain. You should go into a harder science." I said, "I struggle with math sometimes." She said, "Yeah, but you can overcome that. It doesn't have to be a hindrance." I worked as a nurse for about 18 months, and while I greatly respect people in the medical community and wanted to help people, I found that nursing just wasn't a good personality fit for me, so I went back to school. I considered molecular biology or chemistry. After interning at a molecular biology facility at the University of Cincinnati, I found out that the market was pretty flooded at the bachelor's and master's levels. I knew some Ph.Ds. who were struggling to find full-time work, so I thought, "This isn't as lucrative as I want."

I took a pragmatic approach to what I went back to school for. I started enrolling in more chemistry classes. I thought, "This is lucrative at the bachelor's level, and this is at a master's level." Organic chemistry kind of stinks and I'm saying that as a chemist. Once I started getting to a higher level, understanding how everything interacts and how molecules come to be, it's beautiful. The whole universe is based on energy, and it correlates well to symmetry in molecules. It fascinated me.

A lot of people say, "You're smart," but I had to work hard. I took some classes over again because differential equations are difficult. I've been fortunate to work in a heavy R&D position for the past three years with BTG, helping customers solve adhesion issues from a surface chemistry standpoint. I've also been involved with bonded structures, electronics, implantable medical devices, and the automotive world. I love solving problems; I always loved puzzles as a kid. And I learn something new every day. It keeps me on my toes, so I will be starting a Ph.D. program in materials science and engineering in the fall.

Shaughnessy: You'll have two bachelor's degrees and a Ph.D.

Campbell: Yes, if I survive (laughs).

Shaughnessy: Wow. Good for you. You'd be a good one to speak to the girls.

Campbell: I hope so. I have a lot of nieces and nephews, and I try to encourage them. We do science projects together. As I said, it's hard to get in the school system sometimes to get an after-hours class. If people can find a way or have a connection to do that, that's a good outreach as well.

Shaughnessy: Do you have any advice for any young women, or men, who are considering a STEM career?

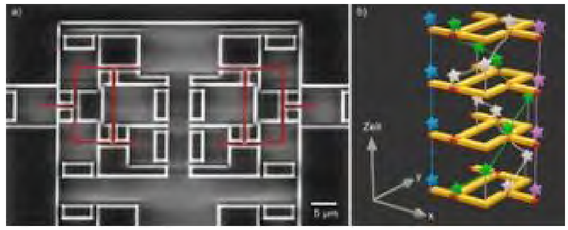
Campbell: If there's a career day at school, ask about it and pursue it. A lot of times, a kid is not going to come to an adult and say, "We're doing this." Be the initiator. Is there a platform where I can speak? Or if you know of local chapters for either youth groups or the Girl Scouts of the USA or the Boy Scouts of America, they usually have an annual workshop every year. Reach out to the local chapter. I would guess that they would be enthusiastic to have someone speak. Also, if you have children in your own family, speak to them, especially the younger girls. I try to share engineering concepts. My nephew once said, "I want to go to the Moon." I responded, "Your aunt works on spaceships."

Further, there are some toys out there that are geared towards engineering for girls and boys. They're fun and teach basic engineering concepts. There are also science museums and other organizations with programs and summer camps geared towards engineering. Those are all good places to start. Look for little ways to be an advocate.

Shaughnessy: Thank you for your time today. This has been great.

Campbell: Thank you. **DESIGN007**

Novel Process for Structuring Quantum Materials



Implementing quantum materials in computer chips provides access to fundamentally new technologies. In order to build powerful and fault-resistant quantum computers, it is possible, for example, to combine topological insulators with superconductors. This process step brings with it some challenges that have now been solved by researchers from Jülich.

Even the ancient Inca used knots in cords to encode and store information in their ancient writing "Quipu." The advantage: unlike ink on a sheet of paper, the information stored in the knot is robust against external destructive influences, such as water. Even novel quantum computers should be able to store information robustly in the form of nodes. For this, however, no cord is knotted, but so-called quasi-particles in space and time.

What is needed to build such a quantum node machine are new materials called quantum materials. Experts speak of topological insulators and superconductors. The so-called "Jülich process" makes it possible to combine superconductors and topological insulators in ultrahigh vacuum to produce complex components.

Initial measurements in their samples indicate evidence of majorana states. "Majoranas" are exactly the promising quasiparticles to be knotted in the shown networks of topological insulators and superconductors to enable robust quantum computing. In a next step, the researchers of the Peter Grünberg Institute, together with their colleagues from Aachen, the Netherlands and China, will provide their networks with readout and control electronics in order to make the quantum materials available.

(Source: Forschungszentrum Jülich)

The Curse of the Golden Board

Beyond Design

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

At DC and low frequencies, the inductance of transmission line paths can be ignored. However, as the frequency and rise time increase, we soon realize that the multilayer PCB is not an ideal environment to transfer high-speed data. Here, parasitic capacitance and inductance plague the most basic of designs. Inductance, in particular, impacts on virtually all signal and power integrity issues.

To optimize the physical layout for acceptable performance, inductance must be minimized:

1. The mutual inductance between signal paths intensifies switching noise
2. The inductance of the power distribution network (PDN) bypass and decoupling capacitors dramatically affects product performance and reliability
3. The effective loop inductance of the return current paths impacts on electromagnetic (EM) emissions

By understanding how the physical PCB layout influences the degree of inductance, the PCB designer can triumph over their arch-enemy.

Electric fields and magnetic fields play an equal role in moving energy in a multilayer PCB. EM fields also move energy in free space but not at DC. The presence of voltage implies that there is an electric field, and the changing of that electric field creates a magnetic field. What may not be appreciated is that moving a voltage between two components requires moving energy (not a signal), which requires the existence of both electric and magnetic fields. When energy is not moving, the magnetic field is zero.

1. Mutual Inductance Between Signal Paths

When current flows in a conductor, there is a magnetic field. When a second conductor, carrying current, is brought into close proximity, there is a force between the two. If both cur-

rents flow in the same direction, then they are additive (think two parallel trace segments). When the currents flow in the opposite direction (think a trace over a return plane), the currents cancel. This implies that two individual traces should be kept well apart to reduce crosstalk whilst a signal trace should be tightly coupled to its return path (plane) to increase coupling and reduce inductance.

Parasitic inductance is often an afterthought in high-speed design. A substrate consisting of conducting and dielectric materials will have some parasitic inductance, possibly leading to problems like crosstalk, induced currents, noise coupling, and other effects that degrade signal quality.

Unfortunately, parasitic capacitance and inductance in a PCB are unavoidable. A PCB is composed of a number of parallel conducting elements that are separated by an insulator, basically forming a capacitor. Likewise, conductors on a PCB will inevitably form complete loops, creating an equivalent inductor. While making dielectric layers in the stackup thinner will decrease the loop area and the parasitic inductance, it will also increase parasitic capacitance. Therefore, one needs to choose the sweet spot where inductance is minimized, and capacitance is maximized.

In high-speed digital applications where multiple data lines can run at tens of Gbps, parasitic capacitance and inductance can produce impedance mismatch along the signal path. Any mismatch caused by parasitics will produce reflections along the transmission line, ultimately increasing timing jitter and bit error rates.

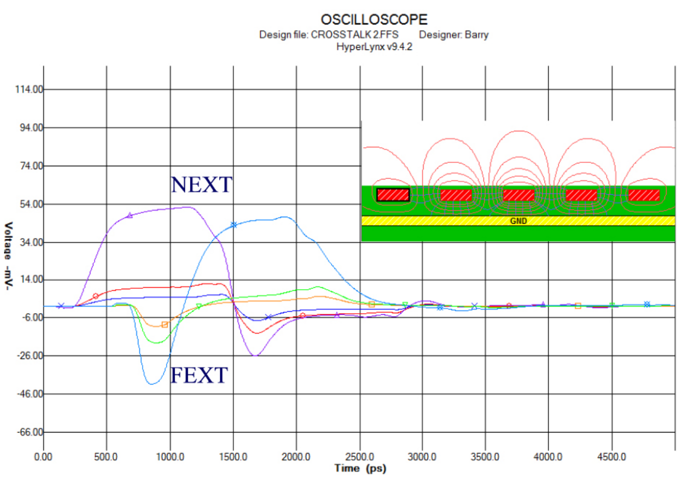


Figure 1: Near and far-end crosstalk for microstrip (simulated in HyperLynx).

Figure 1 shows the near (NEXT) and far-end (FEXT) crosstalk for the victim traces adjacent to the aggressor trace (1.5V at 1 GHz). In this case, the traces are 4 mils wide with 4-mil spacing and have a 40-ohm impedance. As the victim trace gets farther away from the aggressor, the crosstalk decreases. The self-inductance line rings are those field line rings around a trace that arise from its own current only, whilst the mutual inductance line rings are the magnetic field line rings completely surrounding a trace that arise from another trace's current; these cause the crosstalk. Crosstalk creates ringing, which creates electromagnetic radiation.

2. Inductance of the Power Distribution Network

Also, as the frequency and rise times increase, the AC impedance of the PDN increases due to the inductance of the bypass and decoupling capacitors attached to the planes. Every capacitor has an equivalent series inductance (ESL), which causes its impedance to increase at high frequencies. Bulk bypass capacitors provide low impedance up to ~10 MHz. High-frequency decoupling is provided by ceramic capacitors up to several hundred MHz, but above that, only the planar capacitance can reduce the PDN impedance. The power-to-ground plane capacitance of the PCB provides an ideal capacitor in that it has no series lead inductance and little equivalent series resistance (ESR), which helps reduce noise at extremely high frequencies, providing tight coupling (< 5 mils) between planes creates valuable capacitance at high frequencies.

Capacitors reach their minimum impedance at their resonant frequency (Figure 2), which is determined by the capacitance, ESR, and ESL. To meet the PDN target impedance at a particular frequency, a capacitor is chosen so that when mounted on the PCB, it will resonate at the desired frequency and have an impedance

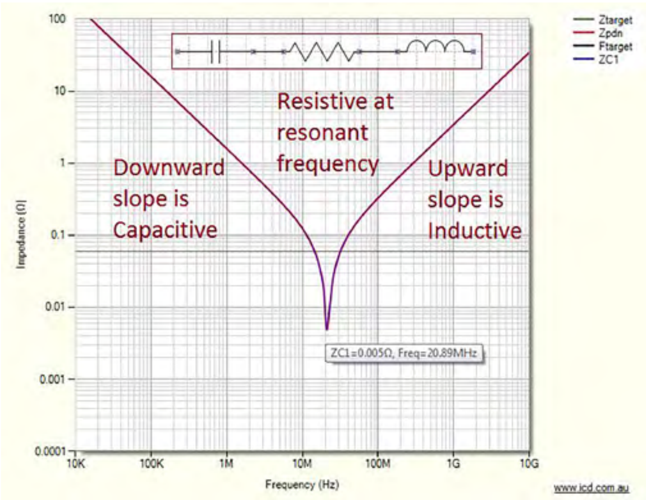


Figure 2: A capacitor has series capacitance, resistance, and inductance (iCD PDN Planner).

that is equal to its ESR. Then, a sufficient number of those capacitors are placed in parallel so that the accumulated ESR's approach the desired target impedance.

In addition, the mounting inductance of a capacitor is comprised of three components: capacitor footprint, capacitor height above or below the plane, and power plane spreading inductance. These three elements describe the loop that current must flow; the bigger the loop, the more the inductance. The footprint (land pattern) for a capacitor dominates the ESL. It consists of via placement with respect to the pad, the length and width of traces connected to the pad, and the way the vias are connected to the power and ground planes. The location of the power/ground planes in the PCB stack-up controls the length of the vias; this is why it is always best to place the decoupling capacitors on the same side of the board as the BGA for high layer-count stackups. Inductance directly depends on the magnetic field, so reducing the energy associated with the loop area reduces overall inductance.

Figure 3 illustrates 0402 capacitors with different fanout patterns. End vias are the worst case where the loop area (inductance) is the largest. This loop area can be reduced by placing the vias closer to the land side and even more so by placing double vias either side of the pad basically halving the inductance. The final case is that of via-in-pad, which reduces the loop area dramatically, but caution should be used as not all assembly shops support this. Placing double vias has approximately the same inductance as using the via-in-pad. Also, vias should be directly connected to the plane rather than by thermal reliefs, which adds to the inductance.

3. Loop Inductance of the Return Paths

PCB designers, generally, take great care to ensure that critical signals are routed exactly to

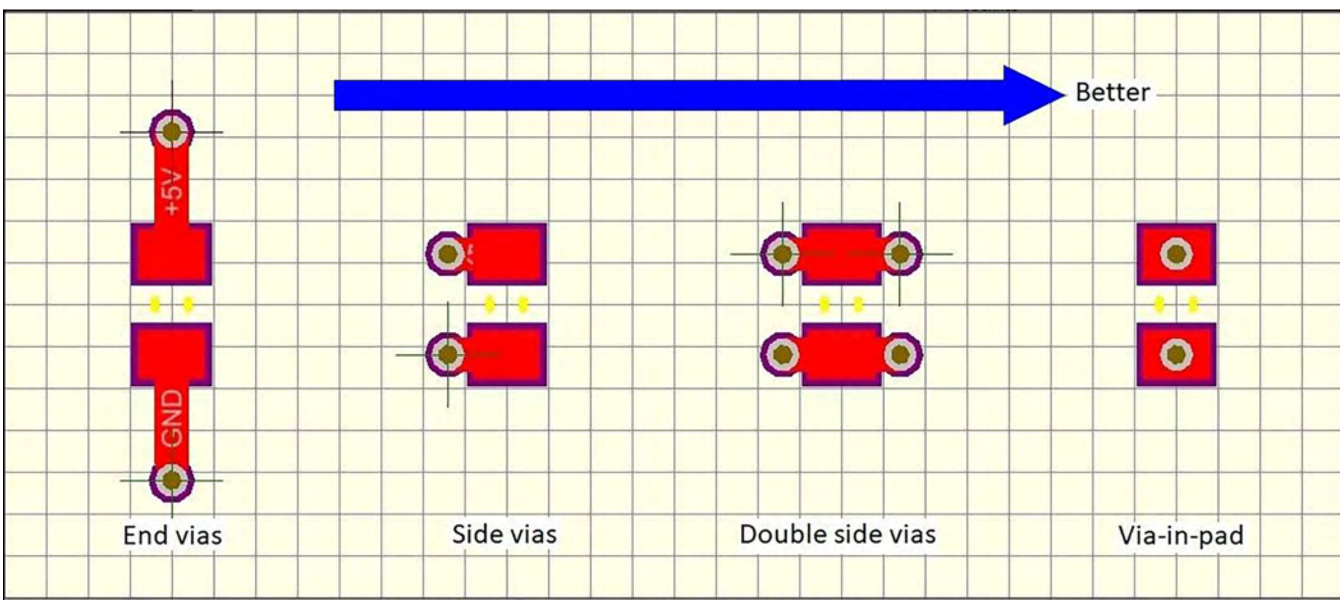


Figure 3: Capacitor mounting and via fanout.

length from the driver to the receiving device pins but take little care of the return current path of the signal. Current flow is a round trip, and the important issue is delay, not length. If it takes one signal longer for the return current to get back to the driver—such as around a gap in the plane—then there will be skew between the critical timing signals. Return path discontinuities (RPDs) can also create large loop areas that increase series inductance, degrade signal integrity, and increase crosstalk and electromagnetic radiation.

Small discontinuities, such as vias and non-uniform return paths on a bus, are becoming an important factor for the signal integrity and timing of high-speed systems. RPDs produce impedance discontinuities due to the local return inductance and capacitive changes. Impedance discontinuities create reflected noise, contribute to differential channel to channel noise, and may promote mode conversion. In the case of differential pairs, the transformation from differential-mode to common-mode typically takes place on bends, and non-symmetrical routing near via and pin obstructions, but can also be caused by small changes in impedance due to RPDs.

Common mode radiation is the result of parasitics in the circuit, which emanate from the unwanted voltage drops in the conductors. As the signal is driven down the transmission line, capacitive coupling between the trace and plane conductors completes the loop and displacement current flows through the capacitance which returns to the source. The common-mode current that flows through the ground impedance produces a voltage drop in the digital logic ground system and generates magnetic radiation.

To control common mode radiation, it is important to minimize the common mode ground voltage at the source. Also, good grounding minimizes noise sources by presenting common mode currents with a low impedance path to ground potential. If the return path of a common mode current is far from the signal path, then the common mode current will radiate. However, if you engineer the return path to be in close proximity to the source current, then the loop area will be small; therefore, the common mode current will not radiate.

In conclusion, parasitic effects can be minimized by separating traces as much as possible, coupling signal traces close to the reference planes, reducing the loop area of return current, using good stackup design practices, and lowering the AC impedance of the PDN by minimizing the decoupling capacitor mounting inductance.

Key Points

- Inductance, in particular, impacts virtually all signal and power integrity issues
- Moving a voltage between two components requires moving energy (not a signal), which requires the existence of both electric and magnetic fields
- Two individual traces should be kept well apart to reduce crosstalk whilst a signal trace should be tightly coupled to its return path (plane) to increase coupling and reduce inductance
- Parasitic capacitance and inductance in a PCB are unavoidable
- Parasitic capacitance and inductance can produce impedance mismatch along the signal path
- As the frequency and rise times increase, the AC impedance of the PDN increases due to the inductance of the bypass and decoupling capacitors attached to the planes
- The power to ground plane capacitance, of the PCB, provides an ideal capacitor in that it has no series lead inductance and little equivalent series resistance, which helps reduce noise at extremely high frequencies
- Capacitors reach their minimum impedance at their resonant frequency
- A capacitor is chosen so that when mounted on the PCB, it will resonate at the desired frequency
- The footprint (land pattern) for a capacitor dominates the ESL
- The location of the power/ground planes in the PCB stackup controls the length of the vias; this is why it is always best to

place the decoupling capacitors on the same side of the board as the BGA for high layer-count stackups
- Capacitor mounting loop area can be reduced by placing the vias closer to the lands
- Vias should be directly connected to the plane rather than by thermal reliefs, which adds to the inductance
- Current flow is a round trip, and the important issue is delay, not length
- Return path discontinuities can also create large loop areas that increase series inductance
- RPDs produce impedance discontinuities due to the local return inductance and capacitive changes
- If the return path of a common mode current is far from the signal path, then the common mode current will radiate

Further Reading

- Olney, B. "Beyond Design: Crosstalk Margins," *Design007 Magazine,* July 2018.
- Olney, B. "Beyond Design: Power Distribution Network Planning," *The PCB Magazine,* May 2012.
- Olney, B. "Beyond Design: PDN Planning and Capacitor Selection, Part 1," *The PCB Design Magazine,* December 2013.
- Olney, B. "Beyond Design: PDN Planning and Capacitor Selection, Part 2," *The PCB Design Magazine,* January 2014.
- Olney, B. "Beyond Design: Return Path Discontinuities," *The PCB Design Magazine,* April 2017.
- Olney, B. "Beyond Design: Common Symptoms of Common-Mode Radiation," *Design007 Magazine,* May 2018.
- Morrison, R. *Fast Circuit Boards: Energy Management,* John Wiley & Sons, 2018.
- Cadence PCB Solutions. "How Parasitic Capacitance and Inductance Affect Your Signals," Cadence Design Systems Inc., May 2, 2019.
- Bogatin, E. *Signal and Power Integrity: Simplified,* Prentice Hall, 2008.
- Johnson, H. W., & Graham, M. *High-Speed Digital Design: A Handbook of Black Magic,* Prentice Hall, 1993. **DESIGN007**

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

Cooling for Quantum Electronics

The startup kiutra GmbH is the first company in the world to have succeeded in developing a permanent magnetic cooling system to reach temperatures close to absolute zero. Such temperatures are, for example, required for the operation of quantum computers. The system was set up by a team of researchers from the Physics Department at the Technical University of Munich (TUM).

TUM researchers Alexander Regnat, Jan Spallek, Tomek Schulz, and Professor Christian Pfleiderer are seeking to meet that demand. All four are currently working on their prototype at the TUM Physics Department. According to Alexander Regnat, there is already the prospect of taking on more staff and setting up separate headquarters.

The team of scientist came up with the idea during their work at the TUM. Again and again, they were faced with the limits of conventional methods for reaching such low temperatures. Therefore, the group developed its own technology to ensure permanent cooling and founded kiutra GmbH in the summer of 2018.

Concepts for permanent magnetic cooling have been around for many years. "However, technical implementation is extremely challenging, and this has previously prevented the development of a product for widespread use," explains Schulz.

"We are the world's first commercial supplier of a cooling system that can magnetically achieve temperatures close to absolute zero (near -273°C) on a permanent basis," says Regnat. (Source: Science and Technology Research News)

A Bright Future: The Sonora Chapter

The Digital Layout

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

Last month's column highlighted the Monterrey Designers Council Chapter in Mexico. As promised, the featured chapter this month is the new Sonora Chapter, also in Mexico. The Sonora Chapter is aiming to establish its presence in the global IPC Designers Council community and set a vision for the future, including educational topics to explore and plans to expand membership.

The core leadership of this newly formed chapter stems from an outstanding group of both young and experienced engineers coming off their successful CID certification. They have formed an excellent group that is very hungry for industry knowledge, eager to take this chapter and run with it, and to make their presence known in the IPC community. They're hyped and ready to go!

Chapter Spotlight: Sonora

by Roberto Ivan Villalba Gonzalez, Chairperson

The Sonora Chapter was established after a CID class that took place in May 2019 in Nogales, Sonora, Mexico. After a recommendation from the instructor (Stephen Chavez, MIT, CID+), class graduates decided to establish an IPC DC Chapter and elected a leadership team (Figure 1).

Our chapter is composed of members from different engineering disciplines, such as manufacturing, testing, and product engineering. We all participate in the different stages of design and manufacturing of electronic assemblies.

Figure 1: Leadership team (L-R), including Rodrigo Martin del Campo Alcocer (assistant chairperson), Roberto Ivan Villalba Gonzalez (chairperson), and Jacob Ivan Castellanos Juarez (technical officer).

The first meeting of the Sonora Chapter included a presentation of team members and a brainstorm of ideas for accomplishing our vision (Figure 2). Our vision is that members of the chapter will find a space to pursue further education, encourage the proliferation of technical knowledge, and broaden their professional network. Some of the topics we would like to cover in the near future include:

- Bringing in experts that could share knowledge to take full advantage of the design tools that are available to us (e.g., Altium)
- Learn about emerging technologies in the world of electronic assemblies
- Share previous experiences and challenges in the design, manufacturing, and testing of electronic assemblies

Currently, all members work in Chamberlain Group's Nogales manufacturing plant, but we are looking for members from all over the state to join us. Please contact me at Roberto.Villalba@chamberlain.com if you're in the area and want to learn more about our chapter.

Recent IPC CID/CID+ Certification Success

by Stephen V. Chavez, MIT, CID+

Well, half of the year has passed us by already. It's amazing how time flies. With that said, we have had many successful IPC CID and CID+ certification classes to date, resulting in many new and seasoned designers successfully achieving their certifications. Each of the instructors, who are led by IPC Hall Of Famer Gary Ferrari, take extreme pride in teaching these courses. The instructors—including Mike Creeden, Cherie Litson, Kelly Dack, Paul Fleming, Dave Seymour, and me—have had great success in each and every class that has been held. Most importantly, I want to highlight the success of all the students who achieved their certifications so far this year.

The student and instructor engagement has been excellent, and in some cases, the class experience has been truly inspiring from much of the student feedback that has been submitted. For us instructors, the best feedback is when a former student contacts us after he/she goes back to their respective companies and ap-

Figure 2: Sonora Chapter members.

plies what they have learned during one of our classes. Knowing that we are making a positive difference and helping others to be successful are just two main reasons why IPC CID/CID+ instructors continue to do what we do.

In the next section, you will find the remaining training sessions to take advantage of, and PCB West is right around the corner. I highly recommend attending these specific CID/CID+ certification classes, along with registering and attending the full PCB West event. This opportunity for both certification and technical content knowledge sharing is always a great combination for us in the industry. I hope you can attend at least one or the other, or even better, both. I hope to see you there this year!

2019 Training and Certification Schedule

IPC Certified Interconnect Designer (CID)

- August 26–29: Markham, ON
- September 6–9: Santa Clara, CA
- September 19–22: Schaumburg, IL
- October 8-11: Carmel, IN
- October 21–24: Anaheim, CA
- November 2–5: Raleigh, NC
- November 5–8: Dallas, TX

IPC Advanced Certified Interconnect Designer CID+

- September 6–9: Santa Clara, CA
- September 10–13: Kirkland, WA
- September 17–20: Schaumburg, IL
- October 21–24: Anaheim, CA
- November 2–5: Raleigh, NC
- December 3–6: Manchester, NH

Note: Dates and locations are subject to change. Contact EPTAC Corporation to check current dates and availability. A minimum enrollment of seven students is required for a class to be held.

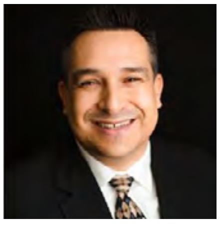
PCB Design Events

- PCB West 2019
- September 9–11: Santa Clara, CA

AltiumLive 2019

- October 9–11: San Diego, CA
- October 21-23: Frankfurt, Germany

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



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

Global Commercial Drone Market to Reach 36% CAGR Over 2018–2022

Technavio analysts forecast the global commercial drone market to grow at a CAGR of more than 36% over a four-year period, according to their latest market research report.

The growing use of artificial intelligence (AI) is one of the major trends being witnessed in the global commercial drones market during the 2018–2022 timeframe. Commercial drone operators are increasingly integrating AI into their drones, which allow drones to capture data, analyze it, and make decisions based on data.

The growing demand for improved data and better data insights has led to a significant increase in the adoption of commercial drones across the world. The Americas held the largest share of the market in 2017, followed by the EMEA and APAC respectively. The region will continue to dominate the shares of the commercial drones market during the forecast period, owing to the increasing investments in commercial drones, and the relaxation of the regulations related to commercial drones by federal agencies such as the FAA. (Source: Business Wire)

High-frequency Material Technical Resources

Lightning Speed Laminates

by John Coonrod, ROGERS CORPORATION

Most of the engineering programs at universities do not include much information on high-frequency printed circuit board materials. There are exceptions, but most trained engineers will enter the industry with minimal background on high-frequency circuit materials. These materials are the core of the high-frequency or high-speed digital PCB, and understanding the many attributes of these materials can be critical for successful PCB-based application implementation. There are many resources in the industry to assist the engineer for learning about high-frequency materials. Before discussing these resources, a brief discussion on what is typically important for high-frequency materials is a good starting point.

Overview

High-frequency circuit materials have many properties, and for some applications, certain properties are more important than others. Here is a quick list of material properties that can be important: dielectric constant (Dk), dissipation factor (Df), bond strength, coefficient of thermal expansion (CTE), moisture absorption, thermal coefficient of Dk (TCDk), substrate thickness control, and thermal conductivity. There are more material properties than those listed here; however, these properties are typically of interest for most high-frequency or high-speed digital applications.

The Dk property of the high-frequency material is usually well-controlled for minimal variation, as opposed to lower-grade circuit materials, which often have large differences in Dk from one piece of material to another. Most high-frequency circuit materials have a documented Dk tolerance, which can be crucial for the designer to consider when trying to account for real-world variations of an application. Along with Dk variation, a related property is often overlooked: TCDk. This property

is a measure of how much the Dk will vary given a change in temperature. Most lower-grade materials have TCDk values in the range of 200–400 ppm/°C. A high-frequency material formulated for good TCDk will usually have values less than 50 ppm/°C.

Another property related to Dk variation is moisture absorption. Water vapor can be absorbed into a circuit by the moisture absorption property of the material. Water has a Dk value of about 70, but that value depends a lot on the frequency of which the data is collected. Regardless, the higher Dk of water will increase the Dk of the high-frequency circuit due to the moisture absorption property of the material, and that change in Dk can certainly alter the RF or high-speed digital performance of the circuit.

Water vapor can be absorbed into a circuit by the moisture absorption property of the material.

In many cases, having a circuit material with tight Dk control is mandatory, but in other cases, Dk control is not as important as a designer may assume. It is common for a PCB fabricator to test a circuit for impedance values as quality control, and it is pretty common for the target impedance to be 50 ohms. Designers often assume that tight Dk control is critical for impedance, but in reality, the Dk value is the lesser concern of several properties that influence the impedance of the circuit.

If designers do simple impedance modeling for a basic microstrip circuit, they will find that the hierarchy of influences on impedance is (1) the circuit conductor width, (2) substrate thickness, (3) copper thickness, and (4) Dk. The Dk value and its variation is the least impactful variable for the impedance of the circuit. However, if designers did the same modeling and looked at these four influences on the impact of phase angle response, they would find that Dk is a much more important property for phase angle consistency than it is for impedance. Designers should be aware of the circuit property that is most important and consider the appropriate influences which can impact variation of that property.

Resources

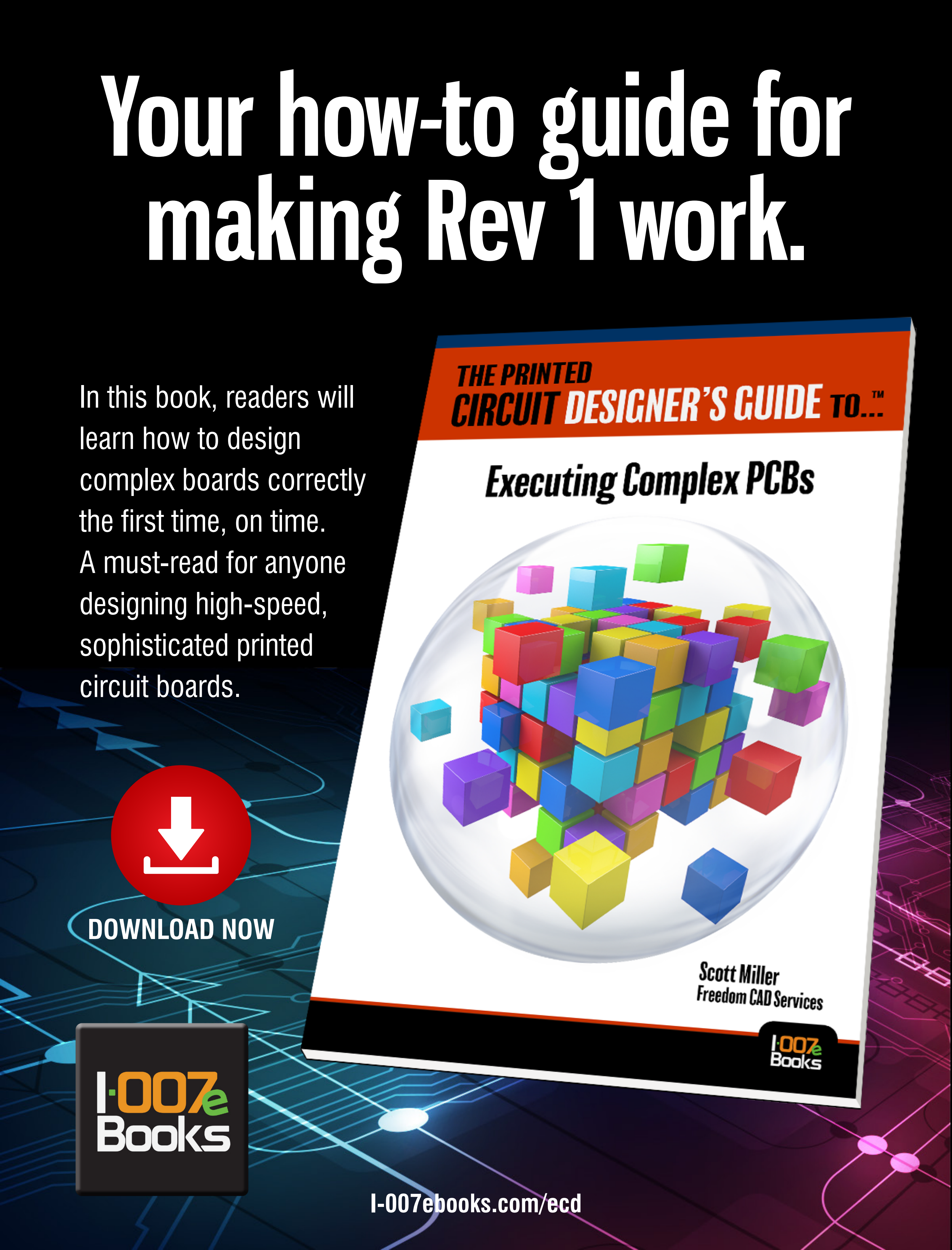
There are a lot of excellent resources in the industry for high-frequency circuit materials. For example, the www.IPC.org website has a lot of information about these materials in terms of specifications and test methods. Also, IPC will often offer training that can include material properties and the influence these properties have on circuit fabrication and performance.

The www.IEEE.org website is another excellent resource for RF and high-speed digital applications. This website is more focused on the performance of electrical structures than materials. Although, there are many papers on the IEEE website related to material characterization of high-frequency circuit materials, and these references can be very useful to understand electrical properties of these materials.

The high-frequency material suppliers in the industry also provide technical information, which is typically more tailored to their products and less of a general education about these materials.

Rogers Corporation provides educational information about high-frequency materials and how they are used in specific applications and general information as well. Rogers has a Technology Support Hub with free calculators, videos, articles, and other content, along with the ability to contact a local engineer for technical assistance, at www.rogerscorp.com/techub. **DESIGN007**

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



Your how-to guide for making Rev 1 work.
In this book, readers will learn how to design complex boards correctly the first time, on time. A must-read for anyone designing high-speed, sophisticated printed circuit boards.
DOWNLOAD NOW
THE PRINTED CIRCUIT DESIGNER'S GUIDE TO...™
Executing Complex PCBs
Scott Miller
Freedom CAD Services
I-007e Books
I-007e Books
I-007ebooks.com/ecd

Managing Global Supply Chain Uncertainty

Connect the Dots

by Bob Tise, SUNSTONE CIRCUITS

We are well into the second year of tariff-centric trade policy [1], and one thing appears certain—uncertainty is here to stay. Though most of the media focus has been on cars and steel or consumer prices and corporate profits [2], the enduring challenge for both the electronics and PCB industries has been maintaining reliable global supply chains.

Since July 2018, the PCB industry has been affected by tariffs on key manufacturing components like laser drills, placement machines, and reflow ovens as well as PCB assemblies used in telecommunications equipment, cameras, and ATMs [3]. Like many in the industry, we were hopeful for and confident about a swift resolution.

Figure 1: The future is still uncertain.

A year later, the path forward is still not apparent. Uncertainty hovers in every planning session: will tomorrow bring another import tax increase or a speedy resolution? The unknowns can freeze an organization in place, hoping for the best (Figure 1). In a recent New York Times article [4], Pete Guarraia, the leader of Bain's [5] supply chain practice, was quoted as saying, "Most companies took a wait-and-see-attitude. That was absolutely the mindset."

Hope Is Not a Long-term Strategy

As these trade conflicts drag on, with no one outcome more or less certain than another, it makes sense to consider new supply chain strategies for high-end manufacturing components like PCBs. According to a recent IPC survey [6], 87% of its U.S. members import raw materials, components, or equipment from China [7]. PCBs are among these critical components and are the foundation of any electronic device. Sooner or later, tariffs associated with their manufacture are going to leave a mark on profits, prices, and production volume.

Alternate sourcing for PCBs can get tricky, especially if there exists a long-term relationship with your vendor. Searching for new sources for all your products impacted by tariffs is a big exercise (Figure 2). You have to weigh the cost of the tariffs against quality expectations, lead times, knowledge transfer requirements, and the risk associ-

Figure 2: The cost of tariffs.

ated with terminating—albeit perhaps temporarily—your existing supplier relationships.

There are also few cost-effective supplier alternatives to China for producers of electronics [8]. For high-end manufacturing, China beats nearby Vietnam in terms of available skilled labor, modern production infrastructure, and relatively limited bureaucracy [9]. And even if an organization is prepared to work with all of those variables, the raw materials for production will come from China and be subject to tariffs anyway. Therefore, it is easy to understand why companies are still unwilling to move from the status quo, even if it impacts the bottom line.

Though absorbing costs or passing them on can work in the short term, eventually, forward-looking businesses will seek competitive pricing advantage by adopting a supply chain strategy that avoids tariffs. Determining how to do so internationally is less obvious than it might seem, especially in the electronics industry where products are so complex (Figure 3).

PCB manufacturers work with a relatively finite list of components, at least compared to our customers who are making medical devices, robots, and drones. This presents our industry with a chance to provide value for our customers who are spending precious resources poring over tariff codes in an attempt to figure out which apply to their components. Domestic PCB manufacturers can offer tariff-free boards to their customers, eliminating both long-term cost uncertainty and a supply-chain headache.

The Case for Domestic PCB Manufacturing

Sunstone Circuits has been making the "made in America" PCB case for years [10] to

Figure 3: Complex electronics.

small- and mid-sized electronics manufacturers, understanding that enterprises relying on high-volume, commoditized boards for mass production are a different animal. For these larger enterprises, there are opportunities to renegotiate overseas supplier contracts or even influence where their suppliers buy raw materials.

Smaller organizations needing a lower volume of PCBs face greater relative risk to the bottom line. The added cost and uncertainty associated with international trade can help make domestic manufacturing more appealing. We understand that even PCBs made in the U.S. will likely have raw materials from China but producing boards here does not present risk like a move to Vietnam or China.

Moving PCBs to be made in America offers multiple advantages. You can eliminate the risk associated with uneven quality, delays common with transatlantic shipping, and risks to your intellectual property. We also encourage our customers to look for hidden costs of offshoring and seriously consider its less quantifiable pain points, like the impact on inventory management and burden on the domestic operation.

The longer tariffs remain in place, the more likely cost-benefit analysis tips in favor of U.S.-produced PCBs. Uncertain trade policy causes us to closely examine our hidden assumptions about offshoring paradigms. You might find that a domestic manufacturer that specializes in low-volume, high-mix manufacturing provides a viable alternative to increasingly costly and risky Chinese production. **DESIGN007**

References

1. K. Alaganan and V. Argod, "Supply Chain Strategies to Minimize the Impact of Tariffs," Chainanalytics, May 14, 2019.
2. K. Russ, "The Costs of Tariffs in the U.S.-China Trade War," Econofact, May 14, 2019.
3. M. Buetow, "Pending U.S. Tariffs to Affect Host of PCB Equipment, Components," Circuits Assembly, June 15, 2018.
4. B. Casselman, "Trade War Starts Changing Manufacturers in Hard-to-Reverse Ways," *The New York Times*, May 30, 2019.
5. www.bain.com
6. www.ipc.org
7. IPC, "Electronics Industry Says Trump Administration Tariffs on China Could Harm U.S. Electronics Companies," May 11, 2018.
8. Reuters, "U.S. Firms: China Tariffs Will Raise Costs With Few Sourcing Alternatives" Voice of America News, June 17, 2019.
9. N. Dimitrijevic, "Manufacturing: China vs. Vietnam (Pros and Cons)," LinkedIn, March 19, 2018.
10. "The Case for PCBs Made in America: Why Domestic Prototyping and Manufacturing Delivers Competitive Advantage," Sunstone Circuits.

Bob Tise is an engineer at Sunstone Circuits. To read past columns or contact, click here.

NASA's Mars 2020 Rover Does Biceps Curls

The robotic arm on NASA's Mars 2020 rover can curl heavy weights with the best. The rover's 7-foot-long (2.1-meter-long) arm handily maneuvers 88 pounds' (40 kilograms') worth of sensor-laden turret as it moves from a deployed to a stowed configuration.

On Mars, the arm and turret will work together, allowing the rover to work as a human geologist would: by reaching out to interesting geologic features, abrading, analyzing and even collecting them for further study via Mars 2020's Sample Caching System, which will collect samples of Martian rock and soil.

"Standing there, watching the arm and turret go through their motions, you can't help but marvel that the rover will be in space in less than a year from now and performing these exact movements on Mars in less than two," said Dave Levine, integration engineer for Mars 2020.

Mars 2020 will launch in July 2020. It will land at Jezero Crater on Feb. 18, 2021. (Source: NASA)

How to Avoid Pump-out and Achieve Efficient Heat Transfer

Sensible Design

by Jade Bridges, ELECTROLUBE

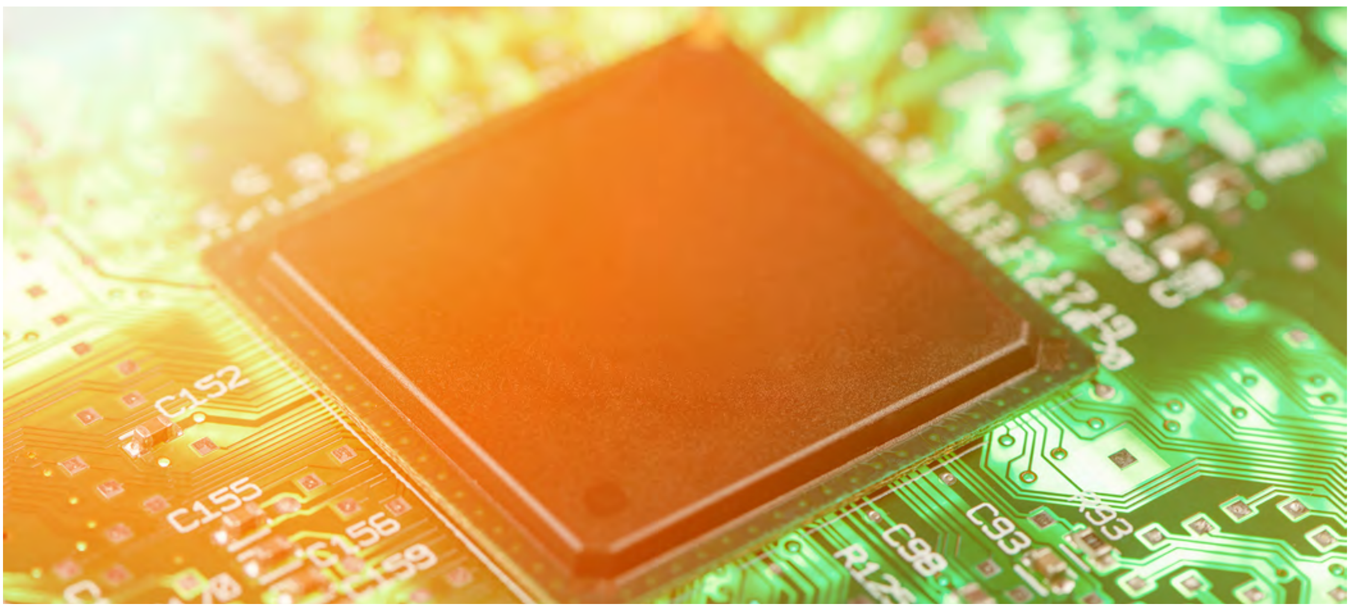
In my previous column, I expressed some of the key issues and questions fielded by our technical support team regarding the all-important subject of thermal management and materials, including gap pads, thermal pastes, and phase-change materials. I cannot emphasise enough how maintaining reliable temperature control of components will significantly extend the performance and lifetime of your devices. It is a critical process that is best factored in during the design stage to avoid unnecessary costs and save time when scaling up to full-scale production.

This month, I will delve deeper into thermal management chemistries and the processes you should follow to achieve a successful outcome and focus on some of the problems you are likely to face when applying a thermal management material. I will also provide an overview of some of the thermal bonding, resin, and encapsulation materials and products that are available, their thermal conductivity, and how to achieve the most efficient heat transfer.

How Can I Avoid Pump-out?

Let's start with an issue that many of you will encounter at some stage in your product development work—pump-out. Pump-out can occur when a device is subject to rapid temperature changes, resulting in the expansion and contraction of the interface surfaces, thus producing a pumping effect. This motion can cause such pastes to be squeezed or pumped out from the interface gap, reducing the thermal transfer performance. To tackle issues with pump-out, it is first important to understand the application conditions and material requirements involved.

Temperature extremes and the rate of change of temperature are important factors that will determine the choice of thermal interface ma-



terial; for example, if operating temperatures are likely to range between -50°C and 200°C, a silicone-based thermal paste would be the go-to option. It is important to consider the type of product used; for instance, a thermal paste compared to a thermal phase change material are entirely different entities. If we take one of our phase change materials—such as the silicone-free TPM350—it is similar to a thermal paste in the sense that it is a non-curing, non-bonding product, but it eliminates the mess and inconsistencies associated with pump-out.

Phase change materials offer efficient thermal transfer along with an enhanced performance with thermal shock cycles and greater thermal protection where temperature spikes can occur due to their ability to store and release thermal energy (latent heat) during the phase change process. Once heated above their softening temperature, the phase change materials alter to a liquid-/gel-like state that perform equally as well as—and sometimes even better than—a thermal grease.

Thermal pastes are often designed to be applied in as thin a layer as possible. They improve the contact between the device and its heat sink by eliminating air gaps and ensuring that the full surface contact area is available for heat transfer. There is, however, a critical thickness that determines maximum thermal transfer with minimal thermal resistance, and while this will depend on the roughness of the substrates and required spacing, it is generally between 30 and 100 microns. It's always advisable to refer to the manufacturer's instructions to apply the correct amount of thermal management product and test thoroughly under realistic accelerated conditions using the actual unit where possible.

Do I Really Need to Use a Thermal Management Product?

The short answer is yes! During use, some electronic components can generate significant amounts of heat. Failure to effectively dissipate this heat away from the component and the device can lead to reliability concerns and reduced operational lifetimes. It is in such cases that thermal management measures need to be taken for heat transfer efficiency to prolong the working life and ensure all components operate within their ideal temperature range for maximum performance.

Thermal Management Products Come in Many Shapes and Sizes

Thermal bonding materials can be used as interface materials either for thermal conductivity alone or to secure the heat sink or electronic component in place. In addition, such curing materials can be used for filling gaps and providing some structural support. By way of example, Electrolube's Thermal Bonding System (TBS) is a two-part epoxy bonding system, which utilises metal oxides to provide high thermal conductivity whilst being electrically insulating. The bonding system is useful in the manufacturing of heat sink assemblies where piggyback arrangements are applied and where the manufacturing design of heat sinks does not allow for welding or brazing techniques to be employed due to complexity or geometry of the fins.

Coming back to the main topic, the fact that these types of products are curing systems means that in general, they do not experience the same issues of pump-out as a non-curing product. With bonding products, it is important to test the strength of the bond during the operational use of the device to ensure the bond strength and physical properties of the product—such as coefficient of thermal expansion—allow the product to perform consistently over the lifetime of the device.

Thermally conductive potting compounds are resins designed to encapsulate components or pot the entire PCB unit, dissipating heat away from the electronic components and in the process and offering additional protection from environmental elements such as water or chemicals. For certain types of heat-generating circuitry, such as power supplies and LEDs, it may be beneficial to encapsulate the device in a heat-sink enclosure using a thermally conductive potting compound. The potting compounds provide excellent resistance to extreme temperatures, chemicals, shock, and vibration. There are a number of thermally conductive encapsulation products available, largely based on epoxy, silicone, and polyurethane technologies and with varying levels of viscosity. These thermal management solutions are particularly suitable for circuitry exposed to the harshest environments and provide high protection whilst enabling even heat distribution to increase the device lifetime.

It's All About Thermal Conductivity, or Is It?

This is commonly misunderstood, but it's not all about thermal conductivity because thermal conductivity values are measured from bulk material to give a comparison of one product to another; they are not a true reflection of the performance in the final application. That is related to the thermal resistance under the exact conditions of use. Don't be fooled by high thermal conductivity values, which can be misleading due to the many options of testing available. In addition, a high thermal conductivity product may also have a high thermal resistance if it cannot be applied correctly in a thin film at the interface. Any excess material or non-uniform application will result in a variation of heat distribution across the interface, thus leading to inefficient heat transfer.

How Do I Achieve the Most Efficient Heat Transfer?

As a rule of thumb, apply thin, uniform layers with the minimal amount of product required to remove all air gaps but leave no excess of material. Remember, the heat sink is far more conductive than the thermal interface material. The job of the interface material is to remove air so that the heat sink can efficiently transfer heat away from the component/device.

Conclusion

When it comes to the choice and applications of thermal management materials, there's a great deal more to discuss, and over the following months, I hope to provide more useful tips and design advice that will help you in your quest for reliable circuit protection. **DESIGN007**

Jade Bridges is global technical support manager at Electrolube. To read past columns from Electrolube, click here. Also, visit I-007eBooks.com to download your free copy of Electrolube's book, *The Printed Circuit Assembler's Guide to... Conformal Coatings for Harsh Environments*, as well as other educational titles.

Researchers Demonstrate Low Voltage LEDs

The voltage of an LED is usually equal to or larger than the bandgap energy per electron charge. A team of researchers based at The University of Manchester, University of Warsaw, the High Magnetic Field Laboratory in Grenoble and the National Institute for Materials Science in Japan have been able to demonstrate LEDs that turn on at much lower voltages.

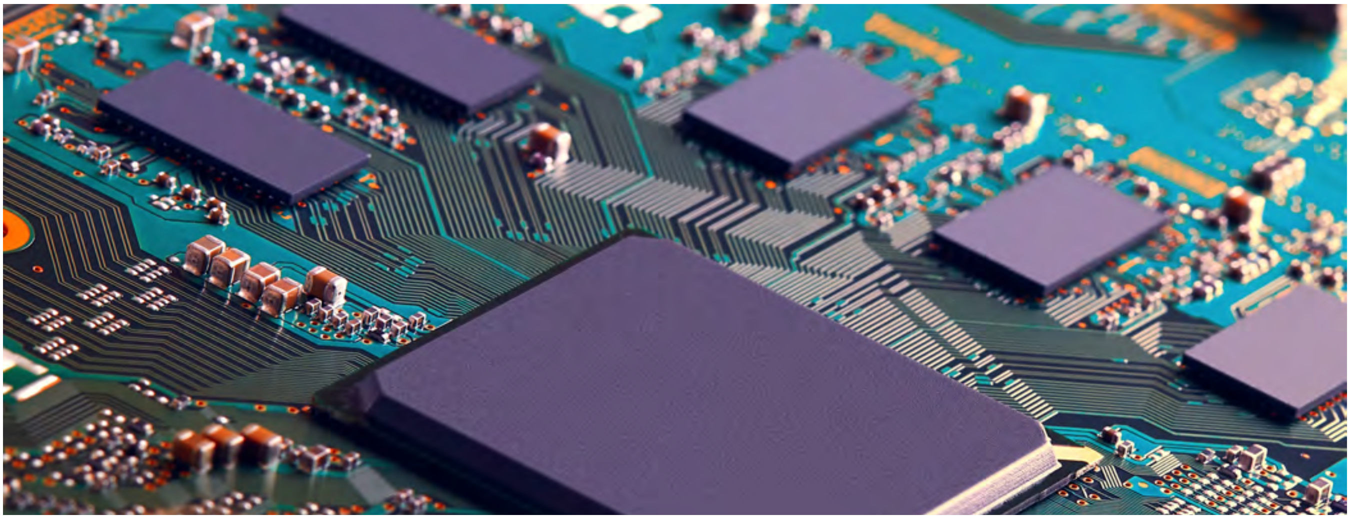
The idea to stack layers of different materials to make so-called heterostructures goes back to the 1960s, when semiconductor gallium arsenide was researched for making miniature lasers—which are now widely used.

Today, heterostructures are common and are used very broadly in semiconductor industry as a tool to design and control electronic and optical properties in devices.

More recently in the era of atomically thin two-dimensional (2D) crystals, such as graphene, new types of heterostructures have emerged, where atomically thin layers are held together by relatively weak van der Waals forces.

The new structures nicknamed 'van der Waals heterostructures' open a huge potential to create numerous designer-materials and novel devices by stacking together any number of atomically thin layers. Hundreds of combinations become possible otherwise inaccessible in traditional three-dimensional materials, potentially giving access to new unexplored optoelectronic device functionality or unusual material properties.

From the fundamental point of view, the observed effects mark an important step towards the realisation of exciton condensation and superfluidity of van der Waals heterostructures. (Source: University of Manchester)



Decoupling Capacitors' Impact on Power and Signal Integrity

Article by Chang Fei Yee
KEYSIGHT TECHNOLOGIES

This article will discuss the effect of decoupling capacitors upon a PCB's power and signal integrity. The study was performed with post-layout co-simulation of power and signal integrity to analyze power distribution network impedance, simultaneous switching noise, and eye diagrams.

Introduction

It is crucial for hardware designers to identify the resonant frequency of each element (e.g., bypass/decoupling capacitor, planar capacitance, and interconnect inductance) of the power distribution network (PDN) on a PCB and its impact on power integrity. A PCB with poor power integrity—such as a higher-than-targeted PDN impedance across the wideband range—results in simultaneous switching noise (SSN) and a shrunken eye diagram of the signal transmitted by the IC that draws power from the PDN. This article demonstrates the post-layout co-simulation of power and signal integrity using Mentor HyperLynx to analyze the impact of decoupling capacitors upon PDN impedance, SSN, and eye diagrams.

Analysis and Results

A PCB containing a system-on-a-chip (SoC) with DDR4 memory interface is laid out. In Figure 1a, the PDN named 1.2V on layer 4 supplies power to a memory interface that consists of one memory IC highlighted in blue. Meanwhile, the ground or reference plane, highlighted in green, is laid out on layer 5. The memory IC has 13 BGA power pins. The footprint of the 0.22-uF

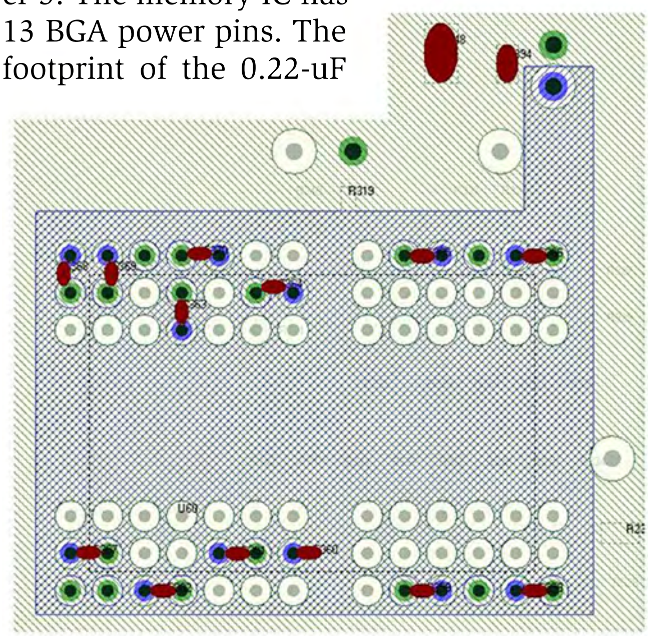
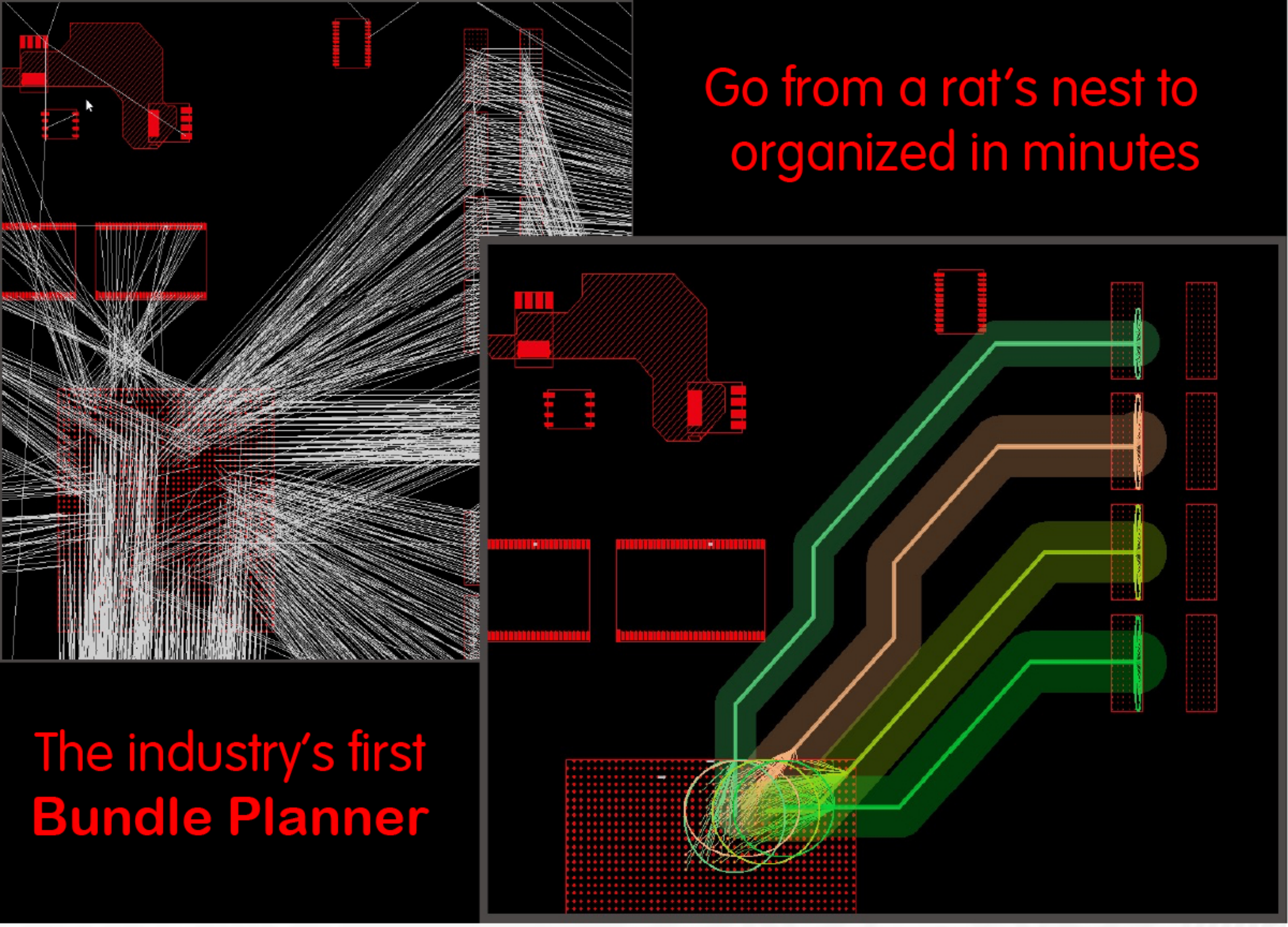


Figure 1a: Top view of the 1.2V power rail supplying power to the DDR4 memory interface.


Go from a rat's nest to organized in minutes
The industry's first
Bundle Planner

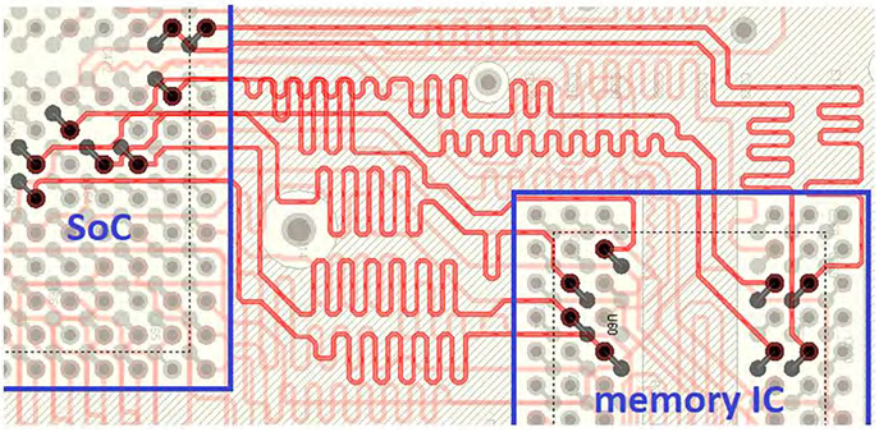


Figure 1b: The eight data signals of the DDR4 memory interface.

decoupling capacitor (highlighted in brown) in a 0201 package dimension is placed across each of the power pins and the ground. Additionally, the footprint of bypass capacitors, 10 uF and a 1 uF respectively (highlighted in brown as well), is placed across the 1.2V power net and ground. The eight data signals of this memory interface are shown in Figure 1b.

To study the impact of a decoupling capacitor on PDN impedance, SSN, and eye diagrams, post-layout co-simulation of power and signal integrity in HyperLynx is performed with PDN conditions listed in Table 1. Each PDN condition has the same PCB layout and stackup depicted in Figure 1a and 1b loaded with one 10uF and one 1uF bypass capacitor. In condition A, there is no 0.22-uF decoupling capacitor. In B, there are four 0.22-uF capacitors. In C, the number of 0.22-uF capacitors is increased to eight. Meanwhile, in D, the quantity of 0.22-uF capacitors is further increased to 13.

Subsequently, the power-aware IBIS v5.0 model of memory IC and SoC, respectively, is imported into the software tool. Memory read operation at 1 Gbps, 500M Hz Nyquist frequency, is set for the mode of co-simulation, whereby the data bus is driven by the memory IC, and the SoC serves as the receiving end (Rx).

First, the PDN impedance that spans from 1,000 Hz to 2 GHz for the four PDN conditions is compared. In theory, the resonant frequency is inversely proportional to the capacitance. With reference to Figure 2, all four PDN conditions have the same impedance profile from 1,000 Hz up to 10 MHz due to the placement of the same number of 10-uF and 1-uF bypass capacitors. Contrast PDN condition A (with no 0.22-uF decoupling capacitors) with conditions B, C, and D (each with at least four 0.22-uF decoupling capacitors) and note the decrease in

PDN Condition	Bypass/Decoupling Capacitor	Quantity
A	10 uF	1
	1 uF	1
	0.22 uF	0
B	10 uF	1
	1 uF	1
	0.22 uF	4
C	10 uF	1
	1 uF	1
	0.22 uF	8
D	10 uF	1
	1 uF	1
	0.22 uF	13

Table 1: PDN conditions being studied.

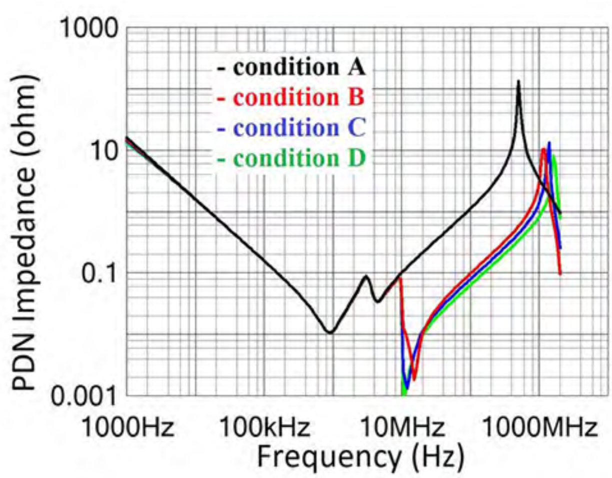


Figure 2: Simulated PDN impedance plots.

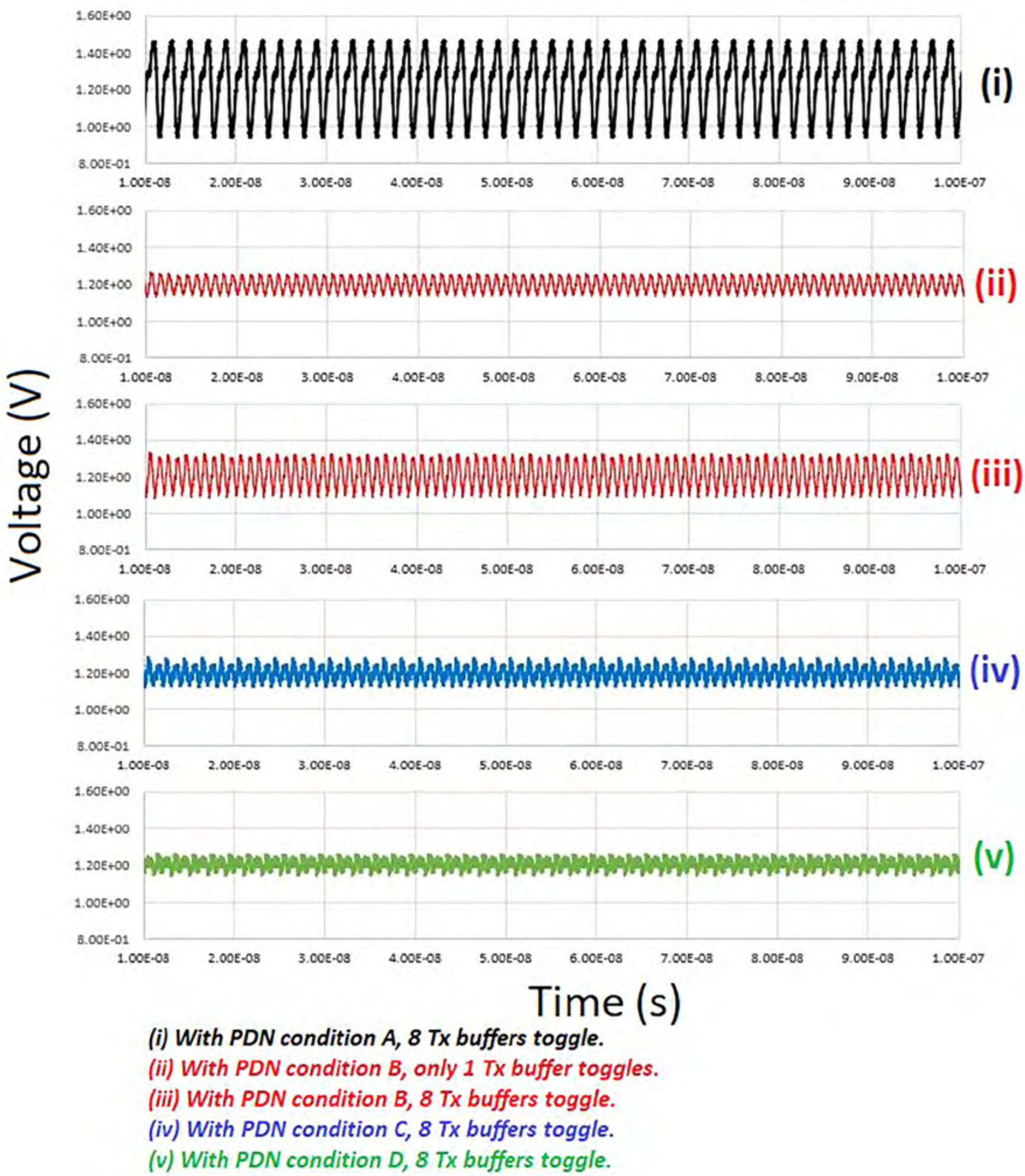


Figure 3: Simulated SSN plots for PDN conditions listed in Table 1.

PDN impedance by at least 0.9 ohms for frequencies above 10 MHz.

Second, the SSN plots of the power nets in the time domain for all PDN conditions are shown in Figure 3. SSN is generated by the interaction between PDN impedance and transient current in the buffer of the transmitting (Tx) IC during signal transmission. The transient current will increase when more Tx buffers in the IC are toggling simultaneously to transmit data signals. In PDN condition B, the amplitude of SSN in Figure 3 (iii) is double of that in Figure 3 (ii) due to larger transient current in the IC with more Tx buffers toggling.

On the other hand, when eight Tx buffers are toggling simultaneously at 1 Gbps, PDN condition A with the poorest decoupling and largest impedance (in the hundreds of MHz) suffers the largest amplitude of SSN: 505 mVpp, shown in Figure 3 (i). On the contrary, PDN condition D with the best decoupling (13 0.22-uF capacitors) and the smallest impedance experience the smallest amplitude of SSN—95mVpp, shown in Figure 3(v) —which fulfils

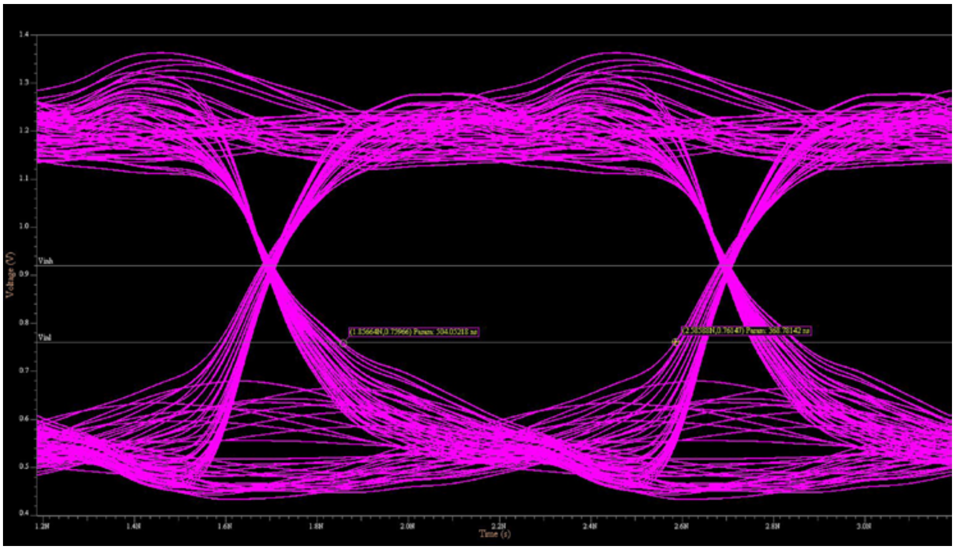


Figure 4a: Simulated eye diagram of the DDR4 data signal for PDN condition A.

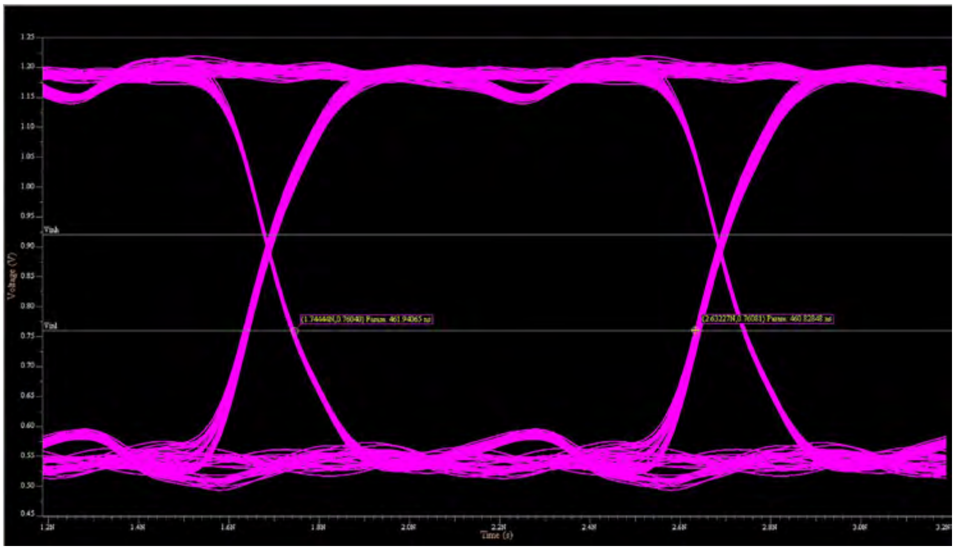


Figure 4b: Simulated eye diagram of DDR4 data signal for PDN condition D.

the ±5% tolerance of the 1.2V power source specified in the data sheet of memory IC.

Thirdly, the eye diagram of the DDR4 data signals at 1 Gbps with PRBS-7 bit sequence (observed at Rx or SoC) is compared for PDN condition A versus D, as shown in Figures 4a and 4b respectively. A much larger eye height and width experienced by data signals in condition D indicates that the suppression of SSN due to improved power integrity helps reduce the jitter in the signals transmitted by the memory IC. The shrunken eye diagram opening suffered in condition A increases the risk of bit error and metastability at Rx IC. The suppression of SSN is achievable by properly placing sufficient decoupling capacitors on power rail with reference to ground.

Summary

The post-layout co-simulation of power and signal integrity demonstrated in this article indicates that placement of sufficient decoupling capacitors on a power net with reference to ground has a positive impact on PDN impedance, SSN, and eye diagrams. Additionally, prerequisite signal integrity simulation must be conducted on the signal lines of interest to ensure the optimal termination scheme and minimal transmission loss before the co-simulation to analyze the mutual effect of power and signal integrity. **DESIGN007**

References

1. B. Olney, "Power Distribution Network Planning," *The PCB Magazine,* May 2012.

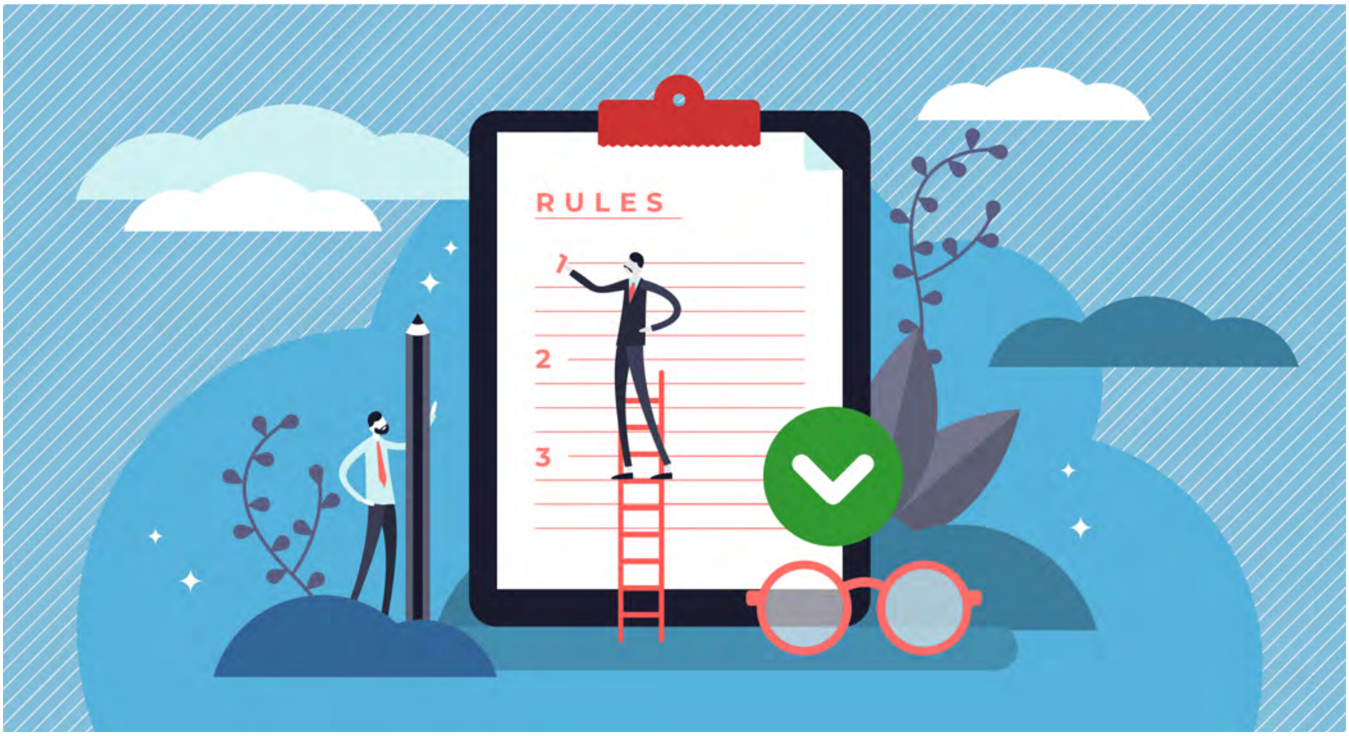
2. F. Carrio, V. Gonzalez, and E. Sanchis, "Basic Concepts of Power Distribution Network Design for High-Speed Transmission," *The Open Optics Journal,* 5, (Suppl 1-M8) pp. 51-61, 2011.

3. E. Bogatin, *Signal and Power Integrity–Simplified: 2nd Edition,* Prentice Hall, 2009.

4. Mentor HyperLynx, "Power-Aware Signal Integrity Analysis."

5. Micron, "Power Integrity Simulation With IBIS 5.0 Models Technical Note."

Chang Fei Yee is a hardware engineer with Keysight Technologies. His responsibilities include embedded system hardware development, and signal and power integrity analysis.



Design Rule Checks Cut Down on Board Respins

Article by Rebecca Lord and John McMillan
MENTOR, A SIEMENS BUSINESS

PCB designs commonly undergo multiple respins as a result of inconspicuous signal integrity (SI), power integrity (PI), and electromagnetic interference (EMI) violations. At an average cost of nearly $28,000 per respin, ensuring that a given design meets its performance, time to market, and cost goals is imperative. To help eliminate complicated and difficult-to-diagnose layout violations, some PCB tool suites offer unique electrical design rule checks (DRC).

When using DRC analysis as part of the PCB design process, engineers can ensure that their PCBs fall within the proper constraints for a multitude of different, advanced electrical design rules. The DRC tool contains fully-customizable SI, PI, EMI, and safety rule checks that enable designers to quickly identify and correct violations well before starting the manufacturing process. By running DRC analysis, it is possible to eliminate error-prone manual inspection and reduce costly design respins that impact the product's time to market and the company's profitability.

To illustrate the power of utilizing a PCB design tool with tightly-integrated DRC analysis, we will use the BeagleBone Black as an example. The BeagleBone Black is a low-power, open-source, single-board computer produced by Texas Instruments and commonly used by both developers and hobby enthusiasts alike. This development board consists of a TI Sitara processor, providing a combination of performance, power, and peripherals to help drive down system cost, simplify design, and expand connectivity within the overall BeagleBone Black design. In addition to the

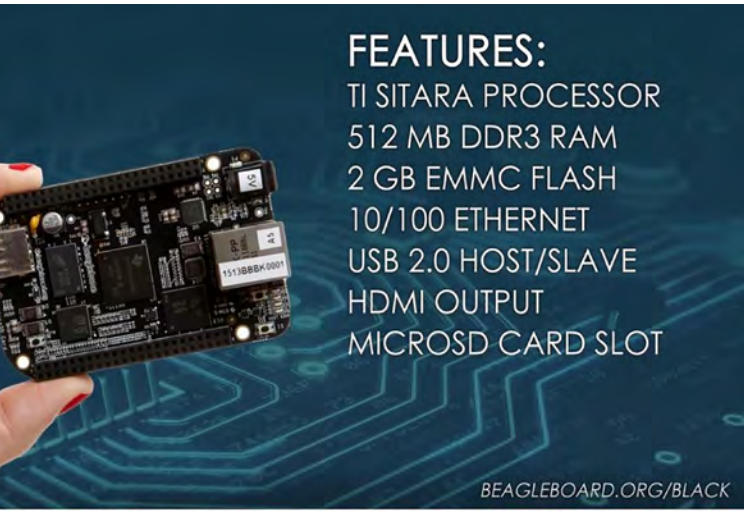


Figure 1: The BeagleBone Black is an open-source development board containing a number of different electrical characteristics suitable for DRC simulation.

TI Sitara processor, the board consists of 512 MB DDR3 RAM and 2GB of flash as well as a number of physical interfaces and other features (Figure 1).

Tightly Linked DRC and PCB Layout Interfaces

Because the layout and DRC tools are fully integrated, a PCB design can be loaded into DRC directly from the PCB layout window. The rules within DRC are sorted into specific categories—SI, PI, EMI, and safety. And each individual rule contains a descriptive overview page, making it easy for designers to select the most important tests to run on their PCB layout. The overview page for the nets crossing gaps rule within DRC is shown in Figure 2.

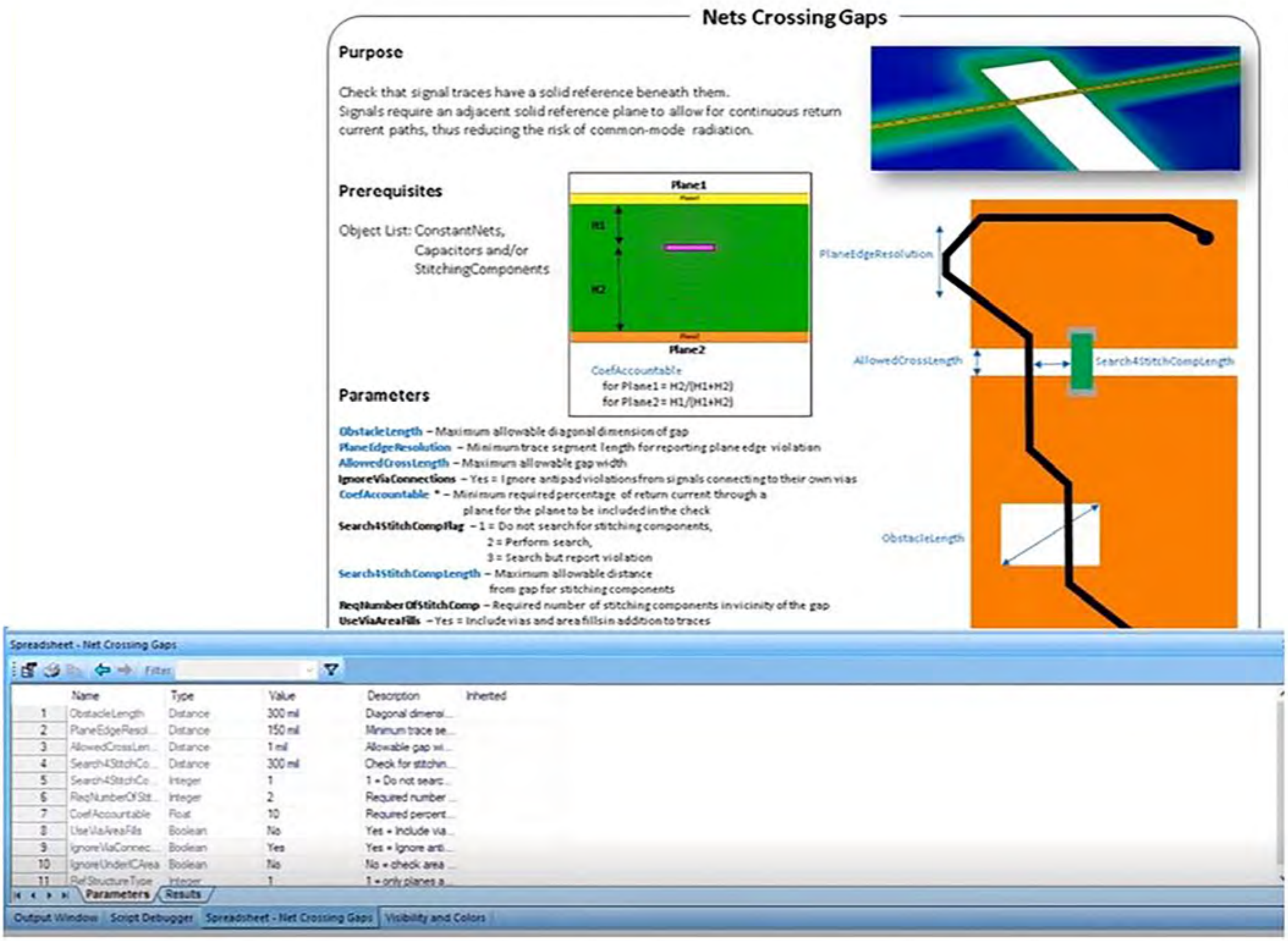


Figure 2: The nets crossing gaps overview page within the DRC tool, which includes a brief description, graphical imagery, and descriptive parameters.

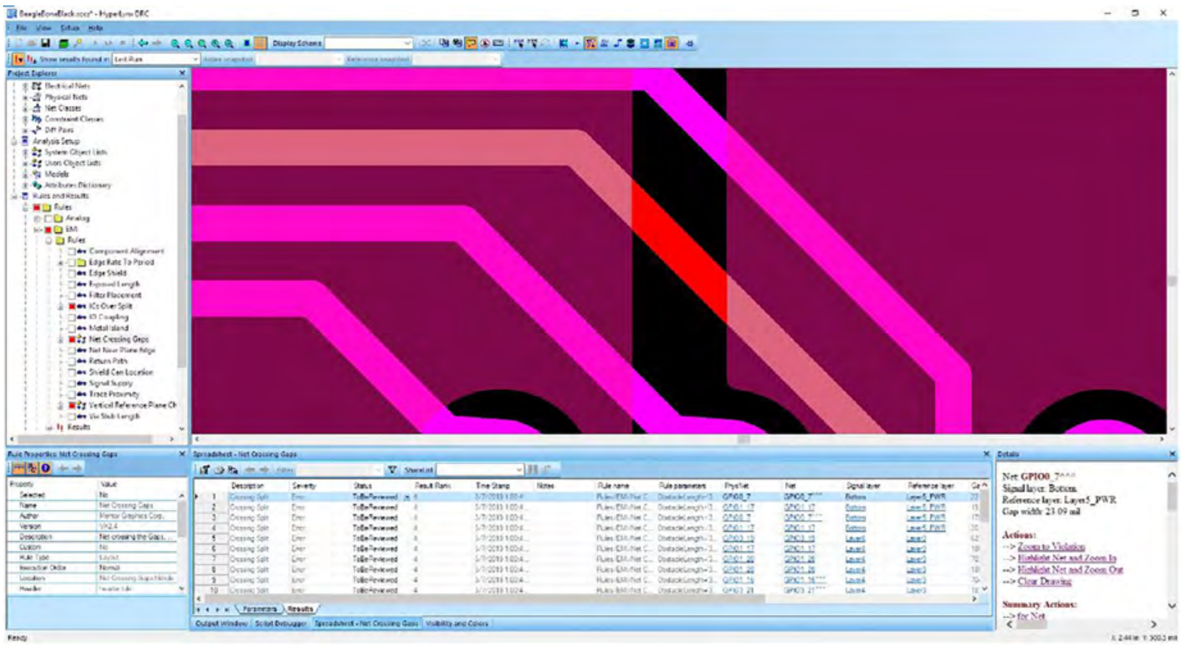


Figure 3: When an instance of a rule violation is selected, the DRC tool will zoom to the location of that violation on the PCB design within the DRC window.

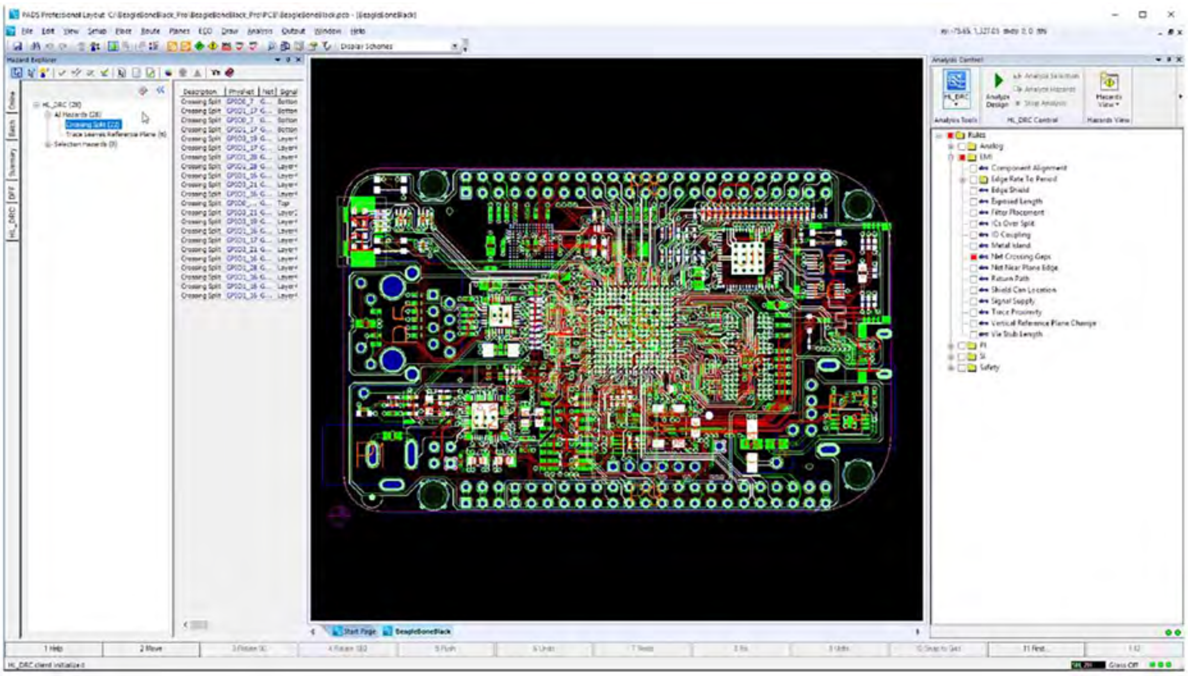


Figure 4: The integrated DRC client is shown inside of the PCB layout tool. This can be used to easily run or rerun rules once PCB changes have been made.

When a high-speed net crosses a split plane, it can create an impedance discontinuity on the signal trace, possibly leading to unwanted reflections, radiation, and crosstalk. With the increased complexity and density of today's PCB designs, finding and reviewing all instances of a net crossing a split plane is a grueling manual process. Additionally, standard simulation tools do not typically check for those occurrences. When the nets crossing gaps rule is run in DRC, it is easy to pinpoint exactly where all instances of these discontinuities may appear.

After running a rule, designers can select the spreadsheet tab to display a list view of results for that test. By clicking on a specific occurrence of a violation within the spreadsheet tab, the tool will jump to the exact location of that error on the PCB design (Figure 3). Any parts or traces associated with that violation will be highlighted.

With tightly linked layout and DRC interfaces, the violation data from the DRC client will automatically load into the PCB layout tool. This allows designers to easily make any necessary changes to their layout without having to manually cross reference between tools. To ensure that violations are cleared once changes have been made to the design, the linked DRC client can be used to run, or rerun, any selected rules from directly within the layout tool window. This feature is shown in Figure 4.

Testing Differential Pair Symmetry With DRC

The BeagleBone Black design contains a number of differential pairs, two of which have a 90-ohm differential impedance. These 90-ohm differential pairs have been structured inside the layout tool's constraint manager to be contained within a separate constraint class. Due to the tight integration between layout and DRC, the constraint class definitions created within the PCB tool will also be defined automatically within the DRC tool. In DRC, designers can quickly create an object list from the constraint class, allowing them to selectively choose to run the next rule on only those two 90-ohm differential nets.

When designing differential impedance traces, symmetry in the length, spacing, and via count/positions of the pairs is imperative for proper functionality. The differential pair rule will check whether these properties are consistent within the proper bounds for all trace segments on a given net. The overview page for the differential pair rule with the user-defined object list selected in the properties area can be seen in Figure 5.

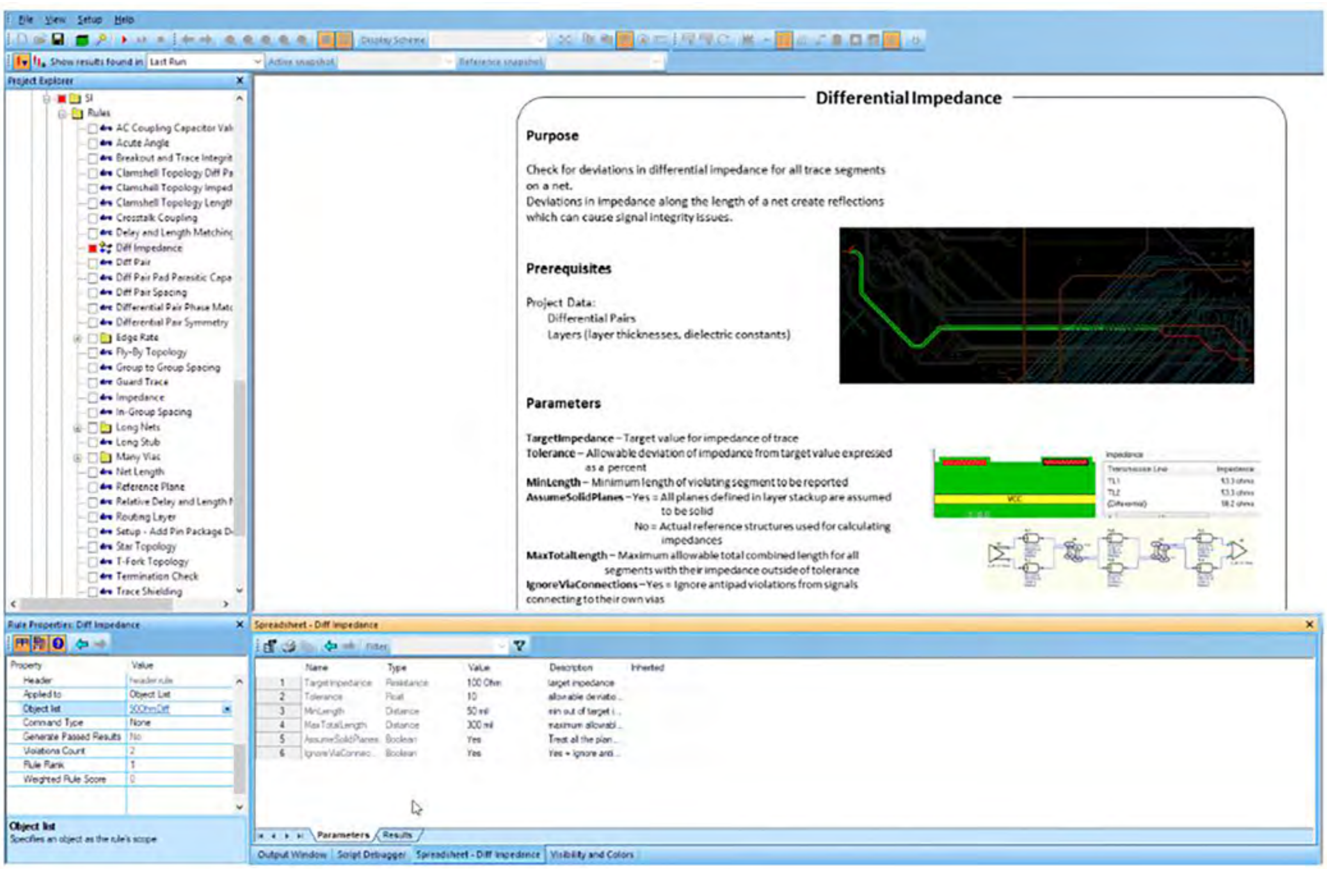


Figure 5: The differential impedance rule in DRC checks for potential impedance violations on differential traces. Rules can be selectively run on specified nets by choosing a user-defined object list in the properties section.

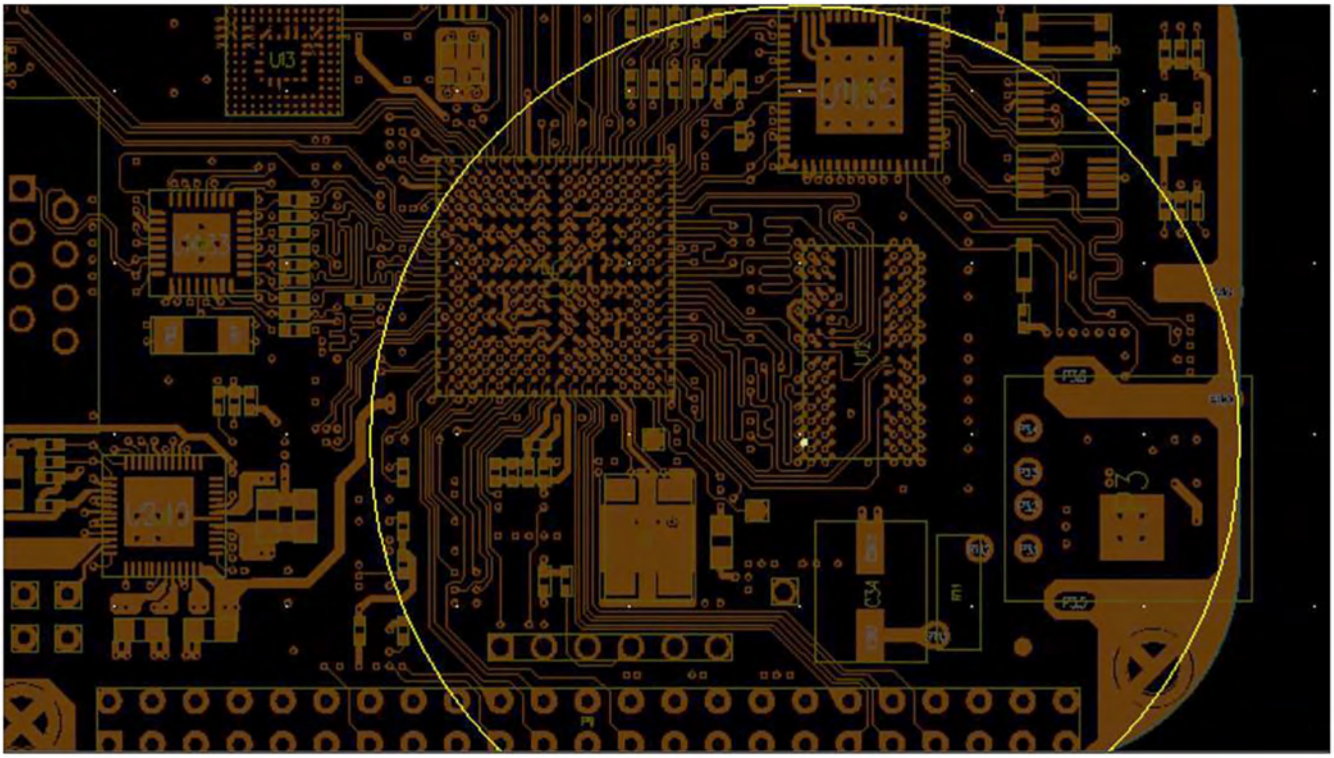


Figure 6: The DRC tool can locate each instance of improper decoupling capacitor placement while showing the defined test radius.

With DRC's customizable properties, designers can easily define rule parameters, such as target impedance on the traces and minimum and maximum allowable trace length. Because the specific differential traces that are being tested have a 90-ohm differential impedance, the target impedance parameter should be defined as 90-ohm before running the rule. DRC provides the flexibility to customize each rule for individual design parameters, thereby reducing false violations and giving designers more control.

Checking Proper Placement of Decoupling Capacitors With DRC

One of the many challenges that engineers face on a regular basis is ensuring that their designs meet the defined specifications for each component. For example, the datasheet for the BeagleBone Black's TI Sitara processor defines the maximum allowable distance for bypass capacitors on the VDDS DDR net as 400 mils. If decoupling capacitors are not placed properly, there is a high chance that the DDR3 interface could malfunction.

Manually checking the placement of decoupling capacitors on each net is a tedious and time-consuming process. Alternatively, the decoupling capacitor placement rule in DRC will locate all improperly placed, and non-existent, decoupling capacitors for a defined net or component. With the information obtained from the TI Sitara's datasheet, the 400-mil placement radius on the VDDS DDR net can be specified within the decoupling capacitor rule. After running the rule and selecting a specific instance of the violation, the tool will highlight where the error occurs, as well as indicate the designated 400-mil test radius (Figure 6).

Additional Rules Available With DRC

Routing a trace from one layer to another is a common design practice used to accommodate today's densely packed PCB layouts; however, care must be taken to reduce the risk of common-mode radiation. Frequently, stitching capacitors or stitching vias are placed near the net when a plane change occurs to allow for a continuous current return path. The vertical reference plane change rule identifies instances of signals transitioning from one layer to another as well as the placement of stitching capacitors or stitching vias near those nets.

Timing on high-speed nets is incredibly important for proper functionality, especially on DDR nets. If DDR signals do not reach their destination within proper timing constraints, the memory may not work properly. Timing issues occur for a multitude of reasons, including transmission line propagation delay due to layer stackup, dielectric properties, and trace routing. Because delay issues are often caused by the unique physical properties of a PCB, the delay and length matching rule can automatically calculate necessary values from the design's layer stackup, and then check for equivalent delays and/or lengths on each net in a specified group.

In DDR designs that use fly-by topology, stub length is important for proper functionality. The fly-by topology rule checks to make sure that nets with fly-by topology are designed within the proper constraints.

The crosstalk coupling rule will help identify areas on a design where unwanted crosstalk occurs on sensitive nets. Crosstalk can cause serious timing and functionality errors, and can also be very difficult to manually diagnose on a manufactured PCB.

The power/ground width rule checks for narrow trace widths on power and ground nets. If power and ground traces are not designed to be wide enough, the resulting current on that net can be insufficient. This may lead to a host of problems including, but not limited to, a lack of adequate power supplied to components as well as unnecessary heat production.

The signal supply rule checks for discontinuities between an integrated component's supply planes and its connected traces' reference plane. These types of violations can lead to potentially strong radiation, and as a result, EMI failures.

The filter placement rule checks for the presence of filters within a close enough proximity to a connector's pins. To protect sensitive signals, as well as prevent radiation, filters are used to suppress noise that may be present on a connector. The absence, or misplacement, of filters on connectors can lead to serious EMI issues and failures.

The return path rule ensures that the tested signals have a sufficiently low-impedance return path. With the increase in today's high-speed circuit design requirements, as well as the decrease in PCB size, adhering to proper return path rules is incredibly important. If the return current on a trace is not able to properly flow underneath the conductor, it can instead take an unintended path through other areas of your circuit, possibly resulting in EMI issues.

DRC for First-pass Success

With tightly linked PCB layout and DRC tools, designers can ensure that their PCBs will not fail as a result of unnoticed SI, PI, EMI, and safety violations. Editable parameters within DRC allow users full control to adapt each test for their specific design requirements. With the aforementioned rules, in addition to many more, DRC can reassure designers that their PCBs will function properly while reducing costly board failures and design respins. When designs meet all advanced electrical rule expectations on the front end, companies can speed up their product's time to market, and ultimately improve profitability. **DESIGN007**

Rebecca Lord is a technical marketing engineer at Mentor, a Siemens business, who specializes in HyperLynx SI/PI.

John McMillan is a member of the PADS Technical Marketing team at Mentor, a Siemens business.

MilAero007 Highlights

U.S. House Approves Measure to Promote Lead-Free R&D in Milaero, Automotive, Medical ▶

IPC is applauding the U.S. House of Representatives for approving a measure that would promote research and development into the performance of lead-free electronics in high-reliability sectors, such as aerospace, defense, automotive, and medical equipment.

NASA Invests $45M in U.S. Small Businesses for Space Tech Development ▶

These selections have an estimated value of more than $45 million and are part of NASA's Small Business Innovation Research (SBIR) and Small Business Technology Transfer (STTR) programs.

SAMI Buys Riyadh-based Advanced Electronics Company (AEC) ▶

The agreement signing took place at a Saudi Arabia–U.K. industry event in London in the presence of senior shareholder representatives, including His Excellency Ahmed Al-Khateeb, chairman of SAMI, and Sir Roger Carr, chairman of BAE Systems.

BAE Systems Calls for Nationwide Effort to Develop Skills for Industry 4.0 ▶

BAE Systems has called for a concentrated effort by the industry, government, and education sectors to ensure that the U.K. can fully benefit from the digital revolution and Industry 4.0.

Of Art and Satellites ▶

A quotation from The Golden Record 2.0—a play written for the NUS Arts Festival—and a high-tech quantum device from the NUS Centre for Quantum Technologies (CQT) is now orbiting in space together.

NASA's Dragonfly Will Fly Around Titan Looking for Origins, Signs of Life ▶

NASA has announced that our next destination in the solar system is the unique, richly organic world Titan. Advancing our search for the building blocks of life, the Dragonfly mission will fly multiple sorties to sample and examine sites around Saturn's icy moon.

Collins Aerospace's David Adams Earns Dieter Bergman IPC Fellowship Award ▶

In recognition of his ongoing leadership in developing and promoting IPC standards on a global basis, IPC—Association Connecting Electronics Industries—bestowed a Dieter Bergman IPC Fellowship Award upon David Adams, Collins Aerospace, at the recent IPC SummerCom.

NASA Invests in 3D Printing for Aviation ▶

Additive manufacturing (AM), also known as 3D printing, is a promising new technology for all areas of aviation manufacturing. Additive manufacturing's flexibility in design and customization, cost reduction, and speed in delivering finished parts make it a perfect fit for creating parts used in aviation contexts.

Raytheon, Northrop Grumman Sign Teaming Agreement on Scramjet-powered Tactical Missile Systems ▶

Building on years of collaboration, Raytheon Company and Northrop Grumman Corporation have signed a teaming agreement to develop, produce, and integrate Northrop Grumman's scramjet combustors to power Raytheon's air-breathing hypersonic weapons.

Interview by the I-Connect007 Editorial Team

Zen Liao, CEO of Simplify Design Automation, recently spoke with the I-Connect007 editorial team about his company's high-level floorplanner for autorouting, which allows engineers to pass their ideas along to PCB designers. Zen discusses the floorplanning technology and his marketing strategies as well as the challenge of getting reluctant PCB designers to embrace autorouters.

Andy Shaughnessy: Zen, can you give us some background on your company?

Zen Liao: Our plan as a company was to provide a tool which could be very low priced, and even free for a certain period of time, that engineers can put their ideas for floorplanning on the PCB. Also, it could be a good entry point for people who want to use our router link. All of the large companies almost gave up the autorouting solution for the whole board. This included the big EDA vendors that have been putting a lot of investment into autorouting.

Now, they say they are focusing on interactive routing instead. A good example is Mentor's sketch routing, but we still believe that routing the whole board is important and doable. However, you need to do some planning on the board first to be able to produce a good result that is desirable for users. That's the whole purpose.

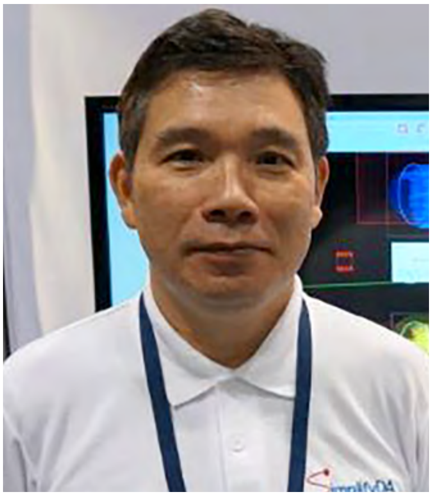
We tried to provide hierarchical steps before doing the detailed autorouting. You do floorplanning on the tool to specify what bundle you want to route, which layer you want to route, which channel you want the bundle to go through.

Shaughnessy: I was looking on your website, and you talk about setting up pin groups.

Liao: Right, a pin group is only one way to specify the bundle. You can select it from tools to form the bundle too because we're also doing pin swap routing. In that case, some pins might not have nets yet or a net ID. In that situation, you cannot select the nets. Instead, you select the pin group to define a bundle. We use the same methodology to define the bundle.

Shaughnessy: So, you're optimizing the routing before you even begin?

Liao: Optimizing, or just letting the user specify the route he or she wants. Later on, the router can follow that path which is specified to do the route. That's particularly useful for some bundles of nets coming from a BGA component that you want to specify or a certain direction or certain layers to route on. We think that certain high-level planning on the bundle and on the nets is necessary, and we try to provide the tool to do that.


Zen Liao

Shaughnessy: Does this come with the router?

Liao: The floorplan capability itself is also included in the router. If you purchase the router, the planning capability is already there. But we separated the floorplanning capability into a single separate software or product so that people can get it to do the planning for almost no charge or a very low price. For a certain period of time, there's no charge to use it.

Shaughnessy: That's pretty cool. And I know some of the big EDA companies have OEMed your router, right?

Liao: Yes.

Shaughnessy: Is this the kind of tool that they would be interested in as well?

Liao: Not yet. We provided the technology for them to enable their tool to do interactive routing. We haven't talked to them about the planning yet, but we will. You need to do the autorouting to make the floorplanner useful. Before, some tools provided a similar bundle planning capability, but I know that few people use it. Because if you're doing planning, it might be useful, but in my opinion, you need to have a good detail router to validate whether the result of your planning is good or not. That's a key issue. If detail routing produces a low completion rate, then it's not very meaningful. Fortunately, our router usually provides high completion rates similar to hand routes.

Dan Feinberg: Is it possible that someone with an older model router than you could use this to update it?

Liao: Theoretically, yes. The floorplanning result is saved in a floor file (or an FLW file). It's easy for anyone who wants to read in this file because the format is open. But as I just mentioned, one purpose is the floorplanning tool lets the engineers pass their idea to layout designers. The electrical engineers put their ideas on how those bundles or the whole board should be routed to floorplanning. Accordingly, they can read it in there. Usually, engineers don't work together with designers. The designers can just look at the result of the floorplanning produced by engineers and follow their idea to finish the design, so that's one idea. To answer your question, they can use any existing layout tool to use the floorplanner's result.

Shaughnessy: And you said before that you do routers not just for the board industry, but also for packaging and ICs.

Liao: Yes. For packaging, it's also useful because the big theme for packaging design is to do the feasibility study. Now, they have to manually route the nets to do the feasibility study or pinout assignment. With this tool, because you can define a bundle when you do the pinout assignment routing, it should be useful for the packaging routing and packaging design also.

Shaughnessy: We talked earlier about how you've been marketing this all around the world and that it's a little bit different when you're presenting this to a customer base in China or the rest of Asia. Could you talk a little bit about that?

Liao: We have been thinking about that for a long time, so we have been talking to our customers regarding these ideas, and they generally express strong interest in it. Some corporate customers don't have to use a planner because our autorouting solution already includes this planning capability. But this time, the planner is targeted to lower-end customers or the customers that are not using our tools yet.

Shaughnessy: What do you think is your biggest market right now?

Liao: PCBs because now, it's prevailing that some large companies have put a lot of R&D effort into autorouting technology and started realizing that autorouting the whole PCB is not easy to do. But the problem is that there isn't a hierarchical approach. Usually, you get the placement of the PCB, and that goes directly to route. We just get the placement and go to do detail route directly. Nowadays, the PCB is more complicated, and this approach usually doesn't produce a good result.

Shaughnessy: How do you plan to get more PCB designers to start using autorouters? Especially in North America, designers are almost proud of the fact that they don't use autorouters.

Liao: Some designers have autorouters, and they don't use them. We still have a long way to go.

Shaughnessy: How are you doing in China? Are you able to get market share there?

Liao: Not yet. We're mainly focusing our resources on development. But we're certainly looking at China's market and are talking to some vendors there to partner with us to do business development as well as technical support. In terms of PCB manufacturing, China is probably the number one market in the world. And design is growing more and more in China.

Shaughnessy: It's amazing how PCB design is almost unknown to college students in the U.S., but in some places, like Romania, it's now considered to be hip to be a PCB designer. Kids want to grow up to be PCB designers in Romania.

Liao: Right. I know a design company that is hiring engineers in Romania. They are happy about that. It's good that this industry is growing globally.

Shaughnessy: And it's considered cool in some regions. It's a good time to be in this industry. Well, what do you have planned next?

Liao: Our main focus is to continue making our customers happy because a lot of companies have started evaluating our technology, but evaluation takes time and resource to do correctly. Right now, our main focus is to make sure the customer doing an evaluation gets good results. In the future, we are looking for a lot of possibilities. Another focus would be expanding our presence through channels in Asia or in other parts of the world.

Shaughnessy: Do you have direct salespeople?

Liao: Not right now other than me. But we do have good partners in different parts of the country. Also, we are looking at more companies to partner with to expand. In Japan, the number-two EDA company, which is a popular listed company, is our main reseller. We are doing well in that region, and it covers other parts of Asia. Further, we expanded our presence into South Korea and are looking at partners in Taiwan, China, etc. Right now, we are doing more and more in Asia, but we are also looking into Europe and India, for example.

Shaughnessy: If you had Cadence or Mentor just agree to bundle your tool in as an option or however they would do it, would that be ideal for you?

Liao: Of course that would be good because they have huge user base. But nowadays, technology partnerships are more difficult than before, such as 20–30 years ago when we successfully did it at CCT. Because all those large companies you mentioned usually have their own activities going on with autorouting technology. It's not that easy, but we look forward to partnering with anyone who would be interested.

Shaughnessy: You were one of the earliest people at CCT, correct?

Liao: Yes.

Shaughnessy: And did CCT develop the first true commercialized gridless router?

Liao: I'm not sure. Some people will argue Racal-Redac was the first.

Shaughnessy: Right, Alan Finch was at Racal-Redac when he designed the first gridless router. I interviewed him once, and he said he was showing it at DAC, and Cooper and Chyan were standing there watching him.

Liao: Oh, really? (laughs)

Shaughnessy: Then, they came out with Speccta a few years later.

Liao: I read Alan Finch's paper, and I think he deserves a lot of respect. He's the pioneer of gridless routing. The business strategy in CCT was right. They made the router a standalone tool, and almost everybody in the industry used that technology.

Shaughnessy: It's a shame that designers have these great routers, and for the most part, they don't want to use them.

Liao: Right. We also try to incorporate the same approach as CCT so that they could plug it into any existing design environment, but it's becoming harder to do.

Shaughnessy: Would you like to add anything else?

Liao: The floorplanner is simple. We want to provide a tool to let the engineer be able to do floorplanning. I hope this idea will prosper in the industry and more and more people will use floorplanners and routers in the future.

Shaughnessy: Thanks for speaking with us, Zen.

Liao: Thank you. DESIGN007

New Nanoantennas to Improve Ultra-Fast Wireless Connections

Researchers of the Polytechnic University of Valencia (UPV), from the Nanophotonic Technology Centre (NTC), have designed new silicon nanoantennas with direct applications in communication and data processing for the next generation of reconfigurable photonic chips.

The work of the UPV researchers has been published in ACS Photonics ("All-Silicon On-Chip Optical Nanoantennas as Efficient Interfaces for Plasmonic Devices").

As Sergio Lechago, researcher at the NTC and co-author of the study, explains, plasmonic devices have enabled the development of important applications in fields such as spectroscopy, near-field and sensing optic microscopy thanks to their unique capability of manipulating light on a nano level.

Within the communications integrated in the chip, plasmonics enable the development of ultra-compact and affordable devices (modulators, detectors or sources) that can function at very high operation speeds with low energy consumption.

This new breakthrough could also be applied to fields such as biochemical or agri-food industries.

(Source: Asociación RUVID)

FLEX007

A SPECIAL DESIGN007 MAGAZINE SECTION

CLICK COVERS TO VIEW PAST ISSUES OF FLEX007 MAGAZINE

Flex Design Education Opportunities Growing—Slowly

What the Flex?

by Andy Shaughnessy, I-CONNECT007

There are no summer doldrums this year. This is a great time to be a part of the electronics manufacturing industry. A board shop owner at a recent conference summed up the last few years this way: "I've never seen it this good for this long."

Yes, the PCB market is hotter than Georgia asphalt in August, and flexible circuits are one of the hottest segments in that market. Demand for flex and rigid-flex circuits continues to grow with some analysts predicting that the global flexible circuit market could reach nearly $25 billion by 2022 with a CAGR of about 12%.

Automotive, healthcare, smartphones, and wearables continue to drive flex innovation. All of the new technologies coming online now fromIoT to 5G spell good news for the flex market.

From a big-picture perspective, the economy is roaring along with slow but steady growth each quarter. The unemployment rate keeps breaking records with a jobless rate of 4% or less for the past 16 months. As many of your managers know, the unemployment rate is so low that it can be hard to find someone to fill a position, and some jobs are going unfilled for six months or more. One manager joked with me that it would take a lot for him to fire a PCB technologist with the labor market so tight.

Try hiring a flex designer right now; you might be looking for quite some time. The flex designer group was always a small universe in the first place, and some of them are headed for retirement. You may have to do what other companies have done: Train your rigid board designers to lay out flex.

There aren't that many flex design classes at trade shows and conferences, but you can find a number of flex design tutorials online. The consortium NextFlex offers a variety of educational resources for flex designers and fabricators. And most flex fabricators and flex material suppliers have a set of guidelines for flex designers, and they'll be more than happy to work with you.

Speaking of material suppliers, we kick off this month's edition with a wide-ranging flex design roundtable. Designer Mike Creeden, the new technical director of design and education at Insulectro, interviews his Insulectro coworker Chris Hunrath and DuPont Circuit Applications Engineer Steven Bowles. The three discuss their work with customers who may be new to flex materials, the trends they see in the flex material market, and some flex design education options, which could include a visit to the customer-centric DuPont Technology and Innovation Center in Sunnyvale, California.

Next, we have an interview with John Michael Pierobon, who developed the free online course Designing Flexible Circuits, available on the website of the flex fabricator All Flex. John Michael, who has been known to teach programming classes in Spanish and Portuguese, discusses his class, which is broken up into seven chapters and covers everything from flex design guidelines to material requirements.

We also have columns by our regular contributors. Joe Fjelstad explains how flexible circuit technology is slowly but surely making its way into the colleges and universities of the world. And Tara Dunn discusses the benefits of an SMTA pilot program for emerging engineers.

Flex and rigid-flex are blowing up now, and Flex007 has the flex information you need. See you next month! FLEX007

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

Industry Experts Talk Flex Design


Mike Creeden

Flex007 Feature Interview by Mike Creeden
INSULECTRO

I recently spoke with Insulectro's Chris Hunrath and DuPont's Steve Bowles at the DuPont Technology and Innovation Center in Sunnyvale, California. We discussed a variety of topics related to flex design, including the support structure that's needed in flex design, the ever-changing world of flex materials, and the need for working with a flex fabricator as early as possible in the flex design cycle.

Mike Creeden: Welcome, gentlemen. Today, I'm with Chris Hunrath of Insulectro, the vice president of technology, and Steven Bowles, flex circuit applications engineer at DuPont. Today, we're visiting the DuPont Silicon Valley Technology and Innovation Center in Sunnyvale, California. This is an impressive facility.

To provide some background on myself, I'm now the technical director of design and education at Insulectro. I'm an EPTAC IPC master instructor teaching CID and CID+ programs, and I'm also the founder of San Diego PCB—a company I sold to Milwaukee Electronics Screaming Circuits in 2016.

Today's designers and engineers are finding the need for using flex, rigid-flex, and printed electronics in all market segments. As we start approaching the next generation of electronics, materials matter. So, the IPC Designers Council Executive Board created and recommended a definition for the profession of design layout that would be published in the first page of the IPC-2200 series designer specifications. The definition explains what a designer needs to do to make revision 1 work. It has been referred to as, "The Designer's Triangle," explaining three perspectives for success. The first perspective is DFS (designing for solvability): Truly solving HDI with micro devices as best you can by using any of the CAD tools in front of you. The next perspective for success that the designer needs to accomplish is DFP (design it for performance). As we route our circuits the copper plays a part, and the materials that exist between the copper play a significant part. I'm hoping Chris can address that. The third perspective is DFM (designing for manufacturability), and much of that is why this Technology and Innovation Center exists at DuPont. I'm hoping Steven can tell us about that because people need to be able to discover what the next step is as they consider flex design and how it can be built.

Steven, can you tell us a little bit about your background and your role at DuPont?

Steven Bowles: I'm a flex circuit applications engineer at DuPont Silicon Valley Technology Center. I've been with DuPont for a little over a year now. Before that, I manufactured flex, rigid, and rigid-flex circuits for about 15 years. I worked at a few different board shops, all do-

mestically, in multiple processes and engineering positions. I've also participated in IPC, am a current member of over a dozen standards development groups, and chair three of them.

Creeden: Next, can you tell us about yourself, Chris?

Chris Hunrath: I've worked both on the fabrication side and on the supply side of the PCB industry. I started with multilayers back in 1983 and never looked back. I have been working with the supply side about two-thirds of my career and with Insulectro for the past 18 years. I work a lot with our customers, especially on the flex circuit side, by helping them select the right materials and with the processes in the fabrication of flexible circuit boards.


Chris Hunrath

Creeden: Although the percentage varies for most designers, the ratio of how many flex circuits to rigid circuits is often low. Therefore, a relearning curve is often a concern. The information about the flex design, manufacture process and correct material selection may not be readily available to them. How could a designer start making some of those decisions, and when should they make those decisions?

Hunrath: That's a great question. If you were to look at flex circuits compared to rigid PCBs at a high level, I like to use the analogy that in flex circuits, the Kapton® material—which most people are familiar with—is the replacement for fiberglass. In other words, if you were to look at rigid PCB stackups, you have copper and prepreg or glass-reinforced layers. In flex, it's Kapton on an adhesive with the copper. In the flex world, Kapton is the base material, and then you add copper and adhesives to it to make the various kinds of stack ups. There are lots of other important factors in doing a stackup in terms of where the bend axis is as well as the neutral axis, etc. But if they keep that concept, the Kapton is the flex equivalent of fiberglass in rigid. I think that helps a lot in how they would visualize a stackup.

Creeden: There are many different materials out there and different types of circuitry getting more into high-speed and RF. All you have to do is say the buzzword "5G" and everyone's ears perk up. Steve, can you give me a brief overview of some of the materials based on what Chris just said?

Bowles: Coming directly from DuPont, we are marketing heavily in our existing commercial materials—Pyralux® AP and TK are some of the best materials for high-speed and high-frequency applications. The Pyralux family of materials are the industry gold standard for reliability and performance.

Creeden: And earlier, we were talking about low-loss materials. Those appeal to an RF engineer.

Hunrath: Flex materials don't have skew because they don't have fiberglass, so they're uniform in their dielectric properties. On the AP, TK, and HT classes, HT adhesives have good performance numbers. And knowing the rigid materials and how they stack up there, they're on the higher end of the rigid material spectrum in terms of loss in dielectric constant.

Creeden: It's quite a challenge when an RF engineer wants to send an RF signal across a flex board, and they're concerned about emissions. You can add a 45-degree ground plane to still retain some flexibility, and perhaps make it in a hatch pattern; that's one way of coaxing these signals.

Hunrath: Yes. There is data available on using those cross-hatch patterns. Even the geometry of the cross-hatch patterns can be optimized to get more performance, and that's an area where DuPont can help. DuPont has some good data on those design characteristics, and I would encourage someone doing a design if they need that to reach out to Insulectro or DuPont. At Insulectro, we'll get you in touch with the right people if we don't have the data in hand, but DuPont has people who have expertise in those areas.

Creeden: A lot of times, the fabricator is going to build it in their supply chain. When should they be contacting them?

Hunrath: Ideally, the earlier, the better because you don't want your design to be backtracked. The earlier that the designer gets the information they need to solve some of these issues, or to meet the performance requirements, the better off everyone is.

Creeden: Right. Steven, what's driving some of your customers into flex right now?

Bowles: Higher speeds, higher frequencies, and smaller devices.

Creeden: Smaller devices in overall housing?

Bowles: In every axis possible. We have customers coming to us, saying, "We have an existing design where we want to reduce the Z, and the total overall stackup thickness needs to be cut in half." With our materials, the films themselves are relatively thin, and we can throw thin copper weights on them too; we can also build a much thinner stack of the flex. There are some thin rigid materials as well, but that's one area that the flex performs for us. Further, we can build and accommodate specific designs that have a lot of bends and kinks in them.

Hunrath: DuPont has a cable and a corresponding flex with the same circuitry and the weight and the performance. That's one of the things you could see at this Innovation Center: the difference between a cable system, or a cable harness, and a flex circuit that replaces that same cable harness. You'd see that there's a huge difference in the fit.

Creeden: That gets back to my original thought about the designer's triangle of three perspectives to look at everything in layout. There's a solvability that with that cable, you're going to replace it with something that is much smaller. You're saying it will typically out-perform because you don't have all the parasitics that the actual pad, cable, and solder have and brings about in that connector.

Hunrath: You save weight. You increase performance, and often, you can save costs too.

You save weight. You increase performance, and often, you can save costs too.

Creeden: Regarding removing connectors, what can I expect would be one of the challenges in moving away from a connector, and going to a flex?

Bowles: From a design perspective, there are many considerations moving to flex. Some people build rigid-flex or try to build a rigid-flex in their traditional rigid shop or vice versa. They might be a traditional flex shop, and now you're trying to have them build a rigid-flex design. There's a learning curve with the difference in materials there.

Hunrath: From a design perspective, there are "do's and don'ts" in flex. If you're going from a cable system—from connector to connector to a pure flex or even a rigid-flex—you don't do things like I-beam circuits, right? You want to make sure that the circuits on both sides of your double-sided flex are offset to improve flex life and flex performance. You have better flexibility and better flex life. Again, a lot of this stuff is well understood; we have that information, and we want to share it and promote it. But that would be one example; if someone went from a cable system not knowing the nuances of flex, they might run into some trouble.

Creeden: Oftentimes, it's the transition from rigid to flex where issues occur. Are there any challenges you'd see there? We've talked about sometimes material oozing out at that junction point.

Bowles: Yes, watch your squeeze-out. But there's also the application of strain relief because of material being flexed. Keep conductors and through vias or any data structure away from that transition zone or limitations to the area, and obviously follow design recommendations.


Steven Bowles

Creeden: We alluded to the fact that there's not a lot of training going on out there in flex. From teaching CID and the CID+, there's some flex information in there, but I think there could be more; it's a bit of a challenge in our industry.

Hunrath: Many fabricators have good flex experience, so they would be good to reach out to. But I also encourage people to come to the Dupont Tech Center in Sunnyvale because we've had several meetings where we've brought in the fabricator and OEM, and that's worked out well.

Creeden: Again, this is an impressive facility. This is the second time I've visited, but now it's fully functional.

Bowles: And you haven't even seen the Innovation Center yet.

Creeden: Right. Can you give our readers a brief overview of what's here, and what service do you hope to provide to OEMs and customers designing boards?

Bowles: DuPont acquired a photovoltaic startup back in 2011 called Innovalight and quickly converted it to a Customer Interfacing Technology Center supporting a few of our different businesses. One of those was the Interconnect Solutions Group that supports Pyralux® and Kapton®, which is the business side I work in. The goal of our site is to work with as many customers in the Bay Area. Obviously, we'll reach further out, but we hope to get in early on prototype applications, help our OEM customers with their designs, and get our material specified in early, as Chris mentioned.

We can bring designers and customers to the site, walk them through the lab space, teach them a little bit about our materials—such as how and when to best use them—and then offer a prototyping service. We can essentially build a flat circuit design, for example, on our materials, and build it on competitive benchmark materials if we need to. Then, we can characterize and analyze the prototype itself and supply data and recommendations to the customer.

Creeden: So, this doesn't have to be a full-blown design. What you're going to do is provide a test vehicle for the transmission line.

Bowles: Correct.

Creeden: They know what their survey is supposed to do, and they know what they want to receive on the load end. How would readers engage with DuPont's Technology Innovation Center?

Bowles: One, come to the Technology Center. Two, you can reach most of us via email or LinkedIn, but our site in Sunnyvale has open doors, and a lot of people connect with us through Insulectro, IPC, and other industry events. You can reach me at Steven.Bowles@DuPont.com.

Creeden: What trends do you see as you look out into the electronics development going toward flex and/or printed electronics?

Hunrath: We're seeing a ton of automotive and medical applications using electronics. Some innovations include weight, heat, reliability, and internet connectivity of the different devices in vehicles. The internet of things (IoT) applies to a car as much as anything else these days. With medical, it's all about being small, thin, and flexible. There are lots of innovations with surgical devices, both single-use and reusable. It's a great space to be in that's helping a lot of people.

Creeden: Regarding printed electronic circuits (PEC), normally, you have copper, which has both a smooth and a rough side. All of those things play a part in the transmission of our signals. What can you tell me about printed electronics?

Hunrath: It's a fully additive process. Instead of applying a sheet of foil to a dielectric and then selectively etching, you're printing your circuitry on a substrate of some sort—usually Mylar or polyester film, but not always—and you're building up your layers on that.

Creeden: And what is that material that you're printing with?

Hunrath: Typically, it's silver—at least there are silver conductors. DuPont has a whole range of conductors for various applications—everything from fired-on to in-mold and membrane touch switch. They have a broad range of products with various binder systems for different applications, but most often, it's silver-based. However, DuPont does have other products as well.

Creeden: Based on most people's understanding, a printed electronic is usually a single-layered circuit; however, it can be a multilayered circuit, and it's not that difficult to achieve. Can you tell us about that?

Bowles: Chris already touched on it with the low-temperature, co-fired ceramic (LTCC) material that DuPont offers; essentially, we print silver and then co-fire a ceramic. We can also print specific gold applications, but we use those in higher frequency applications, and it is a rigid material—not a flexible material like Pyralux.

Creeden: So, it can serve as a way to encapsulate and separate two traces?

Hunrath: You can do many layers in one shot. Basically, you print the individual dielectric green layers all at once. You take unfired substrate, apply your conductors, stack them, squeeze them together, and then fire. You end up with a high-performing multilayer that is very low loss in a lot of cases.

You can do many layers in one shot.

Creeden: And this can all be put into a 3D model to be form-fitted to whatever you want.

Hunrath: They are planar circuits, and they're very rigid when fired, but it is a 3D structure in terms of the layers and conductors.

Creeden: Right. The point I'm trying to convey is that it's not the 2D world we think of for a regular or flex board.

Hunrath: You can have cavities and other features in there. The material doesn't melt and flow like other materials.

Creeden: Is there anything else you would like to add as a closing statement?

Hunrath: We're here to help. We supply materials of all kinds, so let us know what you need.

Creeden: Thank you both. And designers, as you consider doing flex or printed electronics, there's usually a support structure needed. Talk to your fabricator early as well as material suppliers, also if you get the chance, visit the DuPont Technology and Innovation Center.

Hunrath: You're welcome.

Bowles: Thank you. FLEX007

John Michael Pierobon Discusses His Flex Design Class


John Michael Pierobon

Flex007 Feature Interview
by Andy Shaughnessy
I-CONNECT007

We all like free things. Well, if you're new to flex design, you might want to check out an online course offered by All Flex, a flex fabricator. The course, Designing Flexible Circuits, was created by John Michael Pierobon.

John Michael is an engineer who has presented a wide variety of high-tech classes around the world on topics that include C programming, Red Hat Linux, and Z shell scripting. He even has a course titled *How to Succeed in International Business.* He can teach in English, Spanish, and Portuguese, and he's not too shabby at Italian and French either.

I asked John Michael to discuss his flexible circuit design course, and whether online curriculum can ever replace good old-fashioned live instruction.

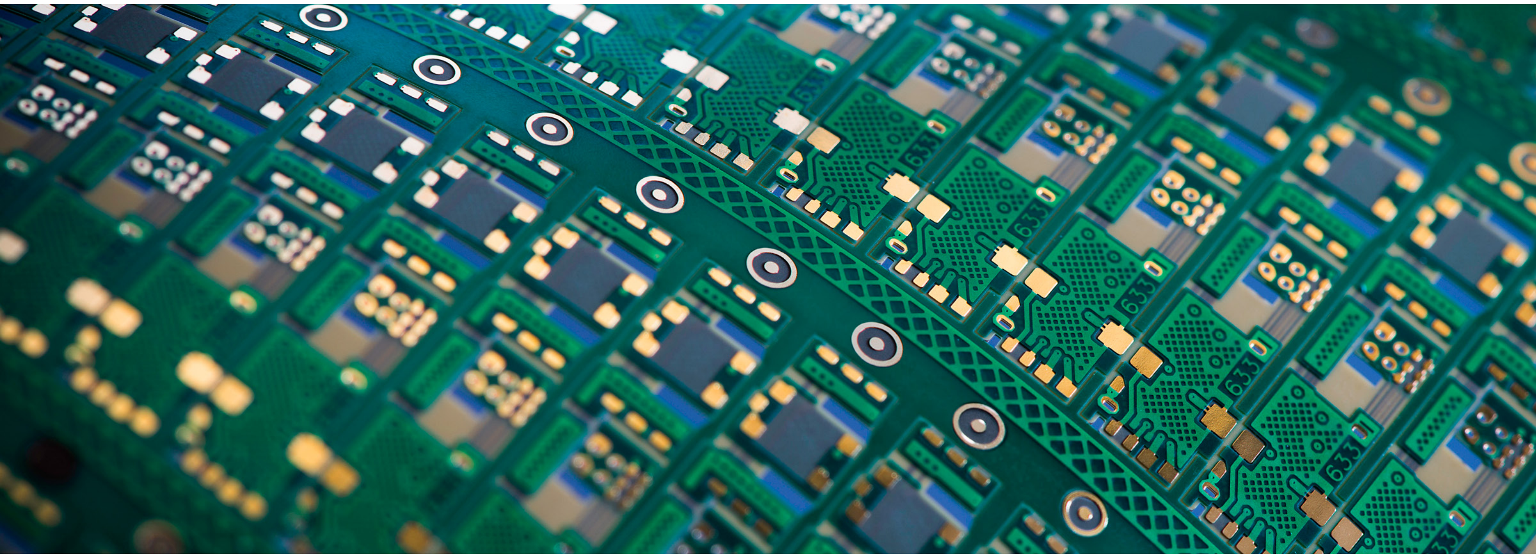
Andy Shaughnessy: Give us a little background about yourself and your company.

John Michael Pierobon: I was born in Minnesota. I grew up in Latin America, having lived five years in Brazil, four years in Mexico, and eight years in Peru. As an undergraduate, I worked as a computer operator to pay for tuition. As a graduate student, I taught Fortran. After I graduated, I worked as an engineer. I got my PE license. I wrote computer programs for engineering applications and then went to work for scientific computer companies. I ran the training department for one of them. Then, the internet exploded. I ran an internet service provider and started teaching courses on building and running the World Wide Web. Now that it is built up, I concentrate on writing and teaching technical courses for companies. I live in Fort Lauderdale, only 148 steps from the beach.

Shaughnessy: How did you start teaching flexible circuit design?

Pierobon: One day, one of my favorite customers asked me to write a course on designing flexible circuits. I had the time to do it, so I did.

Shaughnessy: Can you walk us through the online flex design class that All Flex sponsors? It's a fairly comprehensive class and free to the users, which is really nice.




109C

Pierobon: Yes, it is free. I encourage your readers to sign up at designingflexiblecircuits.com. Hundreds of people from all over the world have taken the class. The course is written in an easy-to-understand language, explaining technical concepts, and each chapter ends with interactive review questions. I have been told that people like the course very much because each page offers a bite-size piece of information, which makes it easy to digest and comprehend. The seven chapters are:

1. Flexible Circuits
2. Design Options
3. Flexible Circuit Materials
4. Flexible Circuit Requirements
5. Design Guidelines for Flexible Circuits
6. Light Emitting Diodes in Flexible Circuits
7. Flexible Heaters

There is an extensive glossary because there are a lot of technical terms in the course. The entire course can be completed in one day, but you can start and stop at any point. Your email address is only used to remember your place in the course. I have added plenty of hyperlinks in the course to make it easy to navigate and to jump around the material as needed.


John Michael lives on the beach in Fort Lauderdale, Florida.

Shaughnessy: Do you ever teach flex design classes in person?

Pierobon: No, but no one has ever asked.

Shaughnessy: What advice would you give someone just starting out designing flex? Are there any other educational resources, books, or webinars that cover flex design?

Pierobon: Take my course. After all, it is free. Also, All Flex has many resources on its website. As for other educational sources, there is plenty of material on flexible circuit design online.

Shaughnessy: What do you see as the pros and cons of online courses? Do you see them as complementing in-person classes or eventually replacing them?

Pierobon: If you are a night owl, online courses are for you. You can take online courses at any time and go at your own pace. If you have the discipline and the self-motivation, online courses are for you. Online courses are for those whose financial budget is extremely constrained.

I prefer to teach classes in person for various reasons. One, I can read the body language of the students and can quickly tell when they are distracted or feel lost. The classes I teach have interactive exercises, and if a student gets stuck, I am there to resolve the issue. Students pay more attention, learn more, and do not have distractions that may plague online courses. But I do not foresee online courses replacing in-person classes.

Shaughnessy: I also noticed that you teach a whole range of other high-tech subjects, such as MySQL, Red Hat Linux, thermal sensors, and dozens of other topics. How did you end up teaching such a wide range of topics?

Pierobon: As an engineer, I enjoy doing research, and I like to learn new things. I learned most of the subjects in engineering school: C programming, UNIX, thermodynamics, material science, structural design, mechanical design, hydrology, statistics, etc. It was just a matter of having the time, interest, experience, and background to fill in the rest.

Customers ask me to write courses for them. If I know the subject, I will do it. For example, I was asked to write an online course on temperature sensors. I know thermodynamics and material science, so I did my research and came up with temperaturesensortraining.com. This particular course may also be of interest to some of your readers.

Shaughnessy: You're quite a polyglot. Other than English, what languages do you teach in?

Pierobon: Spanish and Portuguese. Students prefer to be taught in their mother tongue. As a matter of fact, the last time I taught C programming, it was in Spanish. I have taught technical courses at the university level on six continents. I am truly grateful to All Flex for their continued support.

Shaughnessy: Thanks for your time.

Pierobon: Thank you, Andy. **FLEX007**

A Wearable Device So Thin and Soft You Won't Notice It

Researchers have reported the discovery of a multifunctional ultra-thin wearable electronic device that is imperceptible to the wearer. The device allows the wearer to move naturally and is less noticeable than wearing a Band-Aid, said Cunjiang Yu, Bill D. Cook Associate Professor of Mechanical Engineering at the University of Houston and lead author for the paper, published as the cover story in Science Advances.

"Everything is very thin, just a few microns thick," said Yu, who also is a principal investigator at the Texas Center for Superconductivity at UH. "You will not be able to feel it."

It has the potential to work as a prosthetic skin for a robotic hand or other robotic devices, with a robust human-machine interface that allows it to automatically collect information and relay it back to the wearer.

That has applications for health care (Yu asked, "What if when you shook hands with a robotic hand, it was able to instantly deduce physical condition?") as well as for situations such as chemical spills, which are risky for humans but require human decision-making based on physical inspection.

While current devices are gaining in popularity, the researchers said they can be bulky to wear, offer slow response times and suffer a drop in performance over time. More flexible versions are unable to provide multiple functions at once—sensing, switching, stimulation and data storage, for example—and are generally expensive and complicated to manufacture.

The device described in the paper, a metal oxide semiconductor on a polymer base, offers manufacturing advantages and can be processed at temperatures lower than 300°C.

"We report an ultrathin, mechanically imperceptible, and stretchable (human-machine interface) HMI device, which is worn on human skin to capture multiple physical data and also on a robot to offer intelligent feedback, forming a closed-loop HMI," the researchers wrote.

(Source: University of Houston)

SMTA Pilot Program for Emerging Engineers

Flex Talk

by Tara Dunn, OMNI PCB

The age gap in the electronics industry is something we discuss at nearly all of our industry trade events. We have been talking about this for a few years, and it is now becoming a reality. A significant portion of our PCB designers and most experienced engineers are reaching retirement age and will be exiting the industry in the next 3–5 years. Meanwhile, collectively, there has not been enough done to help train the younger talent we have and to recruit emerging engineers to our industry. Currently, several programs are being launched and supported by industry associations to help bridge this gap.

I had the opportunity to talk to an engineering student interning at a PCB fabricator for her second summer and asked her how she became interested in PCB technology. Her honest answer was that she was not aware of potential career opportunities in the electronics industry until she saw an ad posted at her college. One of the primary reasons she applied was that the opportunity would allow her to stay in the area for the summer. Since then, she has developed a broad knowledge of the industry and is planning to stay in electronics after graduation. She will certainly be an asset wherever she lands.

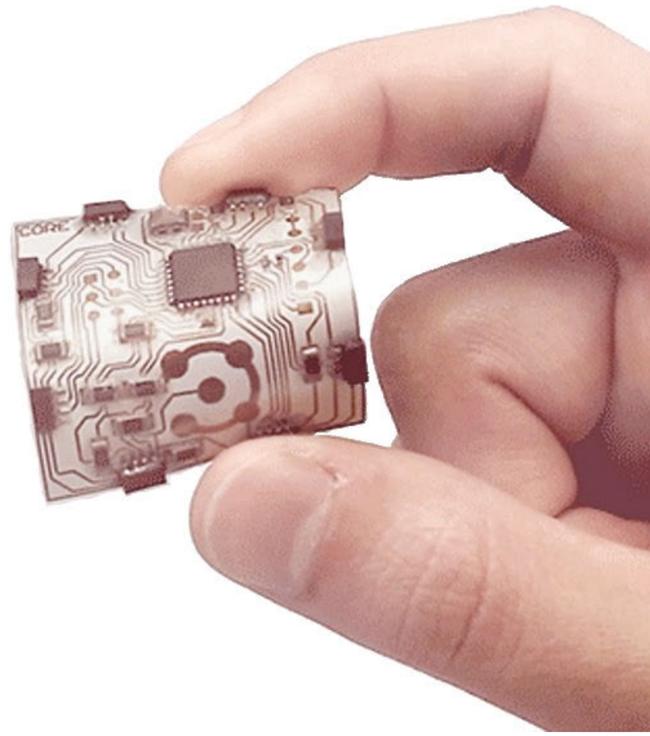
Trying to target engineering students at the post-secondary level, SMTA is piloting a program this fall, aiming to call attention to the potential career paths in the electronics field and mentoring opportunities. I had the opportunity to sit down with Tamara Shephard, membership engagement manager at SMTA, to learn a little more.

Tara Dunn: Hi, Tamara. Thanks for taking a few minutes to talk about this new program. Before we dig into that, for those that are not familiar, can you provide a little background on what SMTA is?

Tamara Shephard: Thank you, Tara. SMTA is an international association for electronics engineering and manufacturing professionals. We offer exclusive access to local, regional, domestic, and global communities of experts as

Tamara Shephard

DUPONT®



PYRALUX® FLEXIBLE CIRCUIT MATERIALS

PYRALUX® AP
high reliability flex & multilayer circuitry

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

PYRALUX® HT
for high temp, high performance flex

PYRALUX®TK
Teflon®/Kapton®

PYRALUX® LF
industry standard for high reliability

PYRALUX® FR
flame-retardant for products requiring UL rating

PYRALUX® APR
reliable resistor, temperature tolerant

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

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

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

PROUDLY DISTRIBUTED BY
INSULECTRO

20362 WINDROW DRIVE • LAKE FOREST, CA 92630 • 949.587.3200 • INSULECTRO.COM well as accumulated research and training materials from thousands of companies dedicated to advancing the electronics industry. Currently, SMTA is comprised of 55 regional chapters around the world and 29 local vendor exhibitions (worldwide), 10 technical conferences (worldwide), and one large annual meeting.

Some of the greatest benefits come from the SMTA's mission of the sharing of knowledge and best processes by bringing educational content and a global network to local regions. It's a wonderful opportunity for students and young professionals to connect to their field domain and potential recruiters.

It's a wonderful opportunity for students and young professionals to connect to their field domain and potential recruiters.

Dunn: Being involved at the local chapter level, I know SMTA has been working diligently to grow student membership and increase the number of student chapters. What is driving that effort?

Shephard: As we are all aware, there is a gap between retiring baby boomers and millennials. Soon, there will be a large number of retirees, and millennials will need to step in and fill those vacancies. SMTA provides both students and young professionals access to those interested in mentoring them. We have seen firsthand that when you combine the hunger of the up-and-coming engineers with experienced industry professionals and tribal knowledge, great things can happen. Sometimes, people just need an open door and a friendly face on the other side to greet them and show them the way.

Dunn: I couldn't agree more. I encourage all of us "more experienced" people to consider mentoring. At the Upper Midwest Chapter Expo, the topic of industry recruitment came up during a breakfast discussion. We spoke at length about this looming crisis in the PCB design, fabrication, and assembly sectors, as so many industry veterans are hitting retirement age. One aspect we discussed was how do we recruit at the college or technical college level and attract emerging engineers to this industry? SMTA is planning to launch a pilot program this fall at a local university. Can you give an overview of the program you are planning?

Shephard: Absolutely! We are very excited about this new program. The intention is to create a program for engineering students that spotlights the electronics industry and gives real-world examples of the types of career paths available and the impact that the electronics field has on our lives. Our industry doesn't rely on only one type of electrical degree—we are in need of mechanical, chemical, and process engineering studies. One of the challenges of recruiting to this industry is helping students realize that there is a career opportunity in this field, giving them the chance to make connections, and making it easy for them to seize that opportunity.

Dunn: There is a lot of competition for emerging engineering talent. What other industries are trying to promote interest in engineering and recruit young engineers? Who is our competition?

Shephard: There are so many programs starting at the high school level—and even earlier—surrounding robotics programs, gaming, coding, etc. Hopefully, we will soon be talking about clubs and courses that dive into the basics of the electronics that power these fun activities.

Dunn: Thinking of our local electronics industry, I understand you are planning to involve both local OEMs and suppliers in this program. In what ways can they participate and become involved?

Shephard: Yes, both OEMs and the local supply chain have an important presence at SMTA. Local organizations have graciously volunteered to provide technical content, participate in panel discussions, and explain the work that their organizations do. In addition to the educational piece, we are also planning to have opportunities to pair students with job shadowing, facility tours, and connections with recruiting functions.

Dunn: In addition to providing technical information, I understand you also plan to provide up and coming engineers with some of the soft skills to help guide them along their career path?

Shephard: Correct. Our programming will also be dedicated to building essential soft skills necessary for success in the industry, such as resume building, making strategic connections, career management, interpersonal skills, etc.

Dunn: Before we wrap up, can you touch on other programs SMTA has in place for post-secondary students and young professionals?

Shephard: We are expanding on Young Professional Programming both at the chapter and headquarter level. We have started providing breakfast briefings for young professionals at select expos to build this initiative. These breakfasts are a group discussion with a knowledgeable industry expert. The goal is to expand their thinking in these areas and facilitate peer connections necessary for the success in the industry. We also recently released an eBook on our website dedicated to crucial soft skills as well as industry-specific knowledge and company HR contacts to aid in internship and job placements.

Further, we will be expanding our Young Processionals Programming globally through our SMTA International Conference and Expo this fall in Rosemont, Illinois. This is a great opportunity for young professionals and engineers new to the industry. The first day will include six free 45-minute technical presentations by industry experts. Day two will focus on soft skill topics and includes Jim Springer, the featured speaker from John Deere, addressing leadership and career management.

Dunn: How can people get in contact with you to learn more?

Shephard: Please reach out to me through our website, www.smta.org, or find us on Facebook, Instagram, or LinkedIn. **FLEX007**

Further Reading

Benefits Book for Young Professionals

Special Events

SMTA International Special Event Calendar

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.

Soft Wearable Health Monitor Uses Stretchable Electronics

A wireless, wearable monitor built with stretchable electronics could allow long-term health monitoring without concern for skin injury or allergic reactions.

"This health monitor has a key advantage for young children who are always moving, since the soft conformal device can accommodate that activity with a gentle integration onto the skin," said Woon-Hong Yeo, assistant professor in the George W. Woodruff School of Mechanical Engineering and Wallace H. Coulter Department of Biomedical Engineering at Georgia Tech.

Because the device conforms to the skin, it avoids signal issues that can be created by the motion of the typical metal-gel electrodes across the skin.

(Source: Georgia Tech)

Flex007 News Highlights

26 Meters of Flex!

Barry Matties spoke with Philip Johnston, managing director of Trackwise Designs, about the company's patented length-unlimited multilayer printed circuits aimed at replacing conventional wire harnesses.

EPTE Newsletter: JPCA Show 2019, Part 2

Many new technologies and products related to flexible circuits were introduced by mid-sized manufacturers at the JPCA Show 2019.

Dissecting the IPC Regional Survey on PCB Technology Trends

Sharon Starr, Denny Fritz, and Mike Carano talk about the global 2018 IPC Technology Trends Report released early this year—the size of the survey, how it was conducted, the general findings, and regional differences.

The PCB Norsemen: From Wooden Huts to Homemade Go-karts–It All Starts With Design!

Whether building the coolest go-kart or the most sophisticated electronic hardware, the story is the same: It starts with design. And for designers and manufacturers, early involvement and commitment between all the involved parties in a product development process diminish the risk for mistakes and misunderstandings.

Flex Talk: Spark an Idea

One of the favorite parts of Tara Dunn's job is the days when she meets with a group of engineers and designers to talk about flex and rigid-flex. They might do a "lunch and learn" with a general overview of the technology or address a specific challenge. It is always helpful to bring samples to pass around and show different features. Usually, looking at a sample will spark an idea and the comment, "I wonder if we could do something like this." From there, the brainstorming begins.

AT&S Marks 20th Anniversary of IPO

AT&S has just achieved two significant milestones: On the one hand, the company is the globally leading supplier of high-end PCBs, according to the figures of the industry expert Prismark for 2018. On the other hand, AT&S is celebrating its successful IPO 20 years ago and its equally positive development over two decades.

The Big Picture: Globalization—What Happened?

Cheap products and services from places like China and India are good, but giving up the position of being the top dog doesn't sit well with most people—especially when you have leaders around the world reminiscing about the past and wanting to make XYZ great again or something similar.

NCAB Group Denmark Names New Managing Director

Jan Kronholm Thomsen takes the role as new managing director for Denmark. Jan was previously the managing director of Multiprint, that NCAB acquired in January 2019.

Aismalibar Increases Manufacturing Capabilities in Barcelona Facility

Aismalibar recently updated its Barcelona facility with three new pieces of equipment to assure their continued leadership in the thermal management laminate market.

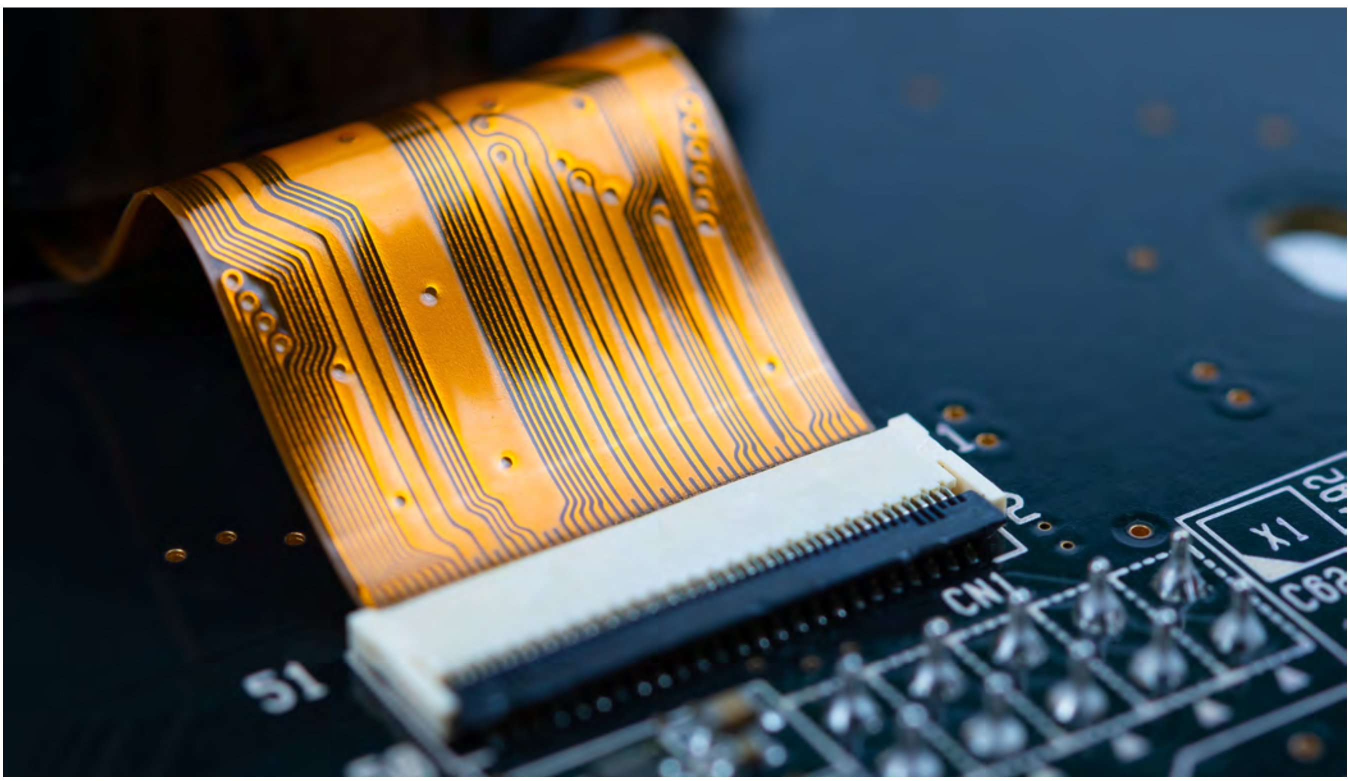
Flexible Circuits Go to College

Flexible Thinking

by Joe Fjelstad, VERDANT ELECTRONICS

Flexible circuits have been in use in various ways for over 100 years, as argued by my friend, Dr. Ken Gilleo, based on a patent search he did many years ago. A flex patent was issued by the British Patent Office to Albert Hansen at the turn of the 20th century. Most of the advances and development, and the promulgation of technology have been through efforts within the electronics industry that were sparked and led by a need for electronic interconnection solutions that allowed product developers to take advantage of flex circuits' unique 3D capabilities, including making connections that were physically dynamic.

This industry provided its own interconnection solutions as printed circuits, which had been long considered low-tech and uninteresting to researchers within academia. There, the focus had primarily been on the more exotic and higher-value semiconductor technology sector where the "big prize money" could be found and where grant money was available to do the kind of expensive research that is generally required for breakthrough solutions in materials and processes to sustain the march of development. Interconnections used to facilitate the use of the magic found on the semiconductor devices were prosaic and of little interest.



However, over the last couple of decades, there has been a slow but steady shift in interest by universities and research institutes around the globe in what comes after the chip, as it is a gating element in the hierarchy of electronic product. Major universities (such as Georgia Tech, Penn State, and the University of Illinois, just to name but a few in the USA), international research institutes (such as Fraunhofer in Germany and IMEC in Belgium), and consortiums (such as NextFlex in the USA, which collaborates with a number of different colleges and universities) are all, and in many cases, have been, engaged in developing products and interconnection solutions that involve flexible circuits including materials, processes, and products.

There are a few things that appear to have sparked the increase in interest. One is the rebranding of some flexible circuit technology as flexible electronics, and another has been the growth in experimentation with stretchable circuits.

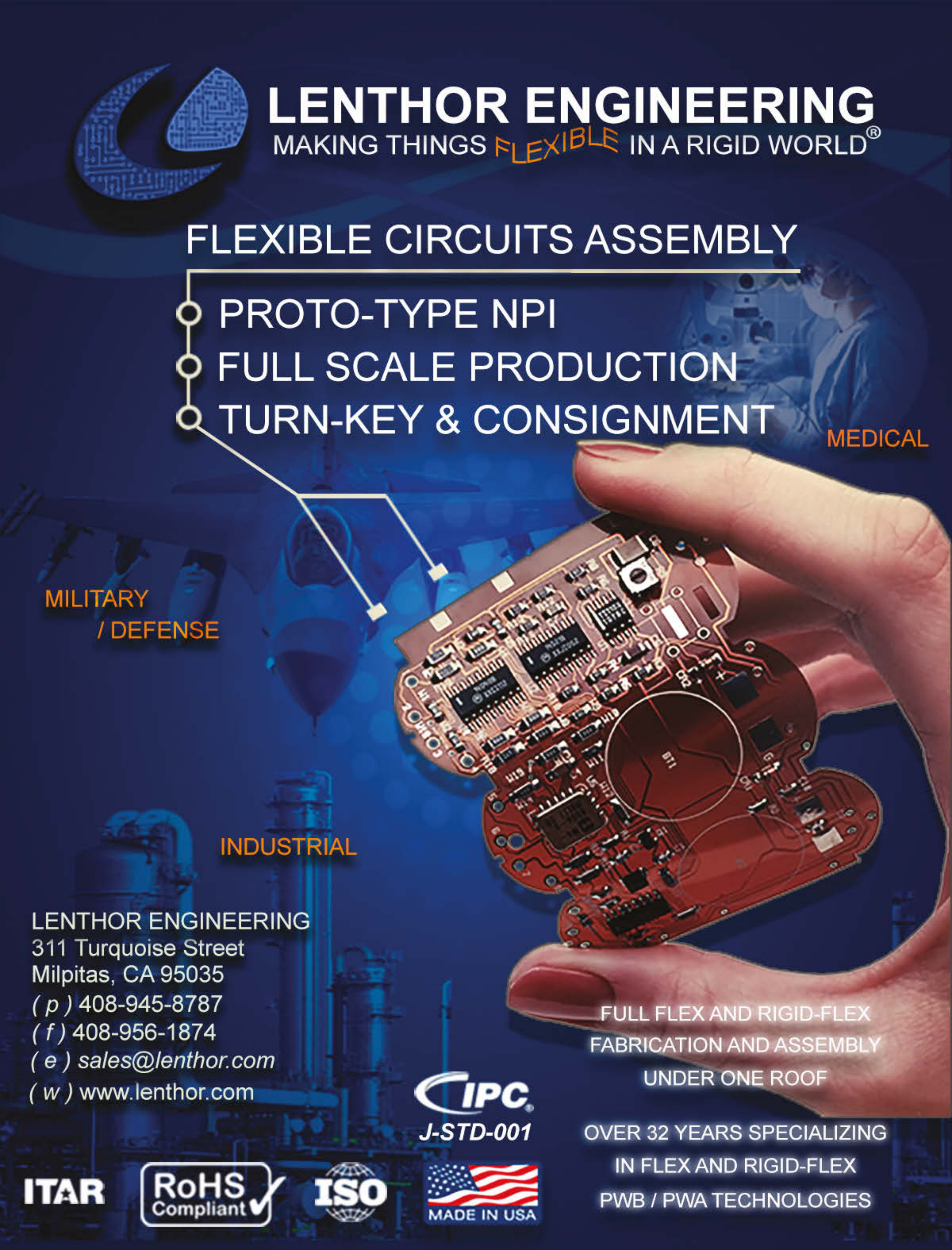
There are a few things that appear to have sparked the increase in interest. One is the rebranding of some flexible circuit technology as flexible electronics, and another has been the growth in experimentation with stretchable circuits. Further, the rise of the internet of things (IoT) has steadily looked to employ flexible circuit materials and processes to create IoT tags, which are poised to connect billions (and ultimately, trillions) of physical objects to the internet.

The interest in flexible circuits within the world of academia is as an enabler to help students and their advisers realize their creative ideas, and in many cases, spin out companies to build and market them. Many universities have a stake in these companies, and they can be lucrative sources of income for both the professors and the schools. It is providing students both with an education in an important technology, and a runway for early success should their ideas succeed in taking off. Even if they do not enjoy immediate success, the knowledge they gain about flexible circuit technology can be put to good use in the industry with established electronic product companies. This is likely to provide value to entry-level engineers as the appreciation of the importance of interconnection technologies continues to rise.

As I first observed and commented a number of years ago, printed circuits were the original "integrated circuit" as they interconnected the various transistors, resistors, and capacitors required for the application. Semiconductor technology allowed for that integration to be radically reduced in size; yet, the printed circuit never went away, and the flexible circuit has vastly improved its potential. Again, this has not been lost on universities around the globe, and flexible circuits are likely to play an increasing role.

In summary, educators and student at institutions of higher education in technological fields have taken note of the unique ability of flexible circuit technology to help solve problems in many different and disparate areas, chasing improvements in computing, communications, medicine, biomechanics, and beyond. Having their attention bodes well for continuing advances for the flexible circuit industry. Expect many great things to come from the partnership of academia and industry in the future. **FLEX007**

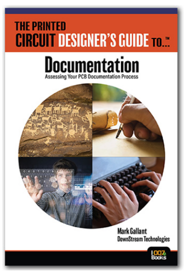
Joe Fjelstad is founder and CEO of Verdant Electronics and an international authority and innovator in the field of electronic interconnection and packaging technologies with more than 150 patents issued or pending. To read past columns or contact Fjelstad, click here.



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
IPC
J-STD-001
OVER 32 YEARS SPECIALIZING
IN FLEX AND RIGID-FLEX
PWB / PWA TECHNOLOGIES
ITAR
RoHS Compliant
ISO
MADE IN USA

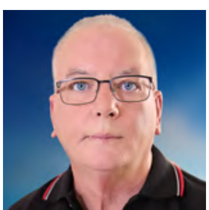
Editor Picks from PCBDesign007

1 New Book Examines Critical Need for Proper PCB Documentation ▶



Bring your knowledge of PCB documentation into the 21st Century with *The Printed Circuit Designer's Guide to… Documentation*—the latest title in our educational library.

2 Beyond Design: The Key to Product Reliability ▶


Barry Olney

With today's rapid product development cycles and time-to-market pressures, PCB designers are pushed to their limit. This situation leaves many developers with the question of how to ensure that their high-speed digital design performs to expectations, is stable given all possible diverse environments, and is reliable over the products projected life cycle.

3 The Shaughnessy Report: Reliability Is a Team Sport ▶

My 2003 Mazda Tribute doesn't look very cool; it's classified as a "cute ute." But it can haul four guitars and a pair of PA speakers with room to spare. It's been paid off for so long that I've been able to put more money away for my rapidly approaching golden years. As the saying goes, "Reliability isn't just an added feature."

4 Design Is a Pivotal Peice of the Puzzle ▶


Julie Ellis

Julie Ellis sees a wide variety of customer design requirements. In this interview with the I-Connect007 Editorial Team, Julie explains how PCB designers can influence the development of the PCB. She shares a variety of tips and tricks that designers can implement early in the design process to help optimize fabrication and assembly later on and keep small issues from becoming big problems downstream.

5 Tim's Takeaways: Clear Communication Takes the Cake ▶


Tim Haag

Whether baking a cake or building a circuit board, it's all about clear communication. If the person writing the recipe had not made the choice to clearly communicate what their intentions were for baking that cake, I would have been lost. A missing ingredient here or an incorrect oven temperature.

6 Connect the Dots: The Future of PCB Manufacturing Doesn't Belong to Robots, but to the Users ▶


Bob Tise

Is the world ready for the consequences of rapid automation? Will the use of robots displace entire categories of workers? Can artificial intelligence really "think"? How will manufacturing, including PCB manufacturing, be affected by all of these smart robots? These questions actually come from a pamphlet published in 1955: "The Age of Automation: Its Effects on Human Welfare."

7 The Pulse: Modelled, Measured, Mindful—Closing the SI Loop ▶

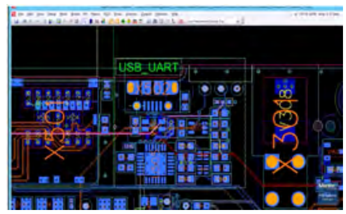

Martyn Gaudion

At speeds up to 2 or 3 GHz, it usually suffices to ensure the transmission line impedance matches the driver and receiver. And a field solver makes light work of the calculation. But push the frequency higher, and other factors come into play.

8 EDA Industry Revenue Up 16.3% in 1Q19 ▶

The electronic system design (ESD) Alliance Market Statistics Service (MSS) announced that the electronic design automation (EDA) industry revenue increased 16.3% for Q1 2019 to $2606.4 million compared to $2241 million in Q1 2018.

9 Mentor Video: Sharing Data Throughout the Lifecycle with EDX ▶



The Xpedition EDM data management suite of tools is designed to provide seamless integration between engineering and external PLM and ERP systems across the enterprise. This integration utilizes the Enterprise Data eXchange (EDX) data format to share data throughout the product lifecycle.

10 Words of Advice: What Feature Would You Like to See in Your CAD Tool? ▶

In a recent survey, we asked the following question: What feature would you like to see in your CAD tool? Here are a few of the answers, edited slightly for clarity.

Career Opportunities

NCAB GROUP
Integrated PCB Production

Key Account Managers/ Outside Sales

Itasca, IL • Hampstead, NH • Ontario, CA

NCAB Group is a global leader in the printed circuit board (PCB) industry, selling over 110 million PCBs yearly from 14 different countries. We are now looking for key account managers for our U.S. teams.

We are looking for sales professionals with at least five years of printed circuit board experience and/or semiconductor experience and knowledge. This is a sales position that requires your ability to convert those cold calls into high-value customer meetings.

What we are looking for:

- A "hunter" mentality
- The ability to create solid customer relationships
- A strong history of working closely with your inside sales teams
- An excellent ability presenting your product and doing the deep dive during your customer visit to identify your customers' pain points
- Knowledge of "SPIN" selling
- A college degree preferred but not required
- Someone who enjoys travel both domestically and globally

Who are we? At NCAB Group, it is the people that make us unique. We work according to our values; quality first, strong relationships and full responsibility, and encourage employee empowerment and initiatives.

apply now

eptac
TRAIN. WORK SMARTER. SUCCEED.

Become a Certified IPC Master Instructor

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

Qualifications and skills

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

Benefits

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

apply now

Career Opportunities

TAIYO AMERICA

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

TAIYO AMERICA

Assistant Department Manager, Operations, Carson City, NV

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

Required Experience/Education:

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

Preferred Skills/Experience:

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

Working Conditions:

- Occasional weekend or overtime work

See complete job listing for more information.

apply now

Career Opportunities

ventec INTERNATIONAL GROUP 騰輝電子

Technical Support/Sales Engineer, U.K.

We are looking to expand our U.K. technical support team. As a technical support/sales engineer (home office/Leamington Spa), you will assist potential customers and current customers in appreciating the benefits of using–and optimizing the use of–Ventec materials in their printed circuit board manufacturing processes, enhance customer loyalty and satisfaction, spread the use of Ventec materials, and grow sales. You will provide a two-way channel of technical communication between Ventec's production facilities and U.K./European customers.

Skills and Abilities Required for the Role

- HNC, HND, degree, or equivalent in a technical/scientific discipline
- Scientific/technical educational background
- Printed circuit board industry experience an advantage
- Good written and verbal communications skills
- Ability to work in an organized proactive and enthusiastic way
- Ability to work well both in a team as well as an individual
- Good user knowledge of common Microsoft Office programs
- Full driving license essential

What's Being Offered

- Excellent salary and benefits commensurate with experience

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

Please forward your resume to humanresource@ventec-europe.com.

apply now

INSULECTRO

Technical Account Manager Chicago/Minneapolis

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

Qualifications:

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

We offer:

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

apply now

Career Opportunities

Analyst Programmer, Hong Kong

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

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

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

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

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

apply now

APCT, Printed Circuit Board Solutions: Opportunities Await

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

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

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

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

apply now

Career Opportunities

KOH YOUNG AMERICA

Field Service Engineer: Multiple U.S. Locations

Reporting to a regional service manager, these customer-focused engineers will uphold the Koh Young culture while delivering professional technical services for our award-winning portfolio of inspection solutions. The role will enthusiastically visit our growing list of customers for installations, training, and evaluations, as well as technical support and maintenance.

We are looking for candidates with a technical degree or equivalent plus three or more years in a production environment with relevant experience. Given our growing customer base, the position will require extensive travel, including some internationally, as well as a collaborative attitude that drives success.

Koh Young is the leading 3D measurement-based inspection equipment and solutions provider. We perform quality control and process optimization across a growing set of industries including PCBA, machining, final assembly, process manufacturing, and semiconductors. In addition to our corporate office in Seoul, our international sales and support offices help us maintain a close relationship with our customers and provide access to a vast network of inspection experts.

Join the industry's leading provider of true 3D inspection solutions. Forward your resume to Michelle.Hayes@KohYoung.com.

apply now

KOH YOUNG AMERICA

Vision and Machine Learning R&D Engineer

Atlanta, GA or San Diego, CA

At Koh Young, we are focused on developing the future and continue to bolster our newly established R&D center near San Diego, California, with top talent focused on vision engineering and machine learning for electronics and medical applications. Currently, we are collaborating with top medical universities and hospitals across the U.S., Korea, and Japan to develop innovative neurosurgical robotic systems. With core technologies developed in-house, we expect to deliver neurosurgical breakthroughs.

The role will develop practical, scalable 3D machine learning solutions to solve complex challenges that detect, recognize, classify, and track medical imagery. Additional focus on the design, implementation, and deployment of full-stack computer vision and machine learning solutions.

The ideal candidates will hold a master's (doctorate preferred) in computer science or electrical engineering with at least three years of relevant experience. We desire a strong understanding of machine learning and computer vision algorithm application within embedded systems, plus significant vision expertise in multi-view geometry, 3D vision, SFM/SAM, and activity recognition.

Koh Young is the leading 3D measurement-based inspection solutions provider. We perform quality control and process optimization across a growing set of industries including electronics, final assembly, semiconductors, and most recently, medical imagery.

Join the 3D inspection leader as we expand. Forward your resume to Michelle.Hayes@KohYoung.com.

apply now

Career Opportunities

TAIYO AMERICA

Technical Sales Engineer

San Jose, CA, USA

The technical sales engineer will perform technical audits and help customers troubleshoot and optimize their solder mask process, prepare and deliver technical presentations explaining products or services to customers and prospective customers, collaborate with sales teams to understand customer requirements and provide sales support, secure and renew orders and arrange delivery, and help in researching and developing new products.

Required Education/Experience:
Applicants must have good "hands-on" knowledge of the printed circuit board (PCB) industry and the liquid photo imageable (LPI) solder mask process. Candidates must be self-motivated, capable of managing key accounts and developing new business opportunities that generate new sales.

- College degree preferred with solid knowledge of chemistry
- 3-5 years of work experience in a technical role within the PCB industry
- 3-5 years of work experience in a sales role
- Computer knowledge, Microsoft Office environment
- Good interpersonal relationship skills
- Good English verbal and written skills are necessary

Working Conditions:
Occasional weekend or overtime work. Travel may be 25-50% or greater.

apply now

INDIUM CORPORATION®

Multiple Positions Available

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

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

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

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

www.indium.com/jobs

apply now

Career Opportunities

THERMAL SYSTEMS

Service Engineer Reflow Soldering Systems (m/f)

To strengthen our service team at Rehm Thermal Systems LLC. in Roswell, Georgia, we are seeking candidates to fill the position of Service Engineer—Reflow Soldering Systems.

Your area of responsibility:

- Installation of Rehm reflow soldering systems at the customers' site
- Maintenance and repair work as well as technical service for our customers in the USA and Mexico
- Execution of machine training

Your profile:

- Completed education studies as an engineer in the field of electrical engineering/mechatronics or comparable education (m/f)
- Basic and specialist knowledge in the field of electronics and electrical engineering/mechatronics
- High willingness to travel and have flexible employment
- Service-oriented and like to work independently

We offer:

- Performance-oriented, attractive compensation
- Comprehensive training
- A safe workplace in one successful group of companies
- Self-responsibility and leeway

Please send application documents online to Natalie Werner at n.werner@rehm-group.com.

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

Sales Personnel, Japan

The Gardien Group is looking to expand the sales team in Tokyo, Japan, and seeking highly motivated team players with a positive attitude. Prior experience in the PCB industry is an advantage but not necessary for the right candidate.

The role involves working closely with the customer to identify their needs and deliver the right solution. The candidate should be able to offer a high level of customer satisfaction to ensure ongoing sales.

Training will be provided along with a competitive benefits package, excellent growth opportunities, and periodic bonuses.

Interested candidates, please contact us at careers.jp@gardien.com with your resume.

Kindly note only shortlisted candidates will be notified.

apply now

Sales Representatives (Specific Territories)

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

- Florida
- Denver
- Washington
- Los Angeles

Experience:

- Candidates must have previous PCB sales experience.

Compensation:

- 7% commission

Contact Mike Fariba for more information.

mfariba@uscircuit.com

apply now

Career Opportunities

ELECTROLUBE
THE SOLUTIONS PEOPLE

We Are Recruiting!

A fantastic opportunity has arisen within Electrolube, a progressive global electro-chemicals manufacturer. This prestigious new role is for a sales development manager with a strong technical sales background (electro-chemicals industry desirable) and great commercial awareness. The key focus of this role is to increase profitable sales of the Electrolube brand within the Midwest area of the United States; this is to be achieved via a strategic program of major account development and progression of new accounts/projects. Monitoring of competitor activity and recognition of new opportunities are also integral to this challenging role. Full product training to be provided.

The successful candidate will benefit from a generous package and report directly to the U.S. general manager.

Applicants should apply with their CV to
melanie.latham@hkw.co.uk
(agencies welcome)

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

Events Calendar

NEPCON South China 2019 ▶
August 28–30, 2019
Shenzhen, China

PCB West 2019 ▶
September 9–11, 2019
Santa Clara, California, USA

World Maker Faire New York ▶
September 21–22, 2019
Corona, Queens, New York, USA

SMTA International 2019 ▶
September 22–26, 2019
Rosemont, Illinois, USA

productronica and electronica India 2019 ▶
September 25–27, 2019
Delhi NCR, India

52nd International Symposium on Microelectronics ▶
September 29–October 3, 2019
Boston, Massachusetts, USA

Altium Live—San Diego ▶
October 9–11, 2019
San Diego, California, USA

Altium Live—Frankfurt ▶
October 8–11, 2019
Frankfurt, Germany

productronica 2019 ▶
November 12–15, 2019
Munich, Germany

PCB Carolina ▶
November 13, 2019
Raleigh, North Carolina, USA

Additional Event Calendars

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

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

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

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

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

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

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

ASSOCIATE EDITOR: **KIERSTEN ROHDE**
kiersten@iconnect007.com

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

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

TECHNOLOGY SPECIALIST: **BRYSON MATTIES**

COVER: **SHELLY STEIN**

COVER IMAGE: **ADOBE STOCK © OLLY**

DESIGN007 MAGAZINE

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

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

ADVERTISER INDEX

Accurate Circuit Engineering........ 45
All Flex........ 107
Altium........ 49
American Standard Circuits........ 7
APCT........ 11
Candor Industries........ 2
Cicor........ 99
CML Eurasia........ 67
Downstream Technologies........ 19, 85
Eagle Electronics........ 59
Elite Materials Co, Ltd........ 39
EMA Design Automation........ 25
Freedom CAD Services........ 63
I-Connect007 eBooks........ 23, 61
Insulectro........ 103
In-Circuit Design Pty Ltd........ 125
Lenthor Engineering........ 111
Mentor, a Siemens Business........ 5
Miraco........ 89
NCAB........ 29, 33, 35
Printed Circuits Inc........ 93
Prototron Circuits........ 75
Pulsonix........ 16, 17
Rogers Corporation........ 77
SimplifyDA........ 71
Taiyo........ 109
US Circuit........ 55
Ventec International Group........ 15