

# THE **pcb** **design** MAGAZINE

February 2016

Technology Outlook with  
Mentor Graphics p.10

What's New at...Pulsonix,  
Zuken, Downstream,  
and Cadence

Also—  
NAMM 2016 Review p.56

an IConnect007 publication

Much More!

## WHAT'S NEW IN PCB DESIGN?



**isola**  
The base for innovation

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



*Quick Search*



*Advanced Search*



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

[www.thepcblist.com](http://www.thepcblist.com)

**ATTENTION BUSINESS OWNERS:**

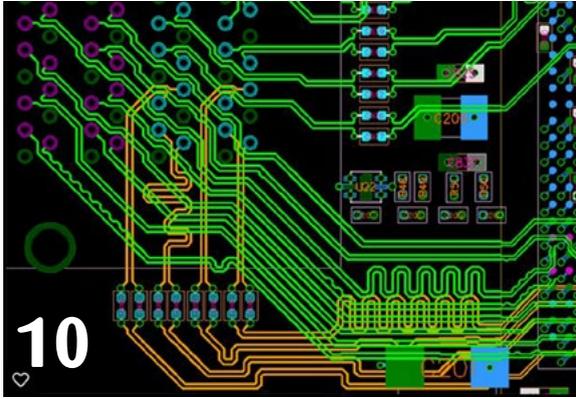
IQ  
TEST  
TODAY



BRING YOUR  
BRAIN

► **CLICK HERE TO TAKE THE BizBrain IQ TEST**





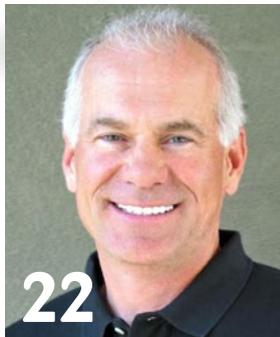
## What's New in PCB Design?

What's new in PCB design, and where is the industry headed? If you've been asking yourself the same question, you've come to the right place. In this issue, technologists from the biggest names in PCB design software tools discuss their companies' latest innovations and the trends they see going forward. Interviews include David Wiens of Mentor Graphics, Brad Griffin of Cadence Design Systems, Bob Williams of Pulsonix, Rick Almeida of DownStream Technologies, and Bob Potock of Zuken.



### 10 Technology Outlook with Mentor Graphics

*Interview with David Wiens*



### 16 What's New at...Pulsonix?

*Interview with Bob Williams*

### 22 What's New at...Zuken?

*Interview with Bob Potock*



### 28 What's New at...Downstream Technologies?

*Interview with Rick Almeida*

### 32 What's New at...Cadence Design Systems?

*Interview with Brad Griffin*



# Low-loss Laminate Materials for RF/Microwave/Millimeter-wave Designs

For over 100 years, Isola has been driving technology forward while improving end-product design with top-performing laminate and prepregs.

Our legacy continues with global R&D pushing our product line to the forefront in demanding RF/microwave designs.

With 10 manufacturing sites around the world, global quick-turn capability, world-class technical service, measurement & modeling capabilities, and industry leading material development capabilities – Isola is well positioned to be your preferred RF/microwave partner.

The newest members of Isola’s growing RF/Microwave laminate family now includes Terra Green™, a halogen-free, very low-loss, thermoset material that is available in a variety of laminate and prepreg offerings. This material is inexpensive to process – improving your company’s bottom line, as well as the environment.

Astra®MT is a revolutionary ultra low-loss thermoset laminate replacement for PTFE, suitable for automotive radar, mm-wave and AESA applications.

I-Tera MT is available in core thicknesses from 0.002” to 0.018” and offers 18 different prepregs for optimal design flexibility.

I-Tera®MT RF is available in 20 and 30 mil cores and a 3.38 and 3.45 Dk.

	TerraGreen™	Astra® MT	I-Tera® MT/ I-Tera MT RF	IS680
Tg	200°C	200°C	200°C	200°C
Td	390°C	360°C	360°C	360°C
DK @ 10 GHz	3.45	3.00	3.45	2.80 - 3.45
Df @ 10 GHz	0.0030	0.0017	0.0031	0.0028 - 0.0036
CTE Z-axis (50 to 260°C)	2.90%	2.90%	2.80%	2.90%
T-260 & T-288	>60	>60	>60	>60
Halogen free	Yes	No	No	No
VLP-2 (2 micron Rz copper)	Standard	Standard	Available	Available
Stable Dk and Df over the temperature range	-55°C to +125°C	-40°C to +140°C	-55°C to +125°C	-55°C to +125°C
Optimized Global constructions for Pb-Free Assembly	Yes	Yes	Yes	Yes
Compatible with other Isola products for hybrid designs	Yes	Yes	Yes	For use in double-sided applications
Low PIM < -155 dBc	Yes	Yes	Yes	Yes

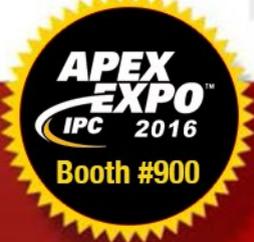
NOTE: Dk Df is at one resin %. The data, while believed to be accurate and based on analytical methods considered to be reliable, is for information purposes only. Any sales of these products will be governed by the terms and conditions of the agreement under which they are sold.



## FREE WEBINAR

### PCB Material Selection for RF/Microwave/Millimeter-wave Design

This webinar will educate you on how to select the appropriate PCB material for RF, microwave and millimeter-wave applications through careful consideration of cost constraints and system-performance parameters, such as the frequency of operation and bandwidth, the electrical size of board and its critical features, loss and uniformity requirements, temperature range of system operation, and thermal and processing considerations.



# www.isola-group.com/RF



- 36** *HIGHLIGHTS*  
**PCB007**
- 54** **MilAero007**
- 66** **Top Ten PCBDesign007**

- 68** *DEPARTMENTS*  
**Events Calendar**
- 69** **Advertisers Index  
& Masthead**

## More Content

- SPECIAL SECTION*
- 56** **NAMM 2016 Review**  
*by Dick Crowe  
and Dan Feinberg*

- ARTICLE*
- 62** **The Gerber Guide, Chapter 6**  
*by Karel Tavernier*

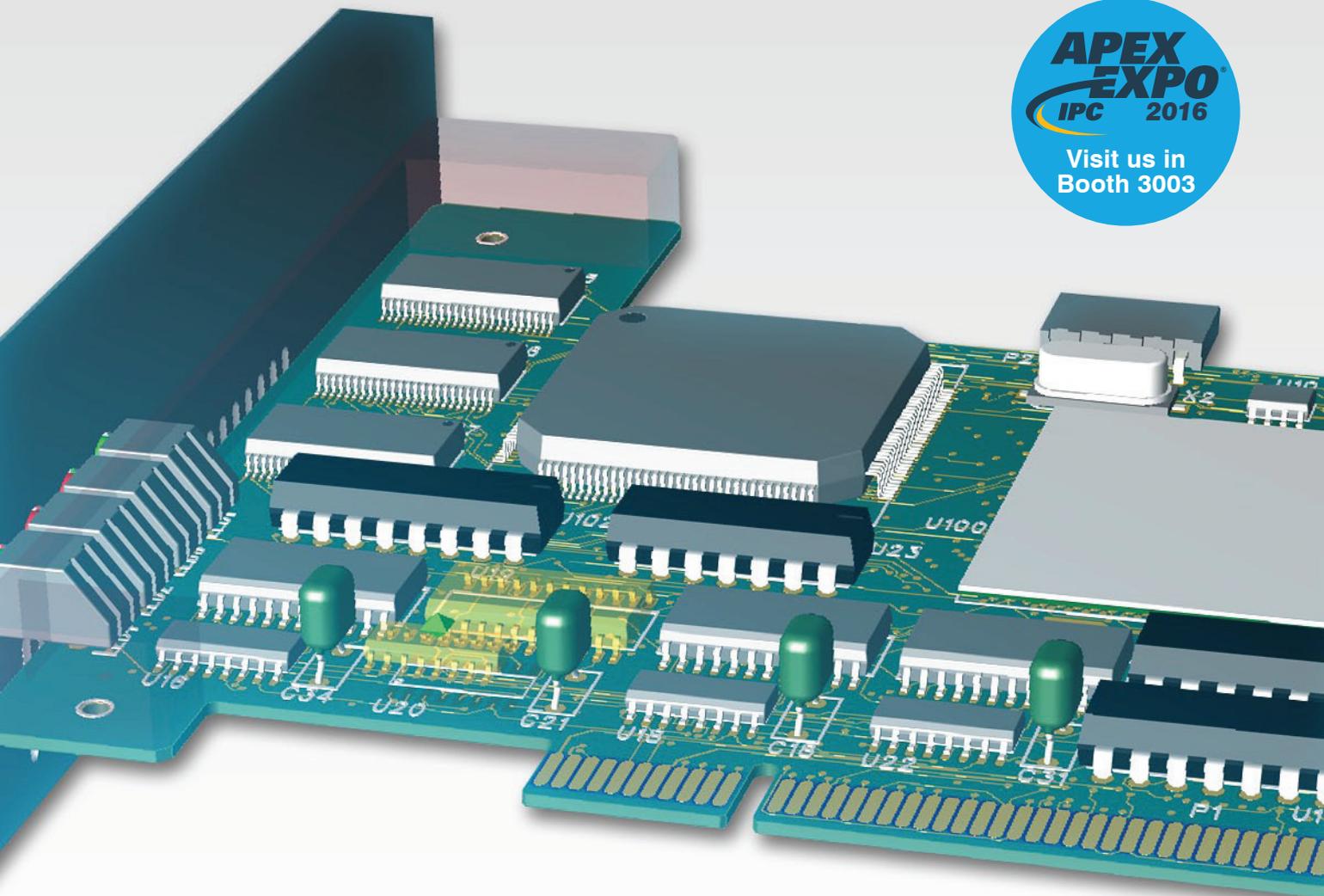
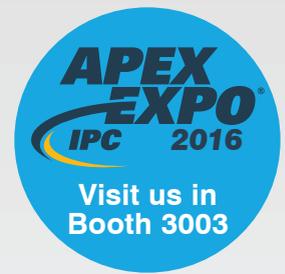
- COLUMNS*
- 8** **A Tale of Two Shows**  
*by Andy Shaughnessy*

- 38** **Faster than a Speeding Bullet**  
*by Barry Olney*

- 46** **How to Design-Out Production Problems**  
*by Phil Kinner*

- 50** **Multilayer PCB Bonding Materials for High-Frequency Applications**  
*by John Coonrod*

- SHORTS*
- 26** **Students Invent Alarm that Protects Your Unattended Bag**
- 30** **Memristors: Making a New Generation for Digital Memory and Computation**
- 34** **You'll Never Be-Leaf What Makes up This Battery**
- 44** **Scientists Bridge Different Materials by Design**



# 3D DESIGN

## More than just a pretty picture

Tackle simple or complex PCB design problems with accuracy and ease. PADS® delivers great value with constraint management, best-in-class simulation and analysis, and powerful interactive and automatic routing. New 3D design with STEP import/export provides a virtual prototype of your electronic assembly. 3D editing with DRC makes the layout process quick and efficient.

Find out more at  
[www.pads.com](http://www.pads.com)



# A Tale of Two Shows

by Andy Shaughnessy

I-CONNECT007

## DesignCon 2016

Every year, I make the trek to Santa Clara for DesignCon. Sure, it's not a "PCB show" by any means. DesignCon usually draws design software companies like Mentor Graphics, SiSoft, Cadence Design Systems, Altium, DownStream Technologies, EMA Design Automation, Simberian, and Polar Instruments.

But every year, PCB has a bigger presence at the show. There are more and more PCB fabricators and laminate suppliers on the DesignCon show floor each year, while PCB design engineering content is the focus of an increasing number of classes and sessions.

This year, DesignCon was bigger than ever. Last year, the show had to move to a new hall at the other end of the Santa Clara Convention Center, and in 2016 it filled that entire space. And unlike just a few years ago, I ran into PCB folks constantly: attendees, exhibitors, and speakers.

Why do PCB types come to DesignCon? As a variety of PCB exhibitors told me, DesignCon is where you want to be to get your product or service spec'd in early in the design process, because this is biggest meeting ground for front-end engineers.

These engineers are walking around the show floor, usually with a bad comb-over or long hair hanging down their back. (There's not much middle ground in engineering hair

styles; it's either freak or geek.) These engineers are seeking out cool new materials and ideas, and they'd love to talk about  $D_k$  and  $T_g$  with you. Every board shop and PCB laminate supplier should exhibit at DesignCon. As one new exhibitor told me, "A booth here may be expensive, but if we make one sale, it's worth it."

One hot topic throughout the week was PAM4 (four-level pulse-amplitude modulation), with nine tutorials and two panels devoted to this "hot new thing." So far, PAM4 is still hard to work with and fairly expensive, but it may provide less signal loss at high speeds and result in fewer traces on super high-speed PCBs. One panel that drew a crowd was "Measurement Challenges of PAM4 Signals," hosted by Doug Burns of SiSoft, Stephen Mueller of Teledyne LeCroy, Luis Boluña of Keysight Technologies, and Mark Guenther of Tektronix. During the show, SiSoft announced that its Quantum Channel Designer now supports PAM4 AMI models.

There were no real slow times at DesignCon this year, even on the last day. Show managers I spoke with said this year's attendance seemed well on its way to beating last year's numbers.

## NAMM Time

Next, I jumped in my trusty new Camaro rental car and high-tailed it to Anaheim for the National Association of Music Merchants show (NAMM). This is the biggest music industry show in the world, and it opened on the last day of DesignCon. Coincidence? I think not.

Have you ever driven from Silicon Valley to Anaheim? It was interesting; I've never spent an hour driving through a storm of tumbleweeds before. I'm glad I got to experience the aromatic wonder that is Bakersfield, but I think I'll fly next time. And if you're taller than 5' 8", don't ever rent a new Camaro. My knees were in my rib cage for six hours. And it's a big car! Who designed the new Camaro?



In Anaheim, I was joined by I-Connect007 contributors Dan Feinberg of Fein-Line Associates and Dick Crowe of Bürkle USA. Dan and Dick have attended NAMM for years, and they served as my guides in this monstrous show that drew over 100,000 attendees this year. NAMM takes up multiple floors in the Anaheim Convention Center. It is just huge; Dan says it's about 1/8 of the size of the CES show, but you get the idea.

I met so many innovators from companies that manufacture musical instruments and amplifiers, but one of the funkiest has to be Pat Quilter of Quilter Labs and QSC Audio Products. Pat designs his own PCBs, and he refuses to “upgrade” from his trusty P-CAD design tool. (He actually dared the EDA companies to build a PCB design tool better than P-CAD.) He wore a giant top hat like a Dr. Seuss character for the entire show, and played lap steel guitar like a session player from Bakersfield.

We also met with John Ack, an engineer with Fishman, a company known for their acoustic guitar amplification products. But now, Fishman has revolutionized electric guitar pickups, which haven't changed much since their invention 80 years ago. Pickups have always been made of magnets wrapped in fine, enameled copper wire that would “pick up” the vibrations of the strings. To change the sound, manufacturers added or subtracted wiring or adjusted the size of the magnets, period.

Fishman threw all of that out the window with the new Fluence pickup. The entire pickup is now one 48-layer PCB. Ack explained that on each layer, one trace of thin copper is etched onto a micro-thin epoxy PCB material, and 48 of these layers are then laminated into a stack and cut into the size of a traditional pickup. Fairly traditional copper vias link the layers together.

The best feature of the Fluence? Manufacturing consistency. The traditional pickup relied on hundreds of turns of wiring around magnets. Constantly changing wire thicknesses, winding machine tensions, and humidity all combined to make two of the same pickup models often sound completely different. The Fluence is a giant step in electric guitar pickups development.

Most importantly: Whoever thought a PCB would be considered a “hot new thing” ever again?



### What's New?

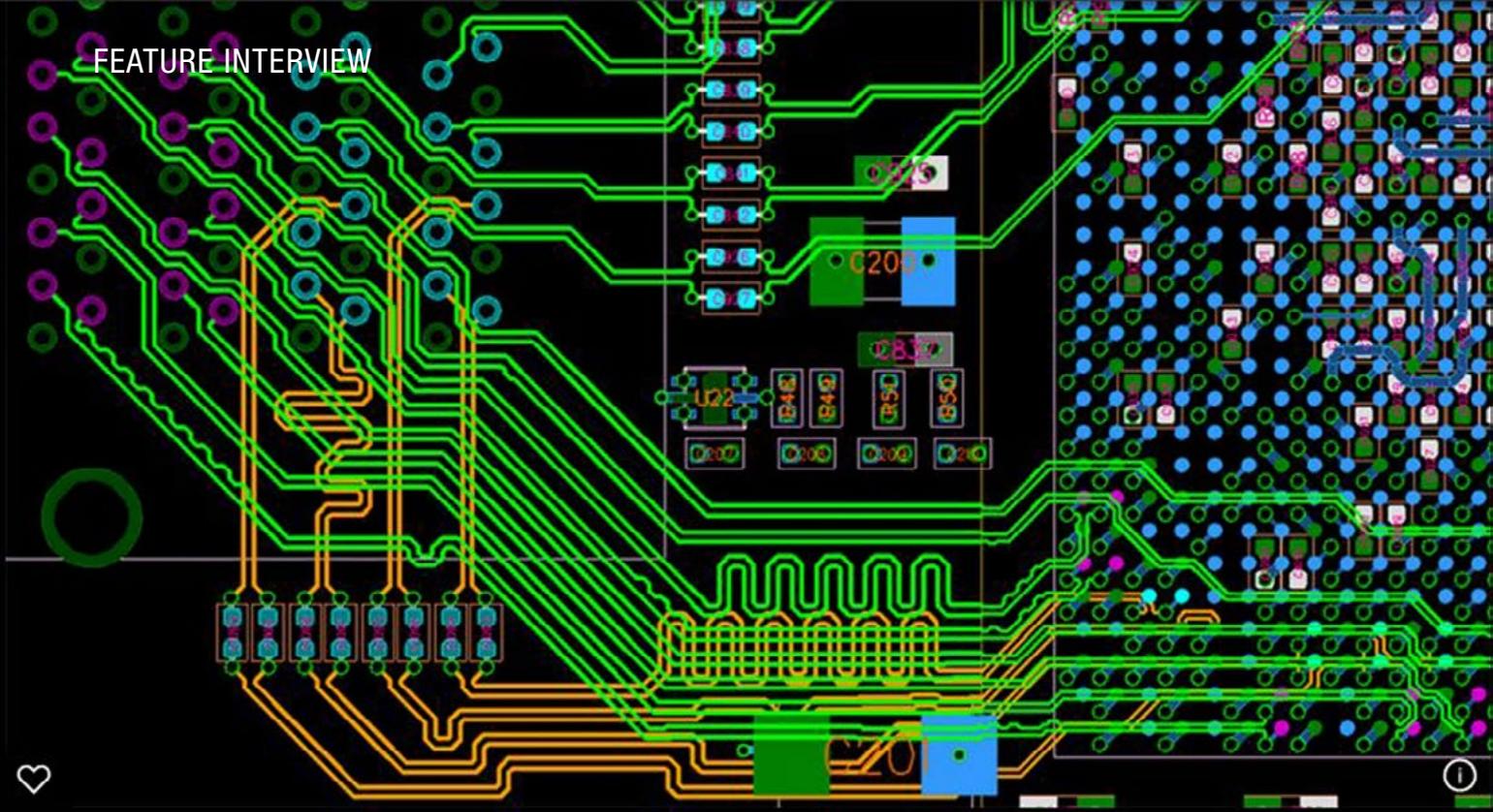
This month, we asked technologists from five companies involved in PCB design tools to tell us what's new. In our feature story, David Wiens of Mentor Graphics gives us a look at some of the trends he sees in the EDA world, based partly on entries in Mentor's Technology Leadership Awards program. Bob Williams of Pulsonix discusses Version 9 and some of its new capabilities, such as the Vault revision and version control feature.

Brad Griffin of Cadence Design Systems breaks down some of the features of the OrCAD Sigrity ERC tool, and the problems customers face with competing design requirements. DownStream Technologies' Rick Almeida discusses a post-processing tool, set for release in late 2016 or 2017, that focuses on integration between documentation and CAM file verification. And Bob Potock of Zuken delves into some newer capabilities, such as engineering data management, and the trend toward product-centric design.

And it's almost time for IPC APEX EXPO and the Design Forum, where we'll be bringing you the latest news from Vegas. I hope to see you there. **PCBDESIGN**



**Andy Shaughnessy** is managing editor of *The PCB Design Magazine*. He has been covering PCB design for 16 years. He can be reached by clicking [here](#).



# Technology Outlook with Mentor Graphics

by **Andy Shaughnessy**

I-CONNECT007

Mentor Graphics recently announced the winners of its PCB Technology Leadership Awards. Now in its 26<sup>th</sup> year, this program provides a great barometer for measuring the newest trends in cutting-edge PCB designs. I caught up with Product Marketing Manager David Wiens and asked him to give us an idea of the trends he's seeing in PCB design and manufacturing, and what the industry has in store for us in the next few years.

**Andy Shaughnessy:** *Let's start off with a look at the industry. What trends are you seeing in PCB design that you believe will continue in 2016 and beyond?*

**David Wiens:** There are several, Andy. A lot of trends are simply a continuation and expansion of what we've seen for several years. As component sizes become smaller and integration more dense, PCBs continue to pack more into

less space, resulting in less real estate for traces. Combine this with the fact that the boards themselves are getting smaller and you see the difficult challenge for PCB designers.

For perspective, over the last 20 years PCB feature sizes have shrunk ~3x while IC features have shrunk more than 40x. This has enabled integration of more and more functionality on silicon, which has in turn driven up on-board pin densities, created more diverse power requirements, and much faster signal speeds. So while the spotlight of the industry remains on silicon, the challenges with board design increase in lockstep.

**Shaughnessy:** *So, PCB designers have to deal with this increasing complexity, while still controlling costs and facing a shortening design cycle. How do designers do it?*



David Wiens

# Getting all the pieces to line up correctly can be hard.



## Problem Solved.

With over 35 years of experience and key partnerships around the world, we have what it takes to make your next project on time, on budget and easy for you.

HOW WE MAKE IT EASY 



**NUJAY** TECHNOLOGIES INC.

Your Global Connection For Interconnects

[www.nujaytech.com](http://www.nujaytech.com)

**Wiens:** A lot of design today boils down to trade-offs. There are trade-offs between performance and cost (e.g., more layers for efficient shielding or power distribution); between performance and manufacturability (e.g., extra steps for via back-drilling); between form factor and thermal management; and of course between product optimization and the ever-present ticking clock; I could list off at least another 20. All of these require the design team to review their options and pick the most efficient path forward. As design tool vendors, we assist in this process with tools that enable the designers to quickly evaluate their options (e.g., look at how different BGA fan-out schemes impact power distribution; or how device vendor guidelines for stack-up, trace geometries/topology can be bent to still meet performance requirements at lower cost).

Resolving these trade-offs also requires collaboration between multiple disciplines (e.g., between electrical and mechanical; PCB engineering and layout; layout and manufacturing; IC, package and PCB design; or SI specialist and PCB engineering). Naturally, collaboration on trade-offs takes time, so here again design tools can assist by streamlining collaboration efficiency (e.g., dynamic, incremental feedback between electrical and mechanical; or multiple engineers or multiple layout designers working concurrently on the same design in real time), or through integration that enables an individual to evaluate their options before needing to

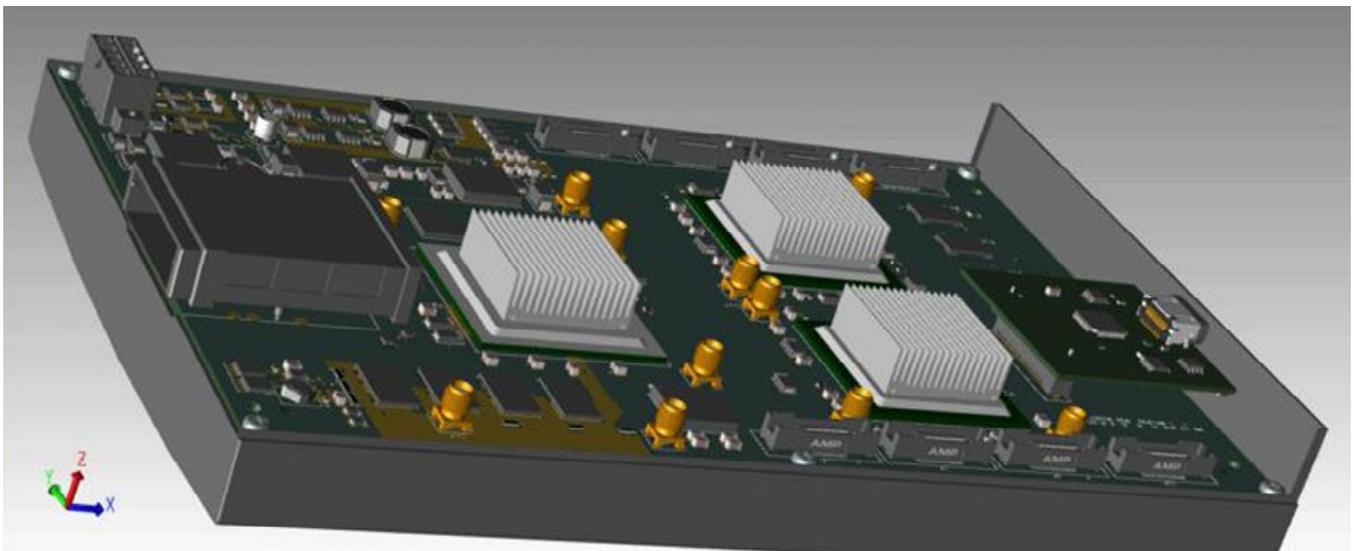
collaborate (e.g., a design engineer running full-board SI screens before needing to review in 3D with the SI specialist; or a layout designer running DC drop analysis to optimize plane area instead of a more extensive PDN review with a specialist).

**Shaughnessy:** *You mentioned simulation. Are you seeing more people simulating?*

**Wiens:** One of the largest opportunities to shorten design time, as well as increase quality, is simulation. There are tools available to allow designers and engineers run simulations before the first component is placed, right through manufacturing handoff.

Simulation used to require signal, power, or thermal specialists to interpret the data and apply it to the design. Now, tools provide simplified interfaces and wizards that enable simulation anytime in the design.

Each year, we hold our PCB Technology Leadership Awards, which give us an awesome view into trends in design. We've been watching a steady increase in simulation. It's done on more disciplines, more often throughout the design process, and it's done by more people on the team. This year, over 80% of the boards were designed for signal integrity and manufacturability (it was over 90% on the winning designs), and 70% designed for power integrity. The number of teams that used analysis vs. conservative constraints or rules of thumb was



# Project Deadlines Keeping You up at Night?



Can't sleep? Trying to figure out how you're going to meet your deadline? Let Freedom CAD Services' experienced team of engineers help to provide the expertise to get your projects done on time and under budget!

## *World-Class PCB Design and Engineering Solutions*

- Complete Engineering Project Management
- Circuit / Schematic Design & Analysis
- Power / Voltage Drop Analysis
- Component Analysis / Evaluation
- Signal Integrity Simulation & Analysis
- Mechanical Engineering
- PCB Layout Team Available 24/7
- Largest Privately Owned, Independent Layout Service Provider in the USA

cā dence®

Mentor  
Graphics®

Altium

pads

ANSYS

daTools  
design easy



EXCEPTIONAL CIRCUIT BOARD  
DESIGN & ENGINEERING SERVICES

[www.freedomcad.com](http://www.freedomcad.com)

REQUEST A QUOTE



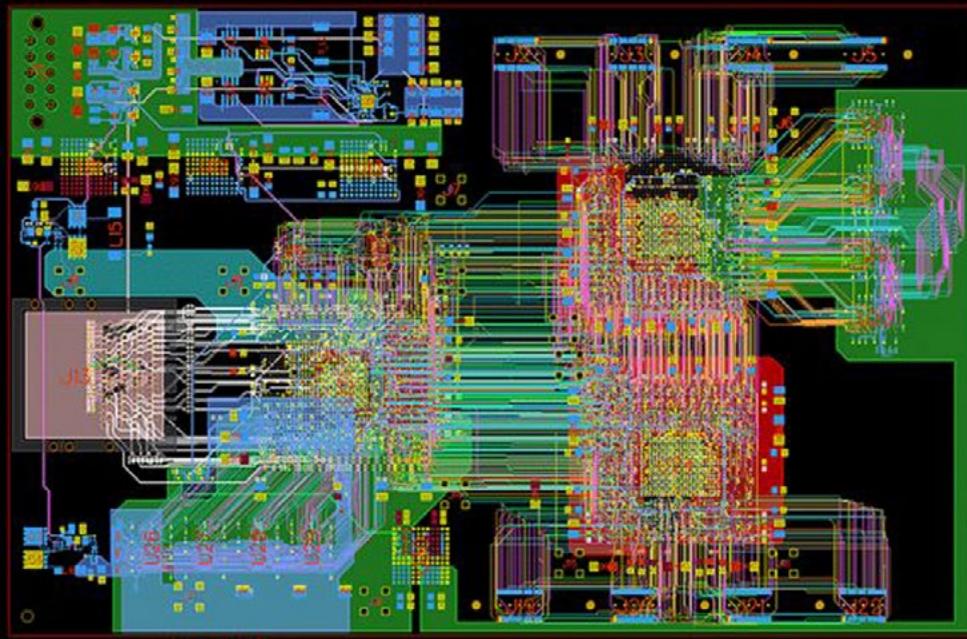
**Freedom CAD Services, Inc.**

HQ, Nashua, NH (603) 864-1300

Sales: • (714) 480-7600 Tustin, CA

• (408) 529-0563 San Jose, CA

• (603) 864-1342 Windermere, FL



within 4% (e.g., 83% designed for SI, and 79% simulated for it). So teams are biting the bullet to do more upfront planning, as well as validation during the design. In return, the number that hit the magical “first-time-right” goal increased, reducing prototype iterations, and significantly reducing the overall design time. So simulation has not only become a best practice, but commonplace in today’s designs.

**Shaughnessy:** *So designers and engineers will be able to run simulations without an expert in the field?*

**Wiens:** Absolutely. While we first think of signal integrity, power integrity is very important as I mentioned earlier. We also have tools for thermal simulation and airflow simulation. The latter has really made an impact on the mechanical design and we’ll be seeing more and more airflow simulation use in the future too.

Another is related to “Lean NPI.” Rather than have the NPI engineer or—worse yet—the fabrication house alter the PCB design to meet their standards, the product can incorporate the required fabricator constraints that become rules that are validated during the design.

**Shaughnessy:** *What do you see happening in other areas of PCB design and manufacturing?*

**Wiens:** Another emerging trend is to treat a design as a system, not a collection of boards, packages, connectors, and even cabinets. This allows a design to more easily be modified, if and when changes are required. To be clear, engineering teams have always designed systems—boards have always gone into enclosures, daughter cards have always plugged onto mother boards, and ICs have always gone into packages which have gone into boards. What’s different is the drive to break down the “black boxes” of each discipline which define and restrict how each component in the system interface with each other. This enables teams to collaborate more efficiently and optimize the entire system, rather than just their specific design component.

The best example is the developing collaboration among chip, package and board designers and vendors. By all three of these disciplines working together, they are able to optimize interconnect from the IC, through the package, onto the board, maximizing performance while minimizing form factor and cost.

**Shaughnessy:** *Do you have anything else to add, David?*

**Wiens:** I’d just like to conclude by saying that I think we’re in for some really exciting times in the next few years. Simulation has really reached the point where it is a best practice with many companies. Increased density and complex circuits have spawned some really incredible tools that will only get better. And the 3D trends in IC and IC packaging present some real challenges to PCB design as well as manufacturing.

**Shaughnessy:** *Thanks, David.*

**Wiens:** Thank you, Andy. **PCBDESIGN**

# 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

[Tour our Facilities](#)

[View our Capabilities](#)



MANUFACTURERS OF QUALITY PRINTED CIRCUIT BOARDS

The Absolute Best Value in  
High Technology Circuit Boards

847.891.5800 • [www.eagle-elec.com](http://www.eagle-elec.com)



Bob Williams

## ...at Pulsonix

by **Andy Shaughnessy**

I-CONNECT007

Since its inception in 2001, the UK-based Pulsonix PCB design tool has become popular across the pond, and parent company West-Dev plans to expand its market share in North America. I caught up with Bob Williams, marketing director of Pulsonix, and asked him to discuss some of the company's newest tools and technologies.

**Andy Shaughnessy:** *Tell us about some of the new PCB design tools at Pulsonix and some new technologies that you're excited about.*

**Bob Williams:** In Q2 of 2016, we will be releasing Version 9 of Pulsonix. The focus in this release has been on two key areas of the product: the interactive high-speed design capability and the addition of a Vault for revision/version control of Pulsonix-related documents. These additions have come directly from listening to our customers and understanding their changing needs. Version 9.0 is the first in a series of large developments for the product which will further accelerate our progress in the USA as well as other markets.

Looking ahead, we have new technologies in the pipeline which will benefit our users and significantly move the product on in terms of its underlying structure and capabilities.

**Shaughnessy:** *What's new in the Pulsonix tool?*

**Williams:** We have quite a few high-speed design enhancements. Engineers will benefit from the expansion of the constraint rules manager within Pulsonix, which incorporates a higher level set of complex rules for today's high-speed designs such as DDR3 and DDR4.

Using new intelligent interactive dialogs, users can define paths that rules will be applied to, ensuring length matching across multiple signal paths. Although these rules can be complex in their nature, the Pulsonix rules manager enables rules to be easily created allowing users to dramatically reduce creation time by applying one rule to multiple nets. Features to interactively create complex signal paths, differential pair chains, matched differential pairs and branched net rules have also been added.

A new dynamic serpentine tool has been added to interactively apply serpentine rules to existing tracks. Rules controlling the amplitude, separation, skew and gap can be con-



# ULTRA HIGH THERMAL PERFORMANCE



***Introducing SH-260, a new Polyimide system with  $T_g > 250^\circ\text{C}$  (TMA)***

***SH-260 IS IDEAL FOR:***

- Burn-in boards***
- Down Hole***
- Aircraft and Aerospace***
- Ultra-high count layer boards***
- Other PCB high temperature/long-duration applications***

Visit us at  
**IPC APEX**  
Booth 741

**CLICK HERE TO  
LEARN MORE** 



Photo Chemical Systems

[www.photochemicalsystems.com](http://www.photochemicalsystems.com)

**PARAMOUNT**

[www.paramount-usa.com](http://www.paramount-usa.com)

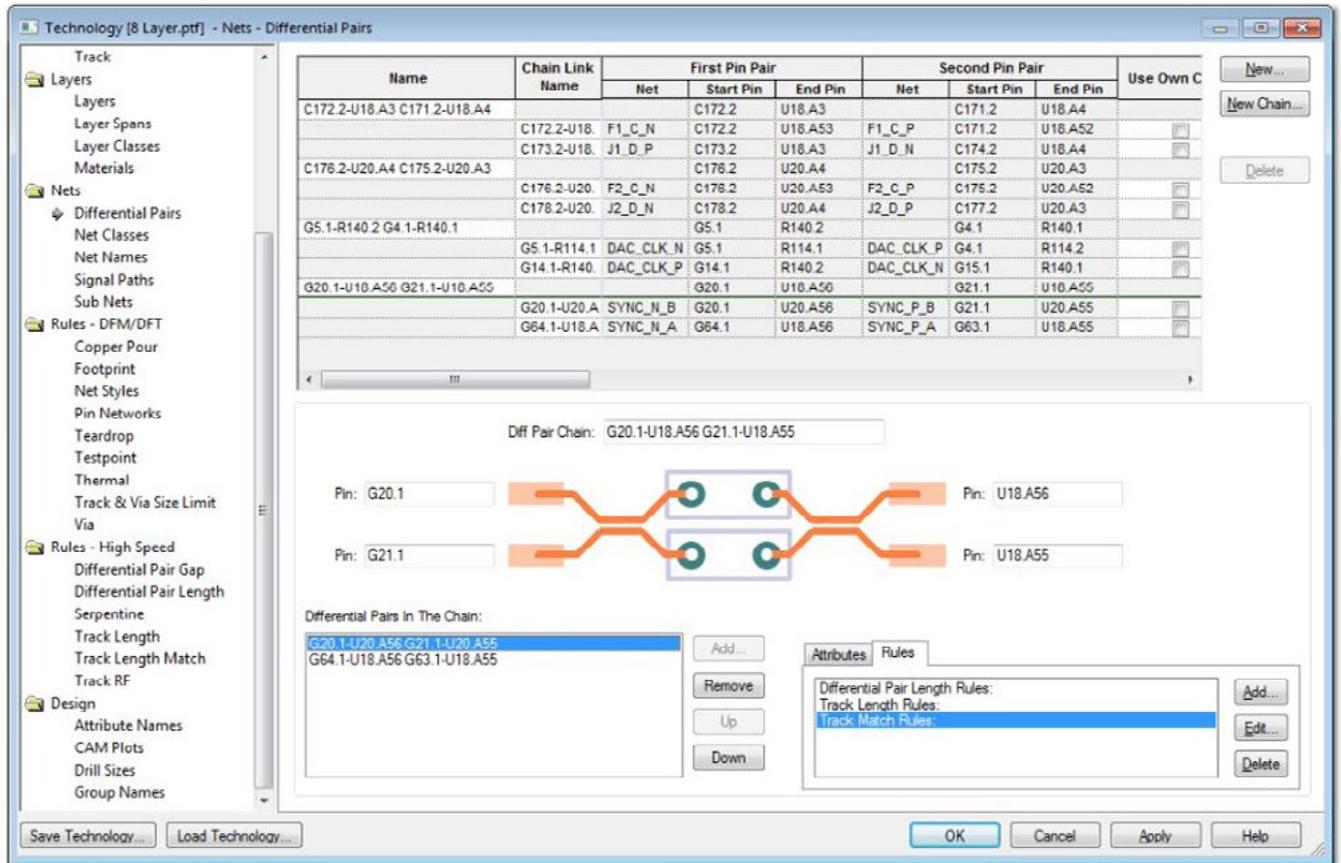


Figure 1: Technology dialog detailing a differential pair chain set-up.

trolled during routing to create precise routing patterns.

Also new is the Pulsonix Vault, a revision and version control feature used within the Pulsonix design editors and library management systems. All items processed through this mechanism are immediately traceable and all movements recorded in a detailed audit trail. With all items now in this controlled environment engineers are able to dramatically reduce design creation time with the knowledge that they are using certified items.

The Vault system can be located locally or remotely with easy access for all users. It offers revision control and versioning of design items, including schematic designs, hierarchical blocks, PCB designs, libraries and technology files. There's full traceability and an audit trail of all Vault items, with checking, verification and reporting of library items used in designs. It also offers system administration tools for Vault

management. The Vault will be provided free of charge to all Version 9.0 users.

**Shaughnessy:** Which customer challenges led you to develop these new products?

**Williams:** Like all vendors, our customers are designing leading-edge technology products as well as “everyday” designs. They demand a product from us that enables them to produce designs with the least amount of effort with the maximum amount of efficiency.

Our latest product is a development based on customer feedback and market trends. Having a very close relationship with our users has been our company ethos from the very start and one we are particularly proud of.

With Pulsonix being adopted by larger organisations, the need for more accountability was required. The Vault enables an audit trail of key design and library movements to be man-



# Engineering And Providing Balanced Interconnect Solutions



Miraco, Inc. • Manchester, NH • [miracoinc.com](http://miracoinc.com) • 603-665-9449

*...connecting technology*

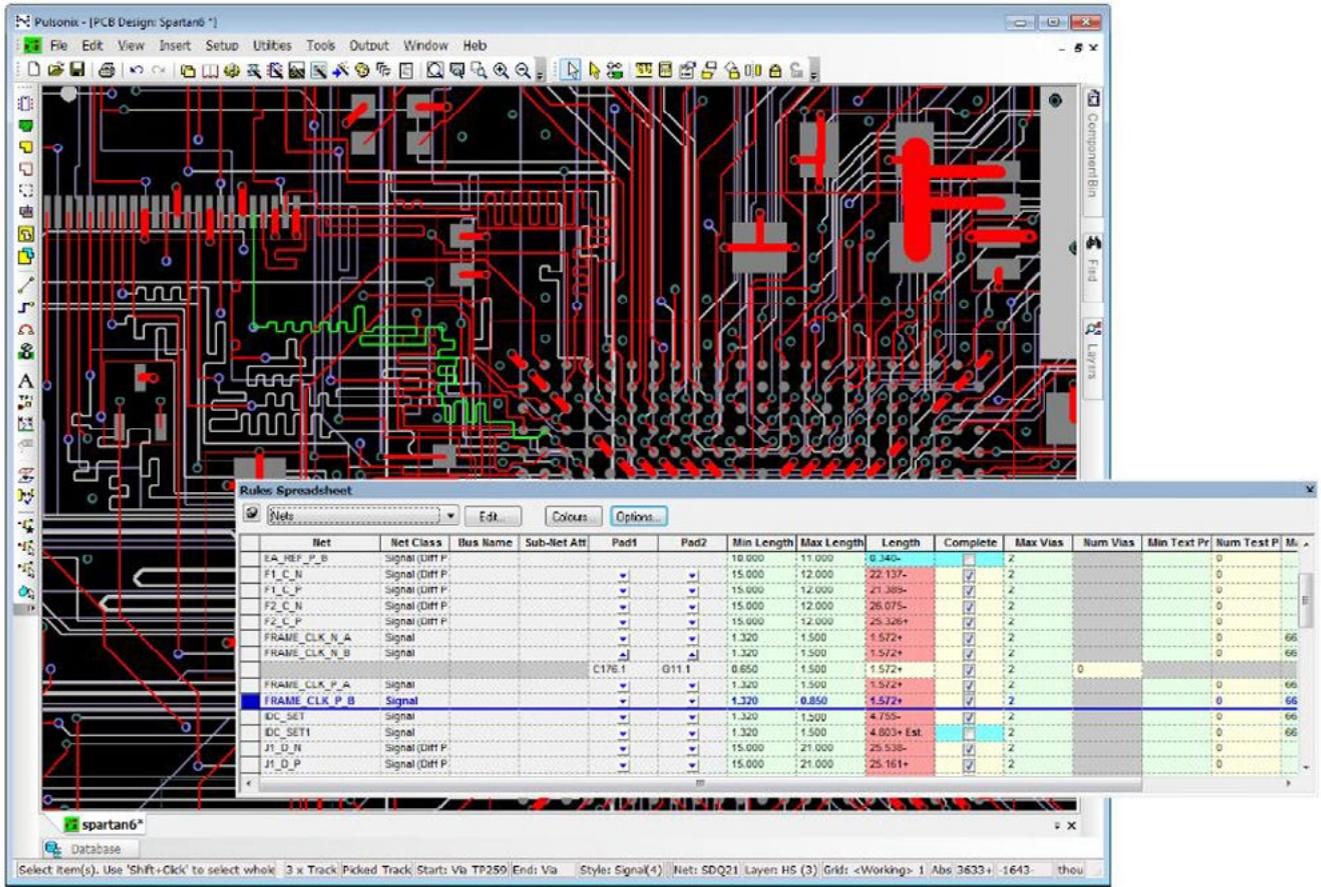


Figure 2: Pulsonix features high-speed routing and a constraint manager.

aged and recorded. It will allow this extra level of accountability to be used where required. Because of its flexibility, Pulsonix users who do not require a Vault facility won't be forced to use it.

We are seeing more and more high-speed designs being created with Pulsonix. In order to maximize its potential, the constraint manager has been significantly enhanced with additional complex rules to aid in the setup. New design capabilities facilitate the routing of these layouts. The new rules form a critical part to ensure the nets and differential pairs match across nets, fundamental in high-speed and DDR routing.

**Shaughnessy:** *What sort of trends do you see in the PCB design tools market?*

**Williams:** We see a future where core competencies in the schematic and PCB design editors

are still paramount but where closer integration with other tools is a necessity. With the Internet of Things being one of the emerging trends, this close integration is still relevant.

What is evident is that the engineers' productivity through any system has to be as painless as possible. Product tool quality, ease of use, fast start-up times and recurrent use for occasional users are also very high on the list. Engineers are becoming more multi-disciplined across tasks that have traditionally been single specialist disciplines, so picking up a tool and remembering how to use it is essential. It's going to be about the usability of the product as well as features and the aftercare service that is provided.

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

**Williams:** Thank you, Andy. **PCBDESIGN**

# Delivering the highest quality standard for **Aerospace and Defense**



Visit us at  
**IPC APEX 2016**  
Las Vegas, NV  
Booth 827

## **Ventec Europe Accredited to AS9100 Revision C**

We are proud to announce that the quality management system at our Leamington Spa, UK, headquarters is now fully accredited to AS9100 Revision C (the two facilities of our parent company, Ventec Electronics Suzhou Co Ltd, have been fully AS9100C certified since 2012).

AS9100 is the quality management standard specifically written for the aerospace and defence industry, to satisfy authorities such as the Federal Aviation Administration, ensuring quality and safety in the "high risk" aerospace industry.

**MORE INFO:**

**POLYIMIDE & HIGH RELIABILITY  
FR4 PRODUCTS**

Wherever technology takes you,  
**we deliver.**



**ventec** 

Ventec Europe  
[www.ventec-europe.com](http://www.ventec-europe.com)

Ventec USA  
[www.ventec-usa.com](http://www.ventec-usa.com)

Ventec International Group  
[www.ventecamines.com](http://www.ventecamines.com)

# What's new?

Feature Interview with



Bob Potock

## ...at Zuken

by **Andy Shaughnessy**

I-CONNECT007

Zuken has developed some innovative PCB design tools in the past few years, and I wanted to find out more about the company's new and upcoming technologies. I caught up with Bob Potock, vice president of marketing for the Americas for Zuken USA, and asked him what was new at Zuken.

**Andy Shaughnessy:** *Tell us about some of the new PCB design tools at Zuken.*

**Bob Potock:** Let's talk about System Planner for hardware architecture design and validation. System Planner is part of the CR-8000 product-centric design platform. As products become more complex, the need for architecture design and validation is growing. Architecture validation sits between product requirements and detailed design. Today, many companies make the leap from requirements to detailed design; years ago that might have worked, but not today. Zuken's System Planner product provides functional, PCB planning, space and parametric

design and validation. Functional design using blocks can be easily mapped back to requirements. And all those architecture decisions roll right into the detailed design tools with the push of a button.

Engineering data management (EDM) is all about data integrity and decision-making in the design process. Supply chain connectivity, dispersed design teams, modular design and version control are examples of competitive requirements. DS-2 Espresso brings these capabilities to the small and medium business segment. A deployment based on configurability and not customization makes deployment fast with all the benefits of an EDM solution.

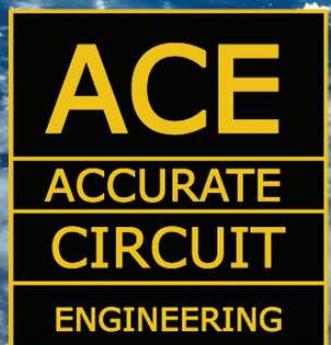
**Shaughnessy:** *What new technologies are you all excited about?*

**Potock:** Rigid-flex designs are entering into more conversations. Many of our customers' products contain multiple PCBs that traditionally required connectors on the rigid PCBs mating to connectors on separate flex PCBs. The ability to design the rigid and the flex as one eliminates the need for mating connectors,

**When failure is not an option, don't  
leave your high tech PCBs to chance.**

**Have an ACE up  
your sleeve!**

- Specializing in high technology quick turn prototypes, as fast as 24 hrs
- MIL-PRF-55110, MIL-PRF-31032A, AS9100, and ITAR certified
- Leading experts in RF, Hybrid PTFE/FR4 and Antenna/Microwave



[www.ace-pcb.com](http://www.ace-pcb.com)

[sales@ace-pcb.com](mailto:sales@ace-pcb.com)



and it's less prone to errors.

System Planner contains both 2D and 3D planning tools that can be applied to rigid-flex designs. In the 2D space, System Planner can help manage component distribution on the rigid PCBs, while in the 3D space, System Planner can ensure the rigid-flex circuit will fit within the proposed enclosure.

**Shaughnessy:** *Which customer challenges led you all to develop these new products?*

**Potock:** Companies need to build more competitive products faster with lower risk. But the process gap between the product requirements and detailed design creates new products with less-than-optimal hardware architectures. Many companies are entering detailed design without validating whether or not the hardware architecture meets both the functional requirements and the cost, weight, power and packaging requirements. An architecture problem discovered in the detailed design phase in many cases is unrecoverable. In a recent survey, we found about 50% of the respondents experienced an architecture flaw during detailed design. A poor hardware architecture ultimately impacts product competitiveness and ROI.

Another challenge we hear about more and more often is the need to be more closely connected to the supply chain. Today's poor connection to the supply chain may cause a lead-time change to be missed delaying a new product introduction. That connectivity will lead to better component selection process, awareness when a component commerce parameter changes and a higher quality BOM.

**Shaughnessy:** *How do these new tools help address your customers' concerns?*

**Potock:** System Planner fills the gap between product requirements and detailed design by providing a hardware architecture design and validation solution. Starting with a functional design built upon reuse modules from your corporate library, a previous generation design or a block with some parts in it, the result is the architectural view of your hardware. The blocks can then be partitioned across multiple PCBs and surface coverage measured. The number, size and shape of the PCBs can be manipulated as well. The space check brings in the enclosure from the MCAD designer to make sure we have a good fit, and the parametric check validates cost, weight, etc. against the requirements. At this point you have validated the design against the requirements and can visualize the product hardware. The design team can have confidence in their architecture as they enter detailed design.

DS-2 Espresso builds that connection to the supply chain and at the same time, improves data integrity throughout the design process. Capabilities like design traceability and where-used are just examples of the benefits an engineering data management solution can bring. What's unique about DS-2 Espresso is that it is designed for configuration and not customization, making it a perfect fit for a broad range of companies.

**Shaughnessy:** *What sort of trends do you see in the PCB design tools market?*

**Potock:** There are a couple of notable trends emerging in the design community. Companies are talking more about a multidiscipline design methodology that we refer to as "product-centric" design. Product-centric design is the next evolutionary step from the current 25+-year-old method of PCB-centric design. Unlike 2D PCB-centric, product-centric includes native 3D, product-level hardware architecture validation, multi-board design, ECAD/MCAD co-design, chip/package/board co-design and product visualization. Another important element of product-centric design is the vertical connectivity from the supply chain through the design elements to manufacturing. A product-centric design approach builds better products faster and with lower risk.

# If you can dream it, we can help you build it.

We provide all the services needed to take your idea into production:

- Board Design
- Product Development
- Embedded Software
- IOT Solutions
- FPGA Design
- Test Engineering

...and deliver them 40-50% less expensively than other service providers! You can use those savings to build the physical product.

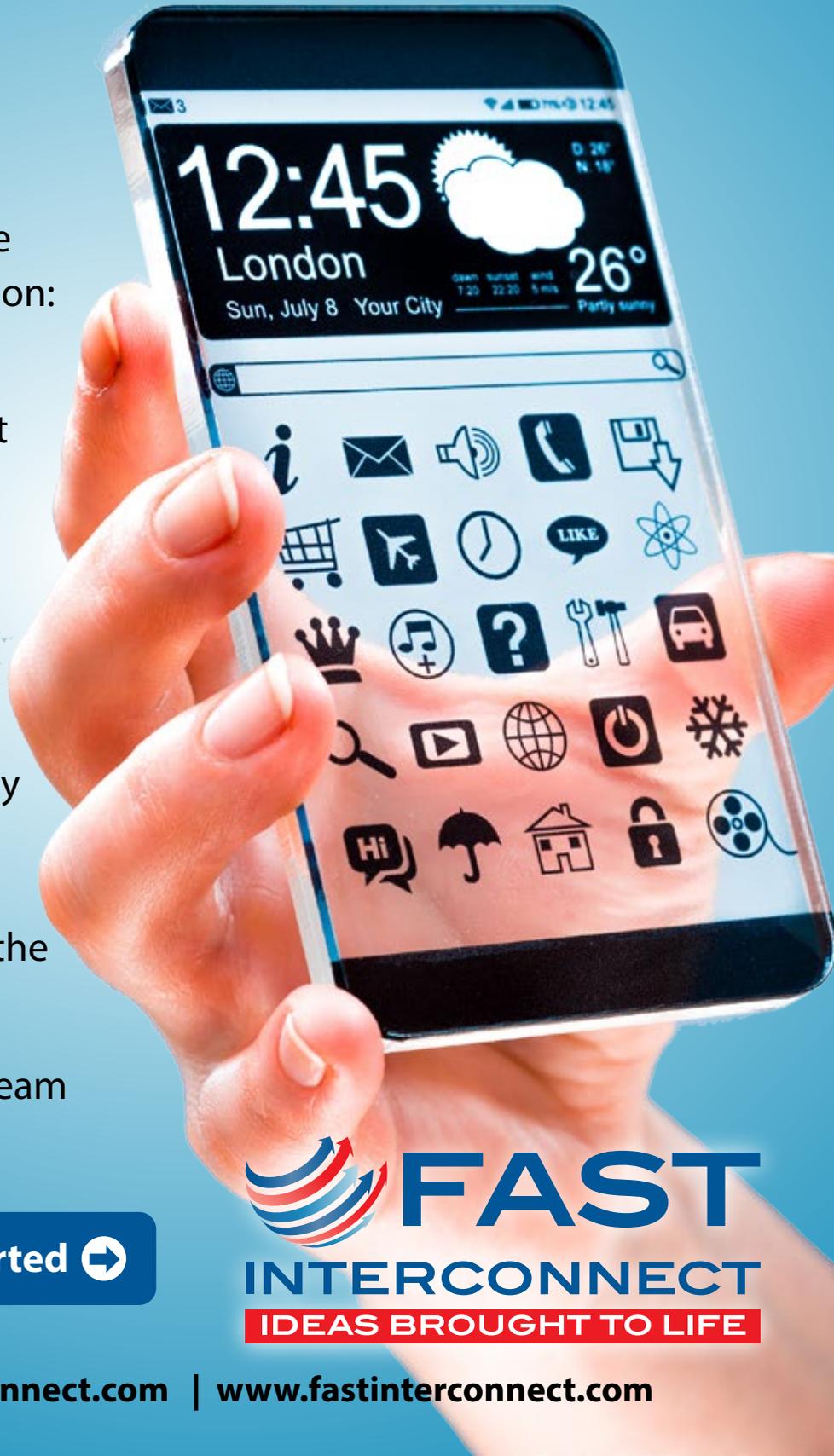
Ready to make your dream a reality?

[Click here to get started](#) 



**FAST**  
**INTERCONNECT**  
**IDEAS BROUGHT TO LIFE**

[sales@fastinterconnect.com](mailto:sales@fastinterconnect.com) | [www.fastinterconnect.com](http://www.fastinterconnect.com)



A second notable trend is the emergence of modular design. Companies are looking to build new products based on trusted functional blocks as opposed to individual components. Imagine that a team of engineers builds functional or modular building blocks and another team assembles the blocks at the product level. This revolutionizes the design process with the benefits of reusability, operating at the functional level and the incorporation of the numerous reference designs being offered by the industry. Taking it a step further, many companies are standardizing on platforms that encompass the hardware and software; new products are simply extensions to the standardized platform.

Finally, we are seeing early signs that IC packaging or multichip modules (MCM) are becoming more mainstream. MCMs have been around for many years but were too expensive for a mainstream product company. Today many silicon vendors are offering bare die and companies like Zuken are offering the capability to package them. The benefits are enormous when you consider size reduction and reliability improvements. And you also get the benefit of a physical “module” that can be mounted on the board as opposed to multiple chips.

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

**Potock:** Thank you, Andy. **PCBDISIGN**

---

## Students Invent Alarm that Protects Your Unattended Bag

Leave your bag unsupervised without running the risk of thieves stealing it? That could be the case as a unique idea of a small, high-tech alarm from Master's students at Lund University is now coming to life.

“I was by myself on a beach in Mexico and had to leave my bag unattended to take a swim. But while in the water I just couldn't relax, as I was constantly worrying about my things,” says Andrew Lentz, Master's student in Entrepreneurship and Innovation and one of the students behind the innovation.

“It was right then and there that I started to think of a solution to this dilemma, of always having to worry about your personal belongings getting stolen.”

In cooperation with Bo Möller and Jiang Qian, both Industrial Design programme students, the group thought of a way to construct an alarm that through using Bluetooth technology recognises the position of the owner via their smart-



phone or wearable.

“If your bag is moving and you aren't—indicating that someone else is taking it—the alarm will go off and won't stop until the bag is dropped,” says Andrew.

The same technology is also used to send push notifications, so if you are leaving your bag behind, you will still know if it is kept safe. The alarm in its current form,

looking very similar to a USB-stick, can be placed inside or outside the bag through a special hook-function.

“You can also put the alarm on “do not open” using your personal code and that way if thieves open and ransack your bag without it being moved, the alarm will also go off.”

The alarm was designed after a lot of research about thieves' behaviour and ways of operating.

The group has started a company called Serenity and launched a Kickstarter campaign for the product.

# 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](http://www.uscircuit.com)





Rick Almeida



## ...at DownStream Technologies

**by Andy Shaughnessy**

I-CONNECT007

Since its founding in the uncertain days of 2002, DownStream Technologies has made a name for itself with its line of PCB design post-processing tools. Founder Rick Almeida gives us an update on the latest innovations at DownStream, and he discusses some of the challenges and trends he sees in the PCB design segment.

**Andy Shaughnessy:** *Rick, why don't you start by telling us about some of the new tools at DownStream.*

**Rick Almeida:** The last few releases of our post-processing tools have been focused in the areas of assembly panel design and DFM. We've added large number of new checking routines in DFM.

We now have a major release in the works planned for release in late 2016 or early 2017 that focuses on integration between documentation and CAM file verification. This release

will modularize our products so that they may be mixed and matched together to create a custom PCB post processing solution based on our customers' requirements. As part of this release we will be bringing out some new tools in support of IPC-2581 PCB stack-up visualization in both DFMStream and our BluePrint documentation tool. We are also investing in re-writes of some of the core CAM350 functions.

**Shaughnessy:** *What new technologies are you particularly excited about?*

**Almeida:** We are in the process of redefining our panel design and documentation capability. We've had a lot of customer input on panel design for both assembly arrays and PCB production and our bringing our panel capabilities together in a new unified panel design tool that understands the various hierarchies associated with PCB panels.

**Shaughnessy:** *Which customer challenges led you to develop these new products?*

# FREE

## OrCAD Capture

### Start Your Next Project The Right Way

Don't let budget concerns keep you from starting your next project with the most popular schematic capture tool on the planet. Get access to the fully scalable PCB design solution that can grow with your business.

[Find Out How >](#)

**OrCAD**<sup>™</sup>  
CADENCE PCB SOLUTIONS

**EMA** | Design  
Automation®

**Almeida:** When engineering groups array PCBs for assembly panels, it's important that they get optimum yield. To do this they must analyze the fit of the assembly panel on the fabrication panel. So the relationship between a one-up PCB, assembly array, and fabrication panel is paramount to determine which trade-offs to make to get the most PCBs on a single panel substrate.

**Shaughnessy:** *How will these new tools help address your customers' concerns?*

**Almeida:** What we see is a tight coupling of manufacturing know-how embedded into software to help PCB designers and engineers make the right decision for optimum manufacturing execution. So the more manufacturing assistance our tools can provide earlier in the PCB design flow, the better for our customers in preparing the designs to be physically built.

**Shaughnessy:** *What sort of trends do you see in the EDA and PCB design tool market?*

**Almeida:** We see a couple of trends actually. With the proliferation of IoT devices hitting the market, flex/rigid-flex and embedded component technology is moving from a few large OEMs into the mainstream part of the market. We see this trend driving new development over the next years. IoT is also making DFM verification paramount as these products tend to supersede themselves in the market very quickly, leaving very small windows for OEMs to realize return on investment for their products. The only way these companies will be successful is by reducing the amount of time and iterations spent in PCB manufacturing. The only way that will be accomplished is by further pulling manufacturing know-how into design engineering. So we expect that the PCB design market will continue to see more development in back-end new product introduction capability.

**Shaughnessy:** *Thank you, Rick.*

**Almeida:** Thanks for the opportunity. **PCBDESIGN**

---

## Memristors: Making a New Generation for Digital Memory and Computation

Memristors are a new class of electrical circuits—and they could end the silicon era and change electronics forever. Now, researchers at Michigan Technological University have made an ideal memristor based on molybdenum disulfide nanosheets.

Yun Hang Hu, the Charles and Carroll McArthur Professor of Materials Science and Engineering, led the research, which was published in *Nano Letters* this January.

Transistors based on silicon, which is the main component of computer chips, work using a flow of electrons. If the flow of electrons is interrupted in a transistor, all information is lost. However, memristors are electrical devices with memory; their resistance is dependent on the dynamic evolution of internal state variables.

"Memristors can be used to create super-fast memory chips with more data at less energy consumption" Hu says.



In reality, memristors usually show lopsided current-voltage characteristics. However, Hu's molybdenum disulfide memristor does show the ideal symmetry. This will make the material more predictable and consistent as it is developed for use in electronics.

To get this symmetry, Hu and his research team started with bulk molybdenum disulfide. They then manipulated the atomic, structural arrangements, referred to as different crystal phases. The bulk material with a 2H phase works well as a regular resistor, and to make it a memristor, the team peeled back the molecular layers. This exfoliation process creates molybdenum disulfide nanosheets with 1T phase. The researchers finally dispersed nanosheets on the two sides of a silver foil to form a symmetric memristor.

Hu says, "These memristor materials will be very versatile, and someday, this white board and that coffee cup could be computers."

# FLEX/RIGID-FLEX? AMERICAN STANDARD CIRCUITS MAKES IT EASY!



No matter what level of Flex/Rigid-Flex technology you require, our team of experts are there to help you get the job done. Need it fast? We've got you covered with quick turns on these and many other high level technologies. American Standard Circuits...let us be your new easy button for PCB applications!

Digital | RF/Microwave | Flex | Metal Clad | RoHS



**American Standard Circuits**

Creative Innovations In Flex, Digital & Microwave Circuits

[www.asc-i.com](http://www.asc-i.com)



Brad Griffin

## ...at Cadence Design Systems

by **Andy Shaughnessy**

I-CONNECT007

Cadence Design Systems has released a variety of PCB design tools lately, and we wanted to find out a little more about what's new at Cadence. I tracked down Product Marketing Director Brad Griffin and asked him to discuss some of the newest technology coming out of Cadence.

**Andy Shaughnessy:** *Tell us about some of the new PCB design tools at Cadence Design Systems.*

**Brad Griffin:** We recently launched a product called OrCAD Sigrity ERC. ERC is an acronym for electrical rule checks. This product represents one of several ways where we have taken advanced Sigrity technology and scaled it to a usable model for folks that are not experts in Signal and Power Integrity. As an example, a PCB designer, without any models or experience with signal integrity tools, is able to screen his design for impedance discontinuities and high levels of crosstalk. This is partic-

ularly valuable on dense designs where routed traces may accidentally get pushed over voids or gaps between power or ground shapes on adjacent layers. Instead of having the signal integrity engineer send the board back for changes, the PCB designer catches the problems on his own and leaves the SI guy to focus on the tough stuff.

**Shaughnessy:** *What new technologies are you excited about?*

**Griffin:** I am particularly excited about how our analysis tool product line is enabling a broad range of design expertise in a streamlined fashion. Everyone knew when we acquired Sigrity in 2012 that we were getting some of the industry's most powerful signal and power integrity tools. Being able to segment that technology for our customer base—from the mainstream PCB designers in the OrCAD space to those creating bleeding-edge designs at some of the largest companies in the world using Allegro—is extremely satisfying. When I meet with customers and hear their ideas of how we can continue



**VERIFY**

**OPTIMIZE**

**ANALYZE**

**PREPARE**

**PANELIZE**

**COMMUNICATE**

**DOCUMENT**

## Ensure PCB Manufacturing Success!



**CAM350<sup>®</sup>**

Verify and Optimize PCB Designs for Successful Manufacturing.



**BluePrint-PCB<sup>®</sup>**

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



**DFMStream<sup>™</sup>**

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](http://downstreamtech.com)  
or call 800-535-3226

to improve our integration between our PCB design and PCB analysis tools, I get even more excited about the future.

**Shaughnessy:** *Which customer challenges led you to develop these new products?*

**Griffin:** Over the past several years, the strategy of most of our customers has been to develop products that are smaller, faster, and cheaper, while consuming less power but providing even greater functionality. This is obviously no small task, but as you can see by the constant improvement in mobile devices, IoT devices, and automotive electrical content, they have been successful. Yet, we constantly hear about the need to be more efficient. Our customers are communicating to us three big business challenges. They want to reduce product cost, they want to accelerate their time to volume production, and they want to reduce the surprises in the design cycle making time to market more predictable.

**Shaughnessy:** *How will these new tools help address your customers' concerns?*

**Griffin:** With what seems like conflicting challenges of adding more functionality to a smaller product that consumes less power yet runs faster, it is clear that signal and power integrity concerns will continue to impact PCB design. We strive to help our customers face these challenges early in the design cycle through a constraint driven flow that enables PCB designers to own a greater share of the analysis burden.

By integrating Sigrity with our OrCAD and Allegro product lines, we believe we are giving just enough technology to the PCB designer so that the iterations that classically take place between design and analysis teams can be dramatically reduced. This can be a big help in helping them meet their product goals

**Shaughnessy:** *What sort of trends do you see in the PCB design tools market?*

**Griffin:** Data transfer speeds are going up and operating voltages are going down. We've seen this trend over multiple decades. However, we have truly reached an inflection point with mainstream interfaces such as USB running at 10 gigabits per second and PCI Express to soon be running at 16 gigabits per second. Modeling certain portions of a design now requires full-wave 3D field solvers. And with the electrical margins so small, optimizing high speed structures is no longer a luxury, but must become part of a standard design flow. I have seen a growing concern about having to take these optimized structures created by external 3D field solvers and having to redraw them in a PCB design tool. Customers tell me the process is error-prone and time-consuming. They are asking us to be able to transfer the optimized structures directly from the 3D full-wave field solver into the PCB design database without having to redraw it. Sounds like a good idea, so stay tuned.

**Shaughnessy:** *Thanks for the insight, Brad.*

**Griffin:** Thank you. **PCBDESIGN**

---

## You'll Never Be-Leaf What Makes up This Battery

Scientists at the University of Maryland have a new recipe for batteries: Bake a leaf, and add sodium. They simply heated the leaf for an hour at 1,000°C to burn off all but the underlying carbon structure. The lower side of the maple leaf is studded with pores for the leaf to absorb water. The pores absorb the sodium electrolyte.

"A leaf is designed by nature to store energy

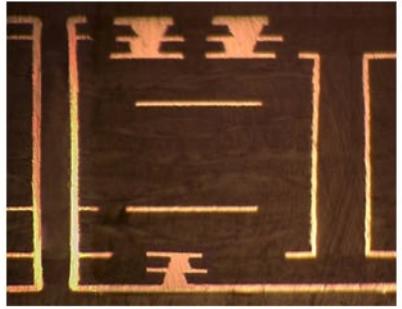


for later use, and using leaves in this way could make large-scale storage environmentally friendly," said Liangbing Hu, an assistant professor of materials science and engineering.

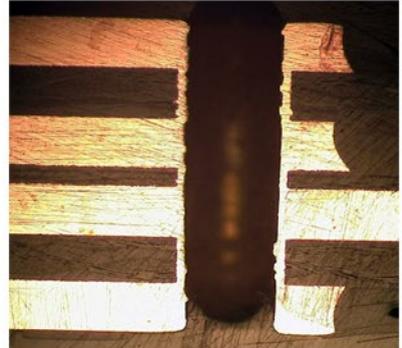
The next step, Hu said, is "to investigate different types of leaves to find the best thickness, structure and flexibility" for electrical energy storage.



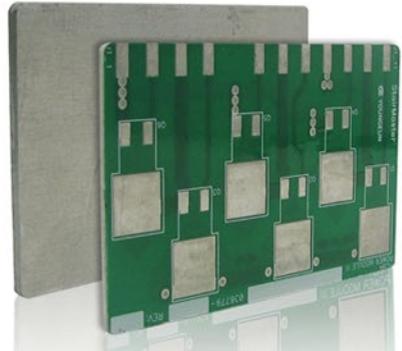
## PRODUCTS



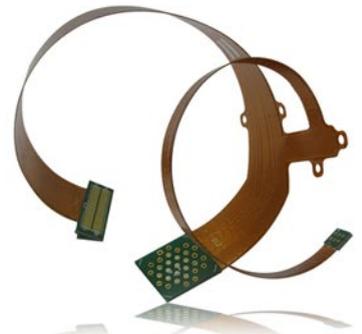
Blind/Buried/Stack via technology



Heavy copper /Pressfit Technology



Multi-layered aluminum board  
lead-free HAL



8 layer rigid-flex  
30mm flexible finished thickness

The ANS team is devoted to technological innovation as well as providing our customers with a high rate of satisfaction and speedy service. With our strengths in R&D, we produce PCBs that meet your specific requirements promptly, and at competitive prices.

The ANS mission is to pursue product and service excellence through continuous improvement in a quest for Total Customer Satisfaction.



**REQUEST A QUOTE TODAY!**

**Head office:**  
ANS Industrial Co. Ltd.  
Kowloon, Hong Kong  
Tel: (852) 2757 1882

**Factory:**  
Remapp PCB (Foshan) Co.,Ltd.  
Guangdong Province, China  
Tel: (86) 757-8862 3888

**www.anspcb.hk**

**Email: sales@anspcb.hk**



### [IoT, Community and Content at the Centre of CES](#)

Phillip Stoten writes about his observations during the recent CES 2016 Show in Las Vegas, including new trends driving the next generation of consumer electronics technologies such as IoT and content.

### [Happy's Essential Skills: The Need for Total Quality Control \(Six Sigma and Statistical Tools\), Part 1](#)

In this first of many columns covering my "Twenty-Five Essential Skills Every Engineer Needs to Learn," I will expand on each of those skills. The introduction to this series published in the January issue of *The PCB Magazine*. As a quick recap, here are the 25 skills that I will be writing about over the next 18 months or so, to publish every three weeks or so in the PCB007 Daily Newsletter.

### [CES 2016 Wrap-Up, Part 1](#)

CES 2016 is now history and most of us are home, or at our next port-of-call. I have seen various attendance numbers, but it's somewhere in the range of 175,000; it was busy, crowded and impossible to see everything. Here is my review of some of the most innovative devices and technologies at CES.

### [CES 2016: Press Day, Showstoppers and Launch Event](#)

Tuesday, January 5 was press day at CES 2016. Some of the largest press conferences are held on press day. Many of them are one- or two-hour advertisements with a number of new product announcements inserted, and some do provide a great deal of market data and trends. The bigger "pressers," such as the Samsung event, fill multiple auditorium-sized rooms as well as overflow rooms.

### [Weiner's World](#)

In this article, industry veteran Gene Weiner looks back at some of the industry highlights in 2015, and ponders on the challenges and opportunities to expect this year. He also highlights recent industry events, including the 2015 International Printed Circuit & APEX South China Fair and SEMI-CON Japan.

### [CES: Day One](#)

CES Unveiled is the official media event for CES. It is the first official happening of what promises to be a very busy and fascinating week. At this event, members of the press get to preview a number of innovative startups as well as some new products from a few established global brands.

### [WKK's Hamed El-Abd on the Current State of China](#)

In the world of real estate, the key term might be "location," but in manufacturing today, it's automation, automation, automation. I had the opportunity to interview WKK's Hamed El-Abd at the recent HKPCA show, who discussed the company's entry into the direct imaging market and how certain areas of the Chinese market are being flooded by a staggering amount of Chinese equipment manufacturers.

### [AT&S Offers Expanded PCB Sales Support in the U.S.](#)

AT&S, one of the global leading manufacturers of high-end PCBs, with headquarters in Leoben, Austria, is committed to absolute customer orientation supporting its vision: "First choice for advanced applications."

### [Digital Imaging Revisited](#)

The advantages of digital circuitization techniques have been described in detail by suppliers of equipment and photoresist. Since phototool generation and conditioning are omitted, there is the advantage of shorter lead time.

### [Catching Up with HSIO's James Rathburn](#)

I have interviewed James Rathburn a number of times in the past few years and he always has something new to say. One of the industry's leading technology inventors, Jim is always finding himself on the cutting edge of our technology. The recent acquisition of the former HEI operation in Tempe, Arizona exemplifies the path his company is taking towards a goal of being the industry's true technology leader.



# TECHNICA, U.S.A.

Fulfilling Manufacturing Needs  
Throughout the Electronics Industry



## Design Challenge:

High-speed applications have you asking **how low can you go?** EMC answers with new **very-low and ultra-low loss materials!**

## Introducing EMC's New Line of Copper Clad Laminates

### EM-891



#### APPLICATIONS:

Very high-speed server, network and telecom PCBs with 25 to 28Gbps per channel design.

#### THERMAL RELIABILITY TEST

- 26L board 130mil
- Pitch Design 0.7, 0.8mm
- Test Condition and Result: IR reflow 260°C 6X with no abnormality

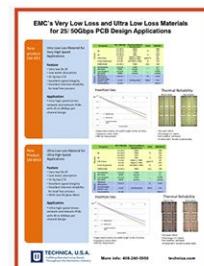


### EM-891K



#### APPLICATIONS:

Ultra high-speed server, network and telecom PCBs with 40 to 50Gbps per channel design.



**DOWNLOAD**  
technical data  
sheet.

# Faster than a Speeding Bullet

by Barry Olney

IN-CIRCUIT DESIGN PTY LTD AUSTRALIA

In a previous *Beyond Design* column, [Transmission Lines](#), I mentioned that a transmission line does not carry the signal itself, but rather guides electromagnetic energy from one point to another. The speed of a computer does not depend intrinsically on the speed of electrons, but rather on the speed of energy transfer between electronic components. Electron flow in a multilayer PCB is extremely slow—about 10 mm per second—so, how does the signal travel so fast, how fast does it actually transfer information and what are the limitations?

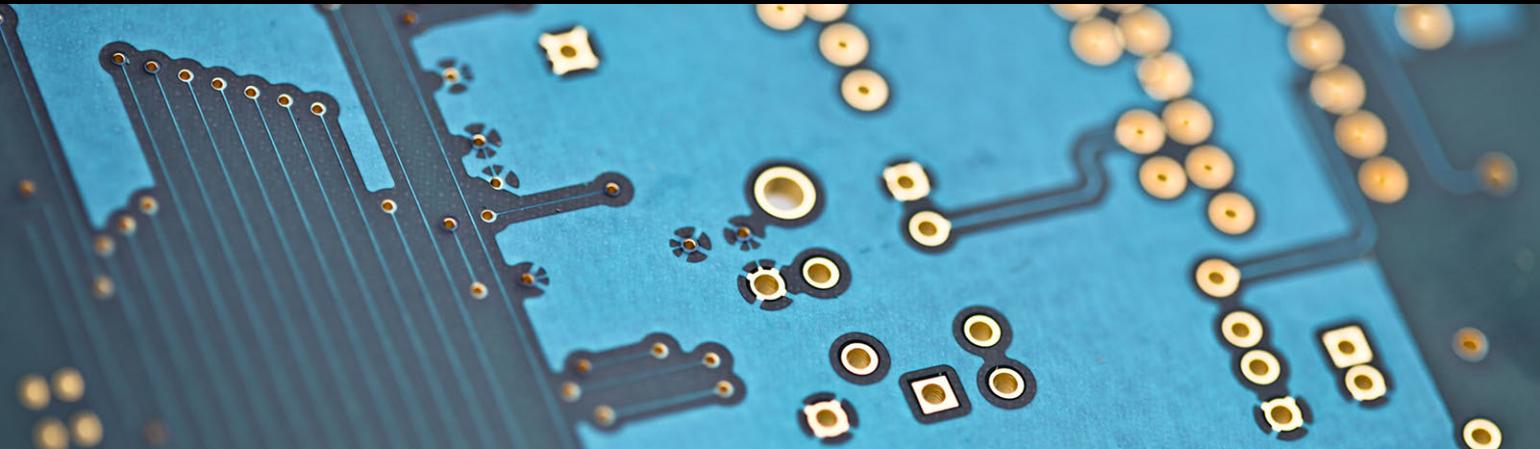
In optical communications, electrons don't carry the signal—photons do. And we all know that photons travel at the speed of light. So

surely, optical fibers must transmit information much faster than copper wires or traces on a multilayer PCB? Actually, photons and electrons transmit data at the same speed. The limiting factor is the relative permittivity (dielectric constant) of the medium in which the signal propagates.

An optical fiber is a cylindrical dielectric waveguide made of low-loss materials such as fused silica glass. It has a central core in which light is guided, and embedded in an outer cladding of slightly lower refractive index. The silica glass used has a dielectric constant ( $\epsilon_r$  or  $D_k$ ) = 3.78 @25GHz. Whereas, for instance, Panasonic's new Megtron 7, low  $D_k$ , glass PCB laminate



Figure 1: An FA-18 approaches the speed of sound. The white halo consists of condensed water droplets formed by the sudden drop in air pressure behind the shock cone around the aircraft. (Courtesy of the U.S. Navy)



# 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](tel:1-800-228-8198) or via e-mail at [support@sunstone.com](mailto:support@sunstone.com)

has an  $\epsilon_r = 3.3$  at the same frequency. The dielectric material determines the velocity ( $v$ ) of propagation of the electromagnetic energy:

$$v = \frac{c}{\sqrt{\epsilon_r}}$$

where the speed of light ( $c$ ) is  $3 \times 10^8$  m/s.

Substituting the numbers, the optical fiber has a velocity of propagation of  $154.6 \times 10^6$  m/s compared to  $164.8 \times 10^6$  m/s for Megtron 7. So believe it or not, the PCB substrate, in this case, will actually transfer the signal faster than an optical fiber. However, the dissipation factor (dielectric loss) of silica is 0.00002 whereas Megtron 7 is only 0.003, which limits the bandwidth. However, this is still fairly low loss compared to standard FR-4 of 0.02.

Fibers have a number of advantages over copper, including higher bandwidth, lower loss, immunity to electromagnetic interference. There is also no crosstalk between signals in different optical cables. Further, non-armored fiber cables do not conduct electricity, which makes them ideal in high-voltage environments or structures prone to lightning strikes and for preventing ground loops.

Figure 2 illustrates a rigid PCB substrate with an embedded optical polymer waveguide. This novel structural design offers potential solutions for low-cost and high-performance semiconductor circuits with optical devices to realize wide bandwidth and low-profile optoelectronic packaging for chip-to-chip optical interconnect applications. The vertical cavity surface emitted laser (VCSEL), driver, and serializer chip are 3D stacked and then attached to one end of the embedded optical polymer waveguide in the PCB. Similarly, the photo-diode detector, trans-impedance amplifier (TIA), and deserializer chips are also stacked and then attached on the other end of the waveguide. Although this system has the same communications speed as a typical trace based interconnect, it potentially exhibits wider bandwidth and much lower noise.

The actual velocity of electrons through a conductor is measured at an average speed called the “drift speed.” The charge carriers (electrons) move very slowly; however, the “knock-on” effect is very fast as it follows the electromagnetic field. When one electron is forced to move, it bumps into its neighbor making it move and so on: the domino effect. The energy propagates as an electromagnetic wave.

The electrical effects that we observe, such as lights coming on immediately when a switch

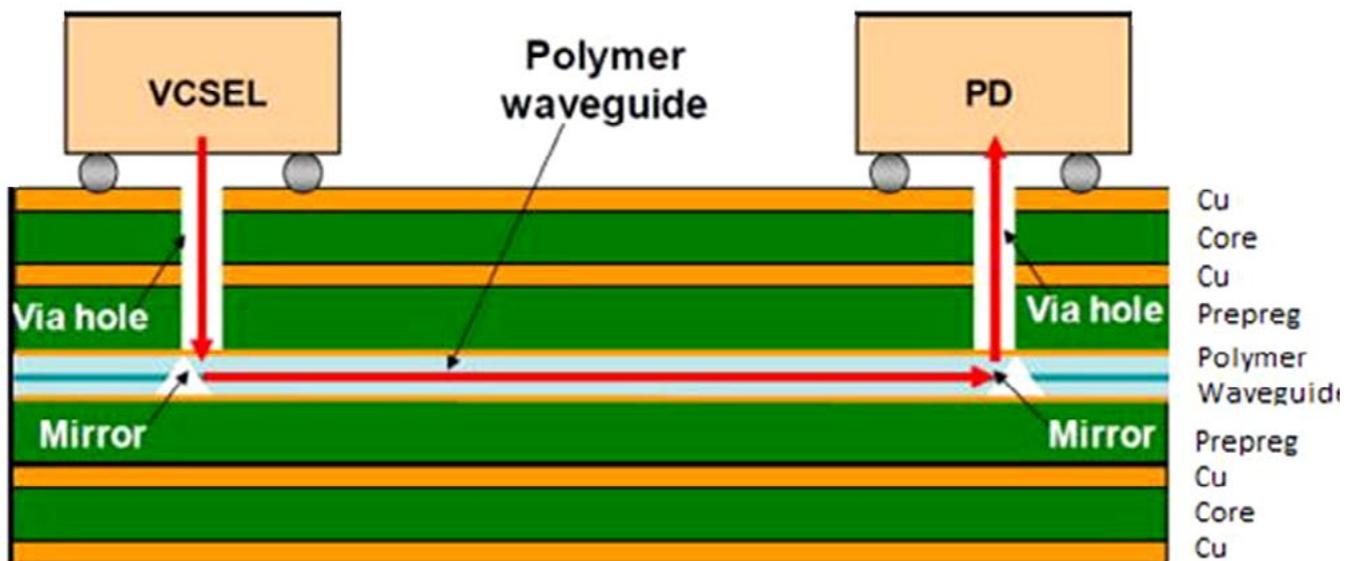


Figure 2: Low-cost, high-performance optoelectronic system embedded into an organic laminated substrate. (Source: Lau, Zhang, Lee)

is closed, are due to this knock-on effect. This explains the observation that a complete circuit is needed for current flow. If one charge carrier cannot cross a gap in the circuit, all the other charge carriers behind cannot move either and current does not flow anywhere in the circuit. This scenario explains DC and low-frequency circuits that have a single point ground reference (as taught in Circuit Theory 101).

However, above 100KHz parasitic capacitance and inductance become significant allowing current to flow in multiple paths. In a multilayer PCB, the electric field charges each section's R-L-C-G elements; in turn, as the rising edge propagates along the transmission line with the return current from each section, flowing back to the source. Then as the pulse passes, the falling edge discharges each section's capacitance. By the time the signal wave reaches the load, it has established multiple paths of return current along the PCB planes.

The speed at which the signal travels down the conductor really has nothing to do with the drift speed of the electrons. The signal is an electromagnetic wave that travels at about half the speed of light. The electrons serve to guide the wave down the wire. It is the movement of the electromagnetic field or energy—not voltage or current that transfers the signal. The voltage and current exist in the conductor, but only as a consequence of the field being present as it moves past.

We have established the fact that signals travel at the same speed, given the same medium, but what limits the bandwidth? Let me use a metaphor to explain: The speed of sound is much slower than that of light at 343.2 m/s in air. However, this isn't the speed of the channel—it is its latency. That is, if you are 343 meters away, you will hear me one second after I speak. That reveals nothing about how fast I can communicate with you, which is limited by

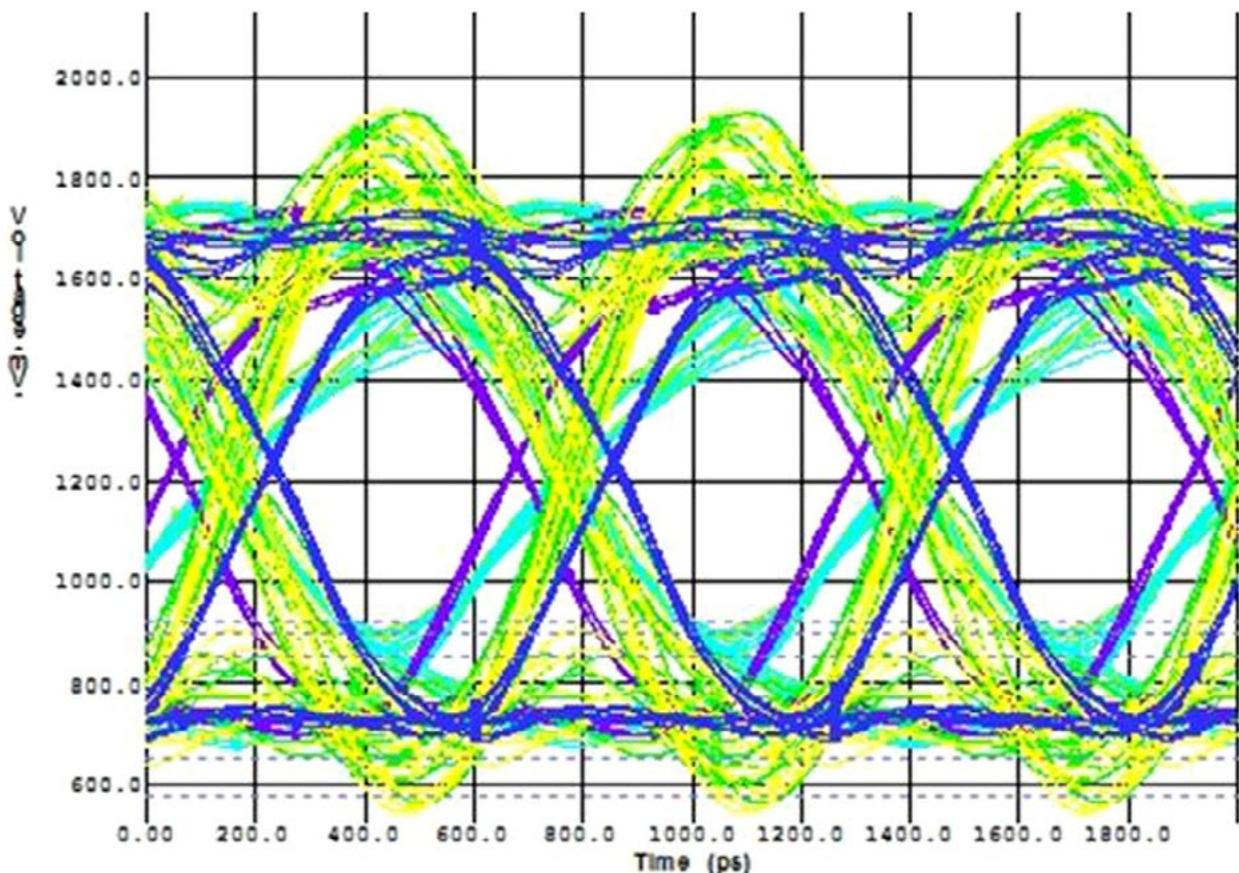


Figure 3: Eye diagram displaying jitter and noise in Mentor Graphics' HyperLynx tool.

how effectively I can speak, and how well you can hear me.

If we are in a quiet room, I can probably speak very quickly and you can still hear me. If we are far apart, or the environment is noisy, I will have to speak louder, more slowly and clearly. With electrical communications the situation is much the same. The speed limit is not due to the latency, but rather how fast one end can transmit with the other end still being able to reliably receive. This is limited by noise picked up from the environment and distortions introduced by the medium—the noise margin.

It is the signal-to-noise ratio (SNR) that bestows optical fibers with a higher bandwidth than other transmission mediums, given the same speed of transmission. One of the most important ways to determine the quality of a digital transmission system is to measure its bit error ratio (BER). The BER is calculated by comparing the transmitted sequence of bits to the received bits and counting the number of errors. Very small changes of the SNR (in the order of a dB) can cause very large changes in the BER.

An eye diagram is a common indicator of the quality of signals in high-speed digital transmissions. An oscilloscope generates an eye diagram by overlaying sweeps of different segments of a long data stream driven by a master clock. In a simulation tool such as Mentor Graphics' HyperLynx, a pseudo-random bit stream is generally used to produce the overlapping sweeps as in Figure 3. In an ideal world, eye diagrams would look like rectangular boxes. In reality, communications are imperfect, so the transitions do not lie precisely on top of each other, and an eye-shaped pattern results. However, in Figure 3, we can see the jitter (horizontal miss-alignment) and the distortion set by the SNR (vertical miss-alignment).

In high-speed multilayer PCBs, we need to select the material with the lowest dielectric constant (Dk) and the lowest dielectric loss (Df) in order to achieve the maximum bandwidth which is the 5<sup>th</sup> harmonic of the fundamental frequency. (The dielectric materials library integrated in to the ICD Stackup Planner, has 20,000 rigid-flex materials up to 100GHz to choose from. This makes selecting the right ma-

terial for your application easy.)

As frequency increases, so does the bandwidth. However, we must select the most efficient frequency for the particular transmission channel. If the frequency is too low or too high, we lose the signal's power. This is due to how the medium responds to different levels of charge energy. In general, the amount of information you can transmit is proportional to the rate the channel can respond. Basically, one has to stay within a certain limit depending on the medium. It just so happens that the higher the operating frequency and the lower the loss, the easier it is to get wider bandwidths and hence more data reliably through the channel.

### Points to Remember:

- A transmission line does not carry the signal itself; it guides electromagnetic energy from one point to another.
- Electron flow in a multilayer PCB, is extremely slow—a few meters per second.
- Photons and electrons transmit data at the same speed. The limiting factor is the relative permittivity of the medium in which the signal propagates.
- Fibers have a number of advantages compared to copper, including higher bandwidth, lower loss, and immunity to electromagnetic interference.
- The charge carriers (electrons) move very slowly; however the knock-on effect is very fast as it follows the electromagnetic field.
- DC and low-frequency circuits have a single point ground reference. However, above 100KHz parasitic capacitance and inductance become significant allowing current to flow in multiple paths.
- The speed at which the signal travels, down the conductor, really has nothing to do with the drift speed of the electrons. The signal is an electromagnetic wave that travels at about half of the speed of light. The electrons serve to guide the wave down the wire.
- It is the signal to noise ratio (SNR) that bestows optical fibers with a higher bandwidth than other transmission mediums, given the same speed of transmission.

# Automotive electronics stay cool with our power electronics PCBs



Solutions in power electronics play a key role in core topics of our times: They enable us to master the future challenges of electro mobility, energy creation and energy distribution.

In order to reduce fuel consumption as well as CO<sub>2</sub> emissions and to increase safety and comfort, it is increasingly necessary to switch higher currents and to dissipate heat.

Amongst all PCB manufactures, SCHWEIZER offers the most comprehensive range of solutions to tackle these challenges. Already today, the automotive industry applies SCHWEIZER PCBs which can operate up to 1,200 Ampere.

**Schweizer Electronic AG**  
Einsteinstraße 10  
78713 Schramberg  
Germany

[www.schweizer.ag](http://www.schweizer.ag)

- One of the most important ways to determine the quality of a digital transmission system is to measure its bit error ratio (BER).
- An eye diagram is a common indicator of the quality of signals in high-speed digital transmissions.
- In high-speed multilayer PCBs, we need to select the material with the lowest dielectric constant (aka Dk) and the lowest dielectric loss (Df) in order to achieve the maximum bandwidth.
- The higher the operating frequency and the lower the loss, the easier it is to get wider bandwidths. **PCBDESIGN**

### References

1. Barry Olney, Beyond Design: [Matched Length Does not Equal Matched Delay, Transmission Lines](#)
2. Robert Dahlgren: Noise in Fiber Optic Communications Links

3. Lau, Zhang, Lee: Opto-electronic Interconnect in Organic Substrate
4. Bruce Archambeault: [PCB Design for Real-World EMI Control](#)
5. Howard Johnson: [High-speed Signal Propagation](#)
6. [www.physics.stackexchange.com](http://www.physics.stackexchange.com)
7. Alan Finkel: Cosmos, Issue 61, "Silicon is King"
8. The ICD Stackup and PDN Planner: [www.icd.com.au](http://www.icd.com.au)



**Barry Olney** is managing director of In-Circuit Design Pty Ltd (ICD), Australia. This PCB design service bureau specializes in board-level simulation, and has developed the ICD Stackup Planner and ICD PDN Planner software. To read past columns, or to contact Olney, [click here](#).

---

## Scientists Bridge Different Materials by Design

In an advance reported in Nature Chemistry, scientists at the University of Liverpool have shown that it is possible to design and construct interfaces between materials with different structures by making a bridge between them.

It is usually possible to make well-controlled interfaces when two materials have similar crystal structures, yet the ability to combine materials with different crystal structures has lacked the accurate design rules that increasingly exists in other areas of materials chemistry.

The design and formation of an atomic-scale bridge between different materials will lead to new and improved physical properties, opening the path to new information technology and energy science applications amongst a myriad of science and engineering possibilities. For example, atoms could move faster at the interface between the materials, enabling better batteries and fuel cells.

Liverpool Materials Chemist Professor Matthew



Rosseinsky said, "When we try to fit materials together at the atomic scale, we are used to using the sizes of the atoms to decide which combinations of materials will "work" i.e. will produce a continuous well-ordered interface.

"The project team added in consideration of the chemical bonding around the atoms involved, as well as their sizes, as a key design step. This allowed the selection of two materials with different crystal structures yet with sufficient chemical flexibility to grow in a completely ordered manner throughout the interface between them.

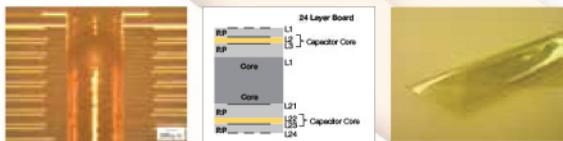
"This was achieved by the formation of a unique ordered structure at the interface which did not correspond to either material but contained features of both of them, an atomic-scale bridge."

It is possible to construct a flexible block, which will fit with both materials, and bridge the gap between them, like the blue blocks bridge the gap between the red and green ones.

# 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](mailto:sales@oakmitsui.com) or call us at 503-313-1905

**APEX  
EXPO**  
IPC 2016

Visit us in booth #924



**FARADFLEX®**

EMBEDDED CAPACITANCE

# How to Design-Out Production Problems

by Phil Kinner  
ELECTROLUBE

## Introduction

In case you're wondering who I am, here's a quick background: I joined Electrolube in May 2014 as technical director for the company's Conformal Coatings Division. I have extensive experience within the conformal coatings industry and I'm well versed in writing papers and presenting at forums around the globe on key issues.

I recently conducted a study that resulted in a paper on increasing end-product lifetime and reliability. I discovered that one key driver is the understanding that a conformal coating only offers effective protection to surfaces when it is applied with perfect coverage. Any voids, bubbles, or cracks in the coating are likely to allow the initiation of corrosion. Understanding this factor is a major step towards developing a reliable product.

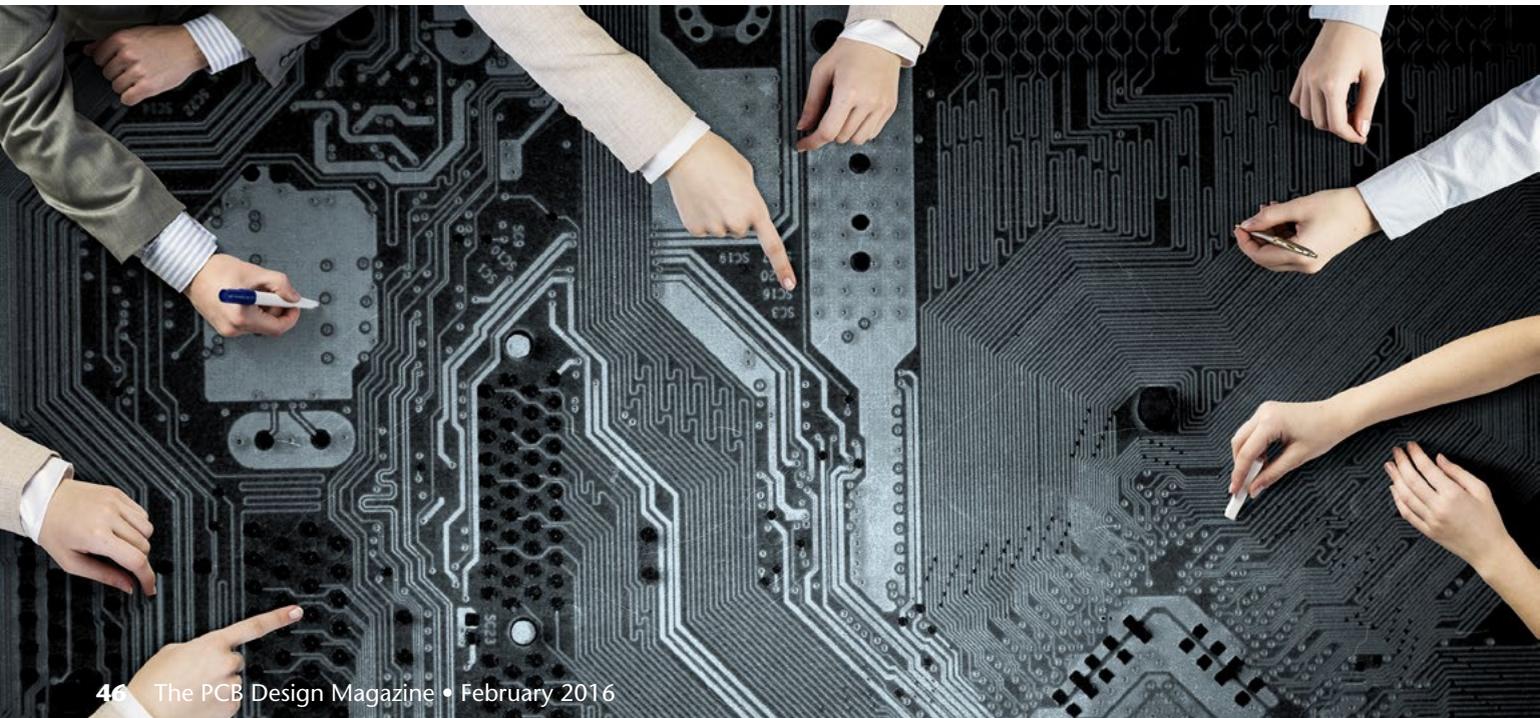
For this column, the first in a series, I will be taking a closer look at production issues, particularly those that circuit designers are in a posi-

tion to address at the earliest stages of a project. I'll provide some design pointers that will help you avoid some common pitfalls when applying conformal coatings. In each column, I will list five essential facts about conformal coatings that will help you to streamline your production processes.

## Designing-Out Coating Challenges

I have quite a few years of experience and knowledge of the coatings business, and I'm still learning. But there are certain production-related issues that cause problems time and time again, many of which could so easily have been resolved earlier at the design stage. In this column, I will highlight these issues and hopefully get the PCB design guys fully on-side with their production colleagues.

Initially, I would just like to give a very short explanation about the types of conformal coating chemistries commonly available and emphasise that they all have their own benefits





Innovation drives the future.  
We drive innovation.

Intercept's complete design flow gives you the power to keep moving, full speed ahead. Use Mozaix for flexible schematic capture and Pantheon for an advanced, intuitive PCB/Hybrid/RF design to move quickly and seamlessly from concept to manufacturing. Our 30+ import translators make it easier than ever to switch tools. Let Pantheon and Mozaix drive your next innovation.

PANTHEON ADVANCED PCB, HYBRID, RF LAYOUT  
MOZAIX NEXT GENERATION SCHEMATIC CAPTURE  
REUSE BLOCK TECHNOLOGY  
MULTI-BOARD & PANEL ARRAYS  
LAYOUT-DRIVEN DESIGN & PROTOTYPING  
SIMULATION & ANALYSIS INTERFACES  
DESIGN AUTOMATION  
DFM & ARTWORK VERIFICATION  
HIGH SPEED DESIGN  
ARTWORK PANELIZATION  
AUTOMATIC DRAWING CREATION  
30+ TRANSLATORS



[www.intercept.com](http://www.intercept.com)

30+ Years of EDA Software Solutions

and drawbacks. Conformal coatings are available in solvent-based, water-based and 100% active materials, as well as vapour-deposited coatings in which monomeric gases are mixed together in a vacuum, where they are polymerized and deposited onto the surface of the PCB as a protective film. Most conformal coatings are either qualified to MIL-I-46058C or meet the requirements of the closely allied IPC-CC-830B specifications. Further, they can additionally be recognised as either a permanent coating, in which case the flammability of the coating is assessed to UL94V0, or as a conformal coating, where the electrical properties will be assessed as part of the UL746E standard.

So, here are five essential facts to consider if you intend to conformally coat your electronics products:

### **FACT 1.**

#### **All solder resists are not created equal!**

The adhesion results with conformal coatings can be very varied when utilising what appears to be the 'same' spec of solder resist from different board suppliers, which can create havoc during the conformal coating process. A quick and very effective solution to this can be to specify a surface energy of >40 dynes/cm on incoming bare boards and ensure that each incoming batch is religiously tested prior to assembly and rejected if they do not meet this minimum value.

### **FACT 2.**

#### **Always leave a buffer**

Conformal coatings are usually liquid when applied, and they will flow with a combination of gravity and the capillary forces present. Whether you are masking or relying specifically on selective conformal coating, a production team will be greatly relieved if you leave a buffer of at least 3 mm clear between the areas to be coated. This small buffer will make the production process easier and prevent future issues in production.

### **FACT 3.**

#### **Simplify, simplify, simplify**

If possible, spend time simplifying the coating process at the PCB design stage. Placing

the connectors and components that must not be coated along one edge of the assembly will greatly simplify the conformal coating application process. This will also allow dip coating to be explored as a potential methodology, and the net result will be quicker application times and reduced costs. If it is not possible to locate all of these connectors and components along one edge, then keeping them to the edges will also optimize the coating process.

### **FACT 4.**

#### **Discrete components: the downside**

Large arrays of discrete components represent a massive coating challenge due to the high levels of capillary forces present, and the result is often quite disastrous, with areas of no coverage/protection on the board and conversely areas of excessive thickness prone to stress-cracking, delamination and other coating defects. Ultimately this will lead to premature failure of the assemblies. Try to avoid this if at all possible!

### **FACT 5.**

#### **Is bigger really better?**

Tall components present challenges of their own by creating shadowed or hard-to-reach areas. Splashing, where the coating splashes into "keep-out areas," is another associated problem. Try to avoid siting tall components next to "must-coat" or "must not coat" components/areas to minimise this.

**Remember:** Thoughtful design strategies will pay huge dividends down the line. And PCB designers will have friends for life among your production colleagues when you make their jobs just a little bit easier!

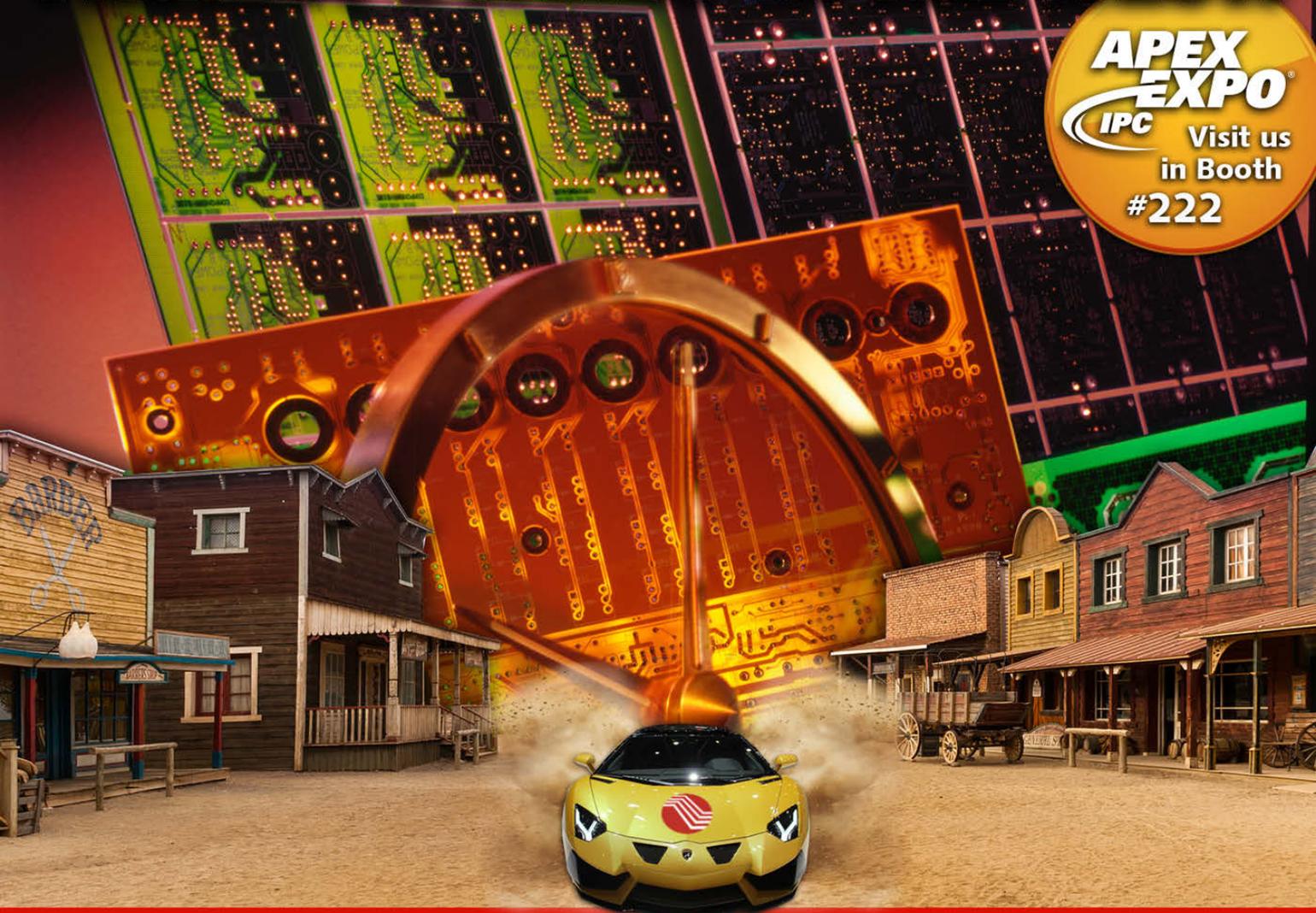
See you next month. **PCBDESIGN**



**Phil Kinner** is the global business/technical director for the coatings division at Electrolube. To contact him, [click here](#).

# FASTEST BOARDS IN THE WEST!

**APEX EXPO**  
IPC Visit us  
in Booth  
#222



When Speed is a Priority  
We're Quick on the ~~Draw~~ *TURN*

MORE INFORMATION

REQUEST A QUOTE

**Prototron**  
 **Circuits**

*High quality PCBs delivered fast, just another reason we are America's Board Shop*

sales@prototron.com | prototron.com

# Multilayer PCB Bonding Materials for High-Frequency Applications

by John Coonrod

ROGERS CORPORATION

When fabricators are choosing a material, they are normally familiar with the bonding materials that are appropriate for their circuit construction. The electrical performance is not always critical, but when it is, the fabricators may not completely understand how the material can impact the material electrical performance. The following is an overview of different bonding materials used in high-frequency multilayer PCB applications.

The materials used to bond high-frequency multilayers may differ greatly in their formulations. Many of these bonding materials are glass-reinforced; however, there are several commonly used bonding materials which are not woven glass-reinforced. The non-reinforced bonding materials are typically a thermoplastic polymer film, however there are some exceptions. Woven glass-reinforced bonding mate-

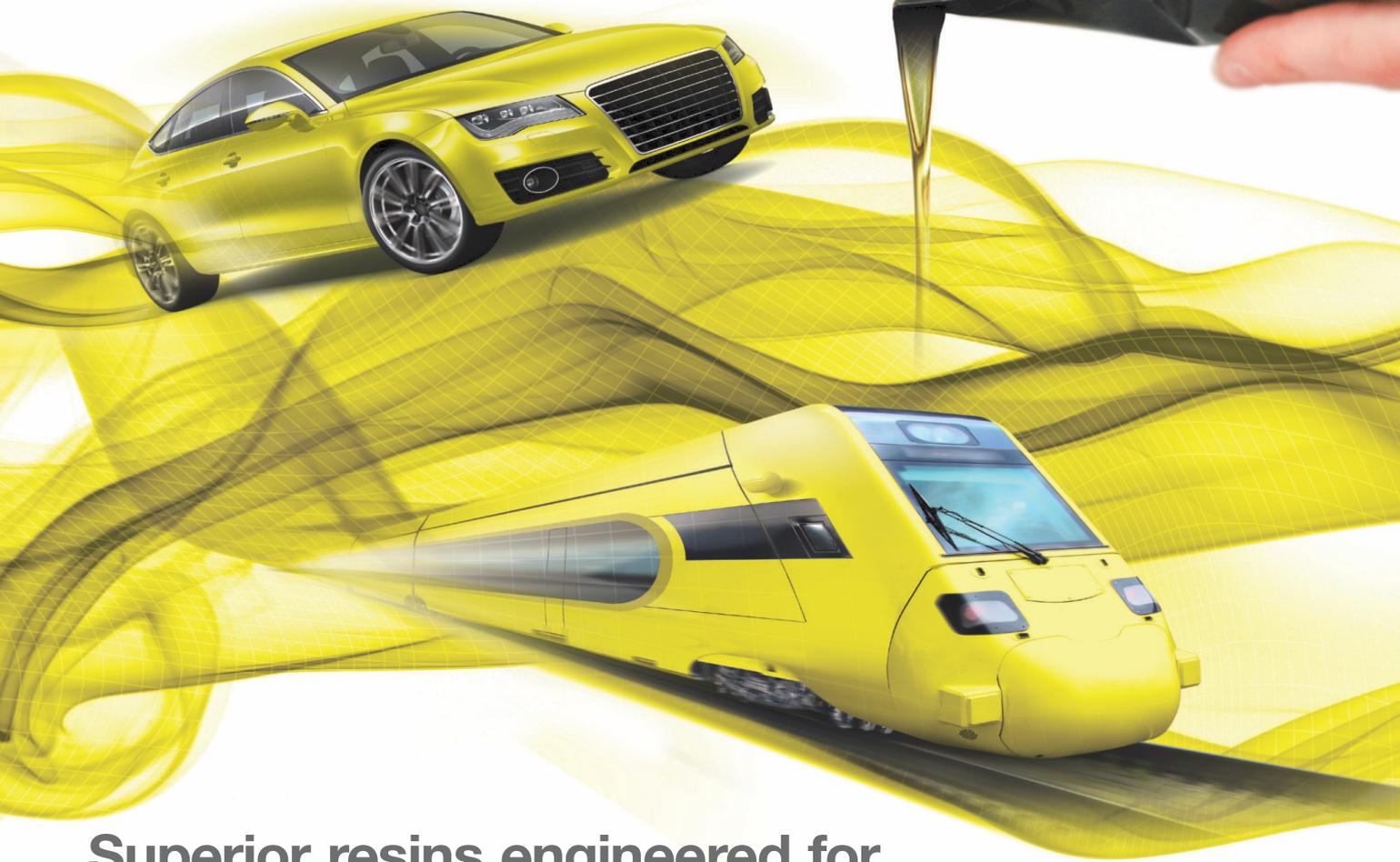
rials are usually thermoset systems and often with special filler to enhance the high frequency performance properties.

The thermoplastic bonding materials are brought to a melt temperature during lamination to achieve adhesion of the layers for the multilayer circuit. These materials can also re-melt after the multilayer has been adhered and re-melting can cause delamination which is why it is typically avoided. The lamination melt temperature and the cautionary re-melt temperatures vary with the type of thermoplastic bonding materials. Some fabricators may not have the equipment necessary to reach these higher temperatures necessary for lamination of a multilayer PCB. The re-melt temperature is typically a concern for the processes following the lamination, which exposes the circuit to elevated temperatures such as soldering.





Visit us on stand 2831  
March 15-17, 2016  
Las Vegas Convention Centre  
Las Vegas Nevada



## Superior resins engineered for challenging environments

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

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

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

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

Tel: 888-501-9203

[www.electrolube.com](http://www.electrolube.com)



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

**ELECTROLUBE**  
THE SOLUTIONS PEOPLE

Electronic & General  
Purpose Cleaning

Conformal  
Coatings

Encapsulation  
Resins

Thermal Management  
Solutions

Contact  
Lubricants

Maintenance  
& Service Aids

The following is a list of thermoplastic non-reinforced bonding materials commonly used in multilayer high-frequency PCBs, with their melt and re-melt temperatures: Rogers 3001 (melt=425°F and re-melt=350°F), CuClad 6700 (melt=425°F and re-melt=350°F), and DuPont Teflon FEP (melt=565°F and re-melt=520°F) bonding films.

The re-melt temperature is lower than the initial melt temperature because of delamination concerns. At the re-melt temperature, the material is soft enough to delaminate. At the initial melt temperature during lamination the material is at its lowest viscosity which allows the material to wet-out and flow for good adhesion in the multilayer while held under pressure in the lamination process. From the temperatures of the different materials, it can be seen that using the 3001 or CuClad 6700 bonding materials would be appropriate for a multilayer which is not exposed to elevated temperatures, such as soldering. The DuPont Teflon FEP material can be used for a multilayer that will be subjected to soldering, assuming the soldering temperature is controlled to below the re-melt temperature. However, some fabricators do not have the capability to reach the initial melt temperature.

There is an exception to the thermoplastic non-reinforced bonding materials, and that is Rogers' 2929 bondply, which is non-reinforced, but it is not a thermoplastic. It is a thermoset material. The thermoset material does not have a melt and re-melt temperature, but it has a cure temperature (during lamination) and a decomposition temperature which is to be avoided due to delamination concerns. In the case of the 2929 bondply, the lamination temperature is 475°F and the decomposition temperature is well beyond lead-free soldering temperatures so it is robust to most elevated temperature processing after the multilayer is bonded.

Each of these non-reinforced bonding materials has different electrical properties. The electrical properties for these bonding materials are as follows: Rogers 3001 (Dk=2.3, Df=0.003), CuClad 6700 (Dk=2.3, Df=0.003), DuPont Teflon FEP (Dk=2.1, Df=0.001) and 2929 (Dk=2.9, Df=0.003). The term Dk refers to dielectric con-

stant or  $\epsilon_r$  and Df refers to dissipation factor or tan-delta.

There are also woven glass reinforced bonding materials. These are typically a combination of woven glass fiber cloth, resin and some filler. Depending on the formulation, the PCB fabrication parameters for lamination can vary significantly. As a general statement, the prepreg material which is highly loaded with filler will typically have much less lateral flow during lamination. The lack of flow can be good and bad. If the prepreg will be used to build a multilayer with cavities where the prepreg needs to be cut back and not flow into the cavity, then these highly filled prepreps may be a good choice. However, if the inner layers that the prepreg is intended to bond have thicker copper, it is sometimes difficult to get a good lamination with these lower flowing prepreps.

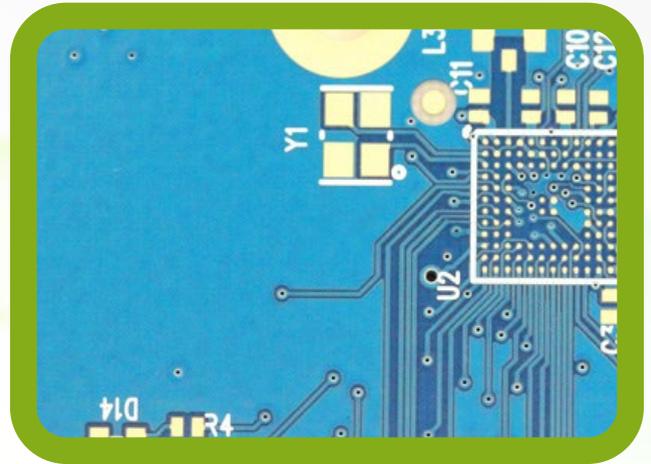
Two woven glass reinforced prepreps which are commonly used in high-frequency fabrication are the RO4450B and RO4450F prepreps (Dk=3.5, Df=0.004). These materials have processing parameters which are relatively similar to FR-4; however, they offer the benefit of very good electrical properties at high frequencies. These materials are highly loaded and will have low lateral flow during lamination. They are a high Tg thermoset materials and very robust to lead free soldering or other elevated processes.

When designing a multilayer PCB for high frequency applications, there are a variety of tradeoffs. The fabrication aspects must be considered along with the electrical performance considerations. It is always highly recommended to contact your material supplier when designing a new multilayer PCB for high-frequency applications so these different tradeoffs can be adequately discussed. **PCBDESIGN**



**John Coonrod** is the technical marketing manager for Rogers Corporation. To read past columns, or to reach Coonrod, [click here](#).

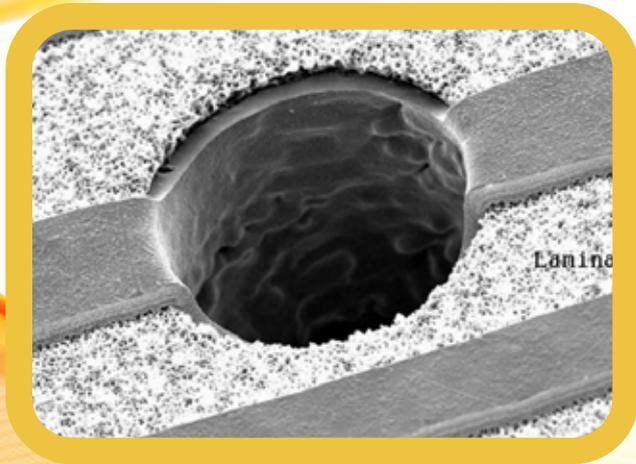
# We deliver Advanced PCB Solutions!



## 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](#)

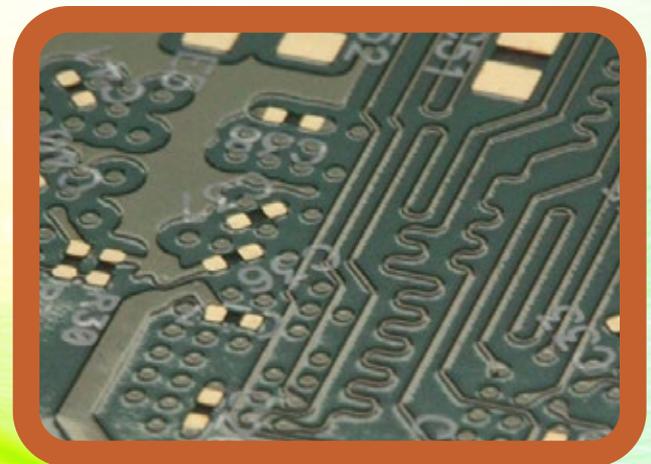


## Landless Via Technology

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](#)

**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](#)



**CANDOR INDUSTRIES, INC.**  
ADVANCED PCB SOLUTIONS

[www.candorind.com](http://www.candorind.com) | [sales@candorind.com](mailto:sales@candorind.com) | (416) 736-6306

# MilAero007 Highlights



## **Mr. Laminate Tells All: Who Would Like a Mil-Spec Audit?**

I remember when IPC-4101 was completed and released in December 1997 and the question came up about whether IPC should create a policeman program to enforce it. To a person that helped create IPC-4101, absolutely no one wanted such an audit program ever again.

## **PCB Technologies Receives NADCAP Accreditation**

PCB Technologies announces that it has received Nadcap accreditation for Electronics.

## **A Conversation with IPC President and CEO John Mitchell**

I-Connect007's Patty Goldman was able to sit down with John Mitchell, president and CEO of IPC, to discuss the organization and where we are going as an industry. We discussed IPC's four aspirational goals—standards, education, advocacy and solutions—as well as short-term goals. We also talked a bit about going virtual and becoming paperless.

## **Raytheon's Development and Testing on Track for DDG 51 Flight III**

Raytheon Company announced its AN/SPY-6(V) Air and Missile Defense Radar (AMDR) team has completed the first full radar array, fully populated with component Line Replaceable Units (LRUs), including more than 5,000 Transmit/Receive elements, in 140 days.

## **NUS Takes the Quantum Leap into Space**

Galassia, a two-kilogramme nanosatellite, was developed by students and researchers from the Faculty of Engineering; Centre for Remote Imaging, Sensing & Processing (CRISP); and Centre for Quantum Technologies (CQT).

## **Satellites Find Sustainable Energy in Cities**

Underground heat islands in cities have an enormous geothermal potential. Warm groundwater can be used to produce sustainable energy for heating and cooling. Researchers of Karlsruhe Institute of Technology (KIT) have now developed a new method to find underground heat islands.

## **Innovative Designs, Smart Manufacturing Deliver Soldier Readiness**

"The Army has called for increased innovation, which is shining a spotlight on prototype designs," said Christopher Manning, Prototype Integration & Testing Division chief, under the Army's Communications-Electronics Research, Development and Engineering Center, or CERDEC. "However, it is imperative that our designs can be leveraged for mass production and sustainment."

## **PNC Purchases Benchtop AOI**

PNC, a provider of rigid, rigid-flex and flex PCBs, has boosted its capabilities by acquiring Nordson YESTECH's BX Benchtop Automated AOI.

## **Northrop Grumman to Design and Produce Shipboard Laser Weapon System Demonstrator**

The U.S. Navy will get a peek at a future where high energy laser weapons could defend its ships against attack under a contract awarded Oct. 22 to Northrop Grumman Corporation by the Office of Naval Research (ONR).

## **Indium Features Gold-Tin Solder Preforms for Precision Die-Attach Applications at AeroDef2016**

Indium will feature its precision gold-tin (AuSn) solder preforms for die-attach at the upcoming AeroDef 2016 event in Long Beach, California. Designed for high-reliability applications, Indium's Pb-free and RoHS-compliant AuSn preforms are available in a variety of standard and custom-engineered designs.



# COOLSPAN® TECA

thermally & electrically conductive adhesive

## Rogers can help by being your reliable conductive adhesive film source

Get the heat out of those high-power PCBs. COOLSPAN® Thermally & Electrically Conductive Adhesive (TECA) Films are ideal for dissipating heat in high-frequency circuits. COOLSPAN adhesives feature outstanding thermal conductivity (6 W/m/K) and reliable thermal stability. Keep things cool, with Rogers and COOLSPAN TECA film.

CONTACT YOUR  
SUPPORT TEAM  
TODAY



Las Vegas Convention Center  
March 15-17 • Booth 607



Advanced Connectivity Solutions

[www.rogerscorp.com](http://www.rogerscorp.com)

## MEET YOUR COOLSPAN® TECA FILM SUPPORT TEAM

### Leading the way in...

- Support
- Service
- Knowledge
- Reputation



**Greg Bull**  
Applications  
Development  
Manager  
Northern &  
Eastern Territory  
(U.S.) & Canada

SCAN THE CODE  
TO GET OUR  
CONTACT INFO.



**Dale Doyle**  
Applications  
Development  
Manager  
Western  
Territory  
(U.S.)



**John Dobrick**  
Applications  
Development  
Manager  
Southern  
Territory  
(U.S.) & South  
America



**John Hendricks**  
Regional  
Sales Manager  
Europe



**Kent Yeung**  
Regional  
Sales Director  
Asia

If you are unable to scan a QR code please visit our  
Support Team website at [www.rogerscorp.com/coolspan](http://www.rogerscorp.com/coolspan)



# NAMM 2016 Review

by Dick Crowe and Dan Feinberg

## Not Your Granddad's Guitar

by Dick Crowe

BURKLE USA

The National Association of Music Merchants (NAMM) held its biannual show in Anaheim in January. PCB Design007 Editor Andy Shaughnessy, columnist Dan Feinberg and I attended this huge event to review the ever-increasing role electronics is playing in the world of musical instruments. It seems as if every instrument has a circuit board now.

Musicians have been using amplification systems for decades. Early guitar amps were used by lap steel guitarists who capitalized on the popularity of Hawaiian music in the '30s and '40s. Today's electric guitars and amps have come a long way technologically, but many guitarists still prefer the sound of vacuum tubes over solid state circuitry. Les Paul and Leo Fender would be proud to see all of the new companies that honor their heritage and contributions.

First, the show itself. Entering the show floor at the Anaheim Convention Center, we were

greeted with a cacophony of sound so loud that Floor Noise patrols walked around with Db meters kept trying to tone the booth noise down. But that is hard when you hear drums, clarinets, trumpets, trombones, synthesizers, electric guitars and all sorts of other instruments competing for attention all at once, and all day long. Exhibitor booths filled the hall with live performances showcasing their particular products. For musicians like us, this was Toy Land. And even after visiting NAMM for the past five years, I still love coming here. It never gets old.

NAMM is a trade show, and not open to the general public. That does not diminish the size of the crowds; attendance was expected to be over 100,000. The show brings people of all types and appearances, many too bizarre to describe in this article. Suffice it to say that my dress and coiffed hair were the exception to the rule.

I walked around the show on Thursday scoping out the manufacturers that we wanted to visit. We selected three companies, but frankly one would need a week to visit and speak with many other exhibitors. Friday was spent looking at instruments and interviewing manufacturers. We tried to speak specifically with those involved in either designing or specifying elec-

The very best in  
PCB supplies  
and equipment

Visit us at  
IPC APEX EXPO  
March 15-17, 2016  
[ BOOTH #247 ]  
Las Vegas, NV

# FELIOS

**Flexible Circuit Material Solutions**  
designed to meet the demanding high performance  
and reliability requirements for commercial,  
industrial and consumer markets

**Felios – RF-775/770**  
Adhesiveless Copper Clad Polyimide  
flexible circuit laminate

Advanced lamination manufacturing technology provides industry leading dimensional stability, flexibility, and consistent lot to lot performance.

Wide selection of copper and film thicknesses. Copper from 2  $\mu$ m to 4 oz., Polyimide from 1/2 mil to 6 mils.

Standard and custom panel sizes and rolls available.

**Felios LCP – RF-705T**  
Adhesiveless Copper Clad Liquid Crystal  
Polymer flexible circuit laminate

Excels in high frequency applications. Cost effective alternative to PTFE based substrates.

Wide selection of copper and film thicknesses – Available in panels and rolls.

LCP Bond-Ply available – RF-606

## INTRODUCING

# PIUS<sup>MATRIX</sup> Coverlay, Bond-Ply and Adhesives

- Excellent dielectric performance, high bond strength and dimensional stability.
- Modified acrylic and epoxy adhesives available. Broad selection of adhesive and film thicknesses.
- UL – 94-0 and halogen free available
- Ultra-wide (48") and long rolls available.

**Extensive Local Inventories Available... Contact Us Today!**



**MATRIX**  
CIRCUIT BOARD MATERIALS

Supplier to the North  
American Circuit Board  
Industry since 1977

**N. California**  
3261 Edward Ave.  
Santa Clara, CA 95054  
P (408) 477-2963

**S. California**  
301 West Dyer Rd, Suite E  
Santa Ana, CA 92707  
P (714) 825-0404

**Canada**  
1124 Mid-way Blvd.  
Mississauga, ON L5T 2C1  
P (905) 670-8400

**800-668-5447 • [www.matrixusa.us](http://www.matrixusa.us)**



Dick Crowe checking out the merchandise in the Taylor suite.

tronics in each company's products. And we were not disappointed. We focused on guitars and amplifiers for this article, but we could just as easily have focused on a host of other products: electric pianos, drums, synthesizers, software—you name it. They were all unique and interesting in their own right.

Our first visit was with Pat Quilter, president of Quilter Labs and the founder of QSC Audio located in Costa Mesa, California. Quilter Labs, formed in 2011, currently designs and manufactures solid state guitar amplifiers, but soon plans to introduce a new bass amplifier. Their guitar amplifiers are recognized as outstanding amplifiers that deliver "tube-like sensitivity."

As founder and chief visionary, Pat Quilter is a flamboyant, enthusiastic advocate for his product, and he remains fully immersed in all facets of its development. In fact, Quilter designs his own circuit boards using an old version of P-CAD, a PCB design tool that he is evangelical about. QSC Audio purchases the bare circuit boards and assembles the product using what those of us involved with printed circuit assemblies would view as simple double-sided SMT circuit boards. QSC Audio uses a more sophisticated multilayer product.

Shaughnessy played guitar through a Quilter amplifier, and Quilter demonstrated a lap steel rendition of "All of Me." Quilter is quite an entertainer.

Our next stop was the Taylor Guitars suite. Arguably the largest manufacturer of acoustic-electric guitars in the U.S. Taylor is located near



From left: Andy Shaughnessy, Taylor Master Luthier Andy Powers, Dick Crowe, and Dan Feinberg.

San Diego, in El Cajon.

(Next year's IPC APEX EXPO show will be in San Diego, only a few miles from the Taylor factory. Factory tours are free and start at 1 pm. I have done it twice and will do it again.)

Taylor manufactures an array of top-of-the-line guitars using the finest wood and tools available. Taylor is a strong advocate for sustainable protection of wood and is very selective about how they procure their wood and ebony fretboards. Taylor also co-owns and operates an ebony mill in Yaoundé, Cameroon and is committed, as they say, to providing "ebony parts used on a guitar (that) has been acquired legally and ethically, with a commitment to long-term sustainability."

That being said, Taylor manufactures over 300 guitars per day in the El Cajon facility, and nearly the same number from their Mexico facility a few miles to the south. While Taylor staffers sometimes joke that the company is really still a "wood shop," Taylor uses sophisticated tooling and equipment to deliver consistent quality, guitar after guitar.

So what does this have to do with electronics? A good question. Taylors are relatively expensive guitars, and like any guitar made from solid wood, they can last many years if they're properly cared for. Humidity and shock often affect the playability and sound of the instrument. Good guitars get better with age, and the sound improves, becoming richer and fuller. But only if the guitar is cared for properly.

Many of us guitar players think only about

changing the guitar strings to keep our instruments sounding good, but humidity, temperature changes, and sudden impacts to the guitar all affect its performance. Now, along comes technology from the experts at Taylor to help you monitor all of these factors. In concert with Blustream, Taylor has developed a patented tool called TaylorSense that monitors humidity, temperature, impact, and battery life and communicates with you through an app (Apple iOS for now) on your phone.

TaylorSense provides more than a visual presentation of the health of the guitar. It has its own accelerometer for measuring shock in G force, as well as a digital tuner, four-track recorder and a metronome, all of which are connected via Bluetooth to your phone. A smart battery box replaces the 9V battery box in most Taylor acoustic-electric guitars. Great care has been taken to ensure that the digital signal does not interfere with the analog signals from the various pickups in the guitar. It's an outstanding technology product from a "wood shop" like Taylor!

Our last formal visit was with Fishman, another innovative supplier of high-quality products for electric and acoustic guitars. We were fortunate to meet founder Larry Fishman, who was celebrating the company's 35<sup>th</sup> anniversary during the show. Fishman is well known for a variety of amplifiers, pre-amplifiers, (the so-called "FishStick" being one interesting amplifier) and a variety of other products. Fishman is located in Andover, Massachusetts, not far from the high-technology innovative section of Boston near Route 28.

Fishman was introducing a new guitar pickup, one that is unlike any wire-wound pickup used on electric guitars. This pickup is a 48-layer flex circuit with an etched circular trace, with plated via holes connecting each layer. This unique pickup, called the Fluence, provides, as Fishman says, the "historical Strat tones without the hum." Electrical engineer John Eck was kind enough to spend time with us explaining the development of the product. Certainly this new pickup will advance the state of the art the same way the humbucker pickup did many years ago.

This is but a short synopsis of NAMM 2016. Suffice it to say NAMM is a show of shows unlike any you may have attended before.

## NAMM 2016: True Technology at Last

by **Dan Feinberg**

FEIN LINE ASSOCIATES

I have been covering the National Association of Music Merchants show (NAMM) for the last five years. As a techie and as a musician who has played blues and classic rock for decades, I have always loved going to NAMM. Wandering from hall to hall, you see hundreds of instruments being played all at once. If you stand in one place it can all sound like noise, but as you wander it blends into a pleasant symphonic cacophony.

You transition from blues to grunge rock,



from drums to brass and strings, and through various musical moods. There are rank amateurs and famous performers playing on stages or in exhibitors' booths. Every year, I see literally thousands of musicians, and frankly, I quite often have no idea if they are famous or rank amateurs.

In addition to the well-known and time-tested instruments like Gibson, Gretsch, and Fender, there are also the newer brands that make absolutely beautiful instruments, almost pieces of art, such as Taylor Guitars. I knew of them but I had never tried one. This year Andy Shaughnessy, Dick Crowe and I decided to team up, and Dick set up an interview with Taylor. The company makes a complete high-end line of guitars using the best available wood and manufacturing processes. They are very serious about protecting the supply of the wood and other materials they use, so that they can guarantee the quality of these precision instruments.

Taylor has developed a monitoring system called TaylorSense that will send a message to an app on your smartphone if your instrument is being subjected to conditions such as humidity, temperatures, or even shock caused by impact outside of normal. My thought as I listened to the briefing Taylor so kindly provided was, "At last, true technology at NAMM."

Our next interview was with Fishman, a company known for producing a number of amplifiers as well as a number of other electronic products. One device I found very interesting. It was a kind of synthesizer that allows musicians to convert the sound into that of any other in-



strument. One young man demonstrated how the sound of his guitar could be converted flawlessly from that of a guitar to a saxophone, a full pipe organ, and anything in between.

Fishman has also designed a guitar pickup made by using a 48-layer PCB instead of countless yards of wound copper wire. So what we have is a 48-layer PCB generating excellent sound, with no detectable hum. Here we have the emergence of advanced PCB manufacturing technology showing its face at NAMM.

When I first started covering NAMM five years ago, I found new/old vacuum tube amplifiers, new solid state amplifiers claiming to duplicate "that vacuum tube sound," and a few amps that actually did just that. I found pre-amps, amps, and recording and interface devices, but not a lot of true technological advances. There were lots of new apps and stands and covers for iPads, but not much in the way of ground breaking new technology.

This year we are starting to see true advances: The IoT is beginning to make inroads into the world of musical instruments. We also saw advances in recording hardware, software, and apps. I expect that in the years to come we will see even more. This year at CES, 3D printing advancements were evident in musical instrument design and manufacturing. What will we see in the way of 3D produced musical instruments and devices at NAMM in future years?

Be sure to look for interviews in future issues of *The PCB Design Magazine* for more about the exciting advancements that cover only part of the growing, expansive and exciting event that NAMM has become. **PCBDESIGN**



# Pulsonix

## Advanced Schematic Capture and PCB Design Software

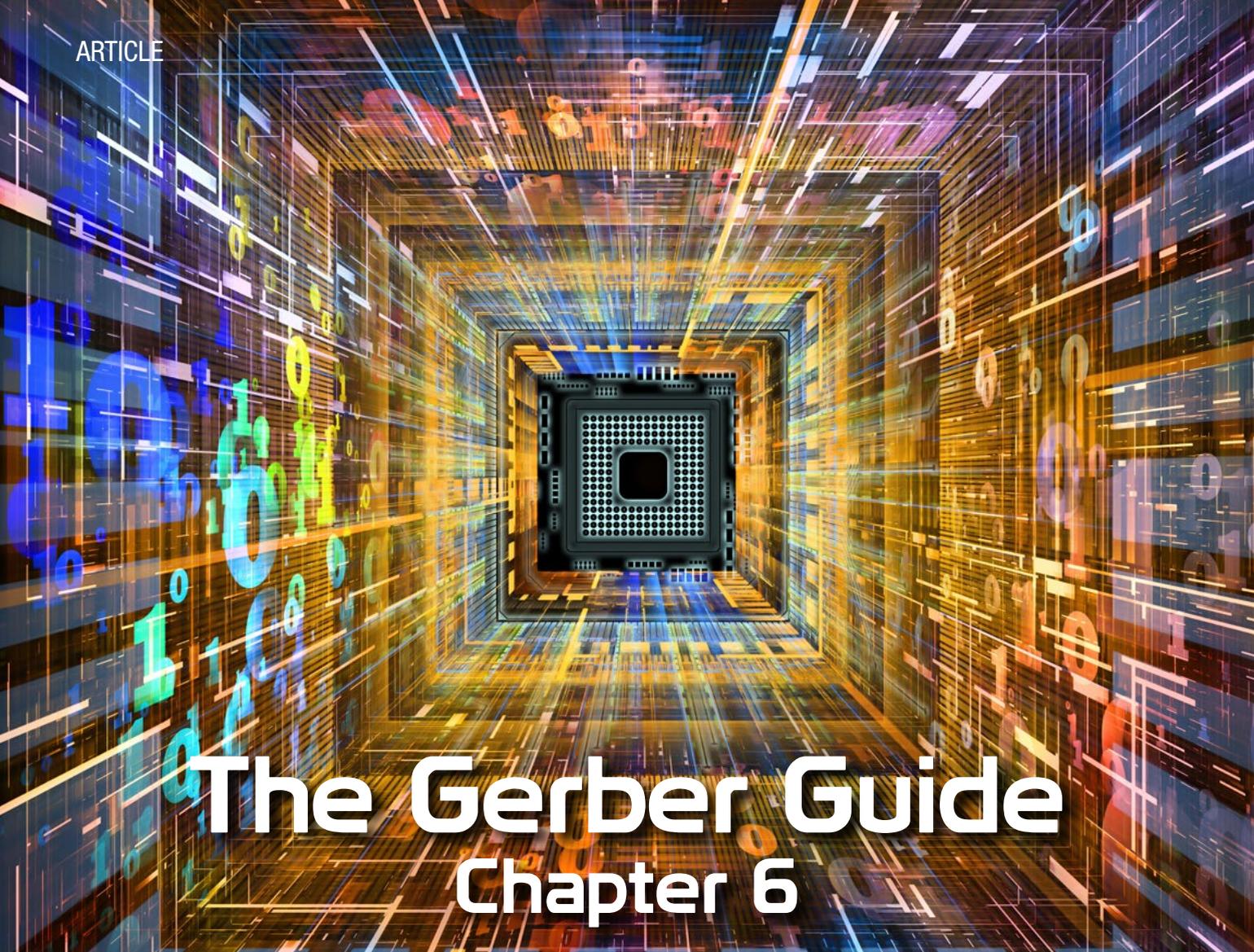
### A refreshing approach to simplifying Schematic Capture and PCB Design

- Import filters for all major tools (designs and libraries)
- Very easy to learn and use
- 5-Star Service and Support

[Download a Free Trial](#)

[info@pulsonix.com](mailto:info@pulsonix.com)  
[www.pulsonix.com](http://www.pulsonix.com)

**PULSONIX**  

# The Gerber Guide

## Chapter 6

**Karel Tavernier**  
UCAMCO

*It is possible to fabricate PCBs from the fabrication data sets currently being used; it's being done innumerable times every day all over the globe. But is it being done in an efficient, reliable, automated and standardized manner? At this moment in time, the honest answer is no, because there is plenty of room for improvement in the way in which PCB fabrication data is currently transferred from design to fabrication.*

*This is not about the Gerber format, which is used for more than 90% of the world's PCB production. There are very rarely problems with Gerber files themselves; they allow images to be transferred without a hitch. In fact, the Gerber format is part of the solution, given that it is the most reliable option in this field. The problems actually lie in which images are transferred, how*

*the format is used and, more often, in how it is not used.*

*Each month we look at a different aspect of the design to fabrication data transfer process. In this monthly column, Karel Tavernier explains in detail how to use the newly revised Gerber data format to communicate with your fabrication partners clearly and simply, using an unequivocal yet versatile language that enables you and them to get the very best out of your design data.*

### **Chapter 6: Drill File Structure**

Drill files specify geometric information, indicating where material is to be removed and where plated copper is to be added. This geometric data must be specified in a standardized and machine-readable way.

Geometrically, drill holes are characterized by their diameter, their span (start and end layers) and whether they are plated or non-

**APEX  
EXPO<sup>®</sup>  
IPC 2016**

# **FORWARD THINKING**

**FOR TOMORROW'S TECHNOLOGY**

## **Unparalleled Access**

What awaits you on the IPC APEX EXPO 2016 show floor? The largest concentration of advanced technologies from the electronics industry supply chain — PCB fabricators, designers, OEMs, EMS companies and more. If you want to experience their forward thinking, you need to be here.

**IPC APEX EXPO 2016 is where ingenuity will be unveiled.**

**Prepare for tomorrow in 2016. Meet customers, network and be inspired by all that you experience on the show floor.**

**Forward thinking begins at [www.IPCAPEXEXPO.org](http://www.IPCAPEXEXPO.org).**

## **LAS VEGAS/NEVADA**

**LAS VEGAS CONVENTION CENTER  
CONFERENCE & EXHIBITION: **March 15-17****

**MEETINGS & COURSES: **March 13-17****

**[www.IPCAPEXEXPO.org](http://www.IPCAPEXEXPO.org)**



.FileFunction value	Usage
Plated,i,j,<type>[,<label>]	<p>Plated drill/rout from layer i to layer j. The mandatory type can take the values <i>PTH</i>, <i>Blind</i>, <i>Buried</i>.</p> <p>The optional label can take the values <i>Drill</i>, <i>Route</i>, <i>Mixed</i>; these labels are not exactly defined; the file creator uses the value he reasonably finds appropriate.</p>
NonPlated,i,j,<type>[,<label>]	<p>Non-plated drill/rout from layer i to layer j. The mandatory type can take the values <i>NPTH</i>, <i>Blind</i>, <i>Buried</i>.</p> <p>The optional label is explained in the row above.</p>

Table 1.

plated. The diameter is easily specified by the aperture (tool) diameter in the Gerber drill file, or the NC drill file for that matter. Span and plating are transferred in Gerber with the ‘.FileFunction’ attribute. To quote from the Gerber format specification (see Table 1).

As can be seen from this excerpt, the Gerber format provides unequivocal language to describe drill span and plating. It should be clear from this that any one file will describe one drill span and one plating instruction, so different spans and plating instructions must be put into separate files. A typical PCB fabrication data set will therefore contain several drill files: one for PTH holes, another for NPTH holes, and others for the different blind and buried spans. By creating data sets in this way, we can ensure that the whole drill file structure is standard and can be read automatically.

If you are not able to add attributes to the file in this way, the CAM operator must manually determine the file function on his CAM system. In this case, you will be providing the information informally, but it should still be as simple and unequivocal as possible. The best way to achieve this is to make the function clear in the file name (e.g., NPTH.GBR). A more indirect method is to list the files and their functions in a text file.

Sometimes both plated and non-plated holes are lumped together in a single file. Some argue that this is OK because the drill map indicates which holes are plated. The drill map does indeed indicate this (usually). But it is not OK.

Drill data must be standardized and machine-readable. Drill maps are neither standardized nor machine readable; they must be read offline by CAM operators, who must work out the drill coordinates visually and then indicate manually which holes are plated and which are not. This terrible practice is bad enough if the plated and non-plated holes are specified using different tool numbers. But where plated and non-plated holes have the same diameter and the file is “optimized” by using the same tool for both, it becomes really excruciating work for the CAM engineer, and risky for the successful outcome of CAM and production. This is why plated and non-plated holes should always be put in separate files. If this is not possible, at least use separate tools for them.

One more thing. In some cases, fabrication data comes in with just a drill map, and no digital drill data at all. This is simply unacceptable. Drill machines cannot read a drill map: drill machines have been using CNC data for decades. Without digital drill data, the CAM operator must pore over the drill map,

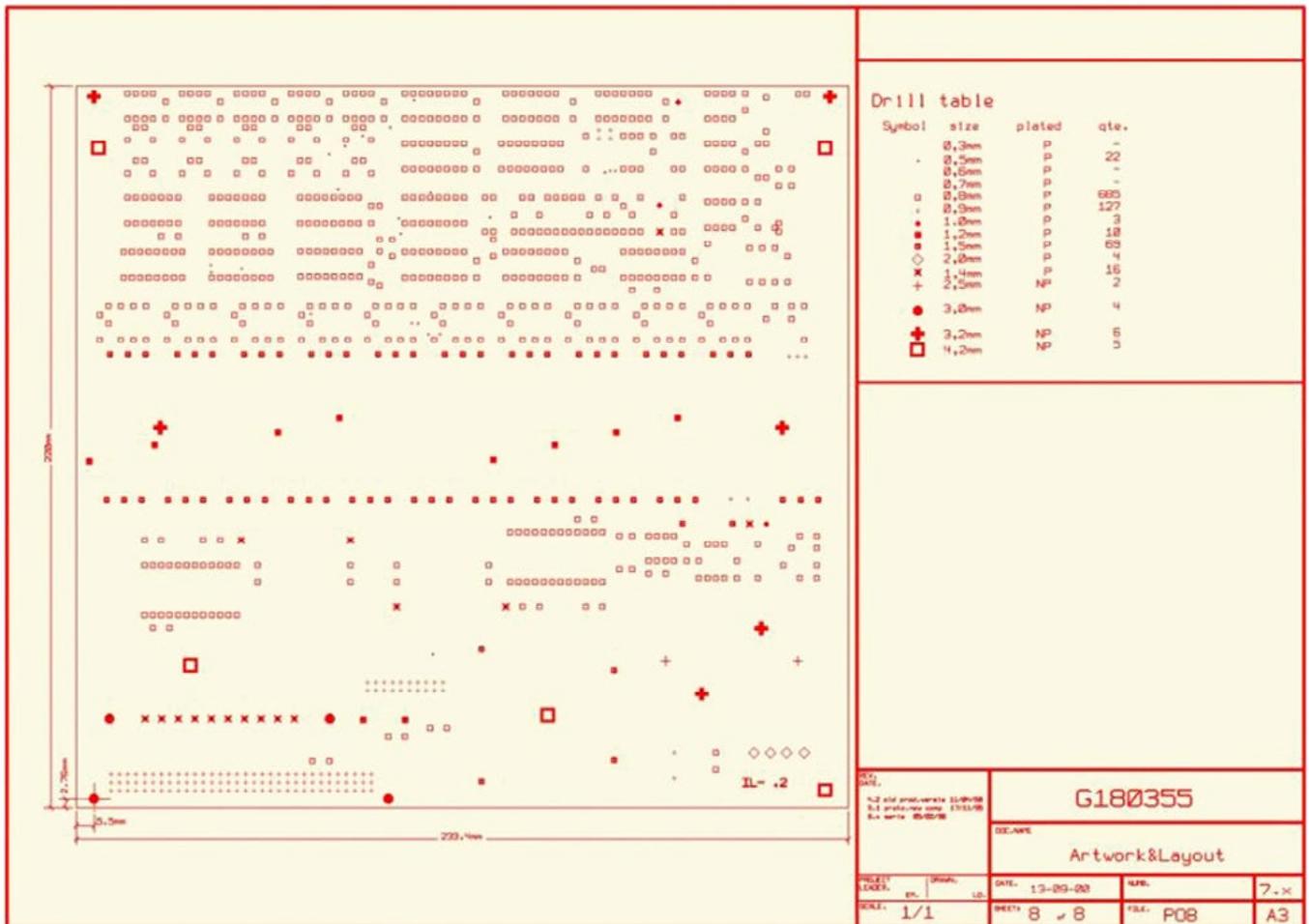


Figure 1: A drill map. It may be useful, but it's not a substitute for drill data.

measuring and manually reconstructing the drill data from it, in a laborious and error-prone process. Don't do this—have mercy on the poor CAM engineer!

Note that the considerations in this chapter apply whether the drill files in your fabrication data sets are in NC or Gerber format. (In Chapter 4 in this series, we argue that Gerber is by far the best choice). Whatever format you choose, the bare minimum is that you create a separate file for each span, and for plated and non-plated holes, and that you clearly indicate which is which.

These requirements for drill data also hold true for rout data. Even though drilling and routing are very different fabrication processes, the difference between them is largely irrelevant when they are viewed as fabrication image data: both simply indicate where material is removed. Indeed, the fabricator may very

well nibble a slot in a rout file, or rout a large hole. In conclusion, the fabrication data must specify what the fabricator must fabricate. The fabricator will decide how to fabricate it.

And remember: Put plated and non-plated holes in separate files.

This column has been excerpted from the [Guide to PCB Fabrication Data: Design to Fabrication Data Transfer](#).



**Karel Tavernier** is managing director of Ucamco. Karel has 30 years' experience in software and imaging equipment for the PCB and electronic printed packaging industry, including sales, service and R&D.

# TOP TEN



## Recent Highlights from PCBDesign007

### 1 Cadence's Steve Chidester Loses Battle with Cancer

Our friend and colleague Steve Chidester has succumbed to cancer. Steve was in marketing for more than 30 years, as product marketing manager for Cadence Design Systems and later as VP of marketing for Zuken, before returning to Cadence. Steve was a great guy, always willing to talk with the media. We'll miss him.



### 2 Designers Notebook: Flex and Rigid-Flex Circuit Design

Flexible circuits represent an advanced approach to total electronics packaging, typically occupying a niche that replaces ordinary printed circuit board assemblies and the hard-wire interface needed to join assemblies. Flex circuits have an advantage over hard-wire interface because they fit only one way, eliminating wire routing errors as well as the time needed for testing and inspection.



Vern Solberg

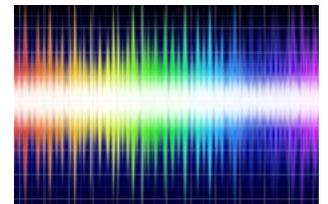
### 3 Mentor Graphics Names 26<sup>th</sup> Annual PCB Technology Leadership Awards Winners

Mentor Graphics Corporation has announced the winners of its 26th annual PCB Technology Leadership Awards. Started in 1988, this program recognizes engineers and designers who use innovative methods and design tools to address today's complex PCB systems design challenges.



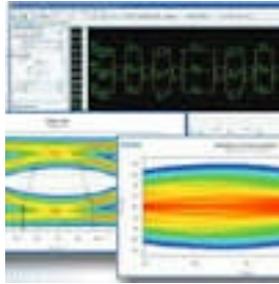
### 4 Systematic Estimation of Worst-Case PDN Noise: Target Impedance and Rogue Waves

In the dark ages of power distribution design, the typical advice was to use a bulk capacitor and one 0.1uF bypass capacitor for every power pin on the digital circuit. This was very unscientific, but served the industry reasonably well in low-density and low-speed circuits. As the designs got more demanding, the target impedance concept was developed.



## 5 SiSoft Announces OptimEYE Technology for AMI Simulations

SiSoft has announced OptimEye, which works with SiSoft's Quantum Channel Designer software to automatically identify the right combination of TX & RX settings to maximize a serial link's operating margin. OptimEye identifies the ideal balance without requiring large "blind sweeps" of settings normally used to search a design space for potential solutions.



## 6 Medical PCB Design: Not Just Another High-Rel Board

Some of the coolest new electronic products have come courtesy of the medical market. I wanted to find out more about this fast-growing segment, so I contacted Kenneth MacCallum, an engineering physicist with StarFish Medical. StarFish is a medical device design company that's created some major electronic medical innovations, and they're about as cutting-edge as you can get.



## 7 New DownStream Training Dates: March 2 and July 20

This technical training course will focus on how to use the powerful features within BluePrint to generate documents and documents sets. Emphasis will be placed on using the Gallery and Document Templates to automate the process. Training courses are always a good investment, in yourself and in your software.



## 8 McCurdy: How to Build a Successful IPC Designers Council Chapter

As a Southern California native and an industry veteran, the name Scott McCurdy is a familiar one for me. Scott has been in the electronics industry for 40+ years. Thirty-two of those years were spent building and running McCurdy Circuits, a successful PCB fabrication facility in Orange County. These days, Scott is focused on the design side of things as the director of sales and marketing for Freedom CAD Services, the largest independent design bureau in the nation.



## 9 Zuken Tempts Engineers with New Year CADSTAR Giveaway

Zuken is offering electronic engineers who made New Year's reso-



lutions to try new design software the chance to achieve this now at no cost. The free download of CADSTAR Schematics Design Tool, worth more than \$500, will allow the whole design team to instantly explore CADSTAR's integrated design flow. The offer runs through March 31, 2016.

## 10 Electrical Design Challenges for Automotive PCBs

A recent article in The PCB Design Magazine by Monica Andrei of Continental Automotive Systems emphasized the systemic nature of an automobile and discussed the characteristics and adoption of software design tools to enable such system-level design. This article will present more detail on signal integrity.



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

# Events



For the 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](#).

**[The Changing Landscape of REACH](#)**

February 10, 2016: Brea, California, USA  
February 17, 2016: Herndon, Virginia, USA

**[FlexTech Alliance](#)**

February 29–March 3, 2016  
Monterey, CA USA

**[ICT-UK Evening Seminar](#)**

March 1, 2016  
Tewksbury, England

**[Dallas Expo & Tech Forum](#)**

March 3, 2016  
Plano, Texas, USA



**[IPC APEX EXPO 2016](#)**

March 13–17, 2016  
Las Vegas, Nevada USA

**[25<sup>th</sup> China International PCB & Assembly Show 2016 \(CPCA\)](#)**

March 15–17, 2016  
Shanghai, China

**[2016 Annual Foundation Course \(ICT\)](#)**

April 11–14, 2016  
Loughborough, England

**[Thailand PCB Expo 2016](#)**

April 19–22, 2016  
Bangkok, Thailand

**[JPCA Show 2016](#)**

June 1–3, 2016  
Tokyo, Japan

**[IPCA EXPO 2016](#)**

August, 2016  
India

**[IPC Fall Meetings](#)**

September 24–30, 2016  
Rosemont, IL USA

**[SMTA International 2016](#)**

September 25–29, 2016  
Rosemont, IL USA

**[electronicAsia](#)**

October 13–16, 2016  
Hong Kong

**[electronica](#)**

November 8–11, 2016  
Munich, Germany

**[International Printed Circuit & Apex South China Fair \(HKPCA\)](#)**

December 7–9, 2016  
Shenzhen, China

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 IMAGE: **LONELY, DOLLAR PHOTO CLUB**



The PCB Design Magazine® is published by BR Publishing, Inc., PO Box 50, Seaside, OR 97138 ©2016 BR Publishing, Inc. does not assume and hereby disclaims any liability to any person for loss or damage caused by errors or omissions in the material contained within this publication, regardless of whether such errors or omissions are caused accidentally, from negligence or any other cause.

February 2016, Volume 5, Number 2 • The PCB Design Magazine© is published monthly, by BR Publishing, Inc.

## ADVERTISER INDEX

Accurate Circuit Engineering....	23	Matrix.....	57
American Standard Circuits.....	31	Mentor Graphics.....	7
ANS.....	35	Miraco.....	19
Candor Industries.....	53	Nujay Technologies.....	11
Dibble Leaders.....	3	Oak Mitsui.....	45
Downstream Technologies.....	33	Prototron Circuits.....	49
Eagle Electronics.....	15	Pulsonix.....	61
Electrolube.....	51	Rogers Corporation.....	55
EMA Design Automation.....	29	Schweizer Electronic AG.....	43
Fast Interconnect.....	25	Shengyi Technology.....	17
Freedom CAD.....	13	Sunstone Circuits.....	39
I-Connect007.....	70	Technica.....	37
Intercept.....	47	The PCB List.....	2
IPC.....	63	US Circuit.....	27
Isola.....	5	Ventec.....	21

**Coming Soon to  
The PCB Design  
Magazine:**

**March:  
Design  
Strategies for  
Increased Profits**

**April:  
Design  
Engineering**

**May:  
Focus on Design  
Automation**

# I-Connect007

GOOD FOR THE INDUSTRY



FREE SUBSCRIPTION



[subscribe.icconnect007.com](http://subscribe.icconnect007.com)

[mediakit.icconnect007.com](http://mediakit.icconnect007.com)

**EDITORIAL CONTACT**

**Andy Shaughnessy**  
andy@icconnect007.com  
+1 404.806.0508 GMT-5

**SALES CONTACT**

**Barb Hockaday**  
barb@icconnect007.com  
+1 916 365-1727 GMT-7



[www.icconnect007.com](http://www.icconnect007.com)