THE pcb design MAGAZINE

May 2017

an I-Connect007 publication

COMPETING IN THE HIRING GAME

Introducing I-Connect007's new micro eBook series

Written for buyers, specifiers, and engineers

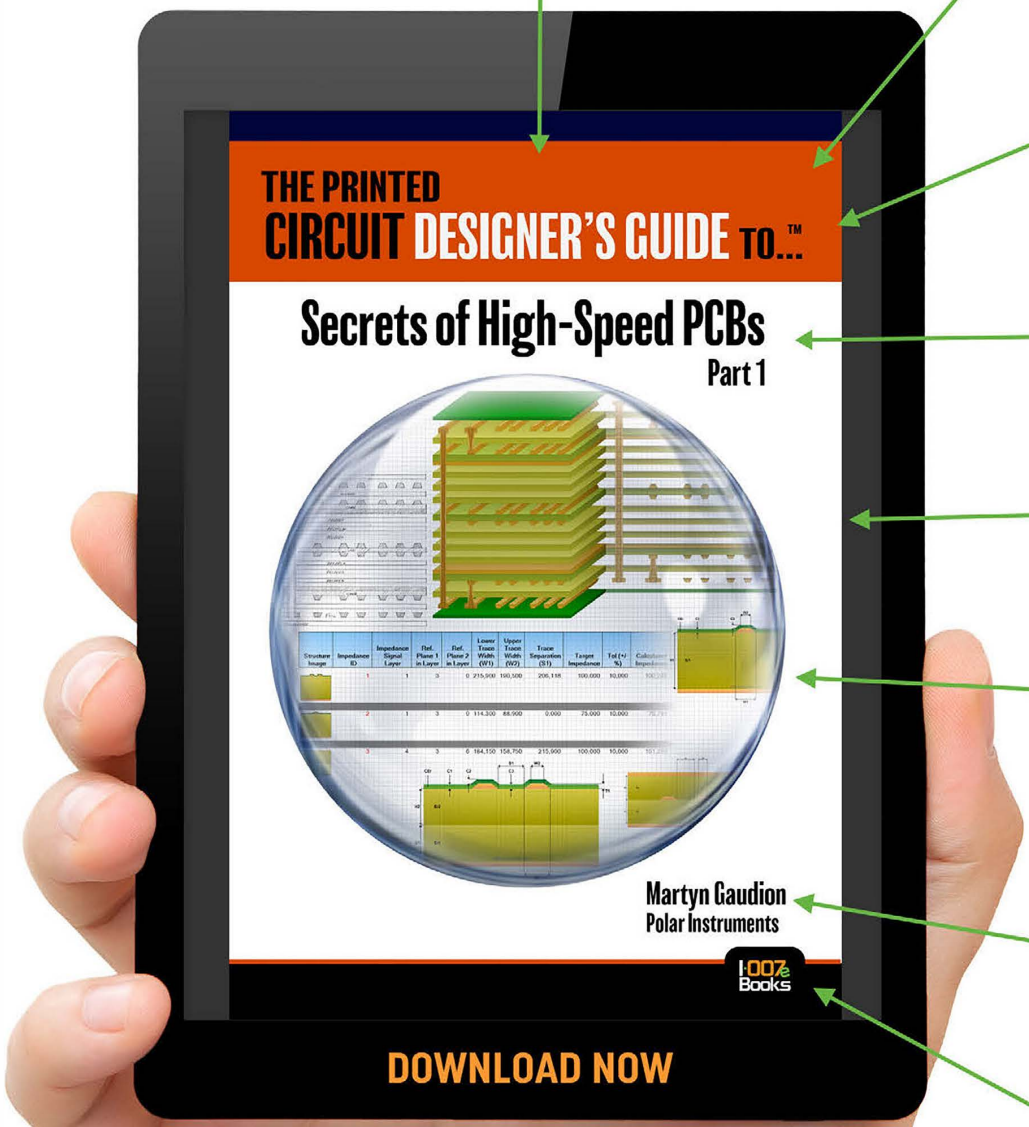
Free download

Highly focused relevent topics

Easy-to-read, 50-60 pages

Great graphics inside and out

Authored by trusted industry experts



I-007e Books
I-007eBooks.com

I-Connect007's expanding eBook library

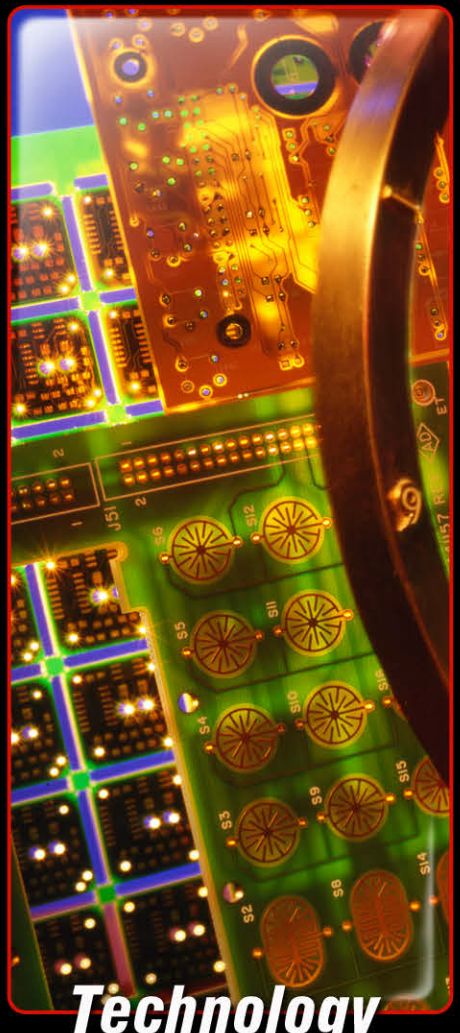
Watch for other exciting titles in our new micro eBook series!

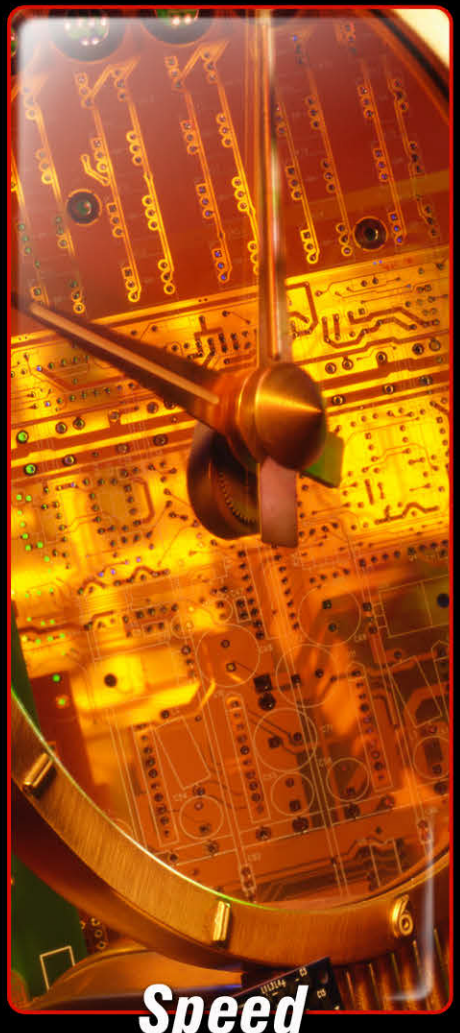
I-Connect007
GOOD FOR THE INDUSTRY

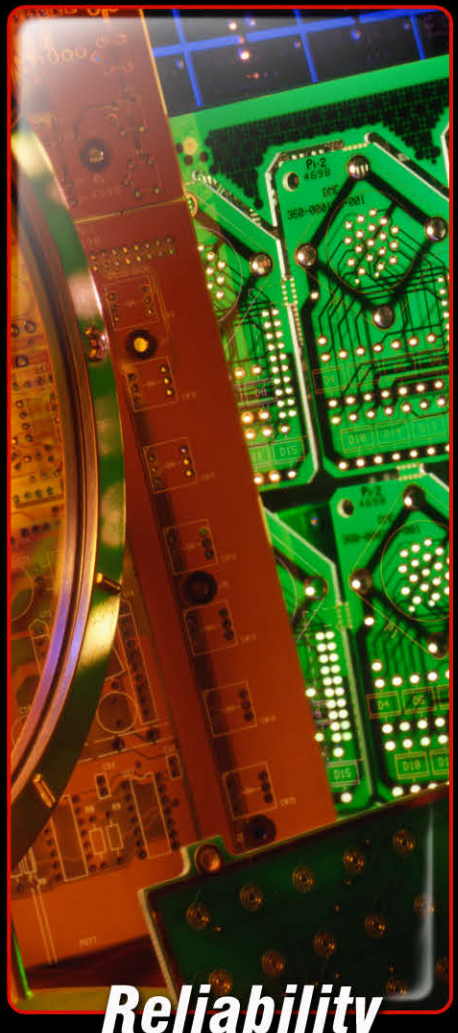
prototron.com

We Take the Time to do it Right

because you don't have the time to do it over.







Prototron Circuits

High quality quick turn manufacturing, just another reason we are

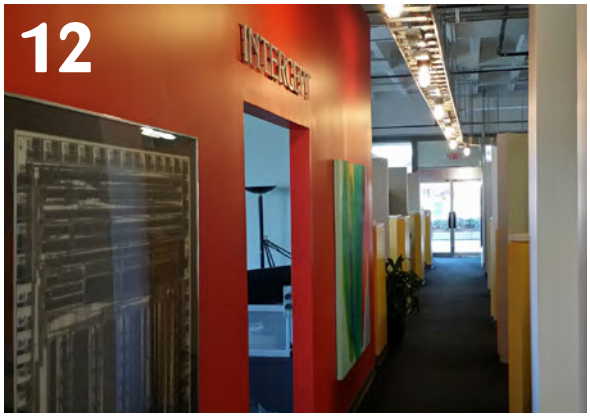
America's Board Shop

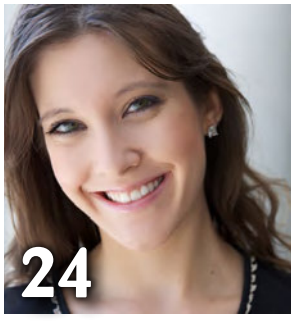
Redmond Facility
15225 NE 95th Street, Redmond, WA 98052
Toll: 888.847.7686 Phone: 425.823.7000

Tucson Facility
3760 E. 43rd Place, Tucson, AZ 85713
Toll: 800.279.5572 Phone: 520.745.8515

THE pcb design MAGAZINE

Feature Content





Help Wanted!

The PCB design segment seems to be doing pretty well right now. But let's face it: Eventually, odds are that you're either going to wind up job-hunting or hiring a new technologist for your firm. In this "Help Wanted" issue, our contributors discuss hiring—from both sides of the desk.

FEATURE COLUMNS

185HR

High-Tg, Low-CTE Materials with High Thermal Reliability

FEATURES

- **High Thermal Performance**
  - Tg: 180°C (DSC) (Base Laminate)
  - Td: 340°C (TGA @ 5% wt loss)
  - Low CTE for reliability
- **T260: 60 minutes**
- **T288: >15 minutes**
- **Lead-free Compatible and RoHS Compliant**
- **UV Blocking and AOI Fluorescence**
  - High throughput and accuracy during PCB fabrication and assembly
- **Superior Processing**
  - Closest to conventional FR-4 processing
- **Industry Approvals**
  - IPC-4101D-WAM1 /98 /99/ 101 /126
  - UL – File Number E41625
  - Qualified to UL's MCIL Program
- **Core Material Standard Availability**
  - Thickness: 0.002″ (0.05 mm) to 0.060″/0.062″ (1.5 mm)
  - Available in full size sheet or panel form
- **Prepreg Standard Availability**
  - Roll or panel form
  - Tooling of prepreg panels available
- **Copper Foil Type Availability**
  - Standard HTE Grade 3
  - RTF (Reverse Treat Foil)
- **Copper Weights**
  - ½, 1 and 2 oz (18, 35 and 70 µm) available
  - Heavier copper available upon request
  - Thinner copper foil available upon request
- **Glass Fabric Availability**
  - Standard E-glass
  - Square weave glass fabric available

KEY APPLICATIONS

- **Heavy copper, 6 & 12 oz automotive applications**
- **Military**
- **Medical**
- **Industrial**

ORDERING INFORMATION:

Contact your local sales representative or visit **www.isola-group.com** for further information.

Isola Group
3100 West Ray Road
Suite 301
Chandler, AZ 85226
Phone: 480-893-6527
info@isola-group.com

Isola Asia Pacific (Hong Kong) Ltd.
Unit 3512 - 3522, 35/F
No. 1 Hung To Road, Kwun Tong,
Kowloon, Hong Kong
Phone: 852-2418-1318
info.hkg@isola-group.com

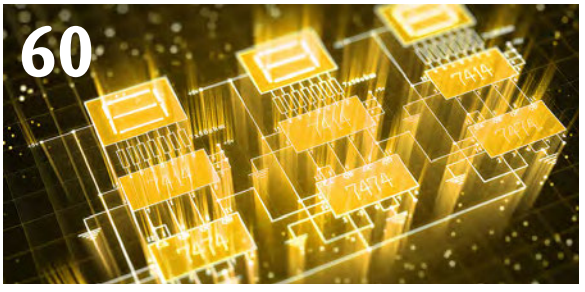
Isola GmbH
Isola Strasse 2
D-52348 Düren, Germany
Phone: 49-2421-8080
info-dur@isola-group.com

THE pcb design MAGAZINE

May 2017

Vol. 6, No. 5

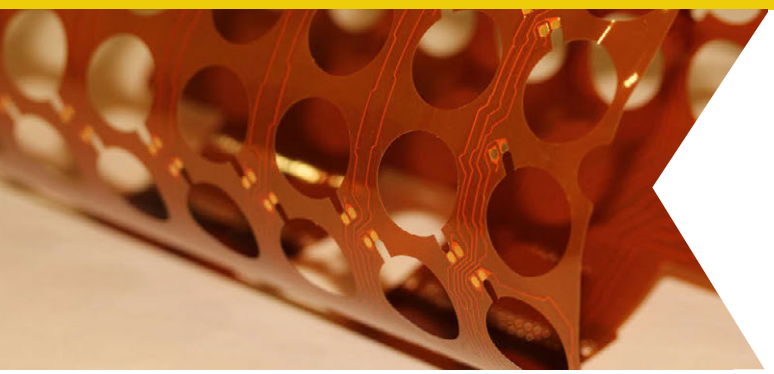
thepcbdesignmagazine.com



More Content

We deliver Advanced PCB Solutions!

Our unique processes enable us to create and deliver Advanced PCB Solutions that meet the needs of the demanding market at competitive prices!



Impedance Tolerance <5%!

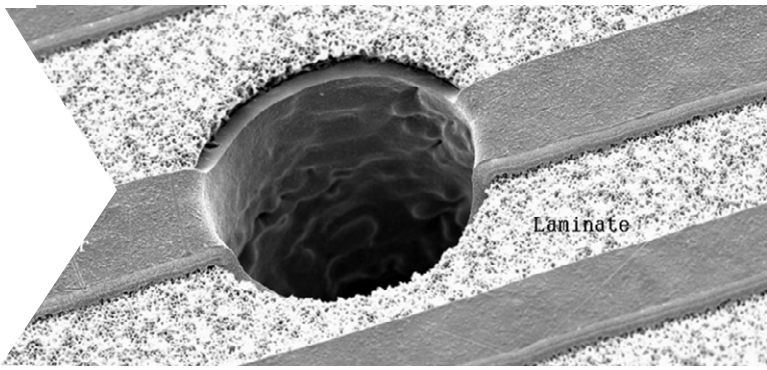
With our panel plating process combined with continuous foil lamination, our impedance tolerances are second to none!

CLICK TO LEARN MORE

Landless Via Technology

We're the only manufacturer in the world to have perfected a liquid photo-imageable resist (PiP) in electrolytic form, used for all our production.

CLICK TO LEARN MORE



Tensile Strength achieved with Candors Panel Plating

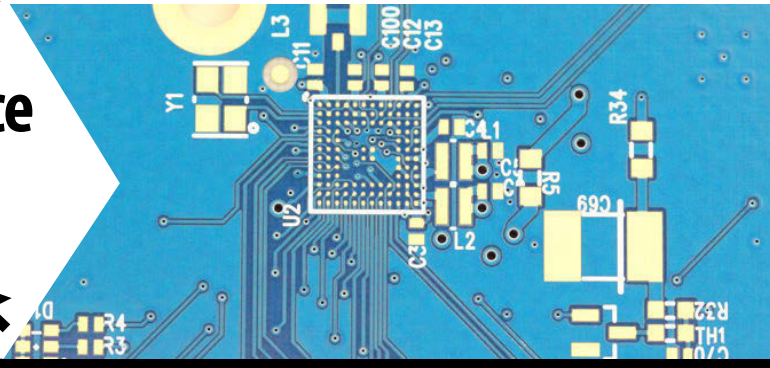
Practically perfect plating exceeds class 3A elongation, purity, and tensile strength

CLICK TO LEARN MORE

Fine line PCB, 2 mil line /space

We have developed a unique set of processes based on electro-depositable liquid photoresist.

CLICK TO LEARN MORE



CANDOR INDUSTRIES INCORPORATED

ADVANCED PCB SOLUTIONS

PHONE: (416) 736-6306

WEB: www.candorind.com

SALES: sales@candorind.com

The "Help Wanted" Issue

by Andy Shaughnessy
I-CONNECT007

SMTA Atlanta is always a treat. This tabletop show in Duluth, Georgia, draws most of the PCB community in Atlanta, and exhibitors from all over the country. These shows are cheap for exhibitors; one good lead more than pays for your exhibit space, and you get to enjoy my neighbors' Southern hospitality along the way.

The show had a pretty decent turnout, despite the extra traffic headaches caused by the "accidental" torching and collapse of part of I-85 last month, which forces motorists onto I-285 and surrounding arteries. (How does a highway burn in the first place?)

The exhibitors were all upbeat, many reporting a better-than-average Q1. One sales manager for a fabricator joked that the company had so many orders that he needed to take a vacation so they could catch up. Of course, he was hitting up potential customers the next day.

Many exhibitors said they were hiring. One capital equipment rep said the firm is constantly searching for new technicians to install and set up their machines. Talk about a small universe: How many people in North America know how to calibrate a pick-and-place machine?

But the highlight of SMTA Atlanta is always the Industry Roundtable. Norcross design bureau owner Albert Gaines once likened the roundtable to a "psychiatrist's couch for PCB designers." This year's roundtable attracted about a dozen designers, electrical engineers

and fabricators from companies like Siemens, NCR and Cisco.

Some interesting points from the very lively conversation:

- Some designers are forced to move up to 16-layer boards in order to fan out BGAs with pin counts of 1,600 and up. As one said, "Designers can drive the cost of the board, unless it's a 1,600 pin-count BGA."
- Every designer present was using FR-4 or a flavor of FR-4. None were designing boards with advanced materials.
- Only one person knew very much about printed electronic circuits. Most wanted to know what the fuss was about, and if anyone was successfully using PEC.
- At one company, the procurement department didn't know that their designers could design .003" lines.
- An "Only in this Industry" story: One Tier 1 CEM convinced its customers that all boards require three re-spins. Why? It's still a prototype if it's still undergoing re-spins, and they charge more for protos. You have to admire thinking like that.
- One great quote: "DFM software isn't perfect. It's still a tool."
- Only one company had plans to hire new designers: When two senior designers retire soon, the company plans to replace them with one PCB designer. Yes, you read that right. At least they won't be farming out their PCB design work.

These SMTA tabletop expos are great regional shows. For all the people who don't go to IPC APEX EXPO and DesignCon, these shows are really a must-attend.

Must Play Well with Others

Which brings us to this month's issue, "Help Wanted." We've heard a lot of rosy talk about how the industry is looking up. Think about it: Do you know any designers who have been out of a job for very long?

So, we asked our expert contributors to discuss what hiring managers are looking for, what skills are hot, and what designers should do if they are forced to freshen up their resumes. Columnist Abby Monaco gives her thoughts on job-seeking in the EDA world, from her point of view as both a hiring manager and a job candidate. Technical Editor Kelly Dack discusses his own job-hunting experience, and how his IPC CID credentials came in quite handy. In an interview, SnapEDA founder Natasha Baker explains how she manages to hire top-notch electrical engineers, and the need to make EDA attractive to such technologists once again. Associate Professor Bojan Jovanovic of the University of Niš, Serbia, discusses his efforts to put together a PCB design curriculum, and what the students should learn in his class. And finally, columnist Tim Haag began this article months ago, not knowing that he would soon be seeking another job in PCB design. He explains the importance of being prepared, as well as sticking to what you're good at and not trying to re-invent yourself to land another job.

We also have articles by Altium Chief Marketing Officer Tim Pawela and Schindler & Schill GmbH CEO Guenther Schindler. And we have columns by our regular contributors Barry Olney of iCD and Alistair Little of Electrolube.

Whether you're hiring or not, we hope you'll enjoy this month's issue. Next month, we delve deep into embedded technology. See you then! **PCBDESIGN**

Andy Shaughnessy is managing editor of *The PCB Design Magazine.* He has been covering PCB design for 18 years. He can be reached by clicking here.

SPECIAL SECTION

HELP WANTED!

COMPETING IN THE HIRING GAME

Go to page 68 to discover the latest job opportunities from:

MacDermid PERFORMANCE SOLUTIONS
TAIYO AMERICA Your Ultimate Solution
eptac TRAIN. WORK SMARTER. SUCCEED.
ZENTECH
LENTHOR ENGINEERING
MivaTek Global
CANDOR
NCAB GROUP Integrated PCB Production
ventec INTERNATIONAL GROUP 騰輝電子
MicroCraft
all flex
SAKI
ARLON

The Hiring Game

by Abby Monaco
INTERCEPT TECHNOLOGY

It's all about who you know. And you'd better know that!

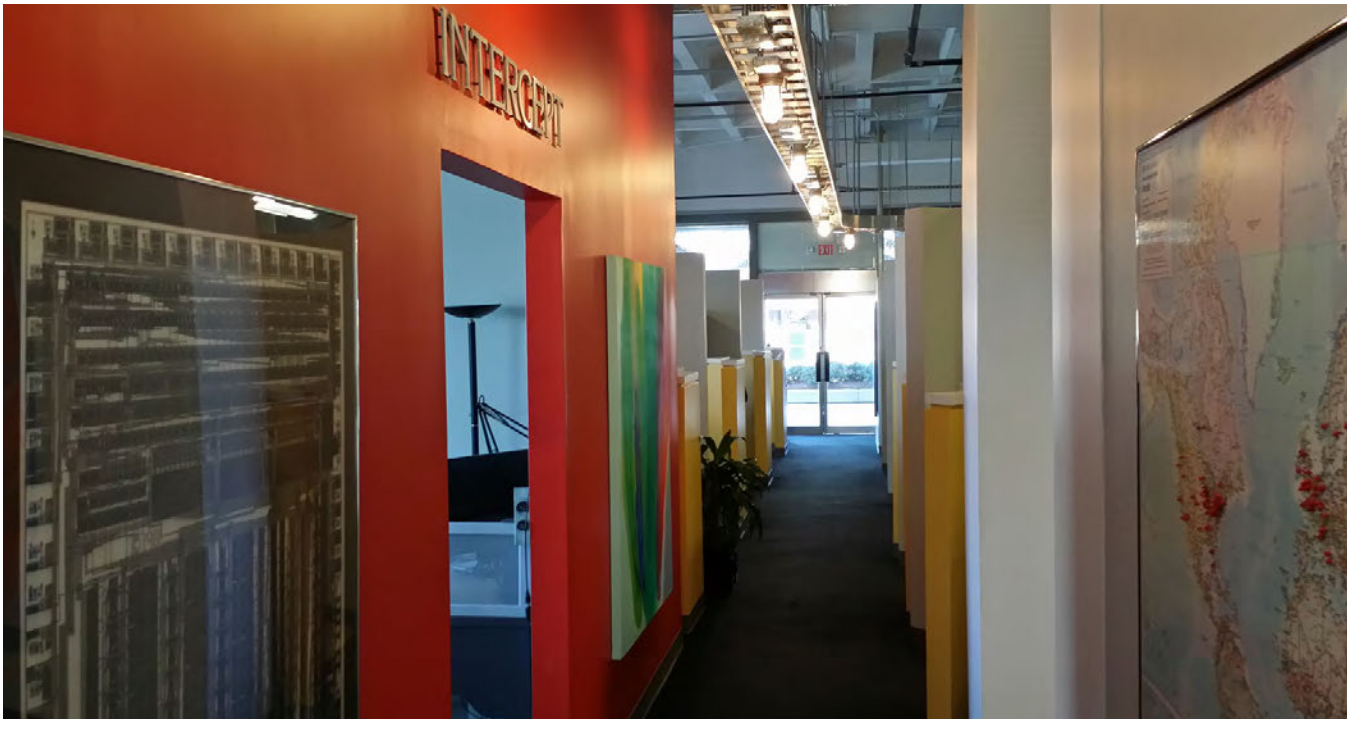
I would be surprised if anyone reading this doesn't know this one major principle of hiring: *Whether you're hiring or looking for a job, nothing happens unless you know the right person.*

In my experience at Intercept Technology, the best way to find new staff is by asking current staff for recommendations. This tends to be a successful approach because someone already on staff doesn't want to make himself look bad, so we can be fairly confident in whoever he recommends. This also helps morale and team-building because a new hire who can be introduced to the team by someone who knows and respects that person creates instant credibility among the entire team.

When this doesn't pan out, the next best thing is to return to the stack of resumes sent in when we were not hiring. So many people in the EDA industry know one another that very often we will have resumes submitted from people we already know, former customers, even people who evaluated our software whose management selected the bigger software vendor just to cover their own backs in the face of the designers' clear choice for our software. So, when one such valuable contact has inquired with us, we keep their information on file. A lot of times these people have moved on into new positions by the time we make contact, but it always makes sense to start with the top picks.

The next option is to contact anyone we know who we would love to have the job. It is a sad fact that if you are already employed, you are more desirable as a candidate because no questions have to be asked about why you are not employed. This is a cold, unfair fact for those of us hard-working folks who have been laid off.

But I have experience hiring people who were "laid off," only to find that they were actu-



ROG breaks FR-4 barrier!

Introducing **Kappa™ 438 Laminates**

Need Better RF Performance? FR-4 No More!

Demand for wireless data is growing exponentially, driving a need for substantially higher levels of mobile network capacity and performance. This demand will grow further in support of the upcoming 5G IoT ecosystem where billions of devices will be communicating with each other, and connectivity is immediate and uninterrupted. FR-4 was historically a material choice for many less demanding RF applications, but changes in the wireless infrastructure related to growing performance requirements, especially in small cells and carrier-grade WiFi/Licensed Assisted Access (LAA), have resulted in instances where the properties of FR-4 are lacking, and RF performance and consistency is compromised. There's no longer a need to sacrifice your PCB performance.

— Now There's Kappa™ 438 Laminates.

Wireless circuit designers can enjoy a true breakthrough with Kappa 438 laminates because they feature the performance of "mid-tier" circuit materials and they shatter the performance limits of FR-4, but at affordable prices. Kappa 438 laminates have low loss, excellent Dk tolerance, and tight thickness control and are engineered for outstanding, repeatable wireless performance.

PROPERTY	TYPICAL VALUE
Dk*	4.38
Df	0.005

*Design Dk; Differential Phase Length Method at 2.5 GHz

By having similar Dk to FR-4, your design transition to Kappa 438 laminates will be effortless! Visit **www.rogerscorp.com/kappa** to learn more.

ROGERS CORPORATION

Advanced Connectivity Solutions

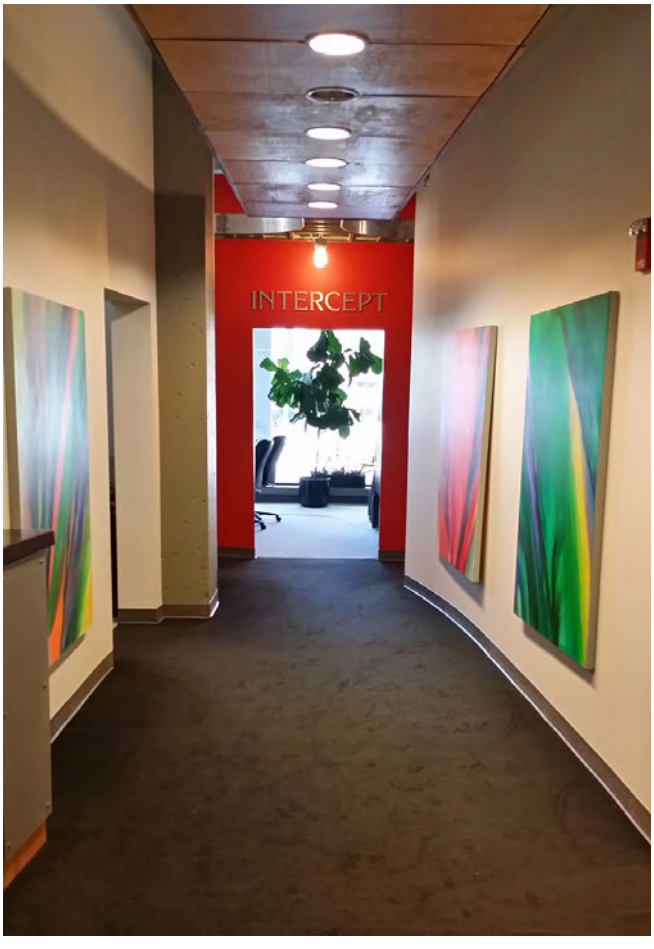
USA - AZ, tel. +1 480-961-1382 • EUROPE - BELGIUM, tel. +32 9 235 3611

www.rogerscorp.com/kappa

ally fired for cause. Many employers try to avoid friction at the end by filling out the papers as though an employee were laid off, which covers up the truth when there is a problem employee. This is essentially kicking the can down the road, causing a very expensive, painful problem for every subsequent employer who is duped into hiring this person only to experience the truth first-hand.

When all else fails, we'll post the job opening on our website. If we are seeking a new software developer, we usually call Georgia Tech University, down the road from us, to see who might like a senior internship that can become a salaried position right out of school, or find a developer who is graduating in the upcoming semester who might like to interview. When it comes to software developers, new talent is always nice to mix into our mid- and senior-level experienced teams.

Over time, we have found that, due to the specialized nature of our industry, posting to job sites isn't helpful. It just creates a tidal wave of resumes, most of which are for candidates who are not relevant, qualified, or even fluent in English. Since Intercept is a smaller company, we don't require a recruitment team, but that also means we don't have a lot of time to wade through useless resumes. For us, it's a good bet that putting something on our careers page will snag someone we'd like to talk to, because anyone looking for a job in EDA will seek us out as opposed to the other way around.



Selecting the Right Person

Whether hiring an account manager or a new software developer, they are always put through our standard software training for both our Mozaix schematic software as well as our Pantheon PCB/hybrid/RF layout software. We do this because they absolutely must have working knowledge of the products that Intercept thrives on. You can't sell something you know nothing about, nor can you write code within an application without knowing how it needs to come out for the person using that application. Even marketing a product that you've never used is an impossible expectation in this market space.

There are plenty of work places where a job is black and white: Make a sales quota, finish coding a project according to the specification, and make hard line release dates. But Intercept expects more. Our staff members need to really understand what they are doing, and have intelligent dialogue with account managers, marketing managers and product managers. We expect collaboration, engaged thought processes, and detailed analysis from all sides.

So, this trial phase of asking a new hire to conduct himself on his own through the training manuals really brings out the personality we will be working with down the road. When a new hire is constantly getting up to ask a question about this or that, we know things are progressing well. Alternatively, when another new hire rips through the manuals without ever getting stuck or asking any questions, that's a red flag. Running advanced electrical engineering and PCB layout software is not simple. Everyone gets stuck at some point, and everyone has at least one question to ask. Everyone.

Getting the Cogs to Fit Together

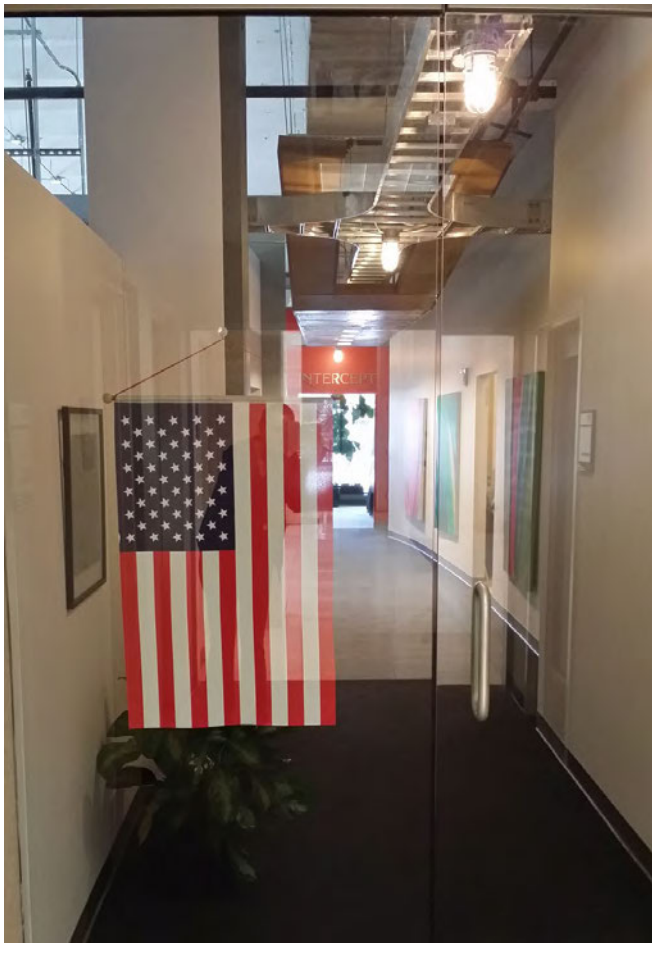
No employer should expect any new hire to be up to top speed right away. My estimation is that after the first three months, it's time to look back and see how far an employee has come. During this time, everyone is getting to know the new person's work habits and personality traits, and there is a bit of time needed to get to a level of familiarity and establish rapport.

Being qualified for a job doesn't necessarily mean a person will fit into the team the way a company needs. For example, I have seen new hires from larger competitors come into Intercept, with everyone in the company feeling very positive. But Intercept is not a large bureaucracy in which people can do very little for large salaries; we expect hard-working, entrepreneurial, self-driven people to join our ranks. We have a small management team that accomplishes big things, meaning that we have no time to micromanage anyone. So there have been times when we were happy to recruit from our larger competitors, only to be disappointed in the lack of output we saw from an individual.

But this isn't to say all is lost. Another very important part of bringing on a team member is to coach them along and give them plenty of opportunities to "get on the bus," so to speak. Getting the new cogs of a team to fit together means giving people opportunity to change course when things might seem to be going poorly. This is accomplished by having a manager sit and chat with that person and offer some friendly insights that might help smooth things out. When I was first employed at Intercept, our engineering manager told me I was taking everything too seriously, and I might be happier if I tried to breathe a little. I didn't realize how uptight I was because I was trying so hard to do my job well.

We like to make sure a new hire is included in lunch outings to help blow off any nervous energy and get to know the rest of us more socially. Often, this leads to questions and answers that ultimately help make that person a better contributor. This also exposes the new person to personalities and team dynamics.

One overriding principle of helping a new hire to be successful is by having carefully developed a good working team of existing employees. Beyond this, by fostering a helpful and open culture among workers, the new person can find ways to fit in without it being stressful.



We spend a large portion of our lives in the office. For employers, it is important to recognize this and to work toward a positive working culture where teamwork comes naturally. But it is also incumbent upon the new hire to also bring positive attitude and willingness to embrace the change in workplace.

Keeping the Good Ones

Our company is built on retaining the good employees. Whatever it takes to keep them happy and motivated, we'll do it. We do our best to offer competitive salaries and excellent benefits, and each employee's motivations are discussed and put into place on projects whenever possible.

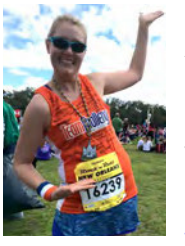
Something that I've always thought of as a nice perk is that we always upgrade the tools

we use, whether it's faster computer hardware or the newest cell phone. Being rewarded for a job well done may not always be in the form of a bonus, but perhaps a new laptop or upgraded processor. While this sounds small, it's actually a big deal given that we are a software company and accustomed to working at a fast pace. It also keeps us using the latest hardware, which we should have working knowledge of since our end users are the ones designing this same kind of hardware.

The majority of employees at Intercept are approaching 20 or more years on the job. We have had some very talented younger people come and go, but the core team has not seen a need to leave. The company gives us all something to believe in, something we feel invested in. We are told that we make a difference, and that we are appreciated. When we demonstrate professionalism and responsibility, we are rewarded with flexibility and trust. It is these small ways that management works that creates such a longevity and loyalty among employees.

I have seen first-hand how difficult it is to have a great team member suddenly put in a resignation notice and leave a gaping hole that the rest of us have to try to fill while looking for a new person. It does happen, and for understandable reasons, but I can say without a doubt that I would never accept a job without my full intention of staying long term.

Remember: Whether you're the interviewer or the one being interviewed, integrity and loyalty are more important than skill level or experience. You can teach a new hire about your software, but we can't teach how to be a good person. **PCBDESIGN**



Abby Monaco is director of Products and Marketing at Intercept Technology Inc. She has been with the company for more than 17 years. She can be reached at abby_monaco@intercept.com.

'Smart Contact Lens Sensor' for Diabetic and Glaucoma Diagnosis

A recent study affiliated with UNIST has proposed the possibility of in situ human health monitoring simply by wearing a contact lens with built-in wireless smart sensors.

This study has been jointly conducted by Professor Jang-Ung Park of Materials Science and Engineering, Professor Chang Young Lee of Life Science, and Professor Franklin Bien of Electrical and Computer Engineering at UNIST in collaboration with Professor Hong Kyun Kim of Ophthalmology and Professor Kwi-Hyun Bae of Internal Medicine at Kyungpook National University.

In the study, the research team unveiled a smart contact lens sensor that could help monitor biomarkers for intraocular pressure (IOP), diabetes mellitus, and other health conditions. The research team expects that this research breakthrough could lead to the development of biosensors capable of detecting and treating various human diseases, and used as a component of next-generation smart contact lens-related electronic devices.

Since blood sugar can be measured with tears, many attempts have been made to monitor diabetes with contact lenses. The biggest drawback with conventional smart contact lenses was thought to be poor wearability. Moreover, because they lens-shaped firm plastic material, many people complain of comfort issues with contact lens wear which made wearing them impossible.

Professor Park and his research team solved these issues by developing a sensor based on transparent and flexible materials. Using this sensor, patients with diabetes and glaucoma may one day be able to self-monitor blood glucose levels and eye pressure. Through the embedded wireless antenna in the contact lens sensor, patients can also transmit their health information, which allows real-time monitoring of their health conditions, as well. In addition, because the system uses wireless antenna to read sensor information, no separate power source is required for the smart contact lens sensors.

HIGH SPEED EXPERTS

Specializing in RF, Hybrid and Microwave PCBs for over 30 years, Accurate Circuit Engineering soars above the rest.

- High-speed impedance tolerances at 5% and 2%
- Prototypes to medium volume production with assembly
- 24 hour, 2, 3 and 4 day lead times available
- PWBs 2 to 50+ layers
- Rigid, Flex and Rigid-flex available
- UL 94V0 Rating up to 8oz copper internal layers

PROUD SPONSOR OF
Geek-a-Palooza
We're *stronger* in numbers!
GEEK-A-PALOOZA.COM

ACE
ACCURATE
CIRCUIT
ENGINEERING

Accurate Circuit Engineering
3019 Kilson Dr
Santa Ana, CA 92707

(714) 546-2162 • sales@ace-pcb.com

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

www.ace-pcb.com

RoHS & WEEE compliant • Certified AS9100, ISO, ITAR, MIL-PRF-31032 & 55110

MUST WORK WELL ON A TEAM; CID A BONUS

by Kelly Dack, CID+
EPTAC

Throughout my decades-long career in PCB design, I have been fortunate. I've only had to search for a job out of desperation once. I had no idea my IPC Certified Interconnect Designer credentials would come in handy when I hit the pavement.

This was during the telecom industry downturn around the year 2000. Suddenly, I found myself laid off for the first and only time in my life.

Prior to being laid off, I'd moved around a bit. I'd worked nine years for an aerospace company; nine years for medical products company; three years for a small product development company; eight months for a PCB design software VAR, and then three years at a telecom company in Southern California. Whew!

During the early years of my career, I'd changed jobs on my own terms, typically by following my engineering peers to new opportunities. My PCB design experience had become suitably diverse. I loved working on design teams and experiencing department camaraderie.

Come performance review time, my evaluations often cited that, "Kelly works well on a team." It was the dotcom era and electronics companies were rolling in cash. I felt on top of the world and I was bringing in more income than ever, until the electronics industry's economic bubble burst. It was unbelievable at the time; maybe you had a similar experience. A lot of designers did.

After a week or two, I realized that I was totally unprepared for a layoff. I had no engineering peers to follow to another local job. The few local companies that were hiring were destined to be under water before long. I had to begin looking for a job on my own. I began reading advice on how to update a resume. "I work well on a team," I wrote. But needless to say, flaunting that you work well on a team is of little value if there is no work. I began searching for jobs out of state.

Soon, I came across a lead for a PCB design job in Reno, Nevada. The job description stated, "Must work well on a team." I thought, "That's me!" I sent off my application.

Before long, my phone rang. It was the human resources manager from the company in Reno. She mentioned that she would like to schedule a phone interview with me. I was hoping she would mention something about the PCB design team in Reno, but she didn't. Great! I responded.

Ensure PCB Manufacturing Success!

CAM350®

Verify and Optimize PCB Designs for Successful Manufacturing.

BluePrint-PCB®

Create comprehensive documentation to drive PCB fabrication, assembly and inspection.

DFMStream™

Verify Design and Manufacturing Rules Any Time During the PCB Design Cycle.

At DownStream Technologies we are committed to delivering comprehensive solutions and workflow process improvements for all your PCB design post processing needs.

Visit downstreamtech.com
or call 800-535-3226

DownStream Technologies

A week or so later, I answered the phone and was introduced to the Reno company's engineering services manager and principle electronics engineer. We talked extensively about my work history and the types of printed boards I had designed, but they seemed particularly interested in a certification I'd received the previous year from IPC, the Certified Interconnect Designer (CID).

I mentioned that I'd paid for the CID training and certification on my own because my company at the time was getting ready for layoffs and probably couldn't afford it. I explained that I considered it valuable—not because I thought I'd be job hunting within the next year, but because I'd heard that it taught PCB design principles based on IPC standard design and manufacturing specifications. I added that the certification program had taught me how to look outside my own bubble to see how my design decisions not only affect the performance of the PCB, but how they affect the success or failure of PCB CAM and fabrication personnel, assembly and testing personnel, and even the inspection and purchasing personnel.

As the phone interview concluded, the engineering manager asked if I had any questions regarding the position. I referred to the job description requirement about working well on

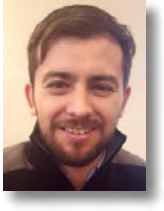
KELLY CHATS WITH HIS CID STUDENTS

While teaching a recent Certified Interconnect Designer (CID) class in Chicago, I got to know my nine students fairly well. They came from all sorts of backgrounds and geographic locations, but they had one thing in common: They wanted to have CID after their names.

Ever the intrepid reporter, I asked if any of them would be interested in talking about their reasons for taking the CID class, and four of them agreed. These are their stories:

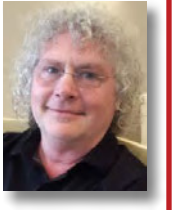
Amy Baker, a PCB designer with Digi International Wireless Design Service in Minnetonka, MN:

"My company supports IPC designer certification because of the credibility it adds to our team. The material is well presented with emphasis on referencing industry approved IPC standards, design methodologies and best practice. The diversity of people attending our class made for interesting class discussion and outstanding network opportunities."



Javier Baca, EE, Yazaki Service, San Nicolas, N.L. Mexico:

"The IPC CID course helped me to think beyond the limited PCB project experience I gained at the university level. The four-day class has helped me better relate to my company's standards and how they compare to IPC standards. I hope to use the knowledge I've gained in this program to increase the quality of our design process back home."



David Moore, PCB design supervisor/manager, Honeywell International, Melville, NY:

"I'm gaining my IPC PCB designer certification to better understand IPC design, documentation and manufacturing standards. I will return to my company with enough foundational knowledge to build upon our PCB design processes and help our department to be accountable for its success."

Julito Tejada, principle engineer, Honeywell International, Melville, NY:

"The benefit of this course is that it gives the attendees command of the PCB design and manufacturing functions by helping us to understand how to move forward using standardization. The workbooks and presentations help us to gain a useful depth of knowledge."

ME AND GERBER THE LIZARD 1986

We love PCBs.
We always have.

We are the geeks. The nerds. The passionate. We all think, talk and dream about that little green masterpiece. We call it **Your Key Component**. Because that's what it is. And our mission is to **produce PCBs for demanding customers, on time, with zero defects, and at the lowest total cost.**

We are the NCAB Group. One of the world's leading producers of PCBs. We have been producing PCBs, and only PCBs since 1993.

Today our U.S. locations are in:
California Florida Illinois New Hampshire Texas Washington

www.ncabgroup.com

NCAB GROUP
Integrated PCB Production

a team. I asked, "What is the PCB design team like?" The principle engineer responded, "Well, it's not really a team. It's just one guy, and he's at the movies with the department."

In the background, I could hear whispering. The engineering services manager said to the HR gal, "Did you tell him we have a team?" And the HR gal replied, "That's a standard qualification on our job template. It's on all our jobs."

So this company in Reno wasn't really looking for a PCB designer who works well on a team. They didn't have a PCB design team—just another independent PCB designer. However, they really liked the insights I'd gained, not only from my diverse PCB employment history, but what I'd learned about considering all the spokes in the manufacturing wheel while gaining my IPC CID certification.

They called me yet again to invite me up to Reno for an in-person interview. I ended up accepting their job offer and developing some really good friendship and teamwork skills with the designer who chose to attend the company movie, rather than be bothered by joining the engineering staff for my initial phone interview.

And during my fourteen-year tenure at that company, a wise engineering manager helped me to realize something about the concept of a team that the IPC CID training and certification had tried to teach me from the start: I had never really worked well on a team. I'd only worked well with a group of like-minded PCB engineers watching out for our own interests.

The team in Reno came together as I was given more opportunities to practice reaching out to all of the other personnel in the PCB manufacturing processes. The team was composed of personnel from all of the various PCB manufacturing disciplines that I'd learned about while gaining my CID.

Now, as a CID instructor for EPTAC, I'm able to give back to the PCB design community by passing on my knowledge. The CID is much more than a piece of paper, or proof that you know how to design a PCB that's manufacturable anywhere in the world.

Bottom line: The CID might just help you get a job. And isn't that what it's all about? **PCBDESIGN**

Kelly Dack is currently a full-time PCB designer for KeyTronicEMS Corp., a dynamic assembly contract manufacturer with facilities worldwide. Additionally, Kelly serves on the executive staff of the IPC Designer Council and is employed by EPTAC Corporation as a CID instructor teaching classes nationwide.

A Touchable Tablet to Guide the Visually Impaired

Navigating in an unfamiliar setting is a major challenge for people with a visual impairment. To make it easier for them to find their way, EPFL researchers have developed a lightweight and reconfigurable touchscreen tablet capable of generating shapes and maps. Users can then "read" the graphic data on the screen with their fingers. The research is part of the European project Blindpad.

Measuring 12 by 15 centimeters, the tablet comprises 192 tiny buttons that can move up and down in just a few milliseconds, almost instantaneously creating patterns such as the layout of a building, street or conference room. Users can also zoom in on a specific part of the map. The actuators are fast enough to make individual buttons vibrate.

The underlying mechanism is straightforward. Each button contains a tiny magnet placed between two coils and two thin layers of steel. Any given button can be moved up or down by generating a local magnetic field by driving current through one of the coils for five milliseconds. The magnetized buttons then remain in the up or down position because they attach to one the two steel plates. "The system requires no power to keep the button in place," said Herbert Shea, director of EPFL's Microsystems for Space Technologies Laboratory.

New ON IPC EDGE

IPC Essentials & IPC ESD Certification

IPC ESSENTIALS

IPC EDGE

- Simple to use online portal
- Convenient for you, open 24/7
- Keeps track of your training transcripts

IPC ESD Certification for Trainers (CET), and IPC ESD Certification for Operators (ECO)

ESD certification control for your company

- Fills a gap in the industry, developed by IPC and EOS/ESD Association Inc.
- Operators are trained by certified ESD trainers
- Certification complies with ANSI/ESD S20.20
- Verifiable and trackable certification record

IPC Essentials

IPC Essentials delivers

- Enhanced knowledge of IPC Standards and their applications
- Modules covered — standards development, impact on the electronics industry, standards training
- Increases knowledge and improves productivity for line operators, managers, executives and all those impacted by IPC standards
- Improvement to the certification programs
- A program requirement for CIT and MIT professionals

Visit www.ipc.org/edge for more details.

IPC EDGE Online Education and Training for the Electronics Industry

IPC EDGE

SnapEDA:
Recruiting Top Engineering Talent in an Amazon World

by Andy Shaughnessy
I-CONNECT007

You don't have to love EDA to work at SnapEDA, but it helps. This startup, founded by Natasha Baker, is on its way to creating the world's largest parts library for PCB designers. Baker leads a small team of young, fiercely talented engineers—the kind of employees that are attractive to companies like Google and Facebook. I asked Natasha to explain her hiring process, and how she ensures that each employee is the right fit for SnapEDA.

***Andy Shaughnessy:** Natasha, tell us a little about SnapEDA and the kind of people you employ.*

Natasha Baker: We're a team of nine, and we're all electrical and computer engineers, except for our graphic designer. Together, we've built the Internet's first library for circuit board design. What inspires us as a team is hearing about the thousands of PCB designs made with our platform each week. Whether it's satellites or drones, getting to assist in the design of interesting new devices is extremely rewarding.

We look for three main things when hiring: First, we look for engineers with strong software and PCB design skills. Software is important because we write software on a daily basis. But to make a great product, we want that software to be guided by a deep understanding of the PCB design flow. Second, we look for passion. As EEs, we want to make life easier for our colleagues. We are passionate about making tools that streamline the PCB design process. Finally, we look for people who are detail-oriented. Our software is simple and easy-to-use, but it's quite sophisticated behind the scenes. We generate critical manufacturing data, like footprints, and develop proprietary software to verify their manufacturability. So, it's crucial that our team members value attention to detail where it matters most—the design data.

***Shaughnessy:** What are some of the challenges you face in hiring skilled, educated workers?*

Baker: One of the biggest challenges is that there seems to be a stigma around the EDA industry. New grads are drawn to companies like

APCT

Passion | Commitment | Trust

Let Us Spell It Out For You

A Leading Manufacturer of High Reliability, Rigid Printed Circuit Boards

Let us spell it out for you. Time is money and the first one to market wins! It's no secret that APCT offers world class cycle times. Not only is our standard lead time consistently faster than the industry norm; our ability to manufacture product in days, rather than weeks is unprecedented. If a design falls behind schedule; or a supplier continually fails to keep their promise; or you simply want to beat your competitor to market, APCT is set up to execute and deliver. They're just not words - *Passion*, *Commitment* & *Trust* are the foundation of our reputation in the industry.

Who are these guys? They're APCT, "where our people differentiate us from the competition."

APCT IS PROUDLY ABLE TO DELIVER

LEAD TIMES World Class Cycle Times

2 - 10 Layers - 24 hours
12 - 24 Layers - 48 hours

We can support Via in Pad
48-72 hours

We can support HDI
3-5 day turns

LAYER COUNT

Standard:	2 - 28	Layers
Advanced:	30 - 38	Layers
Development:	40+	Layers

QUANTITIES From Prototypes to Mid-Production; with Off-Shore solutions through APCT Global

PRICING No tooling charges; No test charges

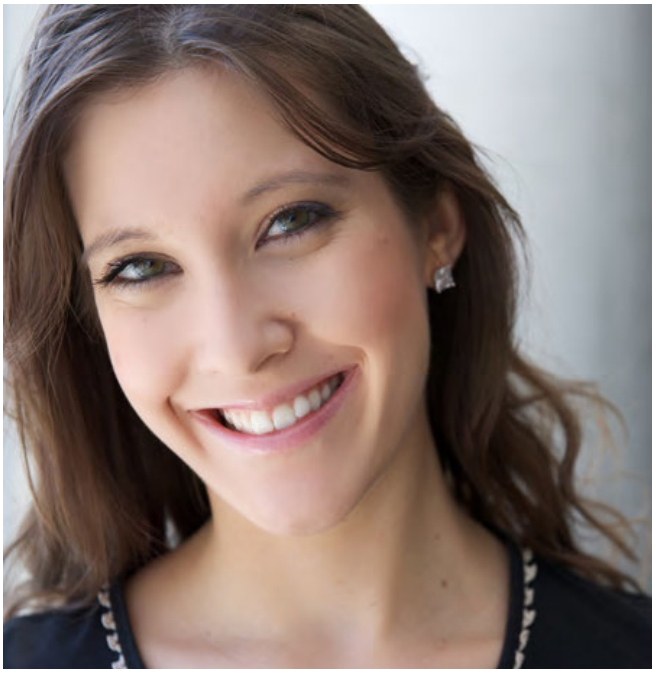
CERTIFICATIONS ISO 9001 Rev 2008 Certified; ITAR Registered; IPC 6012 Class III

www.APCTinc.com

APCT Santa Clara, CA
408.727.6442

APCT Global
203.284.1215

APCT Wallingford, CT
203.269.3311



Natasha Baker

Google, Facebook, Apple, and Amazon, and it seems that EDA has lost some of its luster.

During the recruiting process, I've reached out to several software developers who previously worked in the EDA industry who told me, "Sorry, I've moved on from EDA." I think there's an opportunity to re-invigorate EDA, because it truly is the backbone of the tech industry, which makes all the products that we love possible, from iPhones to Teslas.

Most of our applicants usually have no idea what the EDA acronym stands for before applying. They apply because they know how difficult it has been for them to find the data they needed to make PCBs, so they understand innately how much room there is to innovate. This is a good sign, because it shows with a bit of rebranding, we can as an industry make EDA a magnet for great talent again.

***Shaughnessy:** How do you typically locate potential employees?*

Baker: We're a startup, so it's important that we recruit team members who are a good fit for a smaller team. AngelList has been one of the best sources for this. For those who are unfamiliar with it, AngelList is a website that connects startups, job seekers, and investors. Some of our investors are also from the PCB and EDA industries, so they've made introductions as well. Finally, we post jobs on our website, which gets millions of page views. So that has been an excellent source of applicants as well.

***Shaughnessy:** We've seen company owners offering cars, signing bonuses, and all sorts of things to bring in new staff. Do you find yourself having to "sweeten the pot" to land a really skilled employee?*

Baker: It's a bit too early for us to compete on perks, but we aim to be competitive compensation-wise. We bring people together who believe in our mission to help the PCB design community, and the future growth prospects of the company. They understand that they might not be getting Google-style free lunches today, but that's OK, because they have extremely rewarding work and strong ownership.

***Shaughnessy:** Do you have a training program for new hires?*

Baker: Yes, but it's not at clear-cut as it might be at a large company. While our component engineers have a clearly defined training program in terms of our PCB library standards, and verification processes, our software engineers have much less structure. And that's because our userbase is 50,000 times bigger than our software development team. I am in awe of what they've been able to achieve, but it requires them to wear many hats. One day they're adding new PCB export formats, and the next they're delving into DevOps. Although some of these processes are documented, we're still in startup mode where they're constantly being defined and refined. As we grow our team, I expect we will segment the roles into more specific functions, and as we do that, we'll be able to set clearly defined training plans.

***Shaughnessy:** How do you measure new employees' performance?*

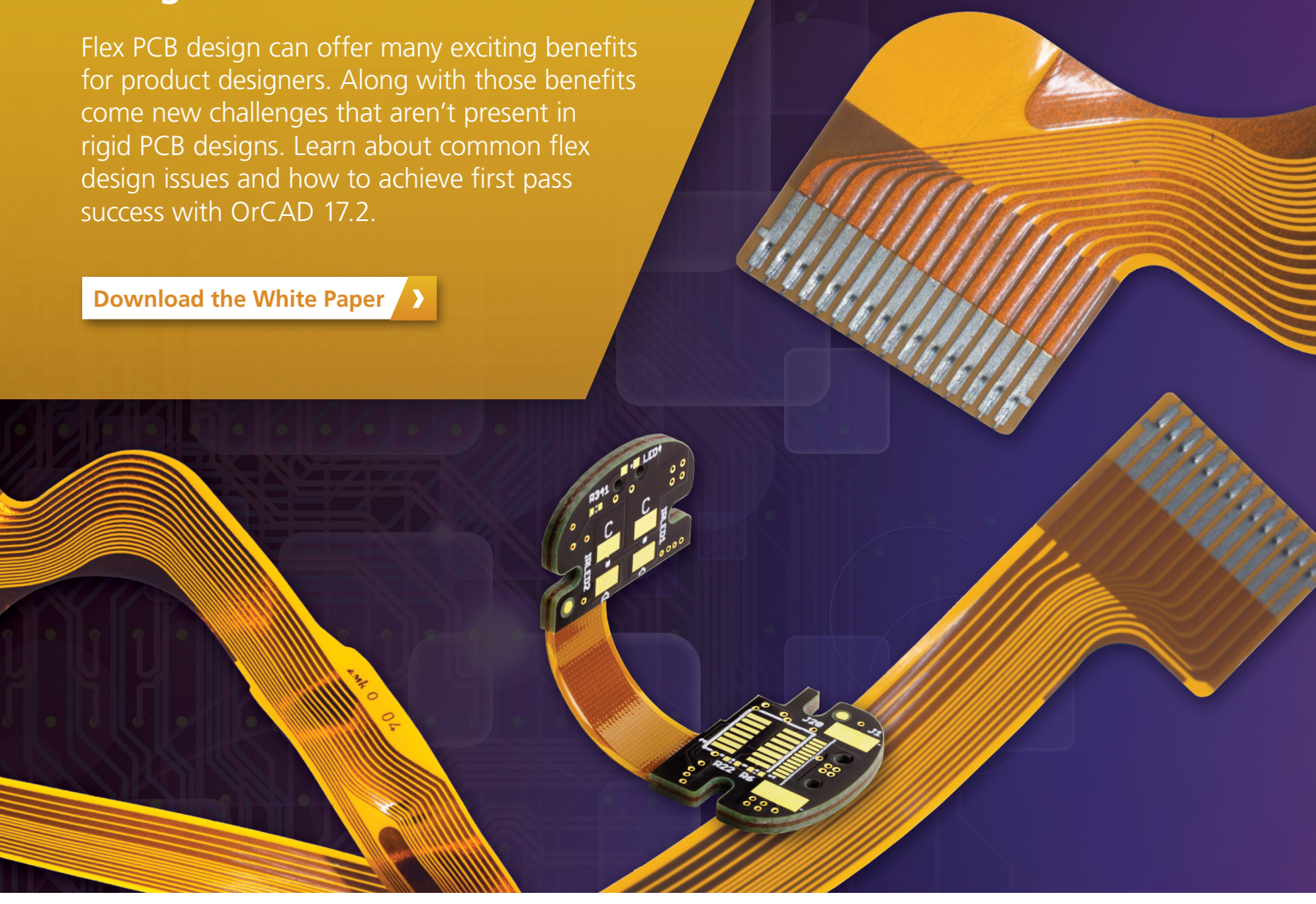
Baker: We have a weekly meeting where we review our metrics. Everyone has ownership of a

Free Flex Design White Paper

Design Flex PCBs with a Tool Designed to Create Flex PCBs

Flex PCB design can offer many exciting benefits for product designers. Along with those benefits come new challenges that aren't present in rigid PCB designs. Learn about common flex design issues and how to achieve first pass success with OrCAD 17.2.

Download the White Paper ›



OrCAD™
CADENCE PCB SOLUTIONS

EMA Design Automation®

metric—including our interns—and we all share how we're seeing those metrics grow. This allows us to monitor performance, but more importantly, work together as a team to help each other perform better.

***Shaughnessy:** One aspect of hiring that companies seem to have trouble with is finding people who fit the culture of that particular company. Do you have any insight on this?*

Baker: Culture is important, and it's something that has been challenging. Do we hire the engineer with decades of experience from large companies, or a recent grad without much experience, but lots of raw intelligence and passion? We realized that our ideal team members are somewhere in the middle, and can almost adapt to the mindsets of both these archetypes depending on the context. Also, as I mentioned earlier, a drive to genuinely be passionate about streamlining the PCB design process is 100% crucial.

***Shaughnessy:** Do you have any advice for other companies in EDA seeking to hire qualified employees?*

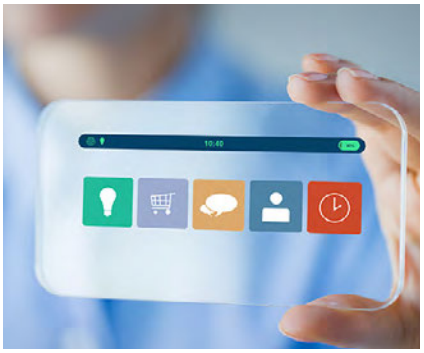
Baker: The more candidates can see how fundamental EDA is to all the amazing products that we love, and how enjoyable and rewarding the technical challenges are, the easier it will be to recruit. If we, as an industry, are going to compete with Google and Amazon, then we need to really show why EDA is great.

***Shaughnessy:** Thanks for your time, Natasha.*

Baker: Thank you. **PCBDESIGN**

Discovery of Thin Transparent Film Could Improve Electronics and Solar Cell

A team of researchers, led by the University of Minnesota, have discovered a new nano-scale thin film material with the highest-ever conductivity in its class. The new material could lead to smaller, faster, and more powerful electronics, as well as more efficient solar cells. The discovery is being published today in Nature Communications.



"The high conductivity and wide bandgap make this an ideal material for making optically transparent conducting films which could be used in a wide variety of electronic devices, including high-power electronics, electronic displays, touch screens and even solar cells in which light needs to pass through the device," said Bharat Jalan, a University of Minnesota chemical engineering and materials science professor and the lead researcher on the study.

Currently, most of the transparent conductors in our electronics use a chemical element called indium. The price of indium has generally gone up over the last two decades, which has added to the cost of current display technology.

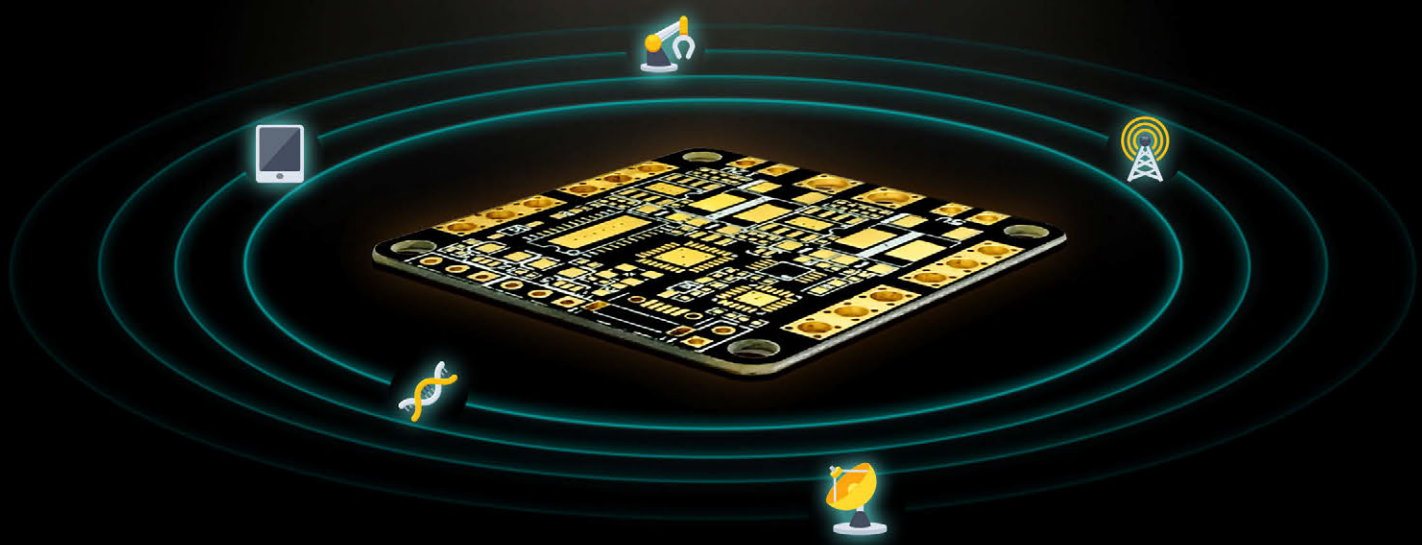
In this study, researchers found a solution. They developed a new transparent conducting thin film using a novel synthesis method, in which they grew a $BaSnO_3$ thin film (a combination of barium, tin and oxygen, called barium stannate), but replaced elemental tin source with a chemical precursor of tin. The chemical precursor of tin has unique, radical properties that enhanced the chemical reactivity and greatly improved the metal oxide formation process. Both barium and tin are significantly cheaper than indium and are abundantly available.

"Even though this material has the highest conductivity within the same materials class, there is much room for improvement in addition, to the outstanding potential for discovering new physics if we decrease the defects. That's our next goal," Jalan said.

Professional PCB Fabrication Service

for Telecommunication, Automation & Test and Measurement Applications

PCBCart has been serving worldwide companies of all sizes with quickturn PCB prototype and mass production services for over 10 years. Each year, we fabricate 23,000+ different PCB designs and we are committed to quality and performance on every circuit board we make.



Our Advantages:

- Easiest online quoting & ordering system
- Best overall PCB quality, customer services & competitive pricing
- Flexible PCB Assembly and Components Sourcing services available
- Advanced techniques including Gold fingers, Blind/Buried hole, Via in pad, etc.
- From FR4 PCBs to Aluminum PCBs, Flex PCBs, Flex-rigid PCBs, Rogers PCBs, etc.
- Free DFM Checking and other custom Electrical Tests (flying probe test, etc.)

ISO 9001 SGCC · LEAD FREE · UL

PCBCART

www.pcbcart.com

sales@pcbcart.com

Launching a New PCB DESIGN CURRICULUM in Serbia

by Associate Professor Bojan Jovanovic
UNIVERSITY OF NIŠ, SERBIA

Let me share with you an experience that I remember from my college days. When I was a student, I had a professor who was too proud of the fact she was an academic.

"You don't need to know how to manually solder electrical parts or how to design printed circuit boards," she lectured. "It is important that you understand the formula for charge carrier currents in a p-n junction."

This was her exciting example:

$$J_p(x \geq x_n) = -qD_p \frac{dp}{dx}$$
$$= -qD_p \frac{p_{n0}}{L_p}(e^{v_D/v_t} - 1)\left[\sinh\frac{(x - x_n)}{L_p} - \coth\frac{w'_n}{L_p}\cosh\frac{(x - x_n)}{L_p}\right]$$

$$J_n(x \leq -x_p) = qD_n \frac{dn}{dx}$$
$$= qD_n \frac{n_{p0}}{L_n}(e^{v_D/v_t} - 1)\left[\sinh\frac{(x + x_p)}{L_n} + \coth\frac{w'_p}{L_n}\cosh\frac{(x + x_p)}{L_n}\right]$$

The allusion is clear. You engineers and researchers should focus on engineering, and let the technicians do the technical jobs! Indeed, from the first semester until the last one, we students did not take a single course covering printed circuit board design, nor did we learn how to manually solder during our laboratory hours.

Right after graduation, I started working as an R&D engineer for a Swiss company that developed and manufactured instruments for measuring magnetic fields and electrical currents. And guess what? Nobody ever asked me about charge carriers in p-n junctions.

Moreover, I was given a heated soldering iron, and from day one, I was asked to design, assemble, test and use many custom printed circuit boards of different shapes and different complexities. We were using the Protel 99 SE CAD tool, the older brother of Altium Designer. You can imagine my frustration and the trembling of my hands while holding a 300°C hot stick in one hand and soldering tweezers with an 0402 resistor in other hand. I felt the same way using my new PCB design tool—I spent months building my designing skills starting from the very basic tutorials (schematic and PCB design of a simple transistor-based multi-vibrator circuit) and finally on to some advanced PCB design techniques.

In the meantime, I moved from industry to academia. I was lucky enough to be part of

ACCELERATING SMART INNOVATION FOR PCB SYSTEMS DESIGN

FIND A LOCATION NEAR YOU

INCREASING DESIGN COMPLEXITY. GROWING GLOBAL MARKET PRESSURES. GREATER DEMANDS FOR MORE DIFFERENTIATED, RELIABLE PRODUCTS. FOR CREATORS OF ELECTRONIC PRODUCTS, IT CAN BE CHALLENGING TO KEEP UP WITH CONSTANT INDUSTRY CHANGES. BUT WHY ADAPT WHEN YOU CAN BE THE DRIVER OF CHANGE?

THIS YEAR, MENTOR GRAPHICS HOSTS A GLOBAL PCB FORUM THAT EXPLORES HOW ORGANIZATIONS CAN ACCELERATE SMART INNOVATION, PUTTING THEMSELVES AHEAD OF THE CURVE IN PCB SYSTEMS DESIGN.

WWW.MENTOR.COM/PCB-FORUM

Mentor®
A Siemens Business

many scientific research projects in many different fields, from FPGA-based digital design to ASIC design of custom hybrid memory cells. Indeed, I found that designing integrated circuits in Cadence tools, for example, was quite similar to PCB design: You have libraries with the components and a stack-up of layers, and vias that you use to properly interconnect them.

During the next few months, I will be promoted to associate professor here at the Faculty of Electronic Engineering, University of Niš, Serbia. I will have the opportunity to organize a few courses in the Embedded Systems module of our Department for Electronics.

My gut feeling, a consequence of the overall engineering experience I gained so far, tells me that today's students—future engineers, deserve to have a course covering many different aspects of PCB design, manufacturing and assembling processes. Let me be clear: I don't at all want to depreciate the importance of an understanding of the deep secrets of electrons and holes inside the silicon. Students certainly need to be quite familiar with transistors, diodes and the rest of the ingredients of any integrated circuit.

Digital electronics, analog electronics, RF electronics, power electronics—everything is important. But let me ask you a couple of questions: What is the final destination of every integrated circuit? What is it worth to master your knowledge of the chips if you are unable to properly interconnect them and benefit from them? Ultimately, a vast majority of graduates will hopefully become engineers. Don't they deserve to have some useful skills in their briefcase while knocking on the doors of industry?

"Digital electronics, analog electronics, RF electronics, power electronics—everything is important."

Truly, the least that faculties can do is to dedicate one of their dozens of courses to the subject of printed circuit board design. Nowadays, in the era of very large scale integration, when electronic devices tend to become smaller yet more powerful, designing printed circuit boards requires more than just technical skills. Placing numerous tiny chips next to each other, dealing with high-speed signals, suppressing the noise, achieving a high-density of interconnections and, primarily, making a printed circuit board that is manufacturable, fully functional and reliable does not seem to be like a trivial job to me.

I have a sincere desire to offer my students some instruction in printed circuit board design. At the moment, I'm in the early stages of establishing a course curriculum. Some PCB design aspects that I intend to cover with the course are as follows:

- Of course, the basics: Students need to master how to create their own libraries of components, schematic symbols and footprints, how to design schematic files, how to appropriately place the components within the available board area and how to properly interconnect them without breaking any design rules. Generating Gerber files, the 2D binary images of all board layers that will be used during the board manufacturing process, is also something that falls under the basic skills category. The same applies for pick-and-place files that contain x,y coordinates of each component on the board as well as its orientation. Those files are necessary during the process of automated PCB assembling. The students should also know how to extract useful information from the component's datasheet, as well as learning about the different through-hole and SMD packages of the chips, etc.

- In addition to the basics, some more advanced and important topics from PCB design world certainly deserve special attention. Students should learn about primary and secondary effects of electrical currents passing through the PCB lines, electromagnetic coupling between the lines, crosstalk, and methods for routing differential pairs of high-speed signals. The notions of blind and buried vias, grounding,

shielding, via stitching, impedance matching, panelizing, and fiducials should also be adopted by the students by the end of the course. Students should be armed with the rules of thumb and guidelines that will help them to achieve high-density interconnect (HDI) and be able to design a board that is both manufacturable and electro-magnetically compatible (EMC). Students should learn to design a board that will be right the first time, easily passing any certification test. Flex and rigid-flex printed circuit boards that ensure flexibility and ergonomics of many electrical devices should not be omitted. Finally, students should learn how to properly document their PCB design project for PCB fabrication and assembling.

Concerning PCB CAD tools, I was thinking of using either Altium Designer or Eagle. Both have been around for decades and are highly adopted by engineering firms worldwide. I myself prefer Altium Designer. It is intuitive to use and has numerous useful options. Nevertheless, there is no free version, and cost is an issue at the university level. But Altium offers special licenses for students and universities, renewable on a yearly basis. On the other hand, CadSoft Eagle, which was recently acquired by Autodesk, has an ideal free educational edition that supports up to six signal layers and 160 x 100 mm routing area. That is why I'm inclined towards Eagle.

I need you, readers of various profiles, to help me in this endeavor. You are invited to contact me and share your experience and suggestions with me. Proposals for the curriculum, useful books, handy CAD tools—literally anything and everything you might suggest is welcome.

It's time to galvanize the students' passion for printed circuit board design! **PCBDESIGN**

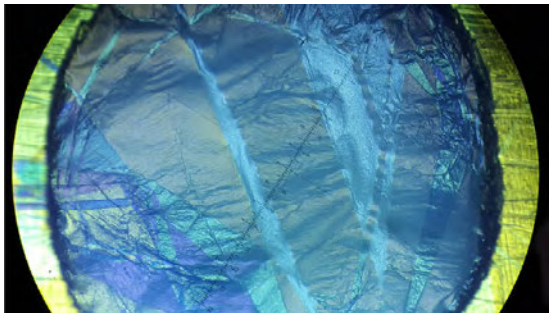
Bojan Jovanovic received his B.S. and Ph.D degrees in electrical engineering from the University of Nis, Serbia in 2006 and 2013, respectively. For any suggestions about his PCB design course, feel free to contact him here. Jovanovic is also available for collaboration on any research or industrial projects.

Semiconductors as Decal Stickers

Scientists at LMU Munich and FSU Jena have developed organic semiconductor nanosheets, which can easily be removed from a growth substrate and placed on other substrates.

Today's computer processors are composed of billions of transistors. These electronic components normally consist of semiconductor material, insulator, substrate, and electrode. A dream of many scientists is to have each of these elements available as transferable sheets, which would allow them to design new electronic devices simply by stacking.

This has now become a reality for the organic semiconductor material pentacene: Dr. Bert Nickel, a physicist at LMU Munich, and Professor Andrey Turchanin (Friedrich Schiller University Jena), together with their teams, have, for the first time, managed to create mechanically stable pentacene nanosheets.



The researchers describe their method in the journal Advanced Materials. They first cover a small silicon wafer with a thin layer of a water-soluble organic film and deposit pentacene molecules upon it until a layer roughly 50 nanometers thick has formed. The next step is crucial: by irradiation with low-energy electrons, the topmost three to four levels of pentacene molecular layers are crosslinked, forming a "skin" that is only about five nanometers thick.

Apart from the ability to transfer them, the new semiconductor nanosheets have other advantages. In addition, after deposition, the nanosheet sticks firmly to the electrical contacts by van der Waals forces, resulting in a low contact resistance of the final electronic devices. Last but not least, organic semiconductor nanosheets can now be deposited onto significantly more technologically relevant substrates than hitherto.

Propelled by Preparation

by Tim Haag
PCB DESIGN CONSULTANT

It's happened to all of us—hitting the pavement in search of a job. Whether as a teenager looking for that first job, as a young man looking for that first career, or as an experienced designer looking for the advancement, I have been there as well.

Granted, our job-hunting methods have changed over the years. When I was a teenager, I was walking or riding my bike to businesses near my home. I remember asking the manager of the local burger joint to give me just two minutes to explain why I would be great to have around. Years later, I traded that bike for a car and spent days driving to different locations to drop off resumes and schedule interviews. These days, it's a matter of networking and filling out online forms just to get your foot in the door. But the anxiety of searching for a job often boils down to the same question: "What can I say and do to become the person that they are looking for?"

I'm sure that if you could add it all up, the amount of time spent by job applicants worldwide anxiously preparing for interviews would be in the billions of hours. Usually the source of the anxiety is based in thinking that we need to become more than who we already are to succeed in getting a job.

Well, I would like to challenge that line of thinking. I believe that instead of focusing on what we think that we need to change into, our time would be better spent highlighting the experiences and strengths that we already have within us. Perhaps if we did a better job exploring who we already are, we might just find that we are more prepared that we would have thought.

Recently my wife and I bought the movie "Sully," and after the movie was over, she went to sleep. I spent the next several hours going through all the extras that came with the disc. I watched several interviews with people re-

STRIVING FOR EXCELLENCE IN EVERYTHING WE SAY AND DO.

- Prototype to Production
- Time Critical Production
- Commitment to Customer Service
- Best Quality & Delivery Performance
- Fair and Competitive Pricing/Costs
- Rapid Response to Quote Requests
- Flexibility Scheduling
- Stock/Consigned Inventory Programs

Review Our Advanced Capabilities Now

EAGLE electronics Inc.

MANUFACTURERS OF QUALITY PRINTED CIRCUIT BOARDS

The Absolute Best Value in High Technology Circuit Boards

847.891.5800 • www.eagle-elec.com

garding the events that happened on January 15, 2009, when Captain Chesley Sullenberger's plane, US Airways Flight 1549, made an unplanned water landing on the Hudson River.

Sully's wife Lorrie and his first officer Jeff Skiles talked about how he spent time making sure that he was always prepared for any eventuality while flying. Lorrie related a story about being a passenger in a small plane that Sully was piloting. She was not happy about being in a puddle-jumper, but Sully calmed her by telling her that during their entire trip, he always had in sight different spots that he had picked out for a forced landing if something had gone wrong.

Skiles talked about how Sully would update the destination flight computer weather report at the 56-minute point of each hour, even though that information wasn't needed until shortly before landing.

> "No one had ever considered the possibility of a dual engine failure at only 3,000 feet caused by simultaneous bird strikes, followed immediately by a forced water landing due to a complete loss of power."

No one had ever considered the possibility of a dual engine failure at only 3,000 feet caused by simultaneous bird strikes, followed immediately by a forced water landing due to a complete loss of power. And there certainly weren't any simulations, training, or procedures for such an event. But because of Captain Sullenberger's habit of rehearsing and thinking through different possible emergency scenarios, he was prepared to calmly deal with this new problem when it occurred. Lorrie said that he doesn't just fly an airplane. He "clothes" himself with the plane so that piloting it isn't just what he does; it's part of who he is.

Do you remember the biblical story of David and Goliath? The giant Goliath had been daring anyone to come out and face him alone in battle, which no one was eager to do. David, who was not a soldier, decided to take up that challenge himself. The soldiers did their best to help him by loaning him their armor, but David declined, saying that he wasn't used to it and it would only slow him down.

David refused to change into something that he wasn't, and told them all that it would be better for him to rely on the skills that he had been perfecting for years as a shepherd tending his father's sheep. When any predator tried to take one of his flock, he would kill it to rescue the sheep. As a result, David had become quite skilled in using the sling to launch a softball-sized stone with deadly accuracy, a point which Goliath would soon find out the hard way.

Sully saved the lives of everyone aboard his airplane because he was prepared. David took on the challenge of battling a giant because he, too, was prepared. Sully didn't wait until the emergency was upon him to suddenly bone up on his emergency procedures; he had been preparing for something like this his whole life. David refused to try to become something that he wasn't in order to succeed, but instead relied on his years of training and experiences to be victorious.

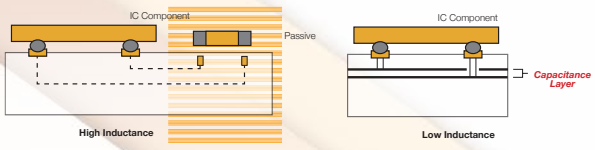
I am not saying that we shouldn't be diligent about learning new skills and adding to our education in order to grow in our respective fields of employment—of course we should. Instead, my point is to encourage you to go into the interview process confident in the skills that you already possess. The things that are important to you now and the skills and abilities that you have been perfecting and achieving success with are the talking points that are going to highlight your abilities during the interview process.

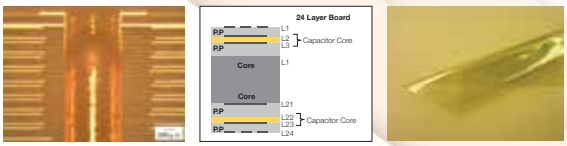
So, stop worrying about who you aren't, and focus more on who you are. Are you a designer with years of experience in different designs? Then don't spend time trying to apologize for areas that you know nothing about, but instead confidently paint a glowing picture of what it is that you have done and how that experience will be a benefit to the new position. Remember

THE SMART SOLUTION TO IMPROVE SYSTEM PERFORMANCE

FARADFLEX ULTRA THIN MATERIALS FOR HIGHER PERFORMANCE PCBS

Faradflex is a high performance, ultra-thin laminate or resin coated copper material designed as an integral part of the PCB, module or chip package.





FaradFlex® Improves System Performance, Price, Quality, and Reliability by:

- Reducing PCB size and/ or thickness
- Utilizing VSP Copper to provide the lowest loss signal for high speed designs
- Increasing functionality
- Improving power distribution (power integrity)
- Reducing the number of SMT Discrete Caps, Traces, Vias, Pads
- Eliminating hundreds (thousands) of SMT and etched and drilled features sites for possible failure on each PCB

For a free sample contact us at:
sales@oakmitsui.com or call us at 503-313-1905

FARADFLEX®

EMBEDDED CAPACITANCE

OAK-MITSUI TECHNOLOGIES
MITSUI KINZOKU CORPORATE GROUP

For more information, visit www.faradflex.com

that in an interview no one else will blow your own horn; you have to do that yourself. And when you are looking for a job, you want your horn to be louder than all the rest that are out there.

Recently I watched comedian Steve Martin in a promotional video for his upcoming online comedy class. In the video, he said, "Remember; you are a thought machine. Everything you see, hear, and experience is usable. Whatever makes you unique as a performer, do it. And know that there's room for you."

That's not bad advice for us non-comedians as well, and it ties right into my point. Stand firm on those experiences and strengths that make you unique as designer, admin, technician, or even as a manager. And while you're at it, stand firm with gusto and confidence. For years, when encouraging someone who is trying to accomplish a difficult task, I've used the expression, "You've got this! We've got our best person on the job!"

I say that because I believe it, and I believe in them, and I want them to believe in themselves. You need to be the person in your life who believes in yourself the most, especially when sitting across from the interviewer for that next big job.

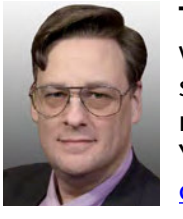
I started writing this column almost three months ago, intending to come back later and polish it up. However, in an ironic turn of events, in this last month I have suddenly found myself in the unexpected position of having to find a new job myself. Talk about life mimicking art!

One of the first hiring managers I spoke with told me that I needed to learn a new skill set to get a job. Immediately, the panic began to set in and I became anxious trying to figure out how I was going to cram many years' worth of education and experience into just a few short weeks. And then it hit me like an Acme frying pan raising a Texas-sized lump on the head of Wile E. Coyote: I was doing EXACTLY what I had recommended in this column not to do. I had become so concerned about my perceived inabilities that I had completely forgotten the wealth of skills and experiences that I already have under my belt.

Boy, did I feel foolish. Since then I have retooled my thinking in my job search to focus on my strengths instead of allowing my fears to distract me.

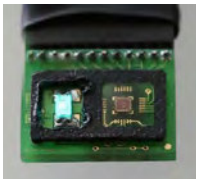
And so if you know of anyone who is looking for a support/training/design manager with years and years of design experience in the PCB design industry, please send me a message. I would love to hear from you.

I was going to end this column by saying the following to all job seekers, but now I am saying it to myself as well: "You've got this!" **PCBDESIGN**



Tim Haag is a senior PCB designer with many years in the industry, supporting and training users, and managing various design groups. You can reach Tim at tim_haag@comcast.net.

A New Sensor Increases Smartwatch Battery Life Five Times



A new generation of sensors developed by ActLight, a startup based in EPFL Innovation Park, can measure the wearer's pulse with the same precision while consuming only one-fifth of the energy. These sensors have been tested by Maher Kayal's laboratory as part of a CTI (Commission for Technology and Innovation) project and are now ready to be used in smartwatches.

"The longer battery life certainly makes things easier for the user, but it also offers major savings in terms of electricity consumption," said Kayal. Thanks to this breakthrough, ActLight was named one of the best Swiss cleantech startups in 2015.

Instead of converting the light into a current and then measuring the current's amplitude, ActLight's dynamic photodiode sensors turn the current into time. The sensors use the pulse of light to identify the moment at which the current is triggered. The result is a small reduction in energy consumption with every heartbeat, but repeated more than 50,000 times per day it adds up to considerable energy savings.

ActLight is in talks with major semiconductor makers and is about to sell the rights to mass produce its innovative sensors.

Cowboy up, Geeks!

Geek-a-Palooza

is coming to Dallas in November.

Details coming soon!

www.geek-a-palooza.com

PCB007 Highlights

Flex Talk: Flex Material Handling—An Inside Peek

As increasingly more designs move to flexible materials to take advantage of space, weight or packaging benefits, it has been clear that flexible circuits require a different set of rules than their rigid counterparts. We spend substantial time working through the design to ensure that flex is as robust as possible.

A Conversation with Gene Weiner

In a discussion following the PCB Executive Forum at IPC APEX EXPO in February, Gene Weiner opened up to Barry Matties and Patty Goldman on the state of the North American electronics industry supply chain and the importance of cooperative efforts up and down that supply chain.

American Standard Circuits Discusses e-Book on Designing Flex and Rigid-Flex

American Standard Circuits is an industry leader when it comes to high-technology printed circuit boards, especially flex and rigid-flex printed circuit boards. So, it was natural that they would write about flex and rigid-flex for I-Connect007's new "Guide to…" e-book series.

PCB Technology Requirements for Millimeter-Wave Interconnect and Antenna

The work done by Optiprint AG in support of MiWaveS substantiates that PCB technology can satisfy the engineering requirements for mmW circuitry providing the manufacturing capabilities can match the positional accuracy, feature tolerance and surface finish requirements.

Elmatica's New Technology Results in an Increase for Flex and Rigid-Flex Printed Circuits

Elmatica has reported an increase in the demand for flex and rigid-flex boards in the Nordics. New advanced technology requires flexible circuits to fulfill challenging form-factor requirements, eliminate connectors and improve performance.

All About Flex: Flex Circuit Stiffeners

Many flexible circuit designs require selectively bonded stiffeners…they're just too flexible! Stiffener materials can be any number of materials, but they are usually polyimide films or FR-4 glass/epoxy substrates and are available in a wide variety of thicknesses.

ACFEST: Benchmarking a New Solderable PCB Finish

New, innovative manufacturing procedures have been developed by the recently completed project, Manufacturing Advanced Coating for Future Electronics SysTems (MACFEST), which has been funded by several partners and the government's Innovate UK.

Printed Circuits LLC Pursues Further Capacity and Technology Upgrades to Start 2017

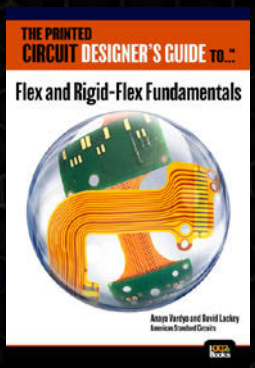
Rigid-flex circuit board manufacturer, Printed Circuits has started 2017 with a few enhancements to their equipment line up, designed to increase capacity, improve technical capability and reduce process time for their customers.

Weiner's World—March 2017

The CPCA show held at the China International PCB And Assembly Show was moderately busy even though the new venue was not quite ready (no escalators, the "water closets" not fully finished, the heat was only on for a few hours one day). It showcased products for PCB Manufacturing, Electronic Assembly Materials and Manufacturing Services.

EIPC Workshop on PCB BioMEMS

The Premier Inn conference centre at Heathrow Airport was the venue for the EIPC workshop on PCB BioMEMS. What, I hear you ask, is a PCB BioMEMS? This is an abbreviation for biomedical (or biological) microelectromechanical systems, otherwise known as lab-on-chip.



Click here to check out our NEW eBook on Flex!

We make Flex and Rigid-Flex easy.

Doing flex boards right can be challenging. We have invested in the right equipment, the right tools and most importantly, the right people, to keep pace with the challenging flex marketplace.

David Lackey, our in-house flex expert, is just one reason why we're the easiest flex supplier to deal with in North America. Let us make flex and rigid-flex easy for you too!

LEARN MORE ▸

American Standard Circuits

Creative Innovations In Flex, Digital & Microwave Circuits

www.asc-i.com | (630) 639-5444

The Dark Side – Return of the Signal

by Barry Olney
IN-CIRCUIT DESIGN PTY LTD / AUSTRALIA

All PCB designers should be aware of the impact of crosstalk on signal integrity. As signal traces come into close proximity, of an aggressor signal, part of that signal is unintentionally electromagnetically coupled into the victim trace as noise. I have mentioned before that current flow is a round trip—the current must return to the source to complete the loop. What about crosstalk in the return path, of the reference planes, as the current weaves its way back through the expansive wasteland of copper? This month's column follows on from my April column, "Return Path Discontinuities," and elaborates on crosstalk in the unseen "dark side" of the signal.

I guess we all think of a copper plane as a thick, solid plate of copper that can basically handle any amount of current we sink into it. It also serves to make the circuit layout easier, allowing the PCB designer to ground anything, anywhere without having to run multiple tracks. That may well be the case with DC or very low-frequency analog circuits, but certainly not in the case of high-speed design. The return current takes the path of least inductance in the nearest plane(s). Returning signal currents tend to stay in close proximity to their signal conductors, falling off in intensity with the square of increasing distance.

However, as the frequency increases, the current is forced into the outer surface of the copper, due to the skin effect, dramatically increasing loss leaving a section of unused copper in the center of the plane. This redistribution

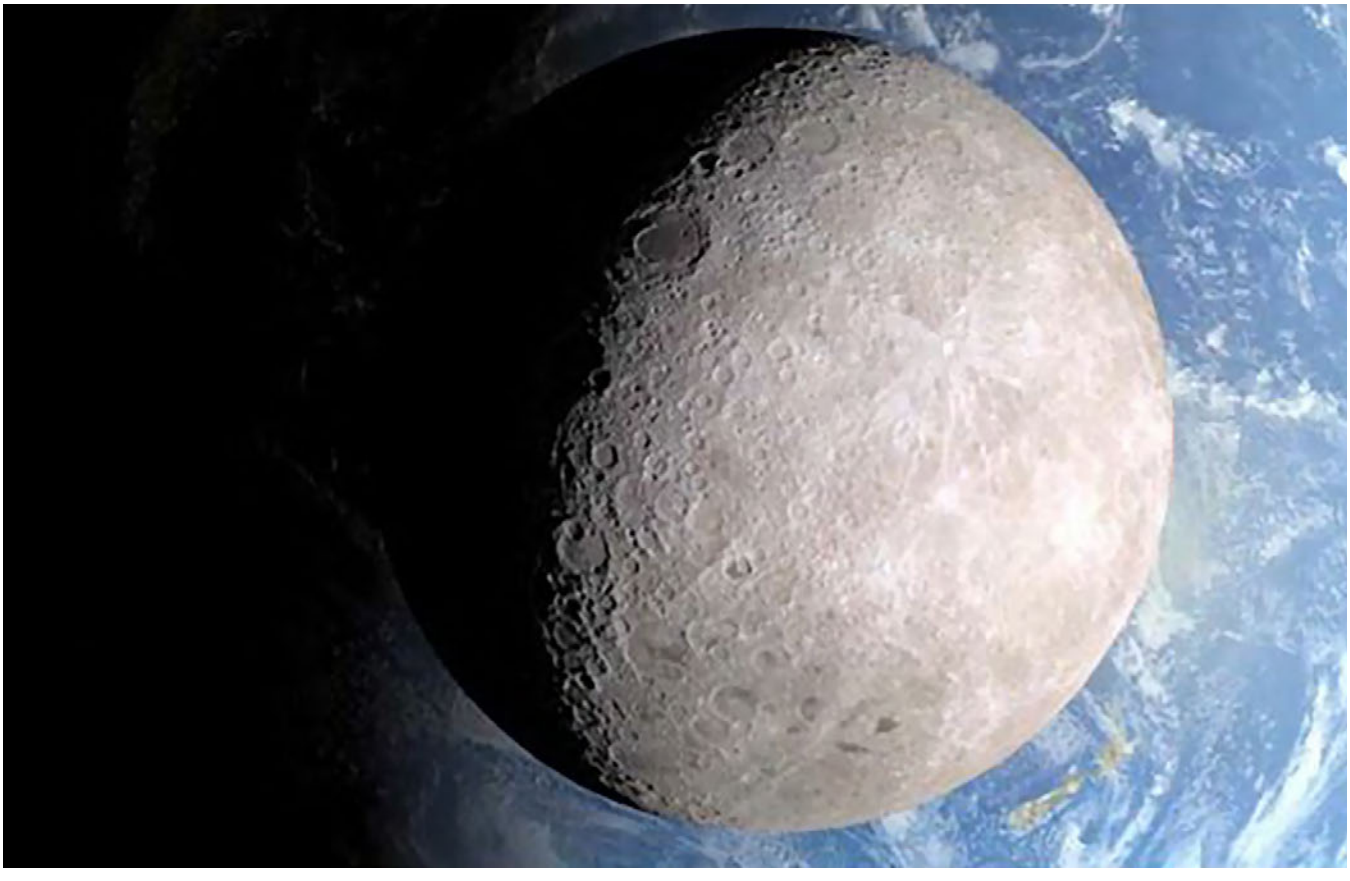
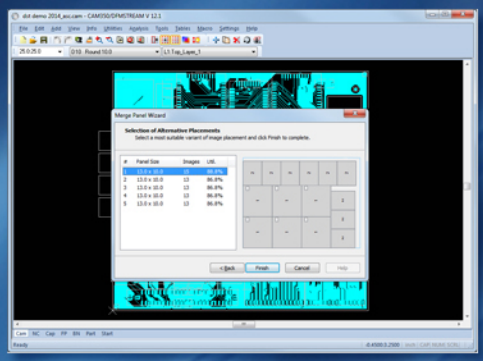


Figure 1: See you on the dark side of the moon (Courtesy of NASA).

Solutions to Ensure PCB Manufacturing Success!

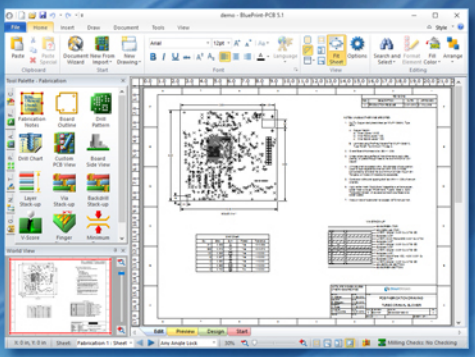
CAM350®

Verify and Optimize PCB Designs for Successful Manufacturing.



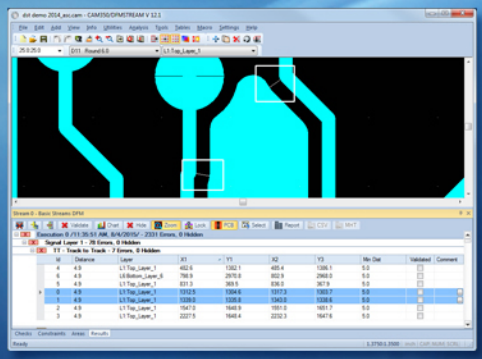
BluePrint·PCB

Create comprehensive documentation to drive PCB fabrication, assembly and inspection.



DFMStream™

Verify Design and Manufacturing Rules Any Time During the PCB Design Cycle.



At DownStream Technologies we are committed to delivering comprehensive solutions and workflow process improvements for all your PCB design post processing needs.

For more information please visit downstreamtech.com or call 800-535-3226.

DownStream Technologies

of current causes the resistance per length to increase and the loop inductance per length to decrease. As frequency increases beyond 1 GHz, the resistance continues to increase while the loop inductance reaches a limiting value. The higher the frequency, the greater the tendency for current to flow in the outer surface of the conductor. The skin depth is given by:

$$\delta = \sqrt{\frac{2}{2\pi f \, \mu \, \sigma}}$$

where δ is the skin depth in microns, f is the frequency in MHz, μ is the magnetic permeability ($4\ \pi \times 10^{-7}$H/m) and σ is the copper conductivity, typically (5.6×10^{7} S/m).

Looking at this equation, it is apparent that skin depth decreases with increased frequency. Figure 2 shows the skin depth compared to frequency. At low frequency (1 MHz), the skin depth is 66 um but this decreases to 0.66 µm at 10 GHz. Above 1 GHz, only the very outer surface of the plane conducts the current. The red horizontal lines represent the plane copper weight and thickness. This shows that at about 30 MHz, a signal traveling in a ½oz (17.78 µm) copper plane would not use the entire plane cross-section but rather the skin effect would begin to have an impact. This implies that at high frequencies, the thickness of the copper plane is irrelevant; ½ oz and 3 oz copper will have the same surface conduction area and hence will only transfer the same amount of current.

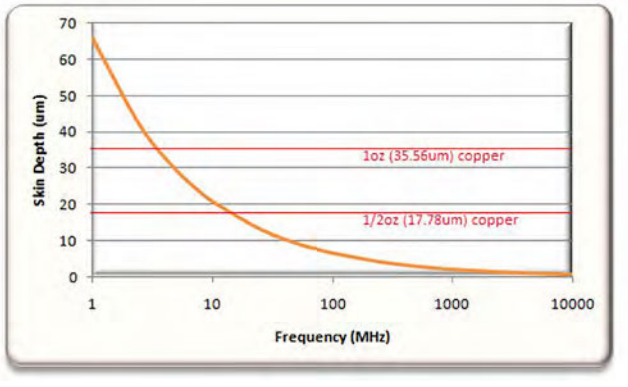


Figure 2: Skin depth (um) vs. frequency (MHz).

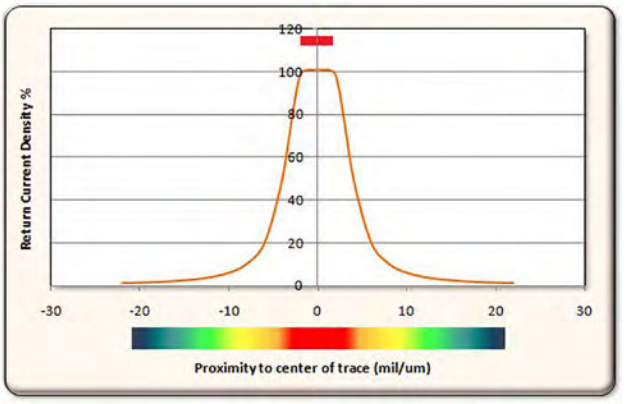


Figure 3: Microstrip return current density.

As seen in Figure 3, the return current distribution, in the plane surface of a microstrip configuration, is reduced as the distance to the center of the trace increases. Here we see a balance of two opposing forces:

- Too narrow a distribution increases inductance as narrow traces have more inductance than broad ones
- Too broad a distribution increases inductance by increasing the loop area

So, there is a sweet spot where the total energy stored in the electromagnetic (EM) field surrounding the trace is optimized. Crosstalk between two or more conductors depends on their mutual inductance and mutual capacitance. The inductance plays the major role in this coupling. The signal return currents will generate EM fields. Those EM fields, in turn, induce voltages (crosstalk) into other signals.

It can be seen in Figure 4 that the differential impedance or the coupling of two parallel traces, levels off at 100 ohms above 12 mils trace clearance (blue curve). This is simulated quickly by multiple passes of the field solver. All other factors being equal, the differential impedance will always be 100 ohms regardless of increased spacing. This also represents the point at which crosstalk (coupling) begins. This curve provides a clear map of the design space and efficiently defines the stackup configuration for single ended and coupled pairs. In this case, once the separation is less than 12 mils, the two traces begin to couple and transfer electromagnetic energy.

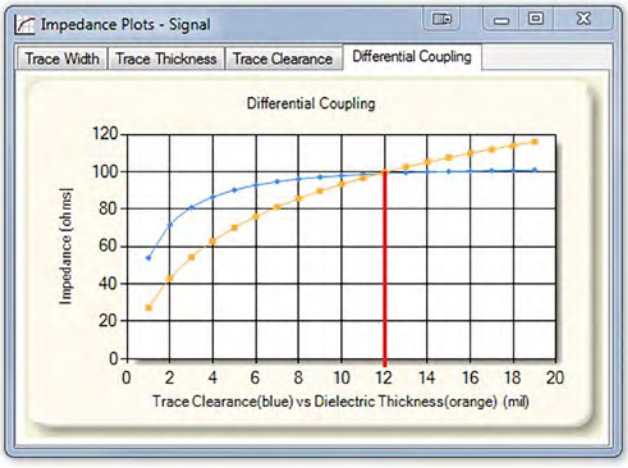


Figure 4: Coupling levels off above 12 mils separation (simulated by the iCD Stackup Planner).

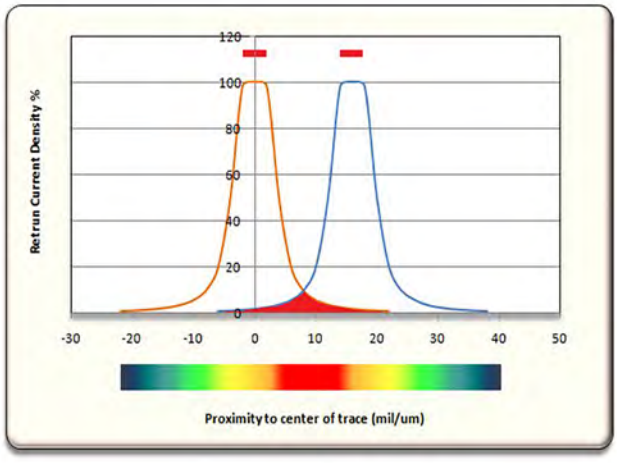


Figure 5: Parallel traces and return path crosstalk.

The easiest way to reduce crosstalk, from a nearby aggressor signal, is of course to increase the spacing between the signals in question. Doubling the spacing cuts the crosstalk to roughly a quarter of its original level. However, crosstalk is determined by the ratio of the trace separation and also the height of the trace above the plane. By varying the trace height, one can also control the coupling–hence crosstalk. If real estate is limited, then this may be a better solution rather than increasing routing density. A tight coupling (less height) results in less crosstalk.

The return current distribution of two parallel traces (Figure 5) shows an overlap of current in the surface of a microstrip plane. In fact, the overlap will be larger at lower frequencies where the return currents tend to spread out and not follow a tight path under the trace. Also, as the voltage increases so does the coupled noise. This is a good reason not to intermingle dissimilar technologies but rather keep them isolated.

If we look into it further, the degree of crosstalk is also dependent on several other factors including driver strength (which can normally be adjusted in the firmware), transmission line length, how far the segments run closely in parallel and signal rise time. In the case of long transmission line lengths, a series terminator slows the signal rise/fall time and reduces reverse-coupled crosstalk at the near end improving crosstalk considerably.

With all of these issues, to take into account, how do we ever get high-speed transmission lines, particularly those with wide parallel buses, to work efficiently? Fortunately, synchronous buses, as typically used in DDRx designs, benefit from an extraordinary immunity to crosstalk. Crosstalk only occurs when the signals are being switched and this crosstalk only has an affect within a small window around the moment of clocking. So, providing the receiver waits sufficiently long enough for the crosstalk to settle before sampling the bus, the crosstalk has no impact on the signal quality at the receiver. It's all about timing—ensuring that the required setup and hold times are provided at the receiver.

Remember: *"Beware the dark side. Anger, fear, aggression; the dark side of the Force are they. Easily they flow, quick to join in a fight. If once you start down the dark path, forever will it dominate your destiny, consume you it will."* —Yoda

Points to Remember

- Current must return to the source to complete the loop.
- Returning signal currents tend to stay in close proximity to their signal conductors, falling off in intensity with the square of increasing distance.

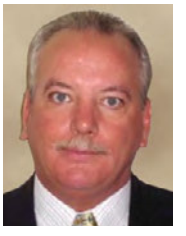
- As the frequency increases, the current is forced into the outer surface of the copper, due to the skin effect.
- Skin depth decreases with increased frequency.
- At high frequencies, the thickness of the copper plane is irrelevant—½ oz. and 3 oz. copper will have the same surface conduction area.
- There is a sweet spot where the total energy stored in the electromagnetic field surrounding the trace is optimized.
- The signal return currents generate EM fields. Those EM fields, in turn, induce voltages (crosstalk) into other signals.
- The easiest way to reduce crosstalk is to increase the spacing between the signals in question.
- Crosstalk can also be controlled by varying the trace height, above the plane. A tight coupling (less height) results in less crosstalk.
- The return current distribution of two parallel traces shows an overlap of current in the surface of a microstrip plane.
- To minimize crosstalk do not to intermingle dissimilar technologies but rather keep them isolated.
- Synchronous buses, as typically used in DDRx designs, benefit from an extraordinary immunity to crosstalk.
- Ensure that the required setup and hold times are provided at the receiver.

References

1. Barry Olney's Beyond Design columns: Return Path Discontinuities, The Dumping Ground, Controlling the Beast, Effects of Surface Roughness on High-speed PCBs, Uncommon Sense.

2. High-Speed Digital System Design, by Steven H. Hall, Garrett W. Hall, and James A. McCall

3. High-Speed Digital Design, by Howard Johnson and Martin Graham



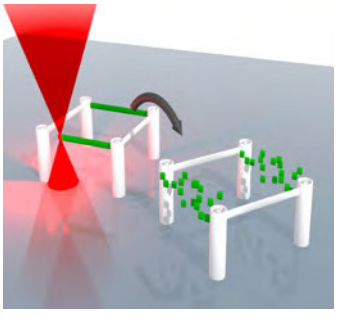
Barry Olney is managing director of In-Circuit Design Pty Ltd (iCD), Australia. The company developed the iCD Design Integrity software incorporating the iCD Stackup, PDN and CPW Planner, is a PCB design service bureau and specializes in board-level simulation. The software can be downloaded from www.icd.com.au.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

Erasable Ink for 3D Printing

Researchers of Karlsruhe Institute of Technology (KIT) have now developed a method to erase the ink used for 3D printing. In this way, small structures of up to 100 nm in size can be erased and rewritten repeatedly. This development opens up many new applications of 3D fabrication in biology or materials sciences.

"Developing an ink that can be erased again was one of the big challenges in direct laser writing," Professor Christopher Barner-Kowollik of KIT's Institute for Chemical Technology and Polymer Chemistry says.

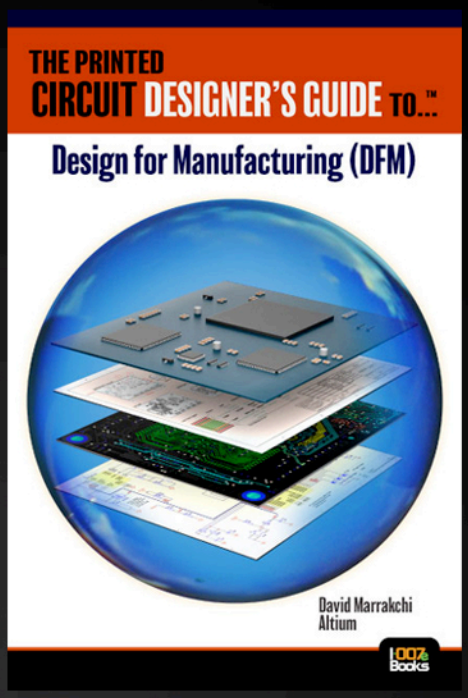
The process was developed in close cooperation with the group of Professor Martin Wegener at the Institute of Applied Physics and the Institute of Nanotechnology of KIT.

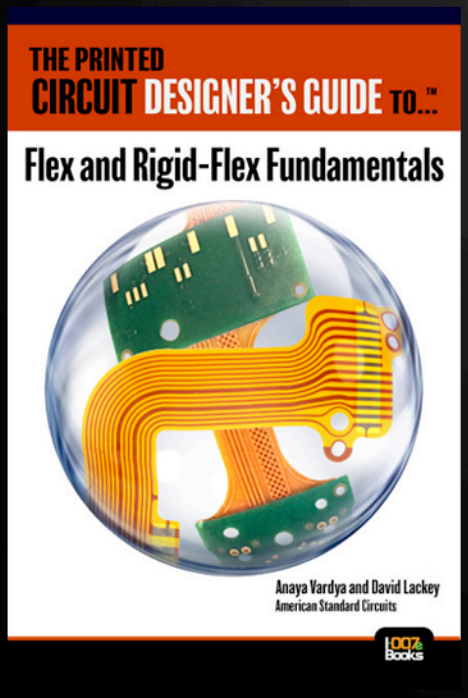


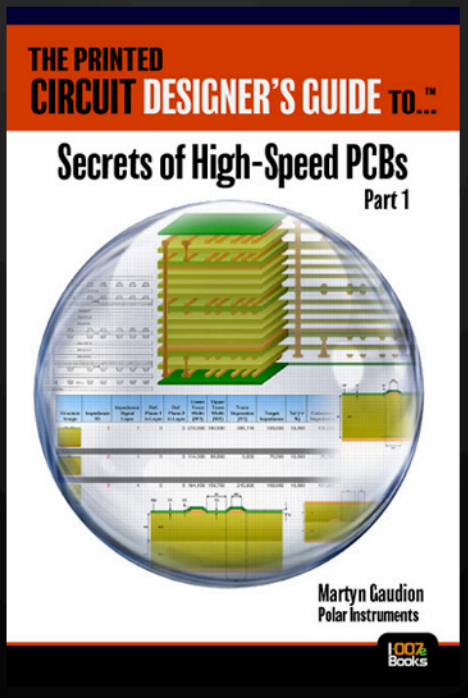
Structures written with erasable ink can be integrated into structures made of non-erasable ink: Support constructions can be produced by 3D printing, which are similar to those used when building bridges and removed later on. Recently, such structures were designed by KIT to grow cell cultures in three dimensions on the laboratory scale.

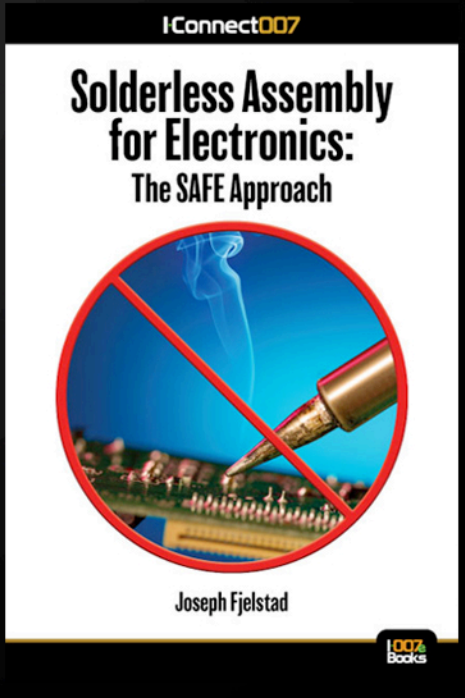
"During cell growth, parts of the 3D microscaffold could be removed again to study how the cells react to the changed environment," Wegener explains. According to the scientists, it is also feasible to produce reversible wire bonds from erasable conducting structures in the future.

Check out the latest titles in our micro eBook library!







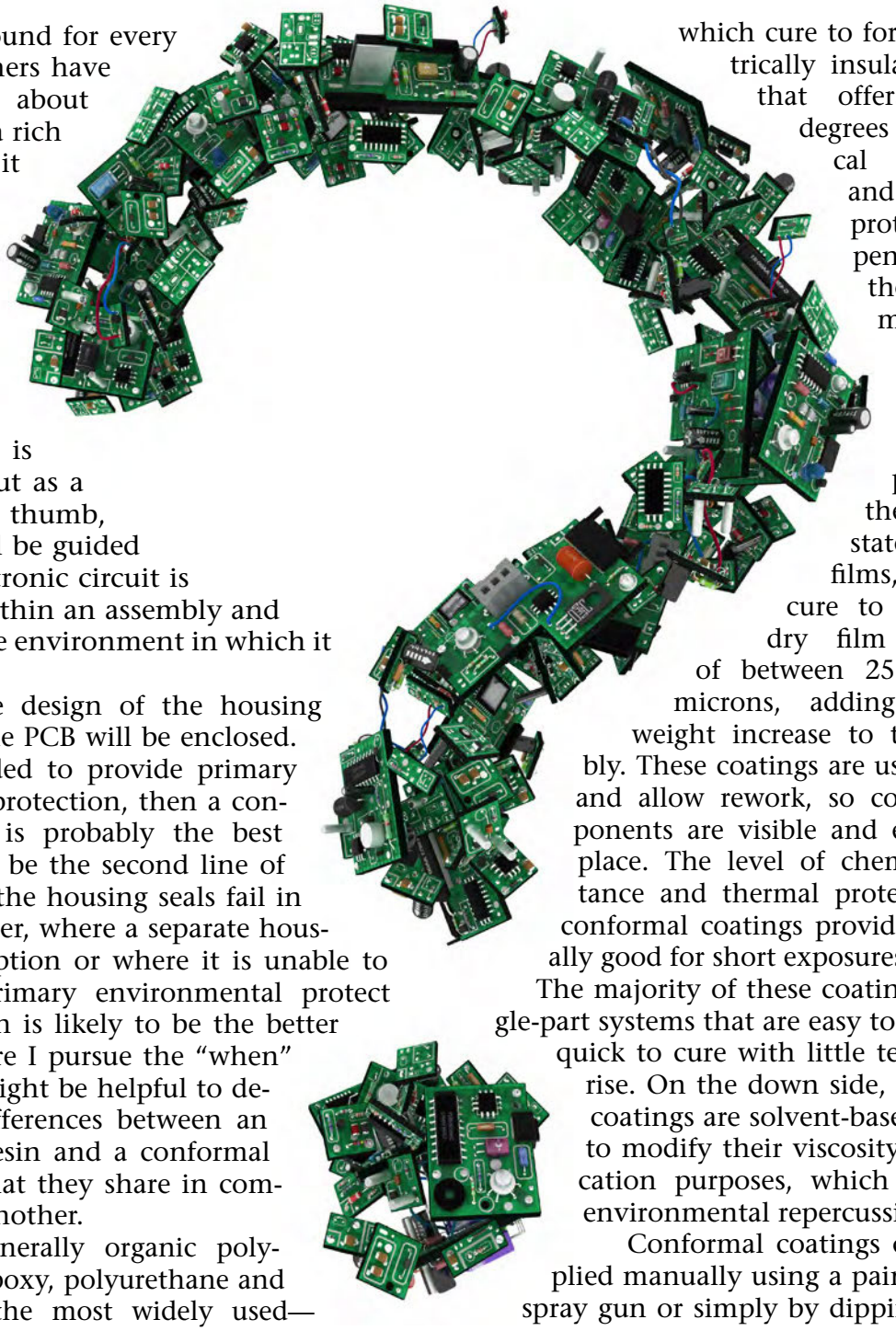


Written by industry experts, these free micro eBooks each contain valuable information on relevant topics in an easy-to-read format. Click on a title above or browse our entire library today!

I·007e Books

To Encapsulate or Coat: That is the Question

by Alistair Little
ELECTROLUBE



If I had a pound for every time our customers have approached me about this one, I'd be a rich man: When is it appropriate to use a conformal coating rather than an encapsulation resin and vice versa?

Well, the short answer is "it depends." But as a general rule of thumb, your choice will be guided by how an electronic circuit is to be housed within an assembly and the nature of the environment in which it must operate.

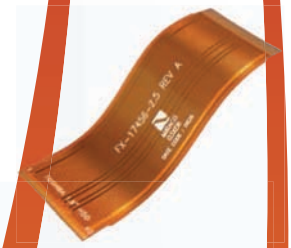
Examine the design of the housing within which the PCB will be enclosed. If this is intended to provide primary environmental protection, then a conformal coating is probably the best choice as it will be the second line of defence should the housing seals fail in any way. However, where a separate housing is not an option or where it is unable to provide that primary environmental protect ion, then a re in is likely to be the better choice But before I pursue the "when" and "why," it might be helpful to define the key differences between an encapsulation resin and a conformal coating, and what they share in common with one another.

Both are generally organic polymers—acrylic, epoxy, polyurethane and silicone being the most widely used—which cure to form an electrically insulating layer that offers varying degrees of chemical resistance and thermal protection, depending upon their formulations. Conformal coatings are applied in their liquid state as thin films, which cure to provide a dry film thickness of between 25 and 100 microns, adding minimal weight increase to the assembly. These coatings are usually clear and allow rework, so coated components are visible and easy to replace. The level of chemical resistance and thermal protection that conformal coatings provide is generally good for short exposures.

The majority of these coatings are single-part systems that are easy to apply and quick to cure with little temperature rise. On the down side, single-part coatings are solvent-based in order to modify their viscosity for application purposes, which can have environmental repercussions.

Conformal coatings can be applied manually using a paint brush, a spray gun or simply by dipping the cir-

Engineering And Providing Balanced Interconnect Solutions



MIRACO

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

...connecting technology

cuit board into a bath containing the coating material. Where large numbers of circuits have to be treated in a fast-moving production line, coatings are more likely to be applied by closely controlled robotic selective coating systems for maximum consistency.

Potting compounds and encapsulation resins can be applied in thicknesses from 0.5 mm, but are generally applied much thicker than this, which can lead to a significant weight gain for the assembly. Weight gain aside, this increased thickness does mean that the PCB is far better protected against chemical attack, particularly in cases of prolonged immersion. A resin can also provide superior protection against physical shock (depending upon the formulation), since its bulk will help to dissipate the forces across the PCB, rather than allowing them to be concentrated. And for those of you who are keen to protect your intellectual property and design advantages, a layer of dark coloured resin can obscure the circuit layout and components from prying eyes. One thing to remember though is that attempting to remove the resin will damage the PCB, severely limiting opportunities for component replacements.

“One thing to remember though is that attempting to remove the resin will damage the PCB, severely limiting opportunities for component replacements.”

Potting compounds and encapsulation resins are generally two-part systems, in which a resin is mixed with a hardener in a precise ratio to form a cross-linked polymer when cured. It's also possible to add mineral substances (fillers) to resins to improve their performance under certain operating conditions. Like conformal coatings, most resins will cure at room temperature, and while this can be a relatively slow process in the case of potting resins, cure time can be reduced by applying heat.

An important thing to be aware of when mixing a resin with its hardener is the inevitable rise in temperature of the mix as the majority of curing reactions are exothermic in nature. The rise in temperature can be controlled by careful judgement of the amount of material that needs to be cast in a single process, as too much resin applied at once may result in components becoming overheated and possibly irreversibly damaged.

Before we get to the criteria for making a choice between potting and coating, it might interest readers to learn that, in certain applications, where a two-part resin formulation may have been the first choice for circuit protection, a two-part conformal coating may turn out to be the better approach, thanks to its superior mechanical properties, compared with one-part coatings.

For example, Electrolube has recently launched the 2K range of solvent-free coating materials, based on two-part chemistry that is similar to resins, but designed to be applied by selective coating equipment in the 200—400 micron range, combining many of the advantages of both technologies and minimising many of the drawbacks of each. Moreover, switching from a resin to a conformal coating will eliminate the weight penalties of the former, which may be critical to some applications.

So, how do we choose which approach is best for a particular application? Clearly, there will be applications where the choice of technology is obvious, for example, where harsh environment duty demands the highest level of protection. In this case, users can be confident that potting and encapsulation will provide the necessary long term protection, so long as the correct resin has been selected, tested, and approved for the prevailing environmental conditions.

As I mentioned, if a PCB housing provides adequate primary protection against adverse environmental conditions, then a conformal coating will protect that PCB should the primary protection fail in any way, as well as protecting against high humidity and condensation within the housing itself. And where ease and speed

of processing are important, conformal coatings will always be the preferred choice, particularly as the thin cured film of a modern, well formulated coating can provide a high level of protection in any case. At this point, I can't resist coming back to the new two-part conformal coatings that I mentioned earlier, as they have rather muddied the waters as far as making that choice between potting and coating is concerned. Two-part conformal coatings can be applied relatively thickly without risk of cracking, giving a sharp edge coverage that performs somewhere between where a conventional conformal coating fails and potting is required. A two-part conformal coating's environmental protection capability is also rather impressive.

For example, in environmental chamber trials simulating highly condensing conditions, while a urethane resin potted assembly gave the highest overall values in terms of circuit protection—and showed the least change during condensing events—the very large difference in thickness between it and a two-part conformal coating didn't show a large increase in performance. Indeed, the two-part coating achieved much the same results as the potting compound at one tenth the thickness.

Despite these advances in conformal coating chemistries, potting and encapsulation resins will always offer the highest level of protection for PCBs, whether used to protect against mechanical shock and vibration, thermal cycling, chemical attack or the presence of high voltages where maximum dielectric strength is needed to avoid damaging discharges and leakage currents. The trade-offs are added weight, loss of rework capability, longer processing times and high cure temperatures.

If you do experience problems choosing between these methods of circuit protection, remember, there are experts out there to help. And it is always worthwhile taking the trouble to test alternative methods of protection, preferably at the prototype stage before you make your final choice; the experts are ready to help you with this task as well. **PCBDESIGN**

Alistair Little is technical director of Electrolube's Resins Division.

Using Sulfur to Store Solar Energy

Researchers of Karlsruhe Institute of Technology (KIT) and their European partners plan to develop an innovative sulfur-based storage system for solar power. The pre-development work under the PEGASUS project will be funded by the EU with about EUR 4.7 million.

The long-term goal of PEGASUS is the development and demonstration of an innovative solar power tower facility. A solar absorber is combined with a thermochemical solar power storage system based on elementary sulfur and sulfuric acid.

The partial project executed by KIT focuses on the technical implementation of combustion. It is planned to develop a lab-scale sulfur burner for stable combustion in the range from 10 to 50 kilowatts at high power densities under atmospheric conditions and temperatures higher than 1400°C.

Elementary sulfur is produced by the disproportionation of sulfur dioxide, i.e., conversion of sulfur dioxide into sulfur and sulfuric acid. The focused sunlight of the solar power plant supplies the process heat with the energy and temperature required to close the sulfur cycle and to convert sulfuric acid back into sulfur dioxide in the presence of suited catalysts.

Use of the stored energy in a burner makes these power plants capable of providing base-load power. In the long term, system costs will be lower than estimated for photovoltaic systems.

MilAero007 Highlights

One World, One Industry: 100 Days In—President Trump and a Better Manufacturing Policy

To truly increase the number of American manufacturing jobs, President Trump should support increased investment in research and development for advanced manufacturing, promote and fund STEM education in primary and secondary schools, and build stronger apprenticeship programs. It is this type of investment—in human capital and technology—that will truly help make American manufacturing great again.

Rep. Lofgren Discusses U.S. Policy Priorities with IPC Member TTM Technologies

Executives and staff at an IPC-member facility held a town hall discussion with Congresswoman Zoe Lofgren (D-CA) on the federal policy issues facing the advanced manufacturing industry. Rep. Lofgren took a tour of TTM Technologies, Inc. in San Jose, getting a first-hand look at the high-tech work being done in TTM's facility.

Capitol Connection: IMPACT Update—To CEOs on Why You Should Attend IMPACT Washington, D.C. 2017

Here at IPC, we place a high priority on making our presence known in the halls of government, because so many policy debates have a direct effect on the electronics manufacturing industry. IMPACT Washington, D.C. 2017 is a chance to join with fellow industry executives in advocating for better public policies for a stronger, more advanced manufacturing economy.

Eagle Electronics Accelerates Production with Two Lenz Drilling Machines

Brett McCoy, COO for Eagle Electronics of Schaumburg, Illinois announced that his company has recently placed a purchase order for two Lenz DLG 615-2 Y765 drills.

Electrotek Receives Nadcap Accreditation for Electronics, Printed Boards

To demonstrate their continued commitment to quality, Electrotek Corporation announces that they have been successful in achieving Nadcap accreditation for electronics, printed circuit boards.

Capitol Connection: Drumroll, Please... Announcing the IPC Government IMPACT Awardees

Government relations are essential to our industry as many of the policy debates taking place will have long-lasting impact on our industry. Tax and regulations reform, immigration, and environmental regulations are up for debate and it is time for our industry to take a seat at the table.

U.S. Army Picks BAE Systems to Design Next-Generation Space and Missile Defense Technologies

BAE Systems has been chosen for a position on a new eight-year, $3 billion indefinite delivery, indefinite quantity (IDIQ) contract to continue supporting the U.S. Army's Space and Missile Defense Command/Army Forces Strategic Command (SMDC/ARSTRAT).

HT Global Circuits Acquires Pho-Tronics

HT Global Circuits, a manufacturer of printed circuit boards, has acquired fellow PCB manufacturer Pho-Tronics for an undisclosed sum. Pho-Tronics will continue to be operated by the existing management team.

UW Security Researchers Show That Google's AI Tool for Video Searching Can Be Easily Deceived

University of Washington researchers have shown that Google's new tool that uses machine learning to automatically analyze and label video content can be deceived by inserting a photograph periodically and at a very low rate into videos.

One World, One Industry: IPC's Global Policy Framework for 2017—Smart Advocacy for the Industry

As President Trump was being sworn in and as the new Congress was getting down to work, IPC released its Global Policy Framework for 2017. As we work to represent more than 3,800 member facilities across the electronics industry's global supply chain, IPC will adhere to this framework to guide our policy work in the coming months.


FAILURE HERE
IS NOT AN OPTION
Our 'Can't-Fail Scenario' Attitude is built on 40+ years of industry-wide reliability and experience.
We specialize in HDI Rigidflex for Mil/Aero and Medical applications, are certified to NADCAP and AS9100 as well as meeting all applicable ITAR Compliancy requirements.
Let us help extend the capabilities, reliability and technology of your next design!
LEARN MORE ABOUT OUR HDI SOLUTIONS
eltek
a Nistec company
www.nisteceltek.com

PCB Design in the Age of IoT

by Ted Pawela
ALTIUM

From the early days of printed circuit boards, the electronics industry has made huge strides in board materials, copper printing methods, miniaturization, rigid-flex, ELIC, EDA, and much more. Many of the devices we use in our homes, our vehicles, and in our workplaces would not be possible without this continuous evolution of PCB design and technology. And yet in 2017, we are poised to shift from evolution to revolution, driven by the idea of the Internet of Things.

By now we've all heard of IoT and have been presented with a multitude of definitions for it. We've also been presented with a set of benefits that sound nice, if not compelling: refrigerators that can tell us when to restock groceries, cars that can avoid traffic, home thermostats and lighting that can be adjusted from our offices, and so much more. But these examples trivialize what the IoT will become and the impact it will have on us. When realized, the IoT will transform our world from a collection of independent "things" into an organized system with logic, reasoning, senses, circulation, and motor skills. In other words, all of the devices and systems in our world will become an organism.

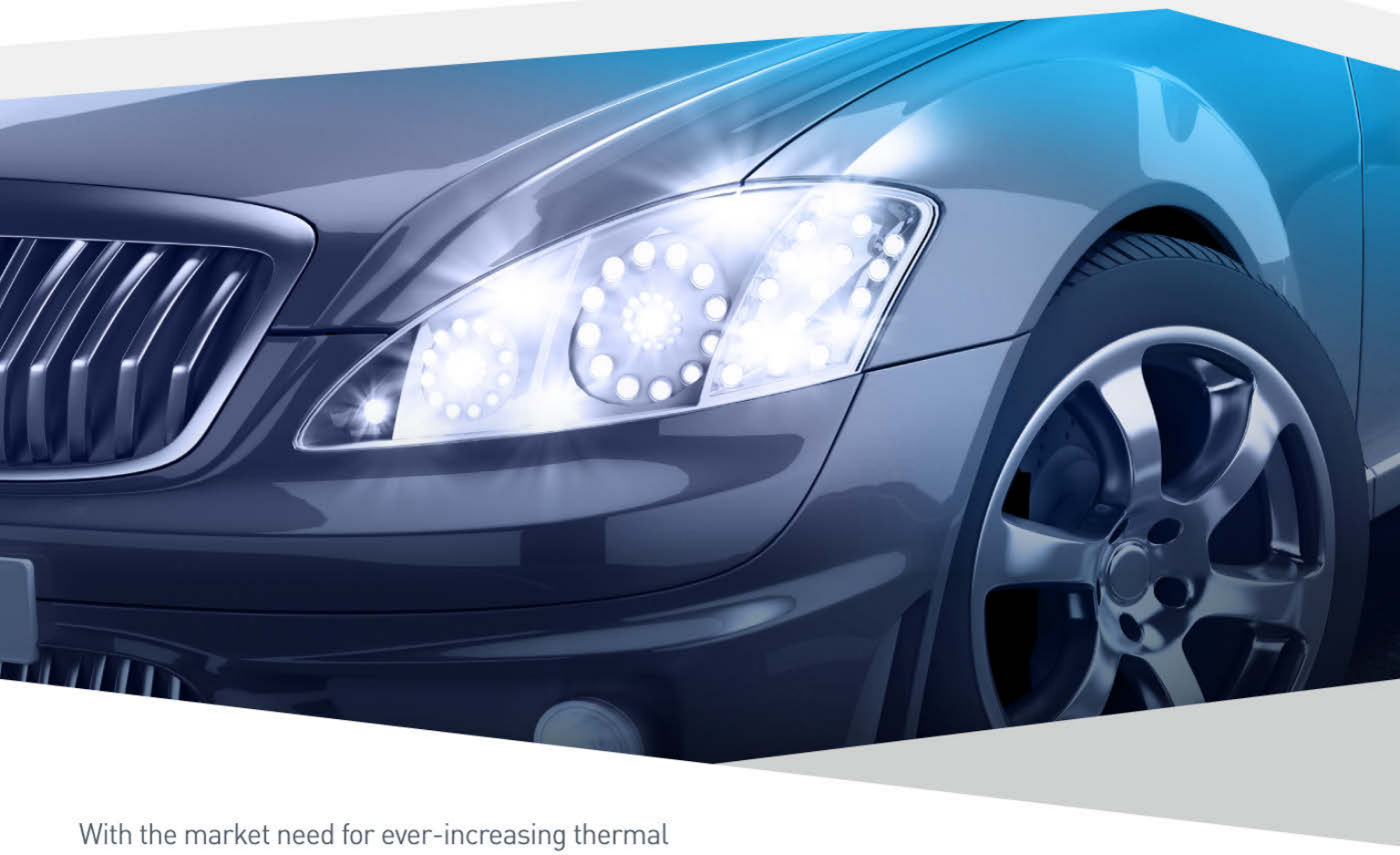
This might sound scary, and will no doubt evoke visions of dystopian societies where machines rule humans, but that's only because movie scripts need a mechanism called an "inciting event" upon which to build an exciting story. In real life, this story doesn't need to be scary; in fact, it holds the promise of a world of possibilities to make life safer, healthier, more convenient, and just plain better.

Imagine vehicles that can sense a problem before it occurs, and arrange for parts to be put into dealer inventory and service to be performed for you, all without you ever making a single phone call, and certainly without the roadside breakdown. Picture a farm with the intelligence to sense an increasing pest insect population and release pheromones that disrupt mating cycles—reducing the need for chemical pesticides and ultimately making our food supply safer, healthier, and more abundant.

Imagine biological sensors that detect the leading indicators of health maladies within your body and alert you and your doctor so that

Meeting the demands of the latest LED lighting designs delivering an unprecedented level of thermal performance

ventec
INTERNATIONAL GROUP
騰輝電子



With the market need for ever-increasing thermal performance, the Ventec VT-4B family of thermally conductive IMS (Insulated Metal Substrates) deliver an unprecedented level of thermal performance through their established ceramic-filled halogen-free dielectric technology. Multilayered constructions are made possible through resin-coated foil and resin-coated film options.

Contact us to discuss your requirements.

Ventec - wherever technology takes you, we deliver

Ventec International Group
179 Ward Hill Ave,
Ward Hill, MA 01835,
United States
T: 978-521-9700
E: ordersCA@ventec-usa.com
Follow @VentecLaminates

www.**venteclaminates**.com

preventative treatment can stop disease before it starts. For each of these examples, connecting into the Internet's vast collection of data and information will be critical. As the body of knowledge in each domain—healthcare, for example—grows and advances, it is crucial that the IoT not be hard-coded to embed the current state of our knowledge. Rather it should be constantly evaluating new information from research around the globe, and adapting through continuous improvement and learning. With all of this in mind, even calling it the Internet of Things is too limiting: It's really the Internet of Everything.

"With all of this in mind, even calling it the Internet of Things is too limiting: It's really the Internet of Everything."

The IoT (or the IoE) isn't likely to happen based solely on the efforts of companies, governments, and organizations. The ideas that come from organizations are shaped and filtered by their own goals and objectives. As a society, however, the scope of creativity from individuals is unbounded. While the infrastructure for IoT can, and probably must, be designed and built by organizations, the applications for IoT will need to be realized through a network effect that enables creative individuals of all shapes and sizes to contribute.

Typically, a network effect is described as a phenomenon in which something becomes more valuable based on the number of people who use it. This is undoubtedly going to be true of the IoT, and yet even more important than the number IoT users will be the number of people who can imagine, design, and deliver hardware and software applications that make the IoT valuable to all of us.

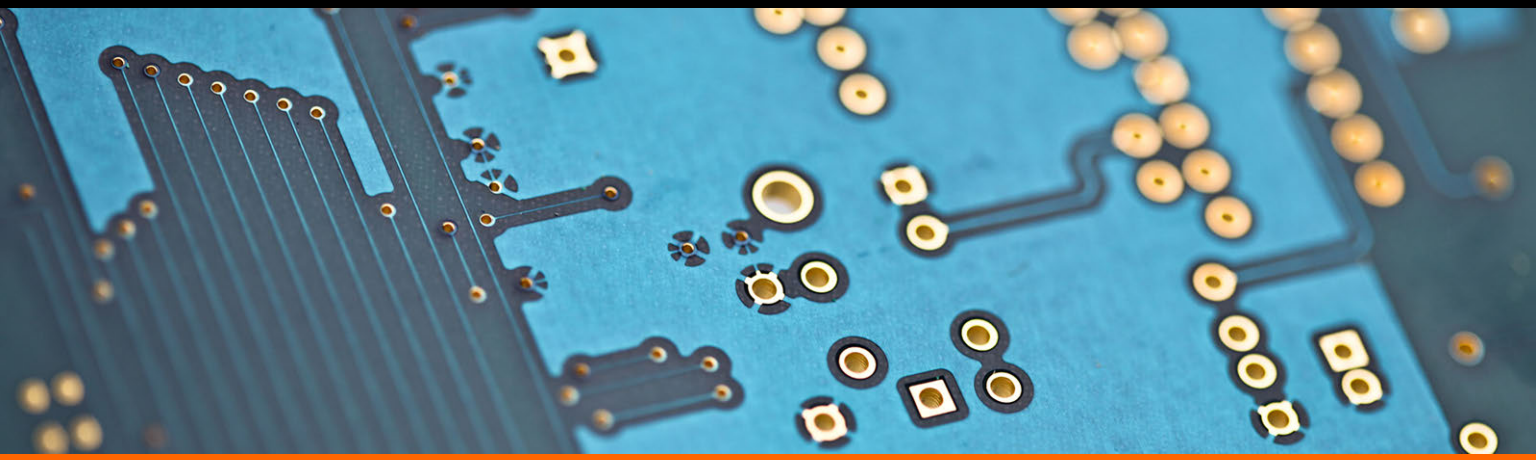
An example of a similar network effect would be for Apple's iPhone. The number of iOS devices is actually significantly smaller than the number of Android devices and users, and yet iOS generates far more revenue. The reason is that the developer community plays a key role in the mobile device network effect. Apple customers choose iPhone in large part due to the number of apps available to them, and there are far more apps available for the iPhone than for Android devices. So who develops iPhone apps? A 2012 study by GigaOM Pro [1] suggested that only about 40% of iPhone app developers work full-time in app development, and nearly the same percentage of app developers do so part-time while holding another job.

The IoT's network of PCB designers will likely need to look much like this, comprised not only of PCB design professionals, but makers, students, and others individuals who have imagination and the will to create, yet lack the resources of their professional counterparts. While they may lack resources today, a series of changes are sweeping through the design and manufacturing worlds that will empower individuals to design and build the apps that will make the IoT valuable for everyone. What sorts of changes?

In design, the open source movement and "freemium" business models have made a new generation of software tools accessible to individuals who traditionally could not afford the up-front investment to purchase them, or the learning curve to use them. For example, schematic design, layout, signal integrity, and more are all available to would-be PCB designers at low to no-cost, with an array of professional features. In manufacturing, digitization has led to simplicity in getting designs realized through very economical outsourced production services, for example, 3D printing. The democratization of design and manufacturing technology has resulted in the ability for individuals, not just companies, to design and manufacture products in batches from small to large affordably and with professional quality.

For a PCB design tool company like Altium, all of the change associated with the Internet of Things causes us to think carefully about what our products need to look like now and in the future. While once we might have focused 100% of our energy and resources on the full-time PCB specialist, in the last 18 months, we

SUNSTONE CIRCUITS *from Prototype to Production*



PCB fabrication, assembly, and free award-winning CAD design software.

Leading the industry for over 40 years as the preferred ***prototype and production PCB solutions provider*** for design engineers.

WE ALSO OFFER:

- Best overall quality, value and competitive pricing
- Design for Manufacturability Tool: **DFMplus**
- Fully supported and free **PCB123®** design software
- Integrated assembly services
- FREE Ground Shipping
- PCBs as low as $28 with **ValueProto®**

100%

On-time Guarantee

QUOTE NOW ›

Questions? Comments? Suggestions?

We appreciate hearing from you!
Contact us at: 1-800-228-8198 or via e-mail at support@sunstone.com

SUNSTONE CIRCUITS

have seen the introduction of new products designed specifically for makers and part-time designers. These tools, based on the different types of people who use them as well as their unique ways of working, cannot be cookie-cutter, "one size fits all" software applications. They need to recognize and enhance the uniqueness of each user type, placing differing levels of priority and emphasis on simplicity vs. feature richness, guided workflow vs. flexibility, and other trade-off scenarios. And most of all, these tools need to be accessible to everyone who has the imagination and desire to create products and applications that will help make the IoT real and an enabler of societal improvements. Accessibility is relative, of course, so I'm not suggesting that everything needs to be free. Rather I would suggest that in the "Age of IoT" we need PCB design tools that fit the end user in terms of design paradigm, capability set, price, and hardware requirements.

The makers market, for example, leans heavily on the open source philosophy for both hardware and software. The open source philosophy suggests a social component of shared designs to support reuse, broad adoption, and technical advancement based on the contributions of many people. The Arduino microcontroller is a great example; while a core team developed Arduino initially, the design was shared openly enabling a broader community to make the design lighter, more powerful, and less expensive. PCB design software for makers needs to be inexpensive (or free), and powerful, yet perhaps most important is that it needs to recognize the open-source philosophy by obviating the effort associated with sharing designs, and making it a part of the PCB design process.

Professional design engineers can afford to pay for tools, yet protecting IP, working in ways that are most productive for them as individuals, and having a rich feature set are most important in their working environment. They are able to pay for the capabilities that they need, but require extreme productivity in return. Individual PCB professionals ultimately need the best set of features for their design types at a fair price—in other words, a value-based approach.

And teams working on complex designs have yet another set of needs. Whereas they have typically looked to enterprise PCB design tools, in the Age of IoT they need less restrictive design processes while still working in ways that ensure consistency in design. Team-based design scenarios involve working concurrently on individual parts of a logically and physically interconnected system; this requires role-based permissions, strict version control, and design conflict resolution to ensure that when individual parts come together, they work as a system. The heavy, restrictive enterprise tools of the past are simply too cumbersome to allow engineers and designers to work at the speeds required in the present and future.

In summary, IoT represents an incredible opportunity for positive transformation in healthcare, transportation, manufacturing, agriculture, energy, and virtually every other element of society. The realization of its vast potential can only come through a network effect, that will be highly dependent on our industry's ability to enable not just professionals, but makers, students, and other would-be creators to design and develop innovative devices that will add value to the IoT infrastructure.

IoT's story won't be a dystopian sci-fi thriller, but rather a feel-good underdog story in which unlikely heroes save the day with their imagination, creativity, and will to make things better. PCB design software will need to evolve to meet their needs if we are to be a part of the story, in ways that support the needs of unique user groups. Altium intends to be a big part of the IoT story by doing what we've always done: making advanced technology accessible to anyone, and everyone who needs it. **PCBDESIGN**

Note: If you're interested in learning more about how we will do that, please consider attending one of our upcoming roadshows for PCB 2020: Altium's Roadmap to the Future.

References

1. The GigaOM Pro 2012 study.

Ted Pawela is chief marketing officer of Altium.

PCB BUYERS - compare nearly 1900 manufacturers now at *The* PCB List.

Quick Search

Advanced Search

thepcblist

The best way to find a PCB fabricator, *anywhere*.

www.thepcblist.com

Shortening Time-to-Market with a Software-Supported Review Process

by Guenther Schindler
SCHINDLER & SCHILL GMBH

Across different industries, manufacturers are achieving competitive advantages through the intelligence of machines. As a result, the role of electronics is becoming increasingly important. Manufacturers no longer have all the time in the world, even in industries that up to now had been exempt from the extremely short product cycles of consumer electronics. One important step towards shortening time-to-market is to make the development process more efficient by improving communication. The EasyLogix software tool, PCB-Investigator, provides a single platform which brings together every aspect of a PCB throughout the production process and life cycle.

During the production process, every step from circuit diagram through to electronic product involves processing, exporting and importing data by various experts using different systems. These transfers, imports and exports are exposed to possible errors. PDF and paper-based review sessions are often the only tools available for correcting errors. With a software-based review solution like PCB-Investigator, however, every participant in the process now has access to a CAD system, which is able to merge PCB data from various sources in order to analyse and visualize that data interactively. As a result, review processes are more reliable, more efficient and paperless.

"Assembly is not possible in this way. Please check the data." Developers are regularly faced with this major problem, which is symptomatic of an industrywide weakness in the process. Development, layout, circuit board production and assembly, quality assurance, purchasing, sales—everyone within these departments will be working on the same product. Yet they all use different systems that focus on various aspects in non-uniform data formats. Monitoring is crucial, given that errors may occur at any stage of the workflow and every time data is exported to different CAD software or a different machine.

The biggest challenge is that due to the lack of a uniform tool chain, reviews usually end up being conducted using PDFs. This means generating a unique copy for every aspect. What's more, the finer details are difficult to recognize, the zoom function is awkward and displaying multiple diagrams side by side is impossible.

Superior Quality Begins Here.

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

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

Take a look at our quality »

COMMERCIAL • MILITARY • MEDICAL • BAREBOARD

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

As a result, paper-based review processes are enormously time consuming and costly and are prone to errors.

A Paperless, Interactive Approach

PCB-Investigator is based on a paperless and interactive platform. It is a user-friendly, flexible CAD/CAM software tool that offers integrated, interactive visualization of every area of development and production. Whilst departments involved in the process continue to work with their respective specialized systems, communication across departmental and process-related boundaries is enabled by PCB-Investigator, and as such offers a real advantage.

The software is based on ODB++, a recognized, common data format designed to transmit PCB design information between engineers and producers as well as between design tools by different CAD and CAM providers. In ODB++, all PCB production data is stored in a single record. PCB-Investigator uses this format as a central database and consolidates all the data for the development process and each step in the production sequence.

PCB-Investigator is unique as that data visualization is not limited to conductor tracks and surfaces. It also stores all available information about components, networks and the stack-up of the layout and makes it accessible to the user. Layers and data objects such as components, pins, networks and copper elements can all be accessed easily for evaluation or modification. Furthermore, the tool is compatible to work with a broad range of file formats: IPC2581, GenCAD, IPC365, Gerber, Excellon, Sieb&Meyer, DPF, DXF, raster (BMP, JPG, PNG, TIFF).

Figure 1: The EasyLogix PCB-Investigator can switch from 2D to 3D views at any time. The housing (blue) is managed in the same way as a component in the ODB++ data collection. The yellow blocks are placeholders showing worst-case scenario space requirements for components for which a choice of different suppliers is still pending.

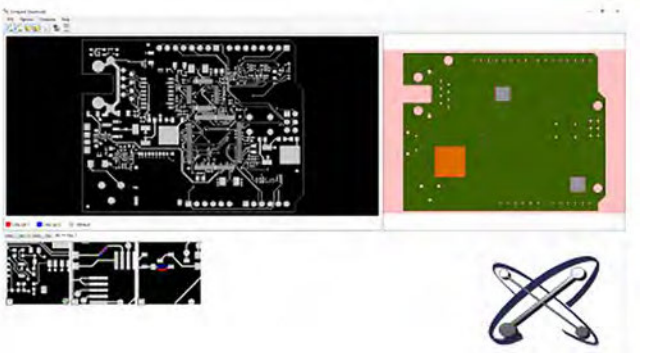


Figure 2: An example of revision comparison and revision history archive.

Whilst the visualization possibilities of ECAD systems are limited, PCB-Investigator displays drill data, exposure data and assembly data at the push of a button in any desired combination of information levels. The software can directly display anything that can be extrapolated from the combined PCB data, whether relating to design rule checking, PCB production, assembly, soldering masks, insulation distances, clearances or leakage currents. If you wish to investigate an individual network path, for example, simply select it within the application and zoom in.

Facilitating Fast, Simple, Version Control

Identifying the differences between two printouts of a PCB layout is not an easy task however has been a necessary process for error identification. PCB-Investigator makes version control easy. The more complex the electronics and the greater the number of teams working on it, the more important it is that every version is not only documented, but also compared and approved by everyone involved. This involves both a graphical comparison and a comparison of assembly data and network connections.

Any electronic product is created by skilfully connecting networks that reproduce the electrical characteristics of a device. PCB-Investigator automatically prepares all of the network information so the user sees an overview of the design of the PCB as quickly as possible. A printed circuit diagram is no longer absolutely necessary to obtain a comprehensive overview.

One important factor in the initial and subsequent development of PCBs is network list comparison. Design changes and version supplements can be incorporated into change lists. The electronic support provided by the PCB-Investigator makes it easy for anyone to compare layers or boards, even for highly complex products. The results are presented in a form that makes every modification easy to follow and provides all of the information needed for effective decision making.

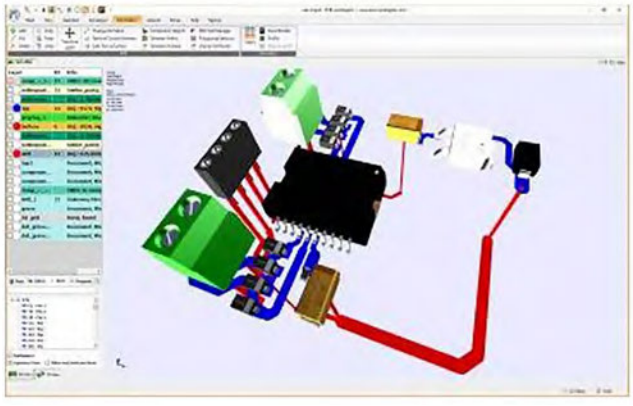


Figure 3: Detailed, highlighted view of power supply and power lead. Individual aspects of the PCB can be filtered and depicted individually.

Comprehensive Component Lists

During the development process, changes are made to the components as a result of optimization. These design changes can be identified and interpreted instantly by comparing the components on two available PCBs. PCB-Investigator standardizes the lists of components and helps the purchasing department to establish an automatic connection between the BOM (bill of materials) and the ERP system. The current price of each component with a part number is determined via the interface, added and displayed in reports. The sales department is responsible for the qualification process for alternative materials or parts. Constant price negotiations and periodic evaluations of delivery capability are part of the material selection process and ultimately pave the way for an affordable, reliable product guaranteed to be available over the long term.

So far it has been necessary to rely on a variety of unconnected BOMs. Every PCB-Investigator user relies on the same, basic, project-specific parts list, so that data specific to each department stays up to date and always remains consistent. A user-specific view enables employees in development, purchasing, production, logistics, sales and other departments to access a database that always represents the most current product development status. At the same time, users can at any time see what changes were made to components and when, whether due to design changes or the selection of better or cheaper parts or parts that will be available for longer.

The Process

How does PCB-Investigator change the review process? First, a decision has to be made to store each change to a product centrally in a valid ODB++ record. Previously a series of reviews involving paper-based formats were needed. Now each participant can load the ODB++ record into their PCB-Investigator for evaluation. This process is aided by four versions of software that reduce the complexity via domain-specific pre-selections: the Basic Edition focuses on graphical representation, the Developer Edition concentrates on electrical properties, the Fabrication Edition prioritizes questions of PCB fabrication and assembly, and the Ultimate Edition is designed for users who need an overview of every aspect.

The ability to compare PCBs with one another and with previous versions in all respects at the push of a button results in savings on development costs and makes it easier to reuse or eliminate parts. The ability to delve interactively into every information layer of an ODB++ record as and when required saves time and reduces the number of prototypes. For example, EMV experts or assembly specialists can highlight an issue at an early stage if they recognize certain layout decisions as problematic.

Data and Processing Sovereignty

Simulating changes and outputting the results in a report form are made possible by the PCB-Investigator. The development status stored centrally in ODB++ does not change in the process. New versions are only generated in the CAD systems of the corresponding departments. Communication regarding the versions and their reviews can be organized via Sharepoint, for example. Everyone involved in the project in any way has the intuitively designed CAD system installed on their computer, and the number of users who can open the PCB data simultaneously is limited only by the purchased number of floating licences. For communication with external offices (such as customers or the EMS provider) a license-free viewer can be created that automatically connects to the data.

Optimization Potential

PCB-Investigator not only helps obtain the best possible evaluations from ODB++ records, it also offers numerous functions for analysing and improving layouts. Automatic calculation of lengths and distances helps designers avoid disadvantageous component arrangements and potential manufacturing problems.

Another strength of the CAD program is its ability to include and depict components. It connects the 2D world of ECAD with the 3D world of MCAD. Components are three-dimensional objects, not just electronic functions and electrical properties stored for a specific region of a two-dimensional circuit diagram. PCB-Investigator bridges the gap between the placement of the circuit diagram and the construction of a housing for the electronics. Given the constant developments towards even further miniaturization, significant advantages may result if a dialogue takes place at a very early stage between the development, production and deployment stations of a PCB.

Temperature Simulation

The PCBi-Physics extension illustrates how thoroughly PCB-Investigator can optimize the development process. Developers can identify problematic heat developments in future products at a very early stage in development and can pro-actively avoid them. The PCBi-Physics interface is powered by ADAM Research's TRM (Thermal Risk Management), a specialized software that has proven its worth over the years and gained industry-wide recognition. TRM and, by

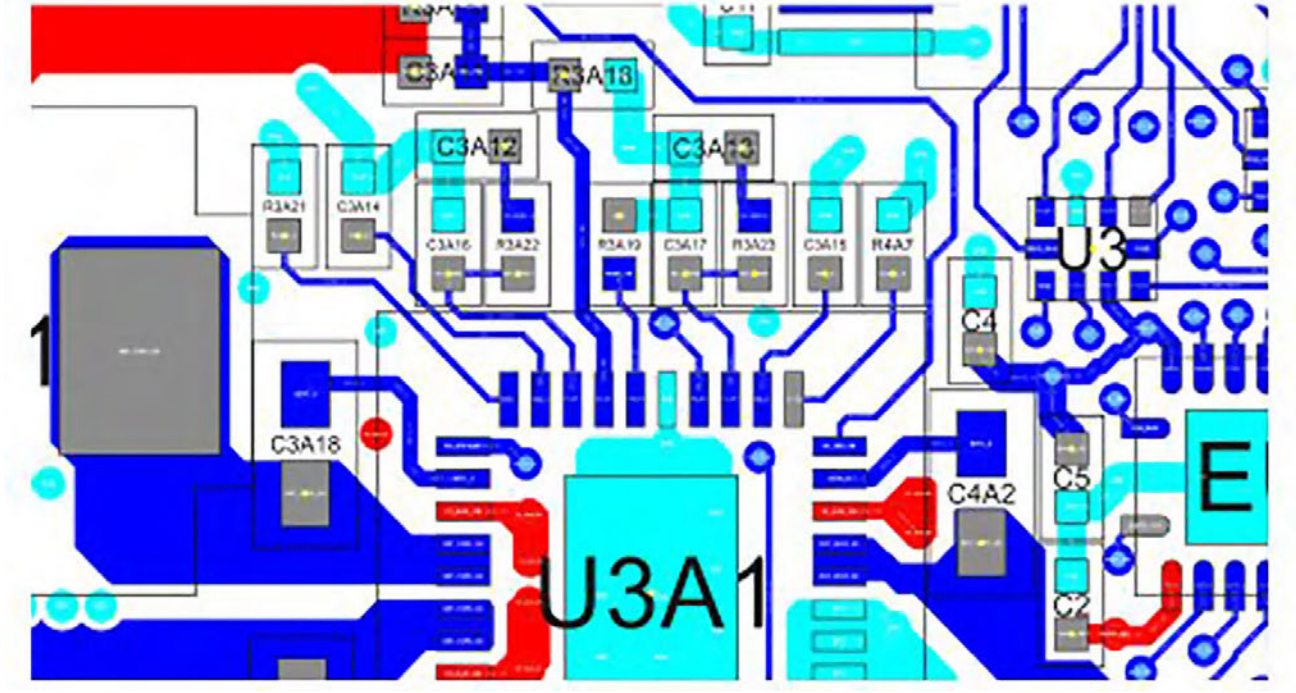


Figure 4: Clarity is ensured through color highlighting.

2017 FLEX

June 19–22
Monterey, California

Flexible Electronics: Accelerating to Manufacturing

REGISTER NOW!

June 19–22, 2017
Monterey, California

Discover why 2017FLEX is widely considered the best technical event of the year for the flexible and printed electronics community.

For more information, please visit www.2017FLEX.com.

semi | FlexTech

extension, the tool from EasyLogix are capable of incorporating multilayer boards, SMD heat sources, embedded components, pins, inlays, power rails, vias, blind vias and buried vias.

Embedded in the CAD/CAM software, thermal simulation can be undertaken by the developer at the push of a button without the need to involve an expert. This reduces the number of necessary prototypes and measurements. It also enables developers to avoid hotspots and the opposite problem, oversizing, thereby reducing development times and costs.

Conclusion

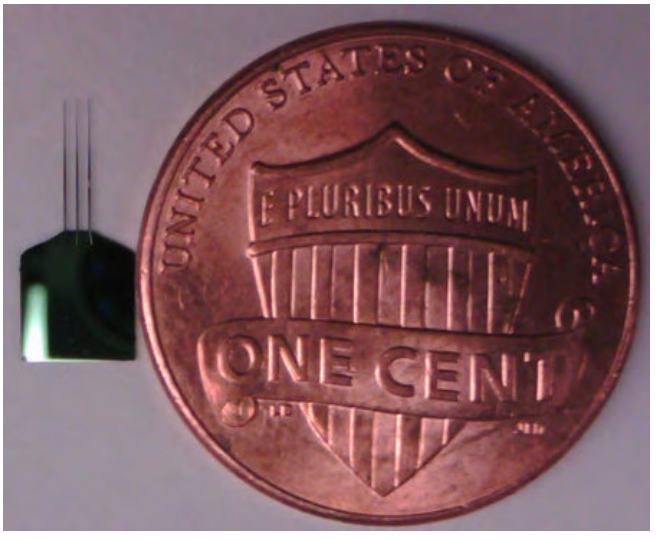
Visualization and analysis of every aspect of a PCB interdepartmentally at the push of a button makes error-recognition and avoidance possible, improves product quality and accelerates time-to-market. Product development and purchasing, production and quality assurance, EMV lab, prototype building, sales—the higher the product complexity, the more isolated is the work in these departments. It is high time that everyone started working with the same information. **PCBDESIGN**

Günther Schindler is CEO of Schindler & Schill GmbH. For more information, visit www.easylogix.de.

Shining Light Deep into the Brain

Researchers at Caltech have developed a new type of optical brain probe based on nanotechnology and the same optical communication technology used to carry internet data around the globe. The new ultrathin silicon-based probes are able to send light deep into the brains of animals to precisely target circuits of neurons that control cognition, behavior, and body functions.

"Tools for delivering light deep into animal brains are still rather primordial," says Michael Roukes, the Robert M. Abbey Professor of Physics, Applied Physics, and Bioengineering at Caltech and principal investigator behind the research.



"So, we decided to develop the next generation of brain probes." The probes were described in a paper that appeared in the journal Neurophotonics.

"These new silicon probes are much smaller than currently used optical fibers and are much gentler on the brain," says the paper's lead author, Eran Segev, a senior postdoctoral fellow in applied physics and materials science at Caltech.

With optogenetic techniques, researchers can use light to specifically activate or silence neurons within the brain and assess their roles. Optogenetic methods genetically engineer neurons to respond to light. For example, blue light is used to activate neurons that have been modified to produce a protein called channelrhodopsin. Conversely, yellow light is used to inhibit activity in neurons engineered to express a protein called halorhodopsin.

As described in the paper, the new probes can target individual neurons while simultaneously minimizing displacement of brain tissue and providing access to areas deeper in the brain than previously possible. Segev says that a subsequent generation of probes, currently in development, will also be able to create patterns of blue, yellow, and other wavelengths of light.

Andrei Faraon (BS ‹04), assistant professor of applied physics and materials science and a coauthor on the study, provided early assistance with the probe designs and laboratory characterization.

iCD Design Integrity

Incorporates the iCD Stackup and PDN Planner software. Offers PCB Designers unprecedented simulation speed, ease of use & accuracy at an affordable price

Dielectric Materials Library
30,700 Rigid & Flex Materials to 100GHz

Termination Planner
Extracts IV Curves from IBIS Models
Calculates Series Terminator of the Distributed System Including Loads

iCD Stackup Planner
Field Solver Accuracy, Characteristic Impedance, Edge & Broadside Coupled Differential Impedance

Allegro, Altium, Excel, HyperLynx, OrCAD, PADS, IPC-2581B Interfaces

Extensive Capacitor Library
5,650 Decaps Derived from SPICE Models

Matched Delay Optimization
Relative Signal Layer Propagation
Ideal DDRx Trace Delay Matching

iCD PDN Planner
AC Impedance Analysis & Plane Resonance

Integrated Stackup Planner, VRM Definition, Plane Resonance, Planar Capacitance & EMI to 100GHz

Heads-up Impedance Plots

PDN EMI Plot with EMC limits (FCC, CISPR) to 100GHz

iCD Stackup Planner - Offers Engineers & PCB Designers unprecedented simulation speed, ease of use and accuracy at an affordable price

- Industry Leading 2D (BEM) Field Solver precision
- Characteristic impedance, edge-coupled & broadside-coupled differential impedance
- Relative Signal Propagation with 'Matched Delay Optimization'–ideal for DDRx design
- Termination Planner - series termination based on IBIS models & distributed system
- Unique Field Solver computation of multiple differential technologies per signal layer
- Extensive Dielectric Materials Library –over 30,700 rigid & flexible materials up to 100GHz
- Interfaces–Allegro, Altium, Excel, HyperLynx, OrCAD, PADS, Zmetrix TDR, Zuken & IPC-2581B

iCD PDN Planner - Analyze multiple power supplies to maintain low AC impedance over entire frequency range dramatically improving product performance

- Fast AC impedance analysis with plane resonance and projected EMI
- Definition of plane size, dielectric constant & plane separation
- Extraction of plane data from the integrated iCD Stackup Planner
- Definition of voltage regulator, bypass/decoupling capacitors, mounting loop inductance
- PDN EMI Plot with FCC, CISPR & VCCI Limits. Frequency range up to 100GHz
- Extensive Capacitor Library –over 5,650 capacitors derived from SPICE models

"iCD Design Integrity software features a myriad of functionality specifically developed for high-speed design."
- Barry Olney

In-Circuit Design Pty Ltd

www.icd.com.au

ZENTECH

Zentech Manufacturing is Hiring!

Looking to excel in your career and grow professionally in a thriving business? Zentech, with locations in Baltimore, MD and Fredericksburg, VA is rapidly growing and is seeking experienced professionals in all areas: engineering, manufacturing engineering, program management, testing, QA and SMT operations. Zentech offers an excellent benefit package including health/dental insurance and an employer matched 401(k) program.

Established in 1998, Zentech holds an ultimate set of certifications relating to the manufacture of mission-critical printed circuit card assemblies, including ISO:9001, AS9100, DD2345, ISO 13485, maintains an IPC 610 Trusted Source QML status, J-STD 001 with Space Certification and is ITAR registered.

Click here to apply

Click for drone video tour of Zentech Facility

ARLON

Arlon EMD located in Rancho Cucamonga, California is currently interviewing candidates for manufacturing and management positions. All interested candidates should contact Arlon's HR department at 909-987-9533 or fax resumes to 866-812-5847.

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

Our facility employs state of the art production equipment engineered to provide cost-effective and flexible manufacturing capacity allowing us to respond quickly to customer requirements while meeting the most stringent quality and tolerance demands. Our manufacturing site is ISO 9001: 2008 registered, and through rigorous quality control practices and commitment to continual improvement, we are dedicated to meeting and exceeding our customer's requirements.

Click here for more details

MacDermid PERFORMANCE SOLUTIONS

Do you have what it takes?

MacDermid Performance Solutions, a Platform Specialty Products Company, and daughter companies manufacture a broad range of specialty chemicals and materials which are used in multi-step technological processes that enhance the products people use every day. Our innovative materials and processes are creating more opportunities and efficiencies for companies across key industries – including electronics, graphic arts, metal & plastic plating, and offshore oil production. Driving sustainable success for companies around the world, and at every step of the supply chain, takes talent. Strategic thinking. Collaboration. Execution.

The people of MacDermid Performance Solutions stand united by a guiding principle: If it doesn't add value, don't do it. This belief inspires a unique culture where each team member has opportunities to imagine, create, hone and optimize. Do you have what it takes? Join our growing team of over 4,000 professionals across more than 50 countries with openings in research, finance, customer service, production and more.

MacDermid Performance Solutions and its affiliates are Equal Opportunity/Affirmative Action Employers.

Apply now!

CANDOR

Experienced Sales Reps Needed

Candor is an advanced printed circuit board manufacturer with a unique manufacturing process. We can genuinely stand out among conventional board shops due to our unique Simplified Process.

Join the Candor team, targeting rapid high quality prototype PCB manufacturing.

We are looking for industry experienced sales reps to manage current customers and find new leads in the USA. Candor is in a unique position to hire onto our team using a very generous sliding commission rate. Contact Sunny Patel for details.

Click here to apply

C A R E E R O P P O R T U N I T I E S

LENTHOR ENGINEERING
MAKING THINGS FLEXIBLE IN A RIGID WORLD®
311 Turquoise Street Milpitas, CA 95035 (Phone) 408-945-8787

PROCESS ENGINEER

Responsible for developing and optimizing Lenthor's manufacturing processes from start up to implementation, reducing cost, improving sustainability and continuous improvement.

Duties include but are not limited to:

- Participate in the evaluation of processes, new equipment, facility improvements and procedures.
- Improve process capability, yields, costs and production volume while maintaining safety and improving quality standards.
- Assist in employee training and certification.
- Work with customers in developing cost effective production processes.
- Engage suppliers in quality improvements and process control issues as required.
- Generate process control plan for manufacturing processes and identify opportunities for capability or process improvement.
- Participate in FMEA activities as required.
- Create detailed plans for IQ, OQ, PQ and maintain validated states as required.
- Participate in existing change control mechanisms such as ECO's and PCR's.
- Perform defect reduction analysis and activities.
- Participate in technology roadmap planning.
- Participate in new materials, processing or other developments as required.

Experience & Education:

- BS degree in Engineering
- 2-5 years of proven work experience
- Excellent technical skills

Competitive salaries based on experience, comprehensive health benefits package, 401(k) and Quarterly Gain Sharing bonus available to eligible employees.

Click here to apply

LENTHOR ENGINEERING
MAKING THINGS FLEXIBLE IN A RIGID WORLD®
311 Turquoise Street Milpitas, CA 95035 (Phone) 408-945-8787

SALES ACCOUNT MANAGER

This is a direct sales position responsible for creating and growing a base of customers. The account manager is in charge of finding and qualifying customers while promoting Lenthor's capabilities to the customer through telephone calls, customer visits and use of electronic communications. Experience with military and medical PWB/PWA a definite plus. Each account manager is responsible for meeting a dollar level of sales per month and is compensated with salary and a sales commission plan.

Duties include:

- Marketing research to identify target customers.
- Initial customer contact (cold calling).
- Identifying the person(s) responsible for purchasing flexible circuits.
- Exploring the customer's needs that fit our capabilities in terms of:
  - Market and product
  - Circuit types used
  - Quantity and delivery requirements
  - Competitive influences
  - Philosophies and finance
  - Quoting and closing orders
  - Bonding
- Submitting quotes and sales orders.
- Providing ongoing service to the customer.
- Problem solving
- Developing customer information profiles.
- Developing long-term customer strategies to increase business.
- Participate in quality/production meetings.
- Assist in customer quality surveys.
- Knowledgeably respond to non-routine or critical conditions and situations.

Competitive salaries based on experience, comprehensive health benefits package and 401(k) Plan.

Click here to apply

C A R E E R O P P O R T U N I T I E S

NCAB GROUP
Integrated PCB Production

Outside Sales/ Key Account Managers

Job Description: NCAB Group USA is looking to add to our existing outside sales team in various U.S. locations: Ontario, California and Itasca, Illinois. We are looking for sales professionals with at least 5 years of printed circuit board experience and/or semiconductor experience and knowledge. This is a sales position that requires the ability to convert those cold calls into high-value customer meetings.

What are we looking for?

- A "hunter" mentality
- The ability to create solid customer relationships
- A desire to excel and not settle for mediocrity
- 5+ years of experience in the printed circuit board or semiconductor industry
- An excellent ability to present your product and do the "deep dive" during your customer visit by asking the open-ended questions and identifying your customer's pain points
- The energy to sell—from prospecting to cold calls to getting the "wins"
- Knowledge of "SPIN" selling
- A college degree
- An enjoyment to travel both domestically and globally
- Must be a U.S. citizen with a valid U.S. Passport

What's next? Interested?
Then send your resume to:

Click here to apply

Visit us at www.NCABGroup.com

MicroCraft

Looking for an energetic
OUTSIDE SALES REPRESENTATIVE
Entry levels are welcomed

Job Description: Grow business by setting up meetings with customers and handling presentations, selling products, and meeting customer needs. Required to meet customers face to face on weekly basis.

Territory: All states including parts of Canada.

Skills/Qualifications: great communication skills; customer hospitality; meeting sales goals; closing skills; territory/account management; prospecting skills; negotiation; travel scheduling; computer and network skills (intermediate level); self-confidence; presentation skills; client relationships; motivation for sales; motivation to learn; loyalty to company; respect to customer and team member; patience; due obligation.

Plus Aspects:

- Bachelor or higher degree
- Experience in outside sales
- Knowledge of printed circuit boards

MicroCraft offers:

- Domestic travel allowance
- International travel allowance
- 401(k) with company match
- 90% coverage on health insurance and dental insurance for the employee
- Life, AD&D, short term and long term disability for the employee
- PTO and holidays
- Work environment/helpful coworkers
- The opportunities to expand your knowledge

Salary: $40,000 to $75,000 or more. Your salary depends on your sales performance. This figure includes sales commission.

Click here to apply

Contact: Kohei Maekawa

CAREER OPPORTUNITIES

TAIYO AMERICA

Business Development Manager

General Purpose/Objectives:
Manage key accounts and develop new business. This person will grow existing accounts and participate in new product development through market analysis to identify new customers.

Essential Duties/Responsibilities:
- Identify significant business opportunities and increase revenue
- Create business and product development plans, do opportunity assessments, initiate product kick offs, and assist with program management until product release
- Lead new product development team and define new products
- Develop product strategies, product positioning, and new product roadmaps
- Release new products to the target market
- Work with our top customers, sales and representative teams, field applications engineers, and distributor partners
- Provide technical training and support for new business
- Interface with customers and field to understand market requirements

Qualifications/Experience:
- Bachelor's degree (Scientific Studies) or equivalent with industry experience and background in chemistry and PWB product marketing
- A combination of 10 years' sales, product marketing, product management, or business development
- US citizenship and fluency in English
- Working knowledge of PWB and IC development
- Strong communication and customer skills
- Must be able to work independently with limited supervision
- Excellent interpersonal communication and organizational skills to work with clients, team members, and management staff
- Multicultural experience in Germany or Japan
- International business

Click here for more details

COAST to COAST CIRCUITS

Advanced technology manufacturer of printed circuit boards in Huntington Beach, CA is looking for manufacturer's representatives in Southern California. Please forward inquiries to Crystal Barrett. (585) 254-2989 ext. 225

Click here to apply

all flex

Flexible Circuits • Heaters • Assemblies

HELP WANTED

In order to support its continued growth and leadership position, All Flex Flexible Circuits and Heaters is seeking a TECHNICAL APPLICATION SALES ENGINEER for our inside sales team. Candidates are to have 5+ years in the PCB industry including a minimum of 2 years with flexible circuitry and/or flexible heaters. Experience in component assembly and flexible heater design is a plus.

The position involves providing extremely high responsiveness and follow-up to assigned accounts and new prospect inquiries. Although primarily an inside sales service provider, the individual must also be able to travel several times per year to support tradeshows and in-person customer support. The position provides technical application knowledge to assist customers in the design and use of flexible circuits, heaters, and assemblies, a key service that All Flex provides.

Background to include:
- Success in a team environment
- Managing large customers – ideally in medical, military, aerospace, and industrial markets
- Proficiency in Microsoft Office Suite with good typing/keyboard skills
- Attention to detail
- Good organization skills in handling multiple activities at the same time
- Professional telephone and e-mail communication skills
- Experience working trade shows
- Good listening and customer management abilities

This position is located in Northfield, MN.

Click here to apply

eptac

TRAIN. WORK SMARTER. SUCCEED.

Become a Certified IPC Master Instructor

We are growing! EPTAC, a leading provider in the Electronics Training Industry is looking for some great people to join our team. If you love teaching and working with great companies in your travels, contact us.

Click here to apply

CAREER OPPORTUNITIES

ventec INTERNATIONAL GROUP 騰輝電子

OEM Account Manager USA
(IMS Materials)

Want to work for a globally successful and growing company and help drive that success? As a US-based key member of the global OEM Marketing Team, your focus will be on IMS (insulated metal substrates) materials that deliver exceptional thermal performance, reliability and quality, particularly demanded by automotive and other LED lighting and DC power conversion applications. Combining your strong technical PCB manufacturing and design knowledge with commercial acumen, you will offer OEM customers (buyers, designers, reliability engineers and the people that liaise directly with the PCB manufacturers) advice and solutions for optimum performance, quality and cost.

Skills and abilities required for the role:

- Technical background in PCB manufacturing/design
- Solid understanding of IMS materials
- Sales knowledge and skills
- Excellent oral and written communication skills in English
- Experience in making compelling presentations to small and large audiences
- Proven relationship building skills with partners and virtual teams

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

Please forward your resume to:

Click here to apply

mention "OEM Account Manager USA" in the subject line.

SAKI The Future in Focus

Field Application Engineer

SAKI America, manufacturer of automated optical, solder paste, and X-ray inspection and measurement systems for the electronics industry, seeks a Field Application Engineer, based in its Fremont, California headquarters. Responsibilities include communicating with customers, sales representatives, and pertinent Saki personnel to understand the customers' applications, assist in equipment selection, perform equipment installation, and provide assembly process assistance, training, support, maintenance, troubleshooting, and services for Saki's products and technologies, at SAKI America's headquarters, regional offices, and customer sites. Perform product and process demonstrations and presentations to customers, agents, and colleagues, assist with trade show activities, and work with factory and development teams to communicate new product ideas and customer suggestions/requests.

Job requirements:

Two-year technical degree (four-year preferred) or equivalent experience. 3-5 years combined experience in customer and technical support with 5-7 years in SMT manufacturing process with SPI and AOI understanding. The ideal candidate will have experience running and programming SPI and AOI systems. Competencies should include excellent verbal and written communication skills, a working knowledge of computer-based business applications, understanding SPC collection and use in a manufacturing environment, problem-solving skills, use of tools such as Six Sigma, and electronics/electro-mechanical troubleshooting capability. The position requires up to 75% travel (3 weeks/month).

Click here to apply

MivaTek Global

Application/SalesEngineer

Positions available East, Midwest and Western United States. Position will focus on supporting sales and applications development for Miva Technologies DLP direct imaging system with the PCB and micro-electronics markets. Experience with photoresist and imaging preferred but not required.

Send resume and contact information for both positions to Brendan Hogan.

Service Technician

Positions available for Eastern and Western United States. Service Technicans will support our rapidly expanding installed base of Miva Technologies DLP imaging systems as well as other systems sold by the company.

Click here to apply

WANTED

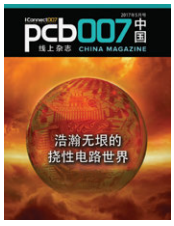
CONTRIBUTORS

Help Wanted: I-Connect007 contributors!

Are you an expert at PCB design, fabrication, assembly, or a related industry skill? Do you have a desire to share your unique perspective and knowledge? If so, we'd love to hear from you. The PCB Design Magazine, The PCB Magazine, PCB007 China Magazine and SMT Magazine are seeking feature contributors and regular columnists.

Being an I-Connect007 contributor does have its benefits: you can share your published articles from our industry-leading technical publications with your byline on websites, blogs, social media, and more.

For more information, contact our editors:
SMT Magazine, The PCB Magazine, The PCB Design Magazine, and PCB007 China Magazine.



I-Connect007
GOOD FOR THE INDUSTRY

TOP TEN

Recent Highlights from PCBDesign007

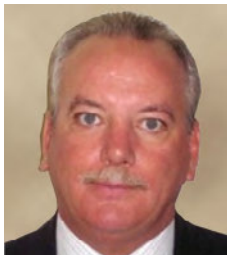
1 'Flexdude' Tom Woznicki Celebrates Company's 25th Anniversary

Twenty-five years ago, Tom "Flexdude" Woznicki got laid off. So, he launched his own flex circuit design bureau and never looked back. Since then, he's designed flex circuitry for everything under the sun, including the Mars Rover; the flex circuits he designed are visible in many of the Rover photos. Tom and I discussed the benefits of flex circuits, the expansion of the flex market, and his company's first quarter-century in operation.

2 Mentor FloTHERM XT Release Simulates Complex Geometries Quickly, Accurately

Mentor has launched the newest release of the FloTHERM XT electronics cooling software product with advanced thermal management capabilities. Based on the electronics cooling DNA of Mentor's FloTHERM technology and the FloEFD CAD-centric philosophy, the award-winning FloTHERM XT product is the industry's first integrated mechanical design automation and electronic design automation electronics cooling solution.

3 Beyond Design: Microstrip Coplanar Waveguides



The classic coplanar waveguide (CPW) is formed by a microstrip conductor strip separated from a pair of ground planes pours, all on the same layer, affixed to a dielectric medium. In the ideal case, the thickness of the dielectric is infinite. But in practice, it is thick enough so that electromagnetic fields die out before they get out of the substrate.

4 New Ultra Librarian Website Provides ECAD/MCAD Models, Part Pricing

EMA Design Automation has created a website for Ultra Librarian giving electronics designers online access to pre-authored and verified parts with pricing and technical information to help them choose the best electronics components for their designs and the ability to download symbols, footprints, and 3D models.

5 Obsessing over Conductor Surface Roughness: What's the Effect on DK?

You know you have an obsession when you are flying six miles over Colorado and you look out the window at the beautiful scenery, and all you can think about is how the rocky mountain topology reminds you of conductor surface roughness! Well, call me obsessed, because that's exactly what I thought on my way to DesignCon 2017

6 European PCB Designer Day to Focus on Critical Layout Factors

The European Institute of Printed Circuits (EIPC) and FED will be holding European PCB Designer Day at the Marivaux Hotel Congress & Seminar Center in Brussels, Belgium on June 21, 2017. Designer Day focuses on some of the more critical design factors including base material selection, performance modelling and testing, signal integrity, component layout tools and 3D structures, and embedded components.

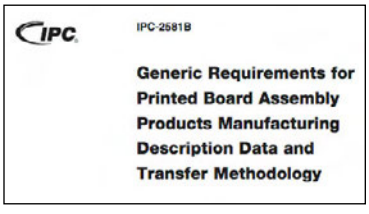
7 DesignCon Announces 2017 Awards for Best Papers

The 2017 DesignCon Best Paper Award winners have been selected. The first-round finalists were judged based on attendee feedback, collected at DesignCon 2017, on the impact of their presentation. Winners include I-Connect007 contributors Istvan Novak and Yuriy Shlepnev.

8 Bruce Mahler Discusses Ohmega's Resistive Material Technology

Bruce Mahler, vice president of Ohmega Technologies, sat down for an interview with Andy Shaughnessy at DesignCon 2017. He discussed the company's latest embedded resistive materials, as well as some of the drivers and challenges in that segment of the materials industry.

9 Wanted: Industry Input on Next Rev of IPC-2581



The IPC 2-16 technical committee is actively soliciting input from industry for the next major revision of the IPC-2581 standard. Regardless of your present IPC-2581 adoption status, we want to hear from each of vou. The objective of this next round of enhancements is to eliminate risk and inefficiency in your day-to-day operations, and streamline your production processes.

10 TTM Shines a Light on Optical Interconnect

Are embedded optics on PCBs set to make a breakthrough in the upcoming years? According to Dr. Craig Davidson, VP of Corporate Technology at TTM, it might be closer than you'd expect. Craig outlines TTM's current pursuit of high-volume manufacturing lines able to deliver embedded optical interconnect, what that would mean for the PCB industry, and why he thinks there will be optical manufacturing production capability by 2020.

PCBDesign007.com for the latest circuit design news and information—anywhere, anytime.

Events

For IPC Calendar of Events, click here.

For the SMTA Calendar of Events, click here.

For a complete listing, check out The PCB Design Magazine's event calendar.

Thailand PCB Expo 2017
May 11–13, 2017
Bangkok, Thailand

JPCA Show 2017
June 7–9, 2017
Tokyo, Japan

IPC Reliability Forum: Emerging Technologies
June 27–28, 2017
Düsseldorf, Germany

SMTA International 2017 Conference and Exhibition
September 17–21, 2017
Rosemont, Illinois, USA

electronicAsia
October 13–16, 2017
Hong Kong

IPC Flexible Circuits: HDI Forum
October 17–19, 2017
Minneapolis, Minnesota, USA

TPCA Show 2017
October 25–27, 2017
Taipei, Taiwan

productronica 2017
November 14–17, 2017
Munich, Germany

HKPCA/IPC International Printed Circuit & South China Fair
December 6–8, 2017
Shenzhen, China

DesignCon 2018
January 31-February 1, 2018
Santa Clara, California, USA

IPC APEX EXPO
February 27-March 1, 2018
San Diego, California, USA

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

EDITORIAL:

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

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

MAGAZINE PRODUCTION CREW:

PRODUCTION MANAGER: **MIKE RADOGNA**
mike@iconnect007.com

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

INNOVATIVE TECHNOLOGY: **BRYSON MATTIES**

COVER: **SHELLY STEIN**

COVER IMAGE: **©ALPHASPIRIT, ADOBE STOCK**

THE pcb design MAGAZINE

The PCB Design Magazine® is published by BR Publishing, Inc., PO Box 50, Seaside, OR 97138

May 2017, Volume 6, Number 5 • The PCB Design Magazine© is published monthly by BR Publishing, Inc.

ADVERTISER INDEX

I-Connect007

GOOD FOR THE INDUSTRY

FREE SUBSCRIPTION

Subscribe

subscribe.iconnect007.com

007 Media Kit

mediakit.iconnect007.com

EDITORIAL CONTACT

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

SALES CONTACT

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

pcb design 007 | pcb 007 | SMT 007 | mil aero 007 | flex 007 | ein 007 | RTW

www.iconnect007.com