

THE **pcb**
design
MAGAZINE

September 2017

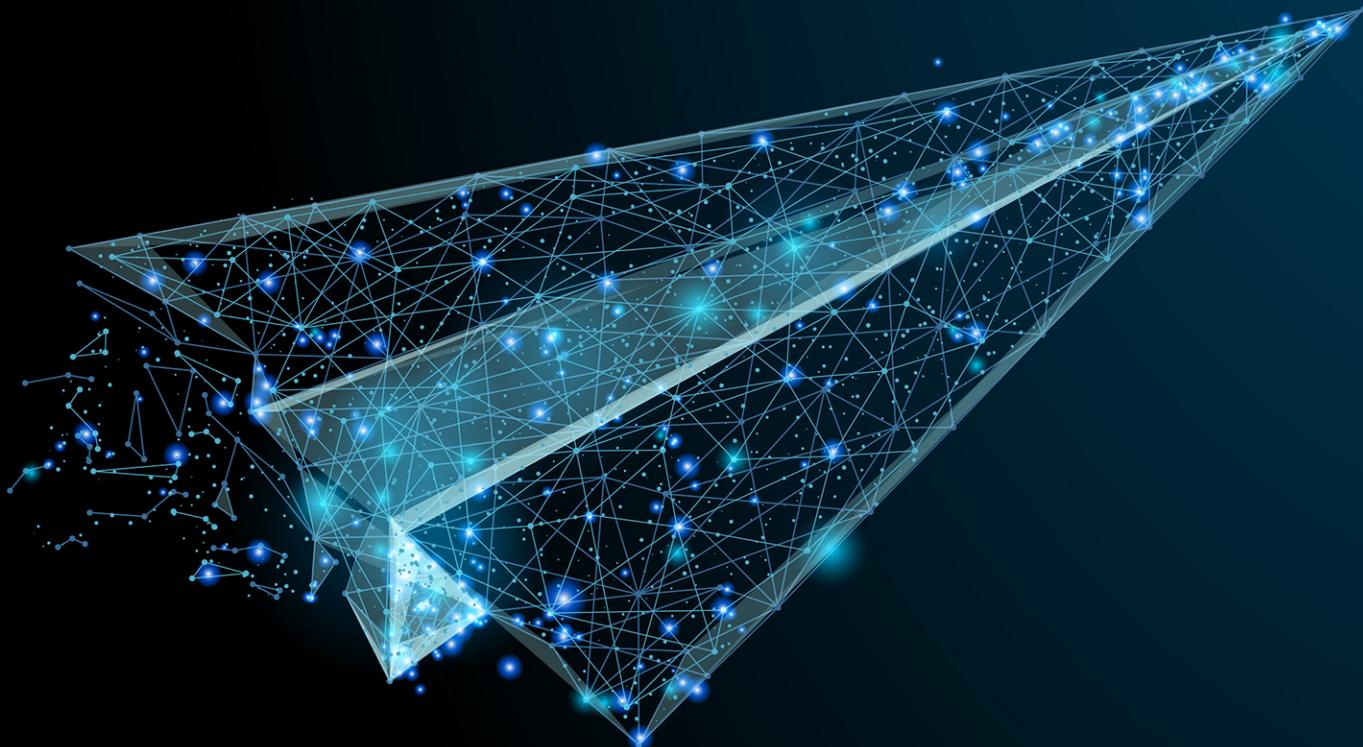
an IConnect007 publication

**Predictive Engineering:
Happy Holden Discusses
True DMF** p.12

**The History of Predictive
Engineering** p.24

**It's Not Your Father's
DFM Anymore** p.30

Much More!

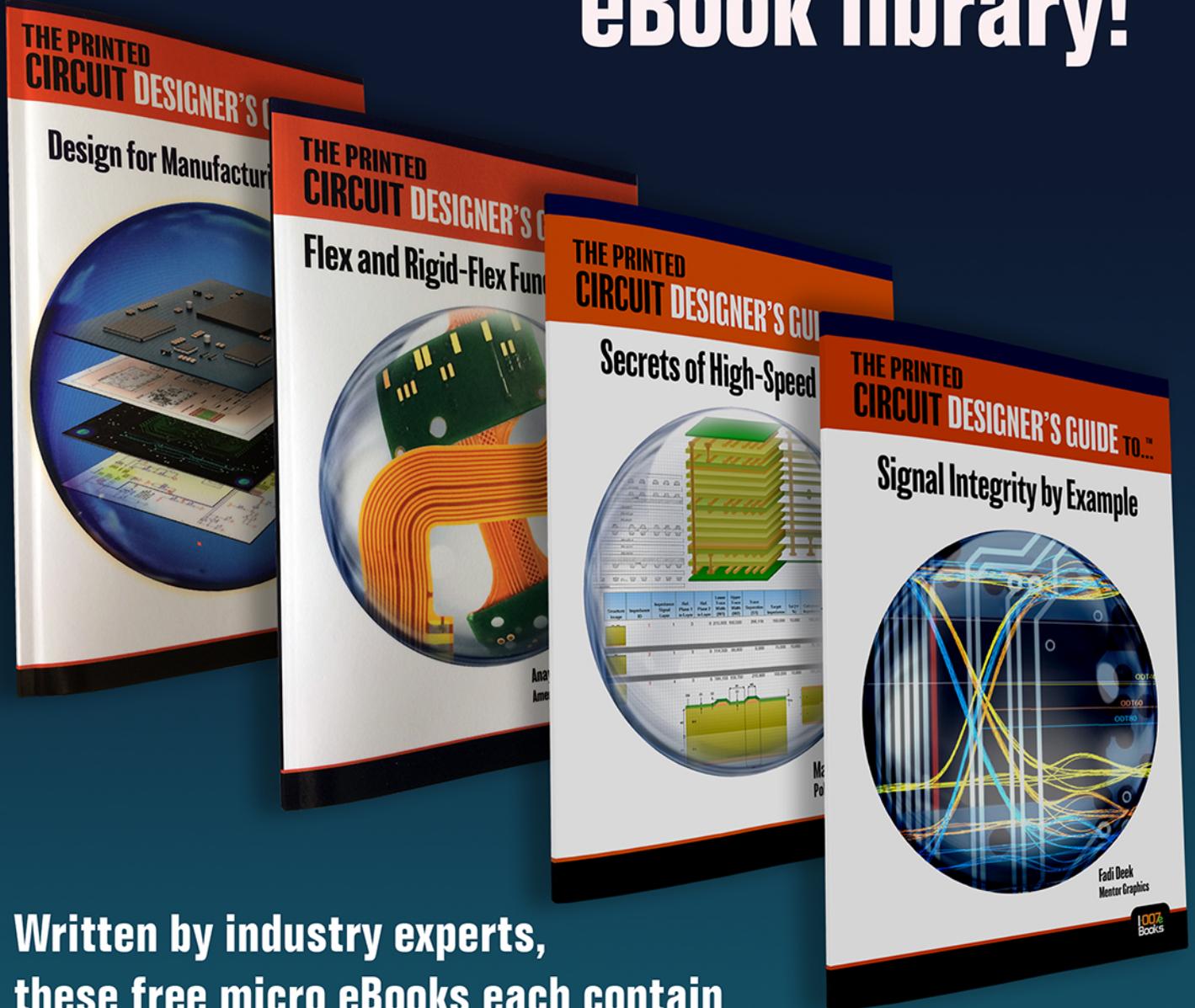


**DFM SIMPLY
REDEFINED**

isola
The base for innovation



Download free books from our micro eBook library!



Written by industry experts, these free micro eBooks each contain valuable information on relevant topics in an easy-to-read format.

Log in or register to access all of our premium content at: myconnect007.com



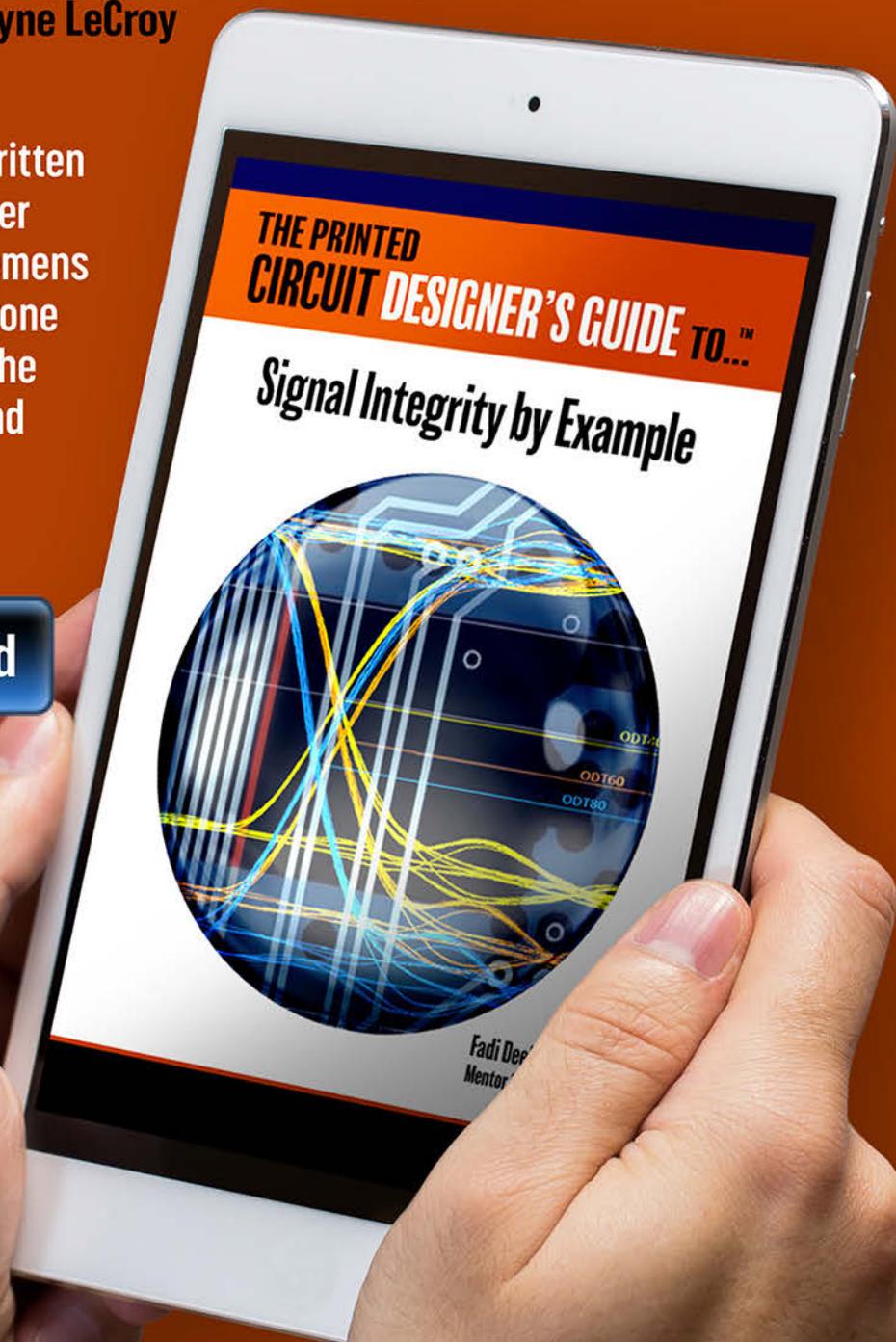
“ If you care about signal integrity, you are sure to pick up a few nuggets of insight from this new eBook. ”



Eric Bogatin
Dean, Signal Integrity Academy
Teledyne LeCroy

This FREE micro eBook, written by signal integrity engineer Fadi Deek of Mentor, A Siemens Business, will benefit anyone interested in identifying the problems, root causes, and solutions surrounding electronic transmissions.

 **Free Download**





DFM

We've heard it for decades: The days of throwing a design over the wall are ancient history. After all, designers have embraced fully optimized DFM processes. Right? Maybe not.

DFM continues to cause designers problems. Even designers seem to realize that they might not know all they need to know about fabrication.



- 12 Predictive Engineering:
Happy Holden Discusses True DFM**
I-Connect007 Interview



- 24 The History of Predictive
Engineering**
by Happy Holden

- 30 It's Not Your Father's
DFM Anymore**
by Patrick McGoff



- 38 An Ounce of DFM Error
Prevention is Worth
a Pound of Cure**
*by Hemant Shah
and Ed Acheson*

Very Low Loss Laminate & Prepreg

Astra® MT77

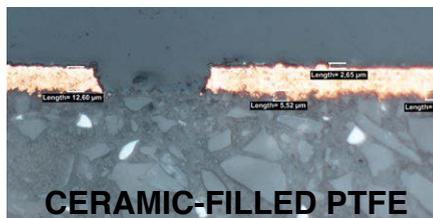
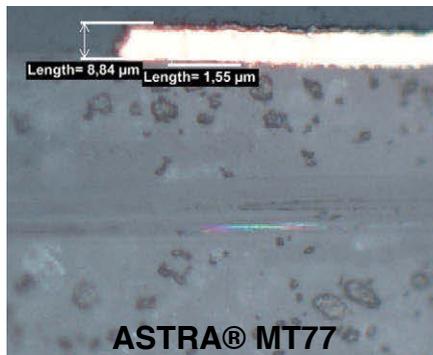
Low Cost
Teflon® Replacement for
Radar Applications

RF/
MICROWAVE

Astra® MT77

Astra® MT77 laminate materials exhibit exceptional electrical properties which are very stable over a broad frequency and temperature range.

© 2017 ISOLA GROUP. ALL RIGHTS RESERVED.



Tg: 200°C Td: 360°C

Ultra-low loss material with
Dk: 3.00 and Df: 0.0017

Comparable high-frequency
properties to Teflon® up to 100 Ghz

Stable Dk over temperature range:
-40 to 140°C and up to 125 GHz

Extremely simple PCB processing
(short press cycle, standard drill
route & no plasma, excellent
prepreg flow, sequential lamination)

Complete design freedom for
RF designers (any layer technology,
HDI, microvia, buried via, blind via,
buried resistor, sequential lamination)
UL RTI 130°C / UL 94 V-0

Best Price/performance Teflon®
replacement material on the market

RF/MICROWAVE • AEROSPACE & DEFENSE • AUTOMOTIVE & TRANSPORTATION

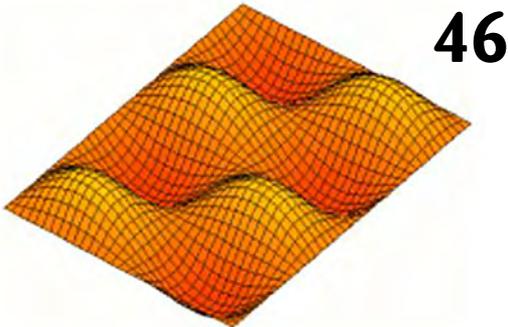
isola

FOR MORE INFORMATION, CONTACT YOUR LOCAL SALES REPRESENTATIVE.

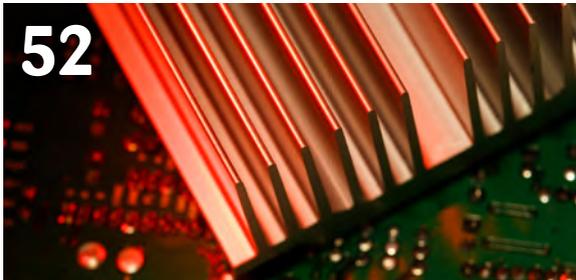
Isola Group
3100 West Ray Road
Suite 301
Chandler, AZ 85226
Phone: 480-893-6527
info@isola-group.com

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

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



46



52



58

COLUMNS

8 **Redefining DFM**
by Andy Shaughnessy

46 **Plane Cavity Resonance**
by Barry Olney

52 **Thermal Management:
Problems and Solutions**
by Jade Bridges

58 **True Design Efficiency:
Think Before You Click**
by Tim Haag

HIGHLIGHTS

44 **PCB007**

56 **MilAero007**



72 **Top Ten PCBDesign007**

DEPARTMENTS

64 **Career Opportunities**

74 **Events Calendar**

75 **Advertisers Index & Masthead**

SHORTS

10 **First On-chip Nanoscale Optical
Quantum Memory Developed**

22 **A New Way to Print
Electrical Circuits**

28 **Integrated Lasers on
Different Surfaces**

62 **Researchers Challenge
Status Quo of Battery
Commercialization**

FAILURE HERE IS NOT AN OPTION

**Our 'Can't-Fail Scenario' Attitude
is built on 40+ years of industry-wide
reliability and experience.**

We specialize in HDI Rigidflex for Mil/Aero and Medical applications, are certified to NADCAP and AS9100 as well as meeting all applicable ITAR Compliancy requirements.

Let us help extend the capabilities, reliability and technology of your next design!

**LEARN MORE ABOUT
OUR HDI SOLUTIONS**

eltek

a  Nistec company

www.nisteceltek.com



Redefining DFM

by Andy Shaughnessy

I-CONNECT007

We've heard it for decades: The days of throwing a design over the wall are ancient history. After all, designers have embraced fully optimized DFM processes. Right?

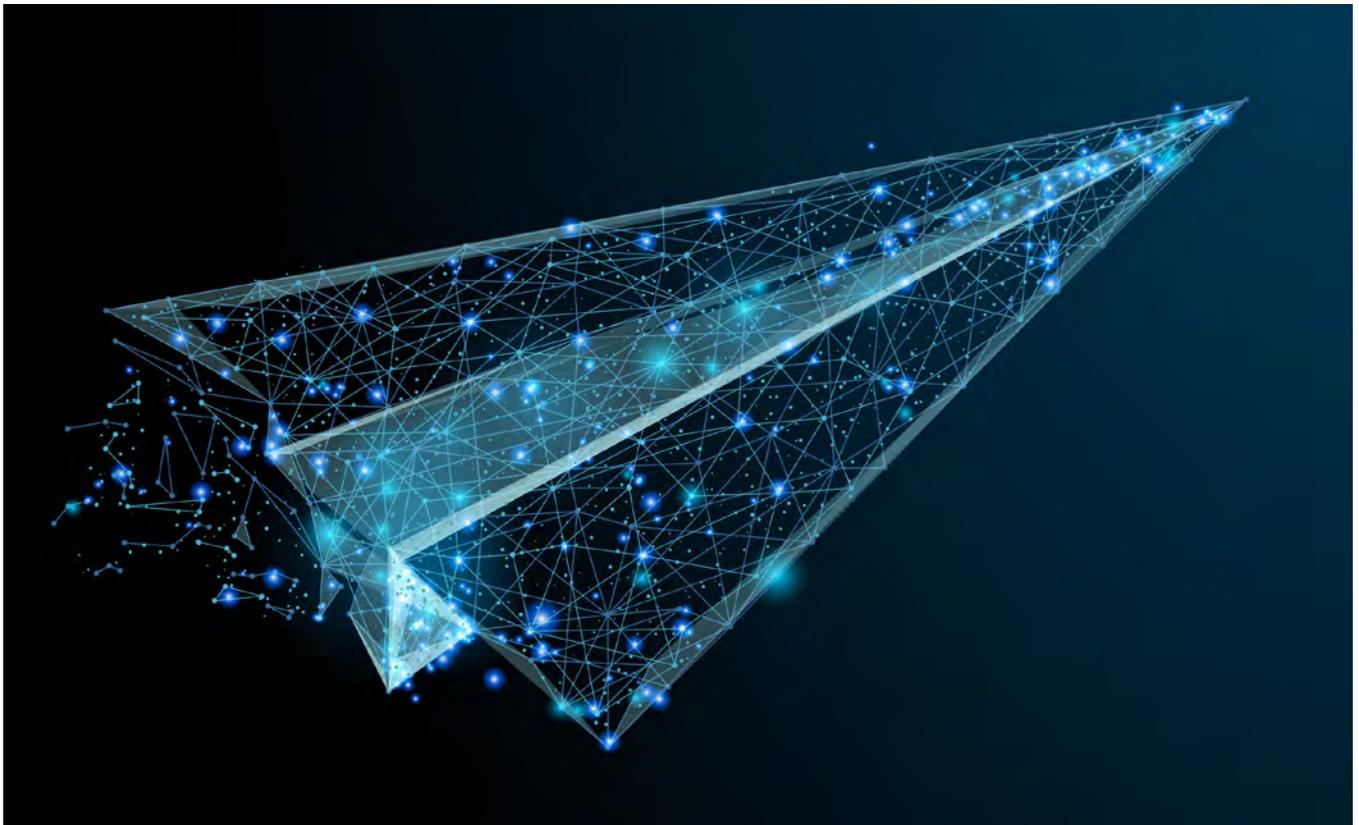
Maybe not. When we survey our readers about the biggest challenges they face, DFM is mentioned almost every time. Whether it's a result of unclear DFM guidelines, a lack of time for DFM review, or outdated design rules, DFM continues to cause designers problems. Even designers—not all of them, of course—seem to realize that they might not know all they need to know about fabrication.

Now, board shops are offering facility tours to help PCB designers and design engineers understand exactly how their designs are transformed into a PCB. Columnist Mark Thompson leads tours of Prototron Circuits, and he's found that even veteran designers are shocked by how

much they didn't know about the fabrication processes of today. Most PCB designers haven't been in a board shop in years, if ever. Fabricators feel that this is a big part of the problem.

Designers are certainly hungry for information about manufacturing. Some of Mark's columns focusing on what happens to PCB design data in the CAM department have garnered over 7,000 views. And DFM classes at trade shows and conferences draw a steady crowd.

But Consulting Technical Editor Happy Holden believes that there's much more to understanding DFM than taking a tour of a board shop. Happy has been involved with DFM for over 45 years, when he first applied engineering knowledge to designing and manufacturing PCBs at Hewlett-Packard. As he points out in this issue, PCB designers need to understand that true DFM is really a form of predictive en-



Altium®

AltiumLive 2017: ANNUAL PCB DESIGN SUMMIT

Training Courses

Technical Sessions

Design Challenge

Industry Keynotes

MEET THE KEYNOTE SPEAKERS

NORTH AMERICA



Happy Holden
The “Father” of HDI



Dan Beeker
Principal Engineer,
NXP Semiconductors

EUROPE



Lee Ritchey
Founder & President,
Speeding Edge



Andreas Doering
Research Scientist
at IBM Research
Laboratory

Learn, connect, and get inspired
at this special 2-day event!

North America

San Diego, California
October 3rd-4th, 2017

Europe

Munich, Germany
October 24th-25th, 2017

Learn more and register at:
www.altium.com/summit

gineering, and designers need to start seeing themselves as the front end of manufacturing if they're going to support true DFM.

How many of you designers consider yourselves to be the front end of manufacturing? Probably not many. You're most likely working at an OEM that may be hundreds or thousands of miles from your fabricator.

But as Happy says, when you get down to brass tacks, this is all part of manufacturing.

In our feature interview, Happy discusses the importance of predictive engineering in DFM, and why some PCB designers equate a design rules check with DFM guidelines. In our second feature, Happy discusses the history of predictive engineering, and how it should be utilized by PCB designers and manufacturers.

Next, Mentor's Pat McGoff discusses the evolution of DFM software, and the importance of good DFM processes during new product introduction (NPI). And Hemant Shah and Ed Acheson of Cadence Design Systems focus on new DFM tools that use predictive engineering to provide feedback to designers so errors can be addressed in real-time, not after the design is completed.

We also bring you columns by our regular contributors Barry Olney of In-Circuit Design, Jade Bridges of Electrolube, and consultant Tim Haag.

On With the Show

It's September, and that means it's show time. Assuming Hurricane Irma doesn't disrupt air travel out of Hartsfield-Jackson Airport in Atlanta, I'll be heading out to PCB West in Silicon Valley, and then on to Chicago for SMTA International. I hope to see you all on the road. But if you can't make it, we'll have all the latest news at I-Connect007.

Next month, we'll focus on signal integrity. As the old saying goes, "If you don't have signal integrity problems yet, you will." If you're not already a subscriber, [click here](#) so you don't miss out! **PCBDESIGN**



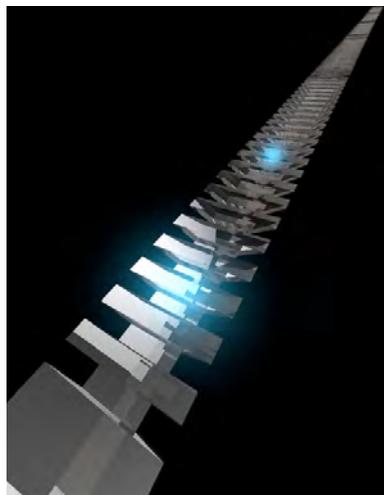
Andy Shaughnessy is managing editor of *The PCB Design Magazine*. He has been covering PCB design for 18 years. He can be reached by clicking [here](#).

First On-chip Nanoscale Optical Quantum Memory Developed

For the first time, an international team led by engineers at Caltech has developed a computer chip with nanoscale optical quantum memory.

"Such a device is an essential component for the future development of optical quantum networks that could be used to transmit quantum information," says Andrei Faraon (BS '04), assistant professor of applied physics and materials science in the Division of Engineering and Applied Science at Caltech, and the corresponding author of a paper describing the new chip.

The new quantum memory chip is analogous to a traditional memory chip in a computer. Both store information in a binary code. With traditional memory,

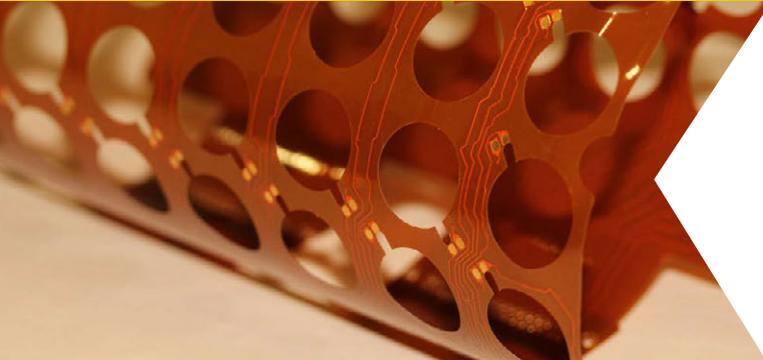


information is stored by flipping billions of tiny electronic switches either on or off, representing either a 1 or a 0. That 1 or 0 is known as a bit. By contrast, quantum memory stores information via the quantum properties of individual elementary particles. A fundamental characteristic of those quantum properties is that they can exist in multiple states at the same time. This means that a quantum bit (a qubit) can represent a 1 and a 0 at the same time.

The team also plans to work on ways to integrate the quantum memory into more complex circuits, taking the first steps toward deploying this technology in quantum networks.

We deliver Advanced PCB Solutions!

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



Impedance Tolerance <5%!

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

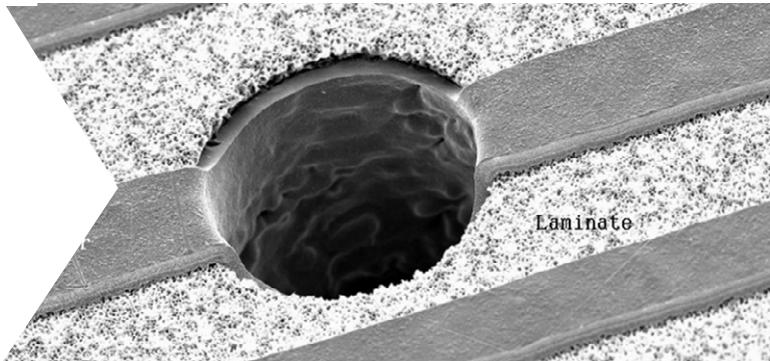
[CLICK TO LEARN MORE](#)



Landless Via Technology

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

[CLICK TO LEARN MORE](#)



Laminate



Tensile Strength achieved with Candors Panel Plating

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

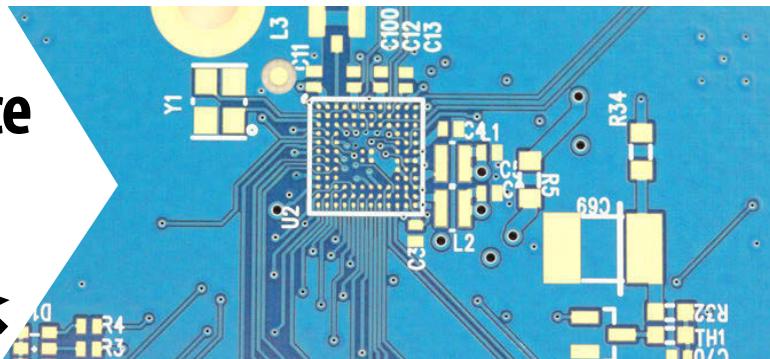
[CLICK TO LEARN MORE](#)



Fine line PCB, 2 mil line /space

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

[CLICK TO LEARN MORE](#)



PHONE: (416) 736-6306



WEB: www.candorind.com

SALES: sales@candorind.com



by I-Connect007

Happy Holden has been involved in DFM for over 45 years, since he first started working at HP and optimized their PCB design and manufacturing processes. Naturally, for this issue, Barry Matties and Andy Shaughnessy made it a priority to get Happy's thoughts on DFM, and what true DFM entails.

Andy Shaughnessy: Happy, why don't you start by telling us about your position on DFM, and why true DFM, to you, means that PCB designers and engineers should utilize predictive engineering.

Happy Holden: If you use the term "predictive engineering," you're not going to get much recognition, because that's a term I use. What most people consider design for manufacturing, or DFX, is software that finds errors in design. Especially in the CAM tooling put out by Valor, the DFX software scans through that and finds

what's cautionary and that all requirements are met. That's all done after the fact.

For me, DFM was introduced when HP took up DFM. It was kind of invented by Professors Peter Dewhurst and Geoffrey Boothroyd. Anyway, DFM and DFMA (design for manufacturing and assembly) were really invented by these two American professors in New England. They wrote a book about it, *Product Design for Manufacture & Assembly*. It later became software.

Their whole philosophy centered on ways to figure out how to do it right the first time. Their book is about the nature of performance during design that allow you to decide if this is going to be easy to assemble. I think the more important component, which Hitachi and the Japanese took it up, was whether the product could be built by automation. That's really significant what Hitachi did with the philosophy. They not only took the 'do it right the first time' approach but then asked how they could do it simple enough that robots and automa-

If Crashes Are Unacceptable

OrCAD is the only acceptable design software.

[Learn More >](#)

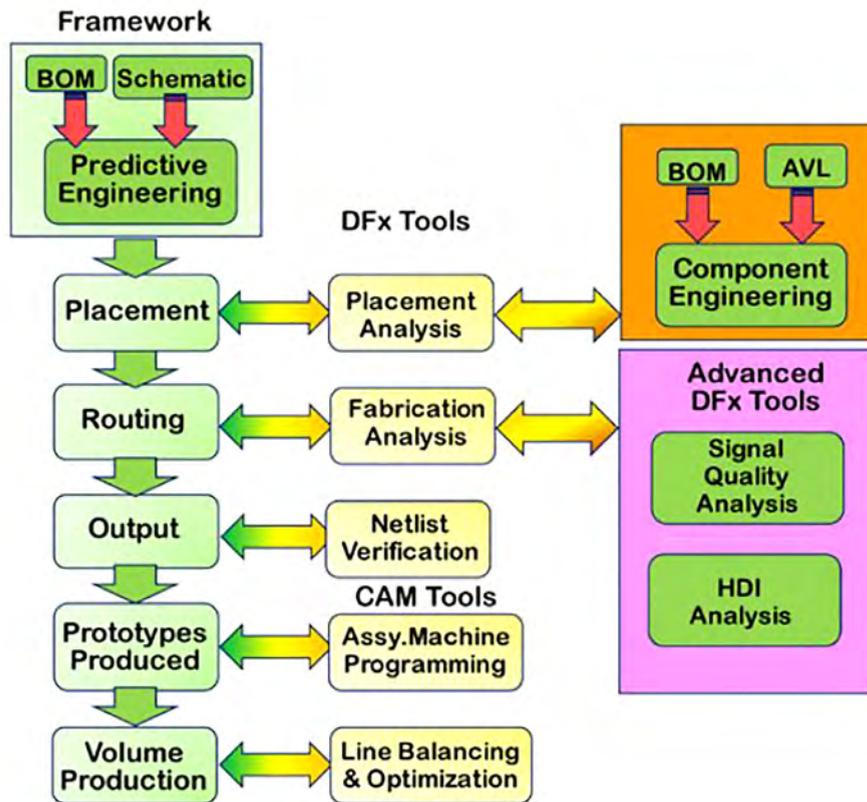


Figure 1: Predictive engineering contains more than just “density modeling.” It provides caution on “failure-prone components” and “suitability for test,” as well as optimizing electrical performance and minimizing signal integrity problems.

tion can build it? The Japanese did this because they wanted to build all these first-generation products in a building next door in Japan and not ship it offshore like Americans were doing.

Contrast that with Apple Inc. Apple designs products that are impossible to automate. They’re so complex. They’re really fanciful, but they have to be built by human hands. You’ve got 86,000 17-year-old teenage girls in Shenzhen building these Apple products. Foxconn put up money to install 1 million robots to replace these girls. Today only about 400 robots have been installed out of that 1 million, because they found out the products they’re building for the Americans can’t be automated.

I refer to DFM as being designed for manufacturing the first time. We’re not just designing it, running software, finding the errors, going back, redesigning it, checking again, going

back and re-spinning it, until it works. I’m alone in this definition unless you happen to be knowledgeable about Dewhurst and Boothroyd, which most people in printed circuits aren’t.

Andy Shaughnessy: I would imagine.

Holden: Manufacturers such as General Electric, Westinghouse, or General Motors, understand Dewhurst and Boothroyd, because it’s taught at universities. The automotive guys are really big on trying to simplify parts in automobiles to make them more reliable at lower costs and easier to be built up by automatic systems. But in electronics, it’s never caught on.

One of the reasons is that Dewhurst and Boothroyd were mechanical engineers. They worked off the kinematics of how many motions it takes to assemble something. How much fixturing, connectors, or screws and bolts it takes to

put something together. That’s their predicting methodology.

The simplest form of automation is the one-axis pneumatic cylinder, or air cylinder. It just goes up and down. If you’re going to assemble something on a conveyor belt, with just up and down motions, that’s the cheapest automation. Some of the most complex assembly is assembling flexible circuits, which I managed for Foxconn.

It’s one of the reasons they kept showering me with money to automate out these 8,000 girls I had in Shenzhen who would do the final non-conformal part assembly of flex circuits, because they would work for six months on Apple products. When Apple stopped ordering after Christmas, the girls would sit around for the next four months and knit, because there were no orders. We couldn’t fire them or lay them off because they were too highly skilled.

They sat around and knitted and painted walls and things like that until Apple started ordering again. Then they all went back to work. One of the highest priorities I was given, with an unlimited budget, was to automate out these girls, so that when Apple stopped ordering we'd just turn the switch off. I told them the bad news was, "Well, maybe in 10 years."

It was difficult to explain to them it wasn't a matter of money, because they thought automation meant just buying robots, putting them on the line, and displacing people. I said no, first you have to have an automation strategy. Then we have to have fixturing and end effectors. Then we have to go back to the original designers and teach them about automation and simplification. We have to work together to simplify this thing because you may not like these girls sitting there knitting and everything, but when it comes to assembly time, they've got these 10 digits on two hands that can manipulate things.

We can teach them to do something in a short amount of time. Whereas with a robot, we have to program it and fixture it and turn it upside down and left and right, etc. It's not that easy to take the human out of the assembly equation.

This is what Hitachi really pioneered, which is why the Japanese products, when they're first invented, stay in Japan for the first three to five years, being built in automated facilities. When everybody's copied them and they start getting price erosion, they start to ship it offshore, preferably to the Philippines or to Vietnam, where they can control the IP. It's not until the product is at least 10 years old that they ever let it into China, because, as they told me, once it goes into China you'll lose all the IP. In the Philippines and Vietnam, you can control that IP. It doesn't leave Japan until the second generation is beginning to be released.

One way they enforce design for manufacturing is that if you don't need the fully auto-

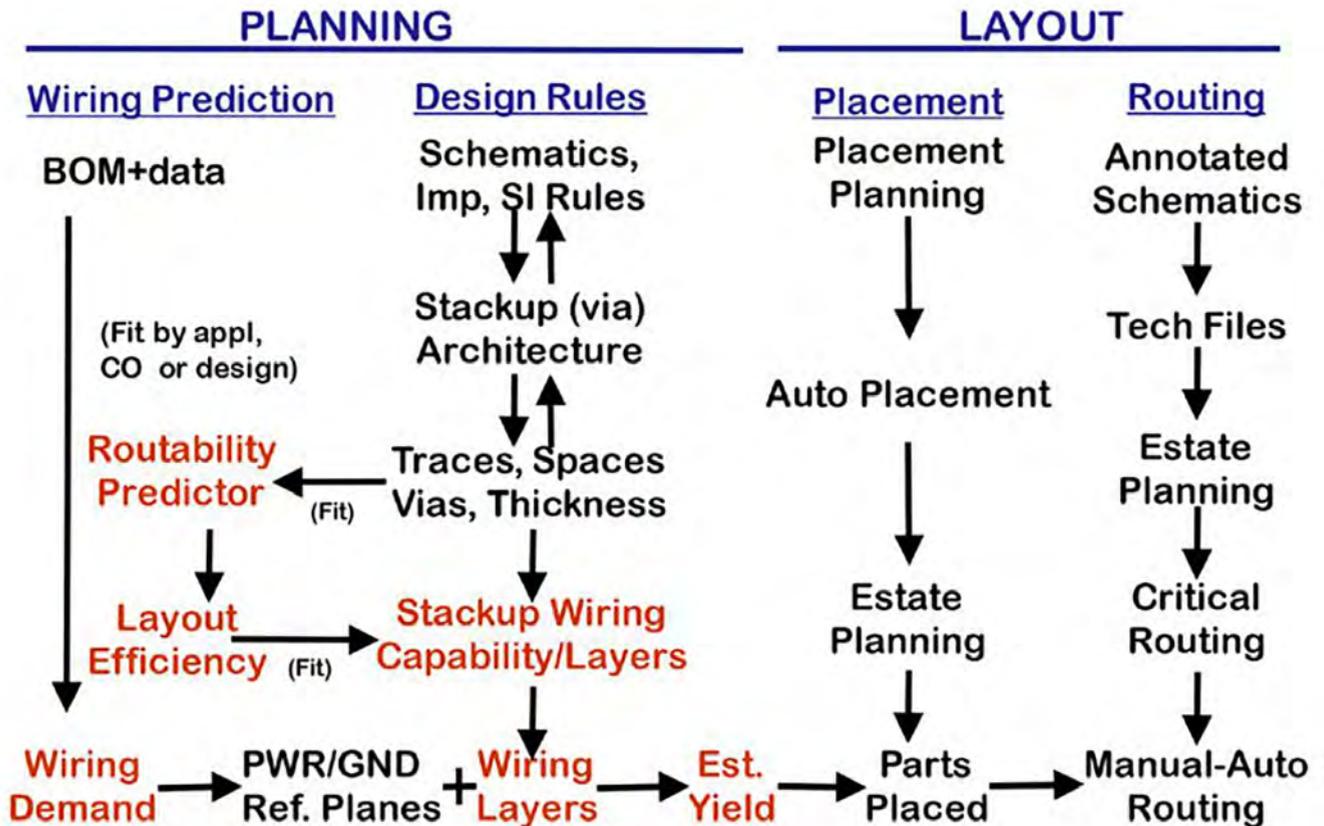


Figure 2: Process flow for the design planning of a printed circuit.

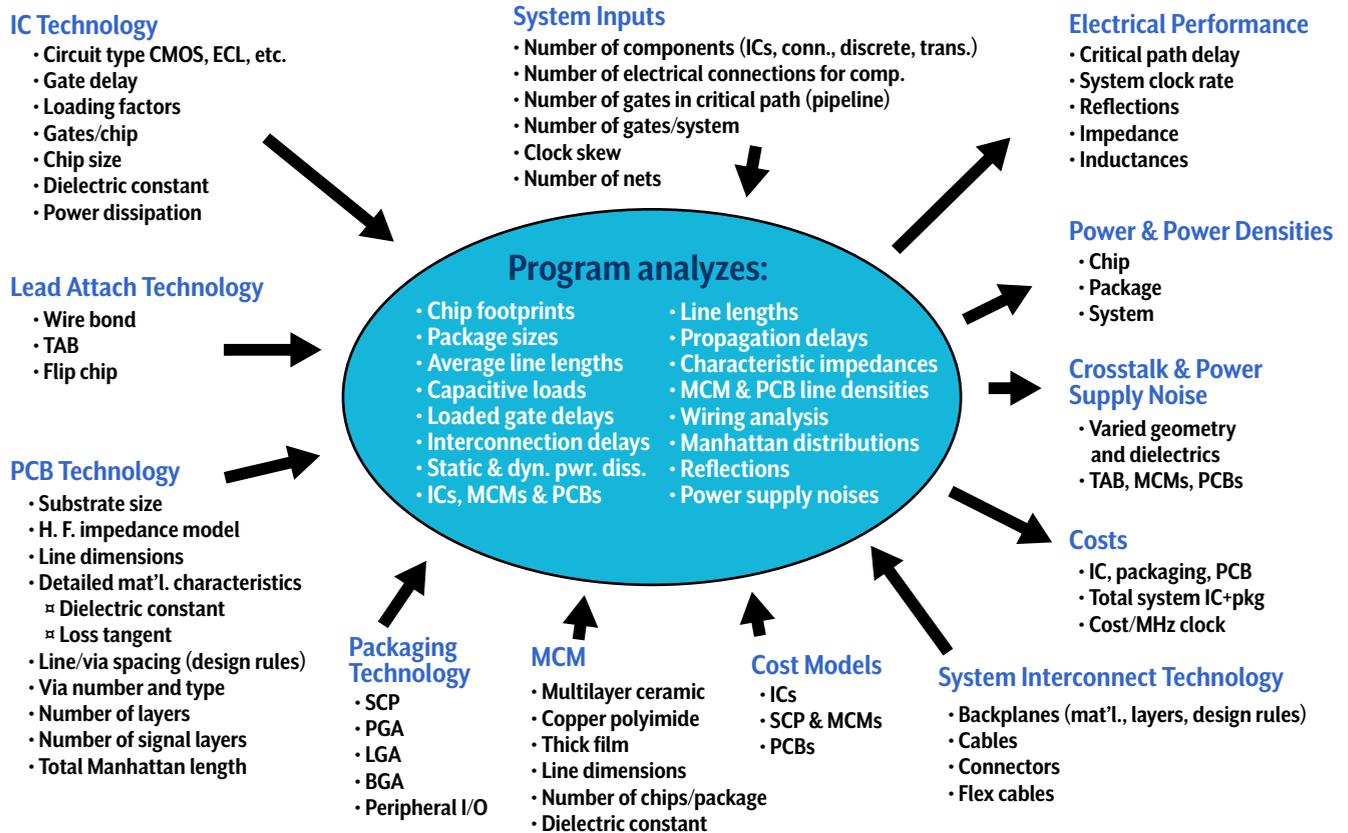


Figure 3: For predictive engineering of a PCB, there should be simulations and tradeoffs that cover all the domains that a user finds critical: costs, manufacturability, density, signal integrity and reliability.

mation assembly, the product will still be released, but you've got to go to the Philippines, or you've got to go to Vietnam or Indonesia and supervise the manual assembly of that, which to a Japanese engineer is like the worse punishment you could even possibly give—to have to go down to Indonesia and supervise 8,000 women building your product.

The Japanese nearly kill themselves in order to simplify these things so that the automatic systems can assemble them. Because DFM and DFX is known by everybody, but is caught up in conjunction with design rule checking, and nobody envisions what it would look like if it was predictive, online or before you start designing this thing so that we're not going to make mistakes. It's going to be easy to manufacture.

Barry Matties: Do the PCB designers of today consider the automation factor?

Holden: No, they don't.

Shaughnessy: I don't think so, no. I don't think PCB designers consider themselves part of the manufacturing process at all. But you believe they should see themselves as the front end of manufacturing.

Holden: If you ask designers who their customers are, they say, "Well it's the lab guys who gave me the schematic and the bill of materials." They pay the bills, but the customer pays for manufacturing.

Matties: With predictive engineering, you're talking more about a lot of simulation prior to a release into the manufacturing environment?

Holden: Right, but also cognitive rules and best practices that you can use as a rule setting so you don't make that mistake of where you place

100% Commitment.

We have expanded the scope of our military qualification to include flex and rigid-flex circuits, having earned MIL-PRF-50884F and MIL-PRF-31032C certifications.

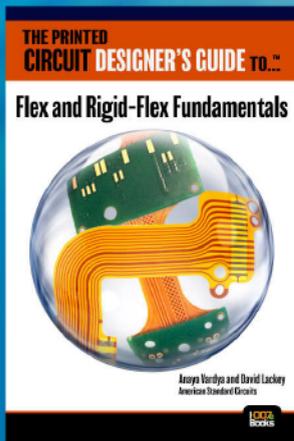
We are also approved for FR-4, polyimide, GF-homogeneous hydrocarbon/ceramic and GF-homogeneous hydrocarbon/ceramic mixed-base materials.

[VIEW ALL OUR CERTIFICATIONS](#)

See us at:



Booth #107



A FREE BOOK FROM THE ASC EXPERTS

We are 100% committed to making flex and rigid-flex as easy as possible, so we wrote an eBook that you can download for free.

[FREE EBOOK DOWNLOAD](#)



American Standard Circuits

Creative Innovations In Flex, Digital & Microwave Circuits



www.asc-i.com

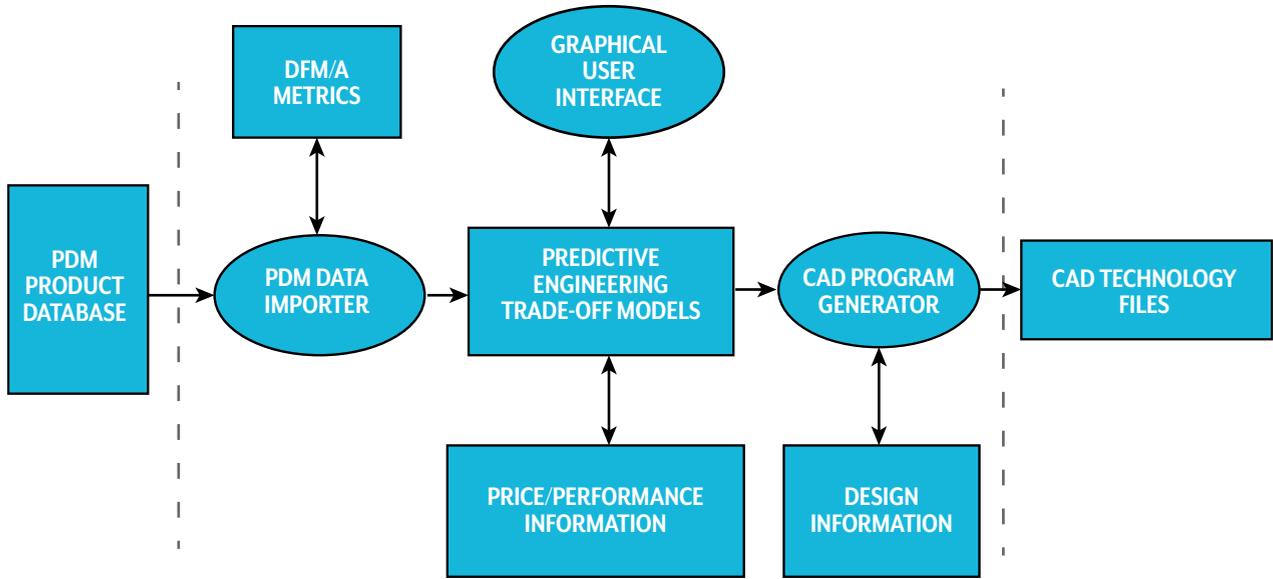


Figure 4: Proposed predictive engineering architecture.

the parts for wave soldering, etc. That’s predictive.

Matties: In the best case, that would be in real time. As you’re designing you’re getting instant feedback.

Holden: Here’s a great example of predictive engineering in an EDA tool. You may not remember Racal Redac’s (later Zuken) Visula, but Racal Redac was really significant in their development of predicting engineer software for signal integrity. There is a set of graphs and widgets on the screen during layout. The simulation would poke information out of the database, so as you placed parts, as you routed traces, and things like that, this thing would run these algorithms and these widgets would go from green to yellow to red or would change, indicating if you’ve just done a dumb thing signal integrity-wise, or thermal-wise. You go up and click on “undo” until it became green. Then you do it a different way so it stayed green.

That’s a beautiful piece of predictive engineering software, because it’s essentially a design coach looking over your shoulder providing you with advice and simulations without you having to run it back twice.

Shaughnessy: Yes, Visula users loved it. But it’s gone now.

Holden: We were using Board Station. We started doing this and defining who is best in class in PC layout. The Mentor software came out fifth out of five. The best was Visula. So we started using Visula with Mentor as the front end. This is how we got into discovering this kind of software they had. Zuken bought Racal Redac. We already had a relationship with Zuken, because HP wrote all the Zuken software. When HP went out of the EDA business, that became Sierra 5000.

Visula, when we were at HP, came back to us in terms of our thoughts. We said, “Well, we really love Visula. We also like Sierra 5000, but when you merge these things together you need to take some of the best things about Visula.”

Shaughnessy: What would you say would be some things that designers should know and understand about predictive engineering?

Holden: Well, I’ve had this lifelong fight with IPC and Dieter Bergman. Dieter and IPC believed that if designers really just understood how printed circuit boards were made, they

wouldn't make these design mistakes. Dieter thought you have to offer tours of board shops so designers can see how PC boards are made. After managing print circuit design, I realized that this was a pipe dream. It's difficult to learn how to design. You can spend your entire life trying to become an expert on design.

How can a one-hour tour of a board shop, where you see all these complex processes and machines, provide any insight on how that board should have been designed? Because it takes an entire lifetime to become an expert in printed circuit board technology. How is a designer going to learn that just by observing a step? We went back and said that we have to provide designers, number one, with the best design process that we can in terms of simulation and tools available to them.

The second thing is that in the design process, there are all these critical decision points that a designer has to choose in order to continue the design. At those critical decision points, they need to know the pros and cons of the alternate path that they can take. We put out a design for manufacturing manual that created what we called the relative cost index.

When they would hit one of these decision points, in the design manufacturing manual, they could find out what the relative costs and a couple other boundary conditions would be if they pulled some particular path on this design. That at least gave them some insight into the decisions they were making. Because, when the design was sent out for a prototype or quoting, they'd say, "Well, why is this quote so high?"

They said, "Well, if you had simply not done this or you've not done that..." The designers said, "That's interesting, but you should have told me this four weeks ago, because this is an early decision, and I can't go back and change that now. I'd have to start all over."

Because we were a design organization owned by manufacturing, not by R&D, our focus was to design the most manufacturable board, which meant we could meet delivery times at the highest calling and the lowest cost. That was our goal with design, not just to meet the schedule. One thing that's interesting, if you ask designers, is "What's the measure of performance?" The measure of performance is meet-

ing the schedule or the price quoted. There's no performance or quality measures when they respond. It's simply that schedule is the only metric we're trying to meet. I said, "Well, we definitely have a problem in that, in design, there's no measure of quality." Design takes absolutely no ownership that this thing is not manufacturable or producible, low-quality manufacturing.

We invented metrics, which I think I elaborated, but we also looked at methods of constantly improving the design process and how we can dig back to first batch yield in electrical test because we discovered that if we physically analyze first test yield in electrical test, the so-called random defects are not random. If you do an F test or T test of significance physically, log after log, you'll find that certain nets always have shorts in them.

When you go back and look at that you'll find some minimum spacing and things like that which moves them, then those things no longer randomly show up as electrical shorts. One of the first papers I ever wrote, in 1971, analyzed electrical tests; even today, nobody analyzes electrical test. They just have a pile of good boards and a pile of bad boards. They don't realize the test data is telling them something about design or something about manufacturing.

You have to understand what a map test is in order to understand that what they're calling random really isn't random. It's designed in. It's a very complex topic. I sympathize with you, Andy, in terms of how to get your arms around it. How do you make this an interesting topic and not just somebody standing up on a soapbox complaining about lead-free soldering?

Matties: Happy, can you share a story you once told me about taking over the design area at HP?

Holden: I complained to the vice president of our process group that Six Sigma and total quality manufacturing were having a profound effect on our manufacturing. We were eliminating final inspection, empowering employees to own quality of the product, and everything else. Everything was going fine except for the front-end and tooling. We were accumulating all of our resources to inspect the incoming design file, because they had so many problems and

things that we couldn't manufacture or manufacture with low yield, that we were defeating the purpose.

I asked when design was going to embrace total quality manufacturing. The vice president said, "Don't hold your breath on that." But we didn't have that problem in integrated circuits because we talked to the integrated circuit manufacturer. He said, "We do all the integrated circuit design. We do not allow anybody except us to design an integrated circuit that goes into our manufacturing."

They create the schematics and all the specs and things like that. We do the physical IC layout. He said, "I never could understand, since we make all the circuit boards, why we aren't designing the circuit boards, so I'm going to promote you to be the design services manager," which is not what I was complaining about.

He told me to fix the problem. I said, "Well, the first thing is that our printed circuit designers aren't paid very well. They're not trained. They don't have the right software tool. They don't have the ability to do research."

.....

“ Well, the first thing is that our printed circuit designers aren't paid very well. They're not trained. They don't have the right software tool. They don't have the ability to do research. ”

.....

He said, "Then give me the budget. I'll approve all of it." The VP said, "And I'd like to see the printed circuit designers put on the same salary level as the IC designers so we can rotate them."

I told the VP how much PCB designers typically made, and he said, "That would essentially double their salary." I said that was great. If I doubled their salary, then they were going to complain a lot less when I demanded that they go back to school. I also demanded that they

all use the same design process so that we had a process that we could constantly improve. If each person has his own design process, there's no way to create tools and get synergy out of it.

Unlike other people, I got the budget and the salary authorization to elevate printed circuit design to where it should be and the budgets to get the best tools and write my own scripts and my own software and standardize as per Six Sigma or quality manufacturing process. They all complained about using the same process, since they were inventing the process by consensus, like this figure of merit. We also brainstormed and came up with these figures. How do I measure your performance? I'm your boss now. How do I know that you deserve a raise? What am I going to measure with? Right now, you say schedule, but that doesn't talk about quality.

I insisted on a quality metric that they can put on each board they designed, so I had a way to compare it. Who needed more training? Who needed more skill?

Matties: It's like making your designers the superstars in the company pays off.

Holden: In the HP product division, everybody in R&D was ranked according to their contribution. You had the PhD gurus who invented wiz bang products in HP that get ranked high. Somebody had to be ranked low and it was always the PC layout people. If you're paying on a bell-shaped curve and everything, that means that PC designers are always the tail of this.

When we moved printed circuit design to manufacturing they became the most important engineers we had in manufacturing, because they determined our delivery, our profitability, our yields, our efficiency. They went from the bottom tail to the top tail of the distribution of contributions, because we valued them more than anything else.

This is something we learned from our Japanese partners. In Japan we were surprised to find that a lot of the PC boards are being designed by print circuit manufacturers. You had to work in manufacturing for at least five years before you could get the opportunity to move up into design. They didn't want anybody designing a



VERIFY

OPTIMIZE

ANALYZE

PREPARE

PANELIZE

COMMUNICATE

DOCUMENT

Ensure PCB Manufacturing Success!



CAM350

Verify and Optimize PCB Designs for Successful Manufacturing.



BluePrint-PCB

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



DFMStream

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

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

Visit downstreamtech.com
or call 800-535-3226

PC board that had not worked in manufacturing first.

Interesting enough, that's the way that Gentex operates here in Zeeland. Everybody from college starts off on the factory floor for at least the first two or three years—absolutely everybody. After the first two or three years and only after you thoroughly understand how the product is made are you allowed to go in R&D, project management, automation, or one of the other areas. Everybody starts on the factory floor.

Shaughnessy: I imagine that most designers do not consider themselves as the front end of manufacturing.

Matties: You're right, too, in that it really ties into what we've been saying about DFP, design for profitability, because that's where it starts.

Shaughnessy: Right. Hard to believe that design can control 75% of the cost.

Holden: In fact, the DFMA evolved at HP, along with DFCA (design for competitive advantage). It's about the last slides I have from my HP days. You can see that the whole "do it right the first

time" idea had permeated the entire design process from conception through the end. We were benchmarking our own design, but also benchmarking competitors. When we were doing the prototypes, we were trying to make our products simple and easy to build, but also better than a competitor's product, including Japanese products.

And we have such great DFM tools for IC design. Why aren't any of these used for the printed circuit design tools? Offshore, they're not using any new software. They're just using more manpower at a lower rate so that they can undercut North America. We can't afford to lose design by outsourcing. We just really can't. I'm prepared to say we can lose PC fabrication. We can lose assembly, but if you've lost design, then it's gone, because design is where the intellectual property of schematic meets the physical world.

Shaughnessy: Hopefully, we can help start a conversation about predictive engineering, and what DFM comprises.

Matties: Thanks for speaking with us, Happy.

Holden: Thank you. **PCBDESIGN**

A New Way to Print Electrical Circuits

Within 15 minutes of meeting Mark Hersam, PhD, a renowned nanotechnology expert and professor of materials science and engineering at Northwestern University, Ethan Secor knew he wanted to work with him. Secor, a fifth-year materials science and engineering PhD candidate, didn't have a project in mind at the time, but when Hersam had an opening in his research group, he jumped right in.

Secor is developing graphene-based inks, which can be printed with traditional methods like inkjet printing.

"For printed electronics, instead of printing red, green and blue inks, we're printing conductors, semiconductors and insulators — different classes of electronically functional materials," says Secor. "So instead of printing pictures, we're printing electronic circuits."



Printing electronics with graphene inks, either on paper or plastic substrates, not only reduces the manufacturing costs of electronic components, but it also allows the use of flexible surfaces. This is critical to integrating electronics into clothing, consumer electronics and products.

Secor is just one of 30 PhD students Hersam has worked with in his research group at Northwestern. Throughout the science and engineering departments, Hersam's dedication to educating the next generation of scientists is something he is known for. "The impact of a university is made by its students," says Hersam. "The beauty of a field like nanotechnology is that it captures the imagination of young people, and will continue attracting them and bringing in the best talent."



My 9th birthday 1978 Montana

We love PCBs. We always have.

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

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

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

www.ncabgroup.com





The History of Predictive Engineering

by **Happy Holden**

It all started in 1983, at HP, when I complained to our groups vice president that our W. Edwards Deming and Total Quality Management (TQM) Six Sigma training was being concentrated in PCB manufacturing. We had eliminated final inspection and instead placed quality in the hands of the operators with a final electrical test. The electrical test was governed by what we learned from Deming.

Our customers' problems with on-time delivery and rejects had gone away, but now our problem was that we were having to put all our engineering at the front end of the process to inspect incoming CAD designs for producibility, as the PCB design groups had no concept of a "quality design." To them their customer was the electronic designer and the schedule. To meet the schedules, they would just throw the design "over the wall," even though they knew the designs had problems.

"Since you understand the problem, I am going to promote you to PCB design services

manager," said the VP, agreeing with me. "In integrated circuits, we design the physical chips and then fabricate them from the specs and circuits the EE gives us. I have never understood why the PCB fab group also did not do PCB layout in the IC model!"

Not the answer I wanted, but at least he gave me the budget and the designers' salary flexibility I needed to recruit and start a PCB layout group as part of PCB manufacturing. So, DFM became the central focus for our design strategy.

DFM/A was just starting out at HP. HP had taken a license from Hitachi and GE to use the Dewhurst and Boothroyd DFM/A methodology. This came after extensive benchmarking review of past HP product designs and visiting other large manufacturing companies like John Deere, Ford, Hitachi, Caterpillar, and Western Electric. The benchmarking clearly showed that just hiring the smartest engineers and giving them the best EDA tools did not guarantee a superior product. The electronic circuits may have been superior, but the physical products were

A BETTER IDEA



Valor[®] NPI DFM JUST GOT EASIER!

Valor NPI software is industry's only automated PCB DFM solution, seamlessly connecting PCB design, fabrication, assembly and test processes.

With the latest version of Valor NPI, setup time for DFM rules can be reduced from weeks to days, and DFM results viewing time can be reduced by up to 50 percent.

LEARN MORE AT [HTTP://WWW.MENTOR.COM/NPI](http://www.mentor.com/npi)

Mentor[®]

A Siemens Business

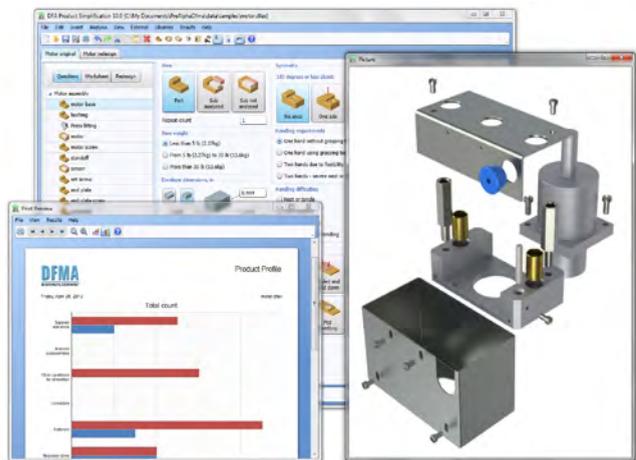


Figure 1: Dewhurst & Boothroyd DFMA software.

far too overdesigned' and complex, thus adding a lot of unnecessary costs and complexity.

The D&B DFM/A methodology was used concurrently with the physical design to point out complexities and provided feedback on how to simplify. It was also used to benchmark competitor's products. The DFM/A methodology was focused on "Doing it right the first time." This was not a design rule checker, and a lot more useful than "best practices," although these are useful tools. This is where I get my definition of DFM or DFX.

CAM software, including DRCs, are activities that take place after the design is finished. We found one EDA supplier, Zuken, that had "design advisor" software for signal integrity, which they gained when they acquired Racal-Redac. This software resided on the screen and provided real-time feedback on SI as they routed traces by taking variables from the database and doing a real-time simulation.

Metrics are an important part of DFX, and D&B created some very useful ones. But we had to develop metrics for the PCB (complexity index, density, connections/sq. inch, wiring capability), PC design (layout efficiency, routability index, wiring demand, first pass yield), and SMT assembly (average leads/sq. inch).

We were focused on taking manufacturing performance in fab and assembly, and feeding this data back to design and layout. This data would then drive the predictive tools advising

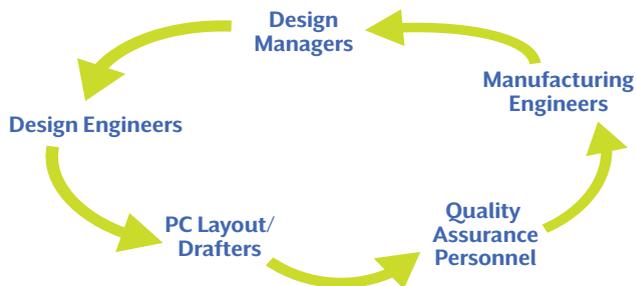


Figure 2: The benefits of metrics as a common design language.

designers on best practices and what-to-do-next. As design's horizon is very short compared to the life of a product. This data had to reside in the product data management (PDM) enterprise software and not in the EDA tools. Like the D&B methodology, the key attribute was "predictive metrics and models" powered by real manufacturing performance.

Thus, predictive engineering or Design for Competitive Advantage (DFCA) emerged as a better definition of DFM or DFX, because everyone was using the term DFM to refer to design rule checks, which occur after the fact and are not predictive.

To support this predictive arena for PCB and assemblies, we developed eight processes:

1. The DFM Manual: A design document that provided: 1. Electrical performance, mate-

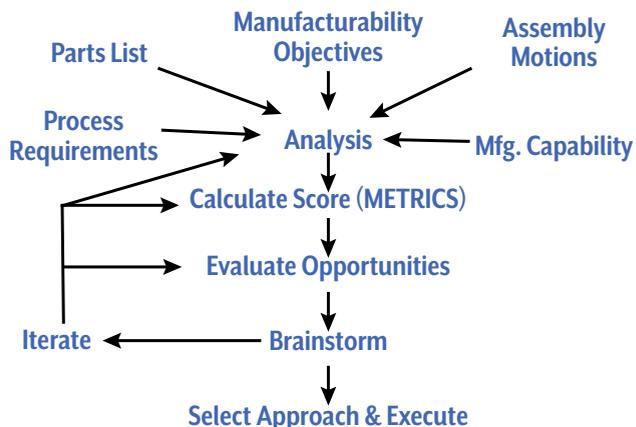


Figure 3: The process to develop a manufacturability metric as a substitute for opinions.



*The little PCB inside
Vance & Hines' Fuelpak3
gives riders a big advantage*

It's the little things.

The flat-track racing champions at Vance & Hines know: when you're rocketing down the racetrack at nearly 200 mph, a lot of little things, like a gust of tailwind or a well-tuned exhaust system, can add up to victory – or defeat.

The same is true for PCB prototyping and production. **Little things, done right, add up to a competitive advantage for your business.**

From our design recommendation that helped get their new fuel management **product to market faster** – to the proactive phone call from one of our meticulous employees that **saved an entire production run** of unusable boards and **prevented a gap in inventory**. These little things are what keeps Vance & Hines coming back to Sunstone Circuits for over a decade.

Learn more at: www.sunstone.com/littlethings

“Instead of robotically manufacturing boards, Sunstone has a thought process behind their work... We have more opportunities to gain competitive advantage because we know our PCB partner can handle anything...”

Larry Hinds

MANUFACTURING MANAGER,
VANCE & HINES



Contact us at: **1-800-228-8198**
or via e-mail at support@sunstone.com

rials and connection density (DEN), first pass yield (FPY) and the PCB Complexity Index (CI), the cost of any design as the Relative Cost Index (RCI); and 2. Cost (RCI) of different materials, layers, holes, final finishes, fabrication, density, and electrical test and performance (controlled impedance).

2. Design for Cost—Report Cards: using a report card of features for both PCB fab and assembly gives and indication of RCIs and if the cost will go up or how to get a discount.

3. First DFM design software tool, 1987: A Lotus Spreadsheet was created for design planning.

- a. Inputs:
 - i. Board features: size, layers, traces, and holes
- b. Outputs:
 - i. DDI, RCI, FPY and electrical performance

4. A second DFM design software tool, 1990: A more complex 55 Excel Spreadsheet was created over a number of years where not just the board and assembly measures are specified but also the system specs and more performance models. This became a virtual prototype called System Performance Analysis (SPA).

- a. Inputs:
 - i. System inputs IC technology, lead attach tech, packaging tech, MCM, PCB technology, system interconnect tech, and cost models.

- b. Outputs:
 - i. Electrical performance, power and power densities, crosstalk and power supply noise and costs.

5. MCC's Multichip Design Advisor, 1991: HP contributed its software and models to MCC for this predictive modeling software that later is spun out as SavanSys.

6. Wiring demand vs. wiring capability trade-offs.

7. HP-LABS Board Construction Advisor—IPDA, 1995: A board planning tool.

8. HP-LABS self-learning Expert System (Explorer), 1996: An automatic search solution for PCB design goals.

I believe that it's critical for today's PCB designers to have a better understanding of predictive engineering. PE is not just for manufacturers. Without predictive engineering, you really can't have a true design for manufacturing process. **PCBDESIGN**



Happy Holden has worked in printed circuit technology since 1970 with Hewlett Packard, NanYa/Westwood, Merix, Foxconn, and Gentex. He is currently a contributing editor with I-Connect007. To read past columns or contact Holden, [click here](#).

Integrated Lasers on Different Surfaces

Fabricating hybrid semiconductor lasers on materials other than silicon-on-insulator (SOI) substrates has proved challenging. Now, A*STAR researchers have developed an innovative technique that can integrate the lasers on to a range of different materials.

Hybrid lasers combine the light-emitting properties of group III-V semiconductors like gallium arsenide and indium phosphide, with conventional silicon technologies.

Doris Keh-Ting Ng and colleagues from the A*STAR Data Storage Institute have developed an innovative technique for bonding III-V lasers on to other substrates, be it silicon, quartz, or metal alloys.

By using an ultrathin layer of silicon oxide to bond the lasers to a silicon substrate, the researchers developed a simpler, safer and more flexible technique than direct bonding, which relies on chemical bonding between the surfaces.



IPC-2581 CONSORTIUM

IPC-2581 is key for Tempo Automation's ability to provide instant quotes for rapid PCB assembly. The rich format of the IPC-2581 standard ensures that our robotic factory can work with all EDA platforms. We believe IPC-2581, as an open manufacturing standard, enables automation and analysis that will accelerate technology advancements in the PCB industry."



Shashank Samala

Tempo Automation Co-Founder
and VP of Product and Sales

To learn more about IPC-2581 Consortium,
please visit www.ipc-2581.com

tempo automation™

It's Not Your Father's DFM Anymore

by **Patrick McGoff**
BY MENTOR

Trivia question for you: What was the first design for manufacturing (DFM) tool used in the PCB industry? Answer: The eye loop (Figure 1).

Yes, back in the day when design organizations sent actual photoplotted films of their



Figure 1: Eye loops were used up until the 1990s to check PCB artwork for manufacturing suitability.

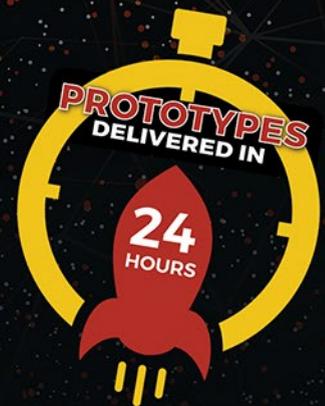
PCB design to the bareboard fabricator, the fabricator would put the received films on a light table and measure feature sizes for line widths, spacing between features and annular ring using an eye loop with a reticle etched onto the glass lens. If the features were beyond the capabilities of the fabricator, the job would be declined and the customer notified. If the film had extraneous features they would be removed with an X-ACTO knife. Voids would be filled in using a black marker.

Simpler times, then. 10 mil lines and spaces and double-sided PCBs were the norm. Manual DFM was feasible.

Needless to say, our industry has advanced in technology since those days. We have PCBs that exceed 64 layers, build-up technology with laser-formed stacked and staggered microvias, embedded devices, and complex rigid-flex circuits. And our design, fabrication, and assembly processes morphed into the new global economy, and in many cases, outsourced and off-shored.

So, what does DFM look like today? Well, for one, it's become more than DFM. We now expect our software tools to serve us more completely

Your signal needs to get around the world fast...
Why go halfway around the world to get your
high-speed PCBs?



**ACE: your high-speed experts right here at home,
specializing in microwave and hybrid PCBs.**



Accurate Circuit Engineering

3019 Kilson Dr
Santa Ana, CA 92707

(714) 546-2162 • sales@ace-pcb.com

For technical assistance, call:

James Hofer
(714) 227-9540

www.ace-pcb.com

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

to take new designs to market. DFM has evolved to become an integral part of new product introductions (NPI). Today's NPI software spans design and manufacturing to accelerate the optimization of a PCB for manufacturing. It looks at the entire design-to-manufacturing release process and deliverables, then streamlines them.

Engineers at best-practice companies use NPI software to help identify ways to optimize their design during the initial release process, with a holistic view of their bareboard fabricators' and assembly suppliers' capabilities. It's not simply a matter of catching mistakes these days. It's about understanding the various challenges and limitations involved in the chosen manufacturing processes and then empowering the product owner to make the trade-off decisions that best serve their objectives.

NPI software must employ expert system logic to automate DFM analysis. Quality DFM analysis is based on a combination of PCB technology classifications and manufacturing constraints. PCB classification can be determined

automatically through intelligent design data. For example, the copper weight of each layer is important in the fabrication DFM analysis because the etch compensation applied and, thus, resultant etch-back are going to be different based on the copper weight used, varying between inner and outer layers (Figure 2).

A 3-mil spacing on an inner layer made of 0.5 oz. copper may be acceptable, whereas the outer layer on the same board may need 4-mil spacing based on the use of 1 oz. copper. Furthermore, a PCB constructed with sequential lamination is going to use different DFM rules than a conventional, single-lamination PCB. Having the DFM software assign those attributes automatically saves engineering time and effort.

DFM systems that use algorithms based on CAD feature shapes can perform analysis in a matter of minutes compared to systems based on vector data, which can take hours to run, and who has hours to wait for DFM analysis these days?

General Copper spacing is the distance between any two adjacent copper features.
 As part of the fabrication process copper features are slightly increased in size to compensate for etching and this will slightly reduce the original design spacing.
 The etching process is achieved through a subtractive chemical reaction where excess copper is removed forming the intended conductors which is why more space is necessary between copper features on thicker copper layers.

Value	Classification	Copper	Layer Side
1 3 ml	Standard	0.5 oz	Inner
2 4 ml	Standard	1 oz	Inner
3 5 ml	Standard	2 oz	Inner
4 4 ml	Standard	0.5 oz	Outer
5 4 ml	Standard	1 oz	Outer
6 4 ml	Standard	2 oz	Outer

Figure 2: An intelligent DFM application will automatically derive the appropriate rules to be used for analysis based on the attributes in the design data.

ROG breaks FR-4 barrier!

Introducing
Kappa™ 438
Laminates



Need Better RF Performance? FR-4 No More!

Demand for wireless data is growing exponentially, driving a need for substantially higher levels of mobile network capacity and performance. This demand will grow further in support of the upcoming 5G IoT ecosystem where

billions of devices will be communicating with each other, and connectivity is immediate and uninterrupted. FR-4 was historically a material choice for many less demanding RF applications, but changes in the wireless infrastructure related to growing performance requirements, especially in small cells and carrier-grade WiFi/Licensed Assisted Access (LAA), have resulted in instances where the properties of FR-4 are lacking, and RF performance and consistency is compromised. There's no longer a need to sacrifice your PCB performance.



— Now There's Kappa™ 438 Laminates.

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

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

PROPERTY	TYPICAL VALUE
Dk*	4.38
Df	0.005

*Design Dk; Differential Phase Length Method at 2.5 GHz



Advanced Connectivity Solutions

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

For years, DFM for flexible and rigid-flex circuits has been ignored. However, as nearly 30% of all electronics companies have flex and rigid-flex circuits within their product portfolios, the segment merits attention. And, with more companies moving to flexible electronics, this trend and this segment's DFM needs will only continue to accelerate. The materials and manufacturing processes are quite different for flex and rigid-flex circuits, and the DFM tools must support those needs. Being able to identify features within a bend area are critical, as is maintaining sufficient clearance around a coverlay zone, for example.

Performing DFM analysis concurrent with the PCB design process increases the competitive advantage. But, to make this practical, the DFM software needs to be well-integrated with the EDA software so that the designer can run the analysis and review the results for basic DFM within the environment of the layout tools. By doing so, they avoid having someone downstream telling them there is an issue. Downstream is no longer adequate. Upstream is required to compete today. As evidence that the electronics industry has evolved beyond just DFM, NPI software is also used to design and optimize PCB panels.

Many companies are inefficient in their panel-design process, even for assembly panels or arrays. Typically, a design organization will use a 2D mechanical CAD tool to create a drawing of what they need for their assembly panels to look like when received from the bareboard fabricator. They will spend an hour or more creating this drawing and adding notes and dimensions, then they output it to their fabricator as a dumb drawing file.

By "dumb," I mean the drawing file has no intelligence to it, no data that the fabricator can key off of in the tooling process. Rather, the fab engineers must recreate the assembly array within their CAM software and submit a proof image back to the designer at the OEM for approval before proceeding with tooling. Needless to say, with good technology, this level of manual processes isn't necessary today.

Instead, the same design organization can easily create their assembly panels using the NPI software and include all the elements needed—

rails, tooling holes, fiducials, and rout or v-score features. Good NPI software will even automatically optimize the boards within the assembly panel to minimize the fabrication material costs. When trying to drive down the costs of electronic products, reducing fabrication material costs is often an overlooked opportunity.

In a study we conducted with four different customer designs, we found panel optimization saved an average of \$126,450 annually per customer, and these savings could be had with as few as four panels. Furthermore, DFM analysis done on the assembly array will identify potential manufacturing issues not able to be identified in the single-up stage. A couple of good examples are having a breakaway tab too close to a SMD pad or a device that overhangs the edge of the PCB obstructing machine vision access to a global fiducial.

Lastly, the NPI process involves creation, validation, and delivery of the design package to the fabrication and assembly providers. Historically, it's delivered as a package comprised of some combination of the following:

- Gerber files
- Drill files
- Netlist files
- Test files
- Centroid data
- Manufacturing BOM
- Drill drawings
- Assembly panel drawings
- Route drawings
- Stackup drawing
- Drawing notes

For an industry that drives the digitalization of the world, we sure are heavily dependent on manual processes with instructions communicated through 2D, non-relational drawings and documents. What is in the drawing or documentation that cannot be represented as a data field (Figure 3)? Nets that are intentionally shorted? Data field tagged to the appropriate nets. Top-side soldermask color? Data field. Dielectric thickness for a given material? Data field.

Dimensions are an interesting topic for digitalization. An electronic product model theoretically does not require a dimensioned drawing.

Professional PCB Fabrication Service

for Telecommunication, Automation & Test and Measurement Applications

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



Our Advantages:

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



www.pcbcarts.com

sales@pcbcarts.com

Product Summary section -> Attribute	Value	Units
Board Requirements		
Board Thickness	0.089200	Inch
Additional Requirements		
Board Outline Tolerance Plus	5.000000	Mil
Board Outline Tolerance Minus	5.000000	Mil
Board Thickness Tol Plus	3.000000	Mil
Board Thickness Tol Minus	3.000000	Mil
Board Thickness Type	over mask on plated copper	
Bottom Legend Color	white	
Bottom Soldermask Color	yellow	
Flammability Rating Standard	UL94V-0	
General PCB Standard	IPC 6012A	
Glass Transition Temperature (Tg)	110.000000	
Legend Sides	Both	
PCB Acceptability Standard	IPC 6012A	
Peelable Mask Side	none	
Plated Edge	Yes	
Plated Slots	No	
Qualification and Performance Standard		
Soldermask Sides	Both	
Top Legend Color	white	
Top Soldermask Color	green	

Figure 3: When information is communicated as a data field, it streamlines the NPI process and reduces the chances for misinterpretation and error.

After all, anyone who wishes can simply measure any feature or distance they wish, snapping to the edges or center of features as necessary. However, in situations such as identifying critical dimensions for incoming inspection, even the dimensions can be communicated as data fields within the electronic product model of the PCB.

I encourage everyone to take a look at their PCB manufacturing deliverables and ask themselves and their suppliers, “What can we do to streamline the communications process?” If you view your suppliers as your manufacturing supply-chain partners, they will be glad to share their perspective with you. Often, you can significantly reduce, if not eliminate, the information communicated in these drawings and documents by using an intelligent data exchange format that supports this information being communicated as data instead of a static

drawing with text. Think how much time your company could save.

We are neither design islands nor manufacturing silos. We are part of an electronics supply chain, and our collective supply chains are the backbone of the digitalization of the electronics industry. Any steps in the design-to-manufacturing process that are not operating at maximum efficiency, both in speed and quality, are hindering the pace of progress. Let’s not be the last link in the chain left holding the modern equivalent of an eye loop. **PCBDESIGN**



Patrick McGoff is responsible for market development of the Valor NPI tools at Mentor, a Siemens Business. McGoff has had several articles published on the subjects of DFM and data exchange.

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



Quick Search



Advanced Search



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



An Ounce of DFM Error Prevention is Worth a Pound of Cure

by **Hemant Shah and Ed Acheson**
 CADENCE DESIGN SYSTEMS

Designers try their best to avoid design for manufacturing (DFM) errors as they lay out the PCB. Experienced designers know common issues to look for and fix, saving them the hassle of a board coming back from manufacturing signoff for rework. They know how painful, time consuming, and frustrating it is to fix errors that are discovered by the manufacturing signoff process. Not only must they stop working on the new, important project they just started, but they are also now both the focus and cause of a delay that probably was unaccounted for in the schedule.

Design First, Ask Questions Later

There is a fundamental problem in the PCB design approach that is used today. Currently, designers hurry up and finish the board, then send to another person or group that uses a different tool to check if the design is ready for

manufacturing handoff. Is it manufacturable? Will it have a high or low yield?

The problem with this approach is that the data is first translated from the native PCB CAD system, and then an intelligent CAD database is recreated. This is essentially a translation process. The industry has gone through many translators and found there is always an interpretation gap. Using a third-party tool provides designers with a second set of eyes, but they must contend with translation mismatches, or if one tool supports a new fabrication method, designers must wait for the signoff tool to catch up and support the same methods.

The industry has seen this with back-drilling, rigid-flex, and embedded components, to name a few such fabrication innovations. Translation and reverse-engineering processes involve many assumptions that cause mismatches and can create false positives, or worse, not find problems at all. False positives cause designers to spin their wheels, change something that is not required and hope no new errors get intro-

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



ventec
INTERNATIONAL GROUP
騰輝電子



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

Contact us to discuss your requirements.

**Ventec - wherever technology
takes you, we deliver**

Ventec International Group

179 Ward Hill Ave,
Ward Hill, MA 01835,
United States

T: 978-521-9700

E: ordersCA@ventec-usa.com

Follow @VentecLaminates

www.ventec laminates.com

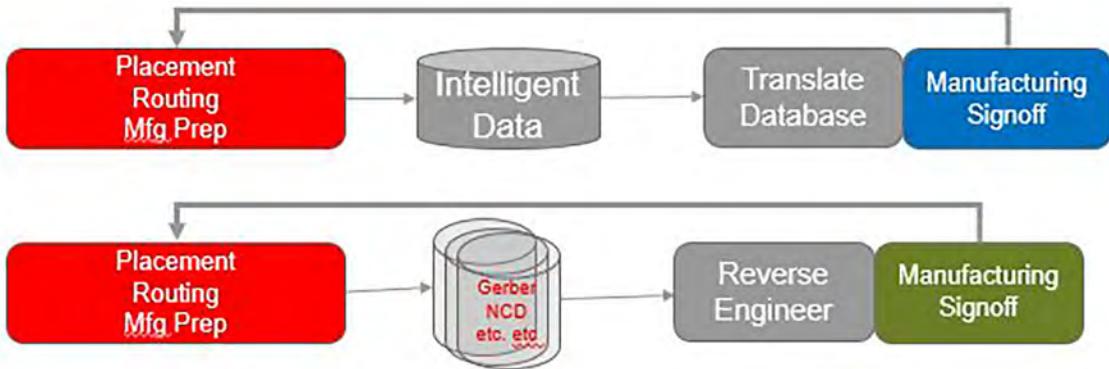


Figure 1: Current new product development and introduction (NPD) process.

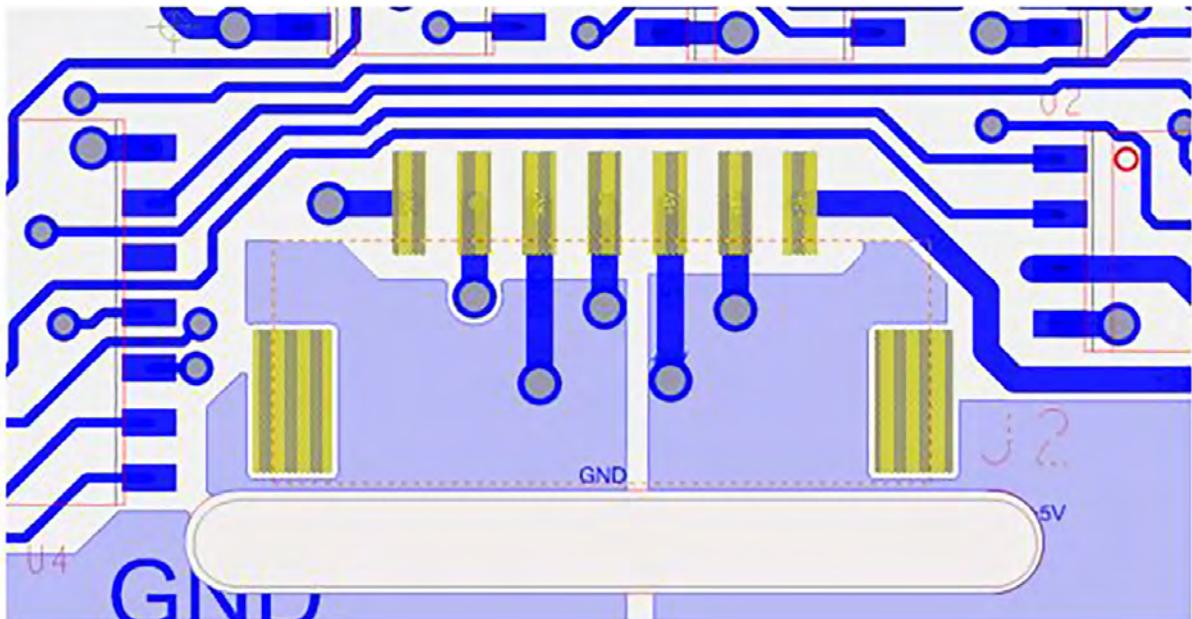


Figure 2: Connector pins are too close to the cutout in the board. Fixing this error after the design is completed—moving traces and other components after they are routed—is too expensive.

duced. If an error is missed and a manufacturing partner finds it, you have lost valuable time and must go back and fix a problem after the designer has moved on to another design.

The second major problem with the current approach is that problems are found after the design is completed. At that point, it takes much longer to fix a simple problem since many other objects—traces, pins, components, shapes—are impacted by an otherwise simple fix.

Some of the top issues that PCB fabricators report can be caught early in the design process with the right manufacturing rules in place.

The Copper-to-board outline is still a top 10 issue. A common challenge comes when the mechanical engineer defines a cutout in the design. A mechanical designer who may not have full knowledge of manufacturing rules often sets the keep-out too small. The PCB designer, trusting the mechanical data, wraps up the design and sends it for signoff.

Unknown to the PCB designer, the mechanical keep-out allows for a 4-mil clearance, but the PCB fabricator requires 10 mils. When signoff receives the PCB design, they detect and report the violation back to the PCB designer, several

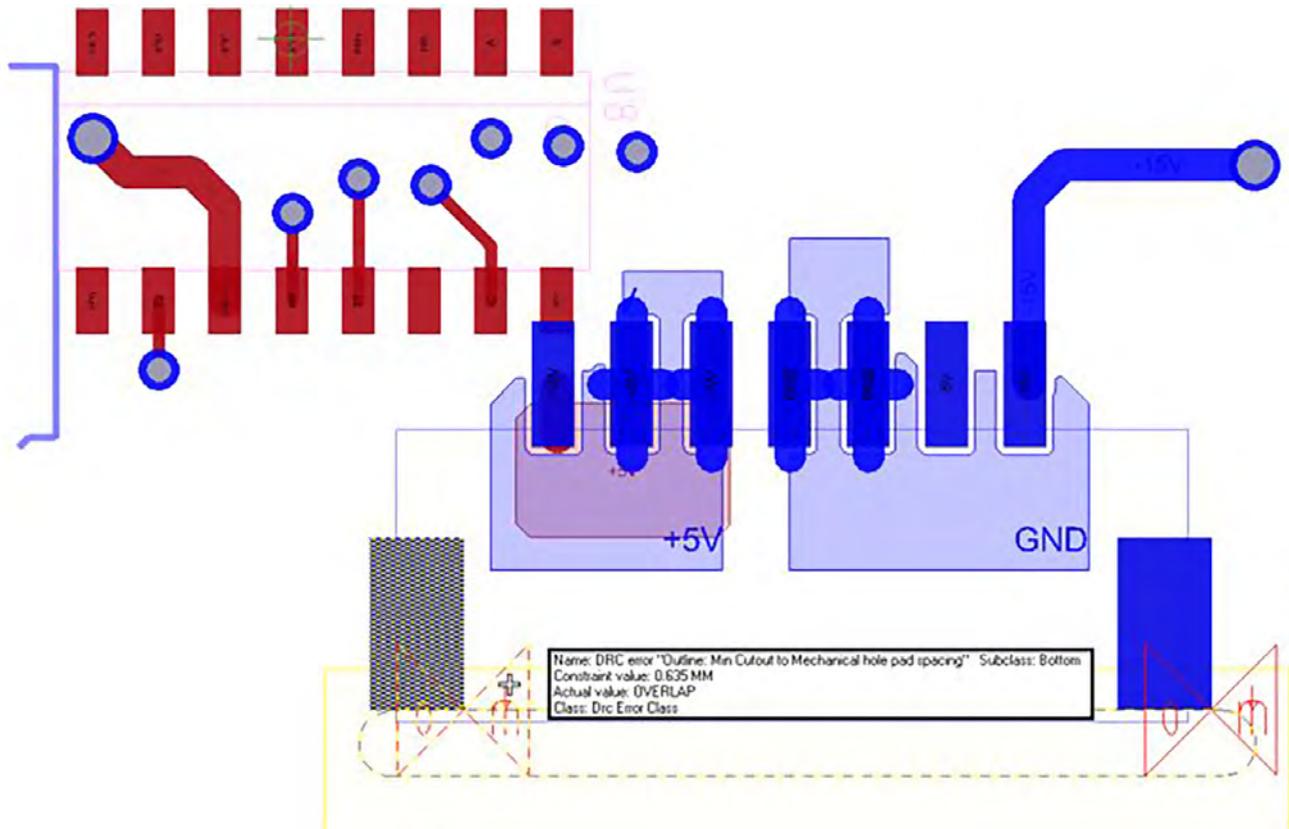


Figure 3: Connector pins are too close to the cutout in the board. It is much less time consuming to move this away from the cutout.

hours or days later. Now the PCB designer must stop the new design, move several components, reconnect the design, and finally, return the fabrication package to signoff with fingers crossed.

Had the PCB designer started with the proper DFM rule set, such as copper-to-cutout checks, the designer would have recognized this issue much earlier, before starting interconnect, and avoided rework altogether.

Enter Predictive Engineering

A better approach is for the PCB CAD tools to identify DFM errors as they are made in real time, not after more work is done. Predictive engineering in-design, in real time. Feedback as you make edits, not later on. Not through an embedded batch engine that must be run after some amount of work is done.

Most PCB CAD tools have accounted for DFM rules in their system, but they are usually a subset. If they were complete, you wouldn't

need third-party manufacturing signoff tools and wouldn't have any iterations with the contract manufacturers that build your boards. Tools have evolved to focus more on electrical and physical rules to ensure the product will function as it is designed and will fit in the enclosure once it is assembled. Manufacturing processes have also evolved, and PCB CAD tools have not kept pace with new fabrication, assembly, inspection, and test methods. Rigid-flex, HDI, and embedded components are some examples of such new approaches that have recently been available to PCB designers.

Detecting DFM issues as they are made allows designers to fix them in the shortest amount of time possible before moving forward with rest of the design. It is also important for designers to avoid spending weeks or months of time to set up the rules. DFM rules should be easy to set up and reuse across multiple designs. Having a real-time DFM arbiter during design ensures

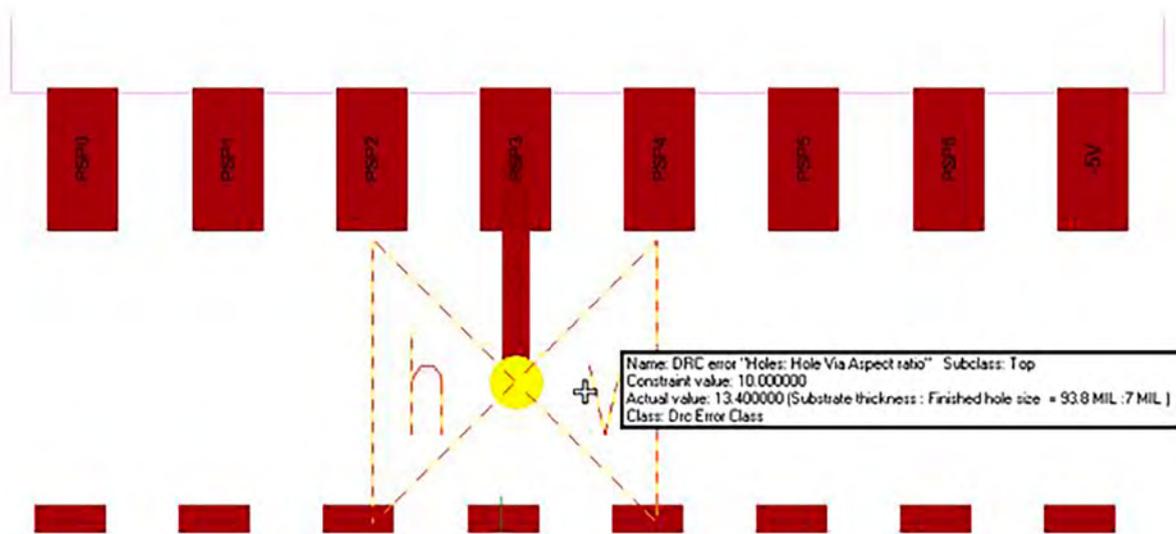


Figure 4: Plated through-hole aspect ratio warning avoids creating many instances of this problem in the design.

that the design can be created correctly for all domains that matter—electrical, mechanical, and manufacturing.

Another example is plated through-hole aspect ratio, which is typically created through improper via selection. The designer has sent the PCB with an overall thickness of .093" and is using a finished hole size of .007" for vias. The PCB fabricator has a best cost-effective plated hole aspect ratio of 10:1. The designer is now informed that the board can be built, but at a higher cost. Now the question is, does the designer spend the time to correct the design, which may take several days or weeks, or eat the extra cost of fabrication?

Had the designer been alerted to the via aspect ratio as the via was placed, this issue would have been recognized the moment the first 7-mil via was placed in the design.

When high-volume production comes into play, many design centers use a defined set of rules that is the best common denominator between many fabricators. Based on IPC producibility classes or specific manufacturer rules, it may be possible to design for a more targeted group of manufacturers that can meet new technology requirements. Innovative new tools allow design teams to create a set of DFM rules targeted for specific technology or manufactur-

ing requirements. The rules should be imported into a design at any time to detect how the design sizes up.

Conclusion

PCB designers are challenged to meet aggressive schedules with the highest quality board possible. They take pride in their work. To avoid delaying the project or creating poor-quality boards that may end up costing their employers more to produce the products, designers need a way to avoid DFM errors earlier in the design cycle. Traditional tools that run as a batch are no longer sufficient to avoid discovering issues late in the design cycle. An ounce of prevention is worth a ton of cure at the end. **PCBDESIGN**



Hemant Shah is management group director of Allegro PCB Products at Cadence Design Systems.



Ed Acheson is senior principal product engineer for Allegro PCB Products at Cadence Design Systems.

STRIVING FOR EXCELLENCE IN EVERYTHING WE SAY AND DO.

EXCELLENCE

FAITH

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

[Review Our Advanced Capabilities Now](#)



MANUFACTURERS OF QUALITY PRINTED CIRCUIT BOARDS

The Absolute Best Value in
High Technology Circuit Boards

847.891.5800 • www.eagle-elec.com

[Victory Giant Technology: The Qualified Forerunner of PCB Smart Factories](#)

The topic of the July issue of PCB007's China Online Magazine is "Automatic PCB Factories," focusing on the pressures of cost, delivery time and process, and the irreversible trend toward automated PCB factories. Centering on this theme, we interviewed Andy Zhou, COO of Victory Giant Technology (HuiZhou) Co. Ltd.

[Catching up with American Standard Circuits' Joe Nickerson](#)

Dan Beaulieu had the chance recently to sit down with Camtech Technologies founder and President Joe Nickerson, now the newly appointed American Standard Circuits' VP of sales, to discuss Camtech Technologies and his new role with American Standard Circuits. It was an interesting talk.

[EPTE Newsletter: M-Tech 2017](#)

Tokyo Big Sight hosted the Mechanical Component & Materials Technology Expo (M-Tech) on June 21. The three-day show included the 21st Design Engineering & Manufacturing Solutions Expo (DMS), the 25th 3D Virtual Reality Expo (IVR) and the Medical Device Development Expo (MEDIX).

[Punching Out! Mid-2017 Report on the State of the N.A. PCB M&A Market](#)

One of the most popular questions we receive concerns the market for M&A. Here is our take on the current market for PCB shops in North America. In general, the PCB market in North America is not growing, which means that to grow, shops either must take market share from others, or grow through acquisitions.

[IPC: Department of Labor Releases RFI to Seek Information on Overtime Regulations](#)

This week, the Department of Labor (DOL) released a request for information (RFI) to gather additional feedback from the public on the overtime regulations, which define and delimit exemptions from the Fair Labor Standards Act's minimum wage and overtime requirements for certain executive, administrative, professional, outside sales and other employees.

[It's Only Common Sense: Getting to No](#)

Does this sound familiar to you? There is a potential customer you have been chasing for a long time. Now, this guy seems to be a pretty good guy because he is always willing to see you, always happy to have a cup of joe with you, and more than happy to go out to lunch with you, especially if you go to his favorite sushi place.

[Catching up with Prototron Circuits' Tucson GM Kim O'Neil](#)

This has been an exciting year for Prototron, which has achieved Mil-PRF-31032 and gone into the flex business. Most recently, Prototron received word that they had achieved their AS9100 Rev D, which made me want to check in with Kim O'Neil and find out more.

[AT&S Concludes Q1 2017/18 with Significant Revenue Growth and Improved Earnings](#)

After more than one year of production at the plants in Chongqing, China, which initially had a significant influence on the profitability of AT&S, a clear upward trend is showing and, consequently, a positive development of revenue and earnings compared with the previous year.

[TTM Technologies to Exhibit at the Electric & Hybrid Vehicle Technology Expo](#)

TTM Technologies, Inc. will be exhibiting at the Electric & Hybrid Vehicle Technology Expo North America 2017 at Booth 1013. The expo runs September 12–14. Technical experts will be available at the booth to answer printed circuit board design, manufacturing, and assembly-related questions.

[IPC Reports Strong PCB Order Growth in June](#)

IPC—Association Connecting Electronics Industries has posted the June 2017 findings from its monthly North American PCB statistical program. Strong PCB order growth moved the book-to-bill ratio up to 1.08 in June, while sales remained sluggish.

The Cadence logo is displayed in a white, lowercase, sans-serif font. The letter 'a' has a red horizontal bar above it. A registered trademark symbol (®) is located to the upper right of the 'e'.

cādence®

Best Time to Find and Fix Errors Is Now, Not Later

Real-time DFM checks while you design

Finally, signoff is smoother with less to fix so you can accelerate your release schedule. Run in-design design for manufacturing (DFM) checks using DesignTrue DFM technology in Allegro® PCB Editor as you design. All while keeping manufacturability above all else.

DesignTrue DFM technology helps you:

- Reduce rework and shorten cycles by running manufacturing checks in-design, while designing
- Ensure manufacturability with a wide range of spacing and copper area requirement rules, independent of electrical requirements
- Configure easily, apply contextually, and reuse manufacturing rules

[Watch Demo ▶](#)

Plane Cavity Resonance

by Barry Olney

IN-CIRCUIT DESIGN PTY LTD / AUSTRALIA

Plane pairs in multilayer PCBs are essentially unterminated transmission lines—just not the usual traces or cables we may be accustomed to. They also provide a very low-impedance path, which means that they can present logic devices with a stable reference voltage at high frequencies. But as with signal traces, if the transmission line is mismatched or unterminated, there will be standing waves: ringing. The bigger the mismatch, the bigger the standing waves and the more the impedance will be location dependent.

Following on from my previous columns, [Return Path Discontinuities](#) and [The Dark Side—Return of the Signal](#), in this month's column, I will cover plane cavity resonance and look at how it impacts electromagnetic radiation.

When return current flows through the impedance of a cavity, between two planes, it generates voltage. Although quite small (typically in the order of 5mV) the accumulated noise from simultaneous switching devices can become significant. And unfortunately, as core voltages drop, noise margins become tighter. Figure 1, illustrates the electromagnetic fields resonating in a cavity. This voltage, emanating from the vicinity of the signal via, injects a propagating wave into the cavity which can excite the cavity resonances or any other parallel

structure (for instance, between copper pours over planes). Other signal vias also passing through this cavity can pick up this transient voltage as crosstalk.

The more switching signals that pass through the cavity, the more noise is induced into other signals; it affects vias all over the cavity, not just the ones in close proximity to the aggressor signal vias. This cavity noise propagates as standing waves spreading across the entire plane pair. This is the primary mechanism by which high frequency noise is injected into cavities—by signals transitioning through cavities, using each plane successively as the signal return path.

At low frequency, the cavity impedance (Figure 2) looks inductive due to the inductance of the voltage regulator module (VRM). However, the VRM inductance, combined with the plane resonance, produces an anti-resonance peak at ~35Mhz, in this case. The plane resonance depends on a number of factors including conductor loss, plane area, dielectric constant and dielectric thickness between the plane pairs. In this example, the planes naturally resonate at ~300MHz with harmonics beginning at ~1GHz. If the AC impedance is high, at the fundamental frequency or at any of the odd harmonics, then the board will tend to radiate from the fringing fields at the edges of the planes.

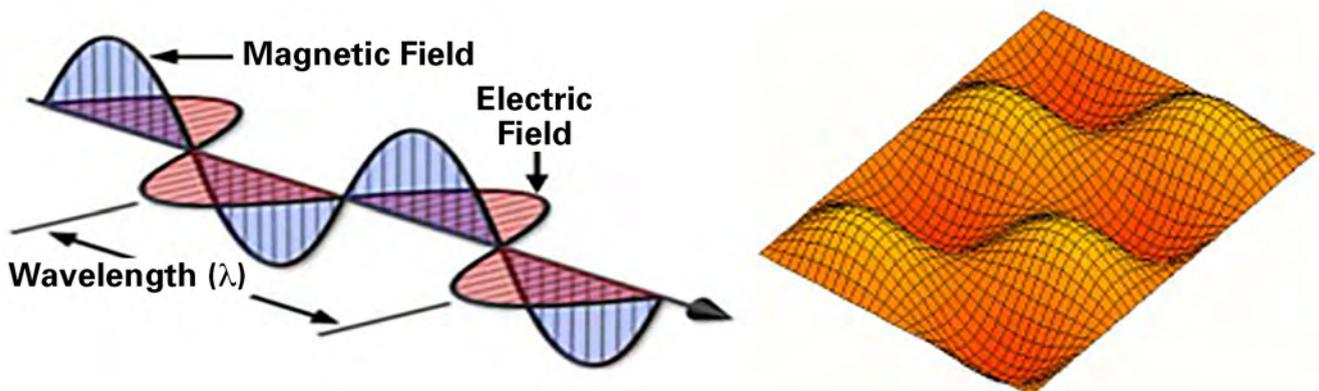


Figure 1: Electromagnetic fields propagating through a plane pair cavity (Source: Wikipedia).

Engineering And Providing Balanced Interconnect Solutions



MIRACO

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

...connecting technology

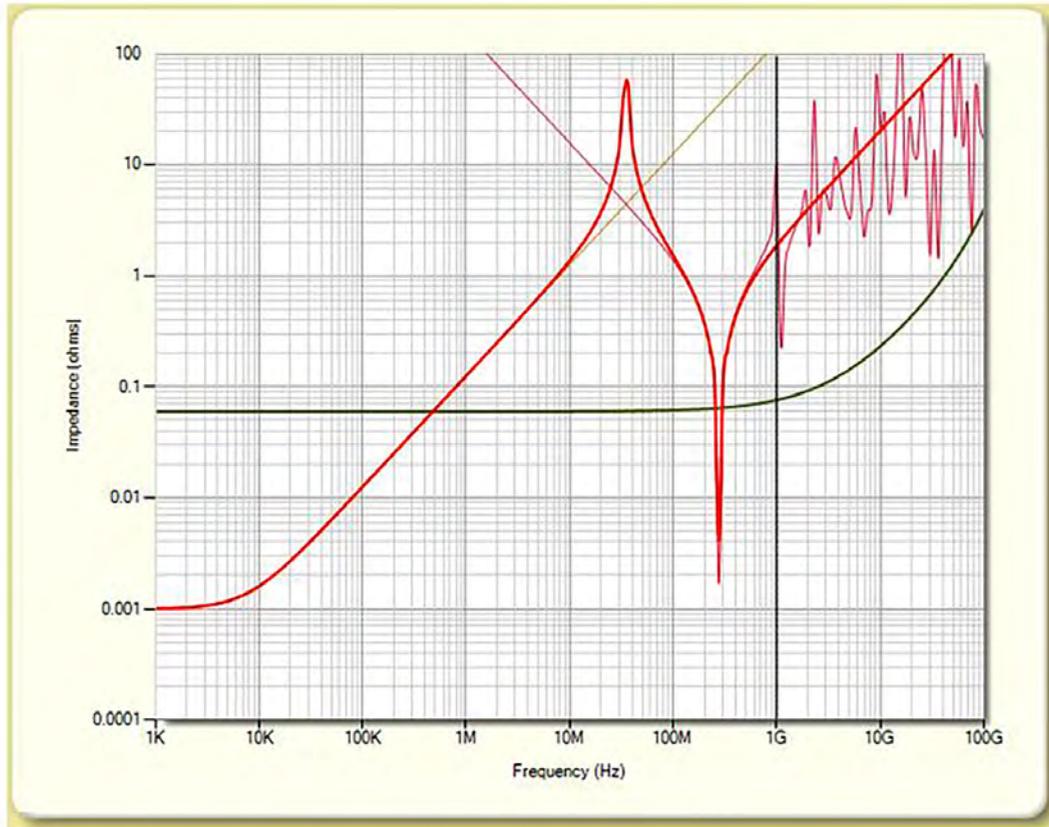


Figure 2: Impedance profile of a plane cavity with VRM (Source: iCD Design Integrity).

Signal paths are designed to be low Q resonators to dampen the ringing and to increase the bandwidth. However, cavities composed of a power and ground plane, can have very high Qs. This means even a slight amount of coupling, from signal paths, can drive resonances and give rise to long range noise voltages within the cavity. Knowing the cavity resonant frequencies, that one might encounter, can highlight potential problems.

When the cavity has open end boundary conditions, resonances arise when a multiple of half wavelengths can fit between the ends of the cavity. Figure 3 shows the cavity resonance of a plane pair with a resonant frequency of 1GHz. If the signal clock frequency (or harmonics) are multiples of 1GHz, then noise can be injected into the plane cavity. When the clock or data harmonics overlap with the cavity resonant frequencies, there is the potential for long range coupling between any signals that run through the cavity. This is one reason

why all return planes should be GND layers, so that stitching vias between GND planes can be placed adjacent to each signal via transition to minimize the possibility of exciting the cavity resonance.

Cavity resonances are (at first) a signal integrity issue but the amplification of cavity resonance excited by fast rise time signals, at high frequencies, can also contribute to electromagnetic emissions (EMI). The frequency components of the voltage noise are related to the peak impedance of the cavity and the frequency components of the return currents. In any complex system, with typical interconnect density, avoiding signal layer transitions is not practicable and is an issue that designers must live with. However, one can learn to avoid injecting excessive noise into the cavity or at least minimize the impact.

The goal of designing a high-performance cavity is to reduce the impedance peaks below the target impedance level and to push the peak

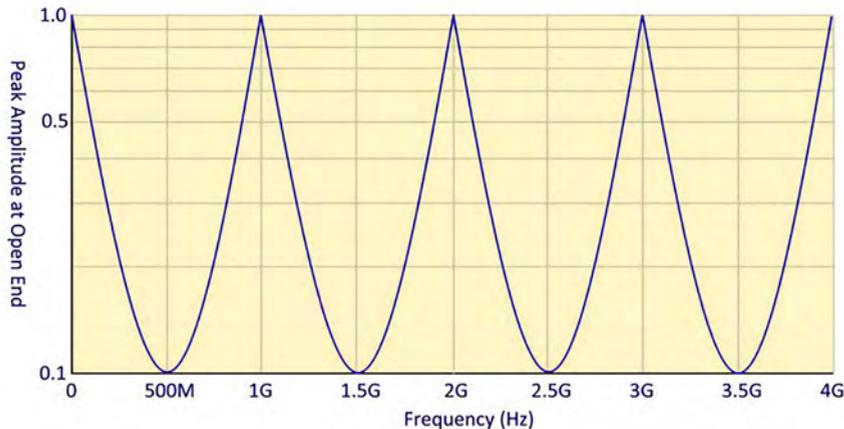


Figure 3: Amplitude at the far end of planes as input frequency is swept (Source: Eric Bogatin).

frequency components, above the bandwidth of the signals.

So, how should engineers and PCB designers go about reducing cavity resonance and emissions?

- A thin dielectric, in the plane cavity, is the most effective way of reducing the peak amplitude of the modal resonance. It reduces spreading inductance and the impedance, of the cavity, and also reduces the resonance peaks by damping the high-frequency components. Thinner plane separation implies less area of equivalent magnetic current at the plane pair edge, or equivalently less local fringing field volume, and therefore lower emissions for a given field strength.

- A dielectric material with a high dielectric constant (Dk) should be selected to add more planar capacitance. This is contrary to the typical choice of high-speed materials that require a low Dk. Remember; we are talking about the dielectric embedded between the planes, which has little impact of the signal properties.

- The parallel resonant frequencies, of the cavity, can be pushed up above the maximum bandwidth of the signals, by reducing the plane size and by adding stitching vias between (similar) planes of a cavity.

- Where the length of a rectangular plane is a simple multiple of its width, such as 1, 1.5 or 2, the resonant frequencies of the length and

width directions will coincide at some frequencies, causing higher-Q peaks—more intense resonances—than usual. So it is best to avoid square planes and simple L:W ratios by choosing irrational numbers.

- When plane pairs resonate, their emissions come from the fringing fields at the board edges. With ground/power plane pairs, edge-fired emissions can be reduced by reducing the plane separation, as described earlier, but this technique cannot generally be used for multiple planes. Alternatively, make the power planes slightly smaller

(~200 mil) than the GND plane. This modifies the pattern of the fringing fields, pulling them back from the edge, and may help reduce emissions to some extent.

Optimization of the PDN is a trial and error process. A combination of modifications to dielectric constant and thickness, of the material, together with an adjustment of plane size can usually establish the minimum resonance for a given configuration. Employing AC PDN analysis software (Figure 4) allows one to integrate the layer stack and dielectric materials with the PDN and enables visualization of these critical adjustments. If you can't see it, you can't fix it!

Points to Remember

- Plane pairs, in multilayer PCBs, are essentially unterminated transmission lines.

- If a transmission line is mismatched or unterminated, there will be standing waves—ringing.

- When return current flows through the impedance of a cavity, between two planes, it generates voltage which can excite the cavity resonances.

- Other signal vias, also passing through this cavity, can pick-up this transient voltage as crosstalk.

- This cavity noise propagates as standing waves spreading across the entire plane pair.

- The slightest amount of coupling, from signal paths, can drive resonances and give rise to long range noise voltages within the cavity.

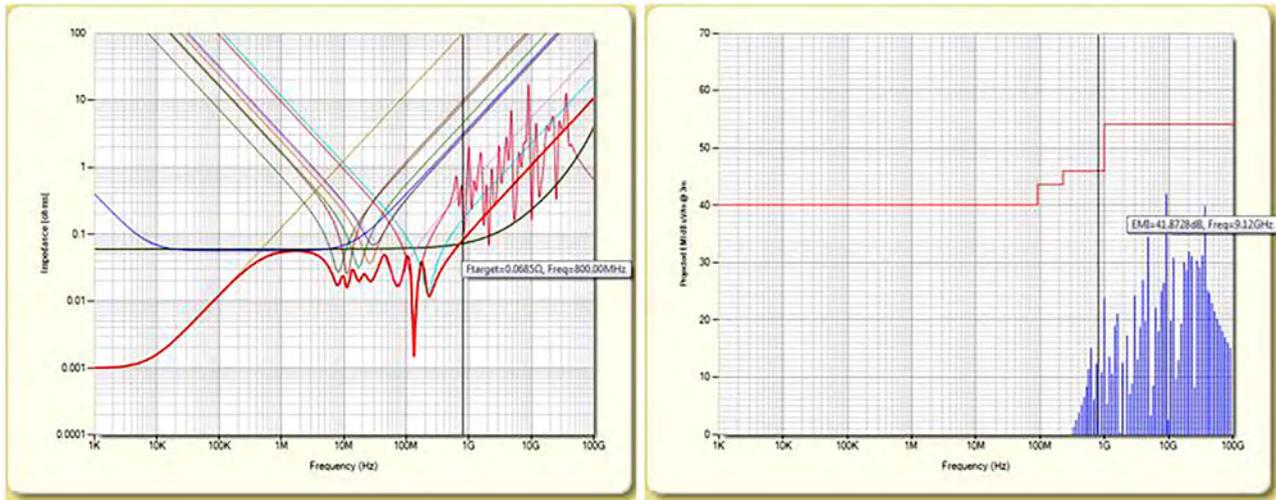


Figure 4: Optimized PDN with projected EMI (Source: iCD Design Integrity).

- When the cavity has open end boundary conditions, resonances arise when an integral multiple of half wavelengths can fit between the ends of the cavity.
- When the clock or data harmonics overlap with the cavity resonant frequencies, there is the potential for long range coupling between any signals that run through the cavity.
- Stitching vias between GND planes can be placed adjacent to each signal via transition to minimize the possibility of exciting the cavity resonance.
- The frequency components of the voltage noise are related to the peak impedance of the cavity and the frequency components of the return currents.
- The goal of designing a high-performance cavity is to push the peak frequency components, above the bandwidth of the signals, and to reduce the impedance peaks below the target impedance level.
- A thin dielectric, in the plane cavity, is the most effective way of reducing the peak amplitude of the modal resonance.
- A dielectric material with a high dielectric constant (Dk) should be selected to add more planar capacitance.
- Effective ways of pushing up the parallel resonant frequencies is by reducing the plane size and by adding stitching vias between (similar) planes of a cavity.

- Avoid square planes and simple L:W ratios by choosing irrational numbers.
- Make the power planes slightly smaller than the GND plane. This modifies the shape of the fringing fields, pulling them back from the edge. **PCBDESIGN**

References

1. Barry Olney's Beyond Design columns: [Learning the Curve](#), [Plane Crazy, Part 2](#)
2. [Principles of Power Integrity for PDN Design](#), by Larry D. Smith, Eric Bogatin
3. [What is the resonant frequency of a cavity?](#), by Eric Bogatin
4. [Advanced PCB Design and Layout for EMC](#), by Keith Armstrong
5. [High-Speed Digital Design](#), by Howard Johnson

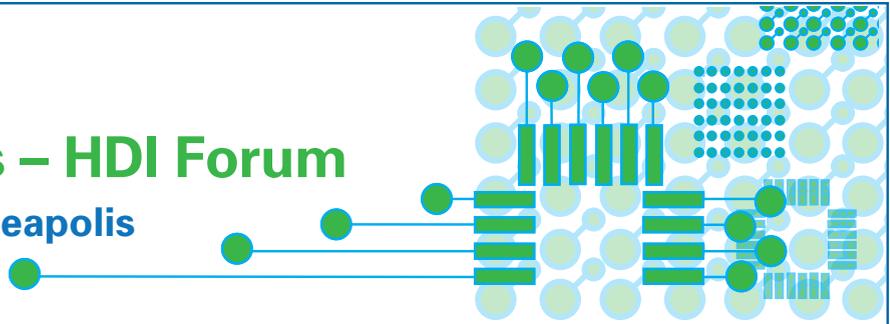


Barry Olney is managing director of In-Circuit Design Pty Ltd (iCD), Australia, a PCB design service bureau that specializes in board-level simulation. The company developed the iCD Design Integrity software incorporating the iCD Stack-up, PDN and CPW Planner. The software can be downloaded from www.icd.com.au.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



IPC Flexible Circuits – HDI Forum

October 17–18, 2017 • Minneapolis



Topics of instruction and tutorials will range from flexible circuit metallization to designing a reliable and cost-effective HDI-circuit — this event promises to provide quality education and networking opportunities.

Subject matter experts from many of the industry's leading companies in this field will share their best practices, and proven methods of success, including:

- TTM Technologies
- Dow Chemical
- NSWC Crane
- Flexible Circuit Technologies
- Pioneer Circuits
- RBP Chemical Technology, Inc.
- Saturn Electronics
- Uyemura
- San Diego PCB
- DuPont
- Nextflex
- HDPUG

[MORE INFORMATION](#)

[REGISTER TODAY](#)

Sponsors



Exhibitors



Thermal Management: Problems and Solutions

by Jade Bridges

ELECTROLUBE

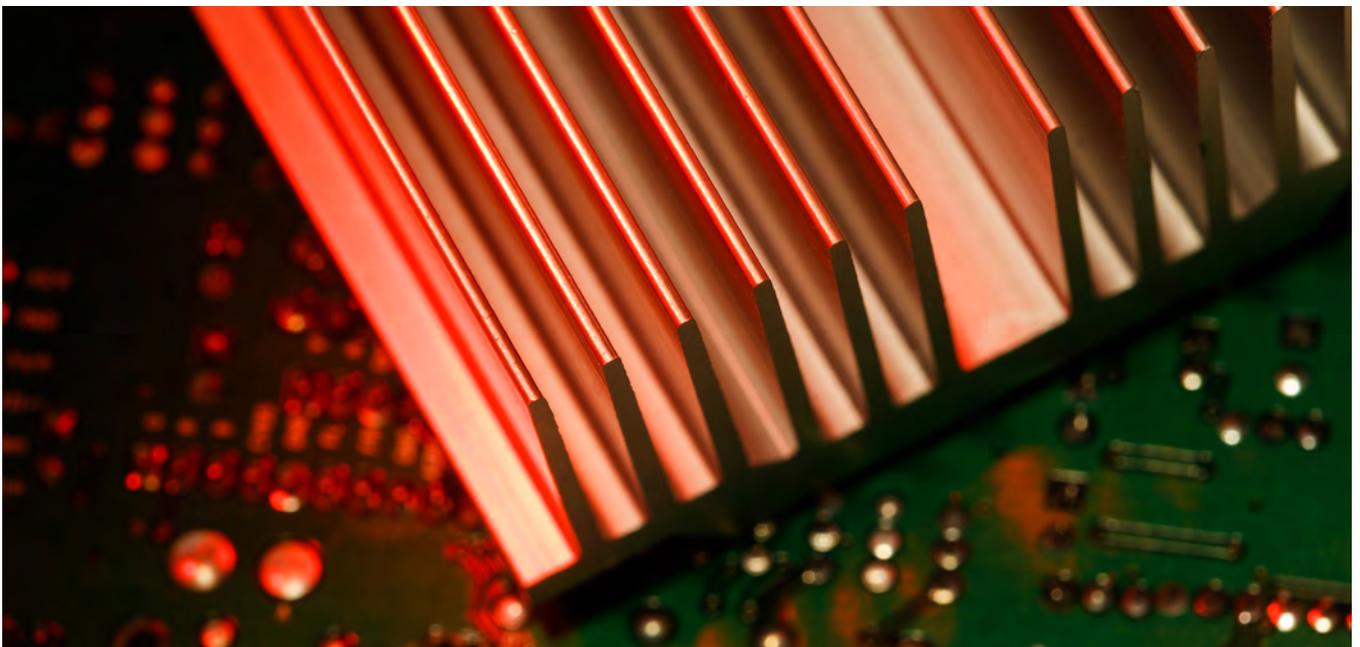
Last month I expressed what I hoped would prove a good starting point for my series of articles on the all-important subject of thermal management. This was a five-point guide, which I compiled based on the many questions that our customer support teams field every day on the telephone, at exhibitions and when visiting customers' premises. This month, I'd like to progress matters further and concentrate on some of the problems you are likely to encounter when applying a thermal interface material, and how to select materials that are appropriate for your electronic assembly and its operating conditions. I'll also be looking a little more closely at bond line thickness and its effects on performance, as well as the alternatives to non-curing thermal pastes. But let's start with a problem that, I'm afraid, most of you will encounter at some stage in your product development work: pump-out.

Pump-out can occur when a device—an insulated-gate bipolar transistor (IGBT), for exam-

ple—is subject to temperature changes resulting in relative motion between the conductor and its heatsink between which a non-curing thermal paste has been applied. This motion can cause such pastes to be squeezed or pumped out from the interface gap, reducing the thermal transfer performance.

To tackle issues with pump-out, it is first important to understand the conditions and materials involved. Temperature extremes and rate of change of temperature are important factors that will determine the choice of thermal interface material; for example, if operating temperatures are likely to range between -50°C and 200°C , a silicone-based thermal paste would be the preferred option.

In addition to these temperature considerations, the materials being used may affect the interface material, particularly with regards to the spacing between the device and its heatsink—otherwise known as the bond line thickness.



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



Thermal pastes are often designed to be applied in as thin a layer as possible. They improve the contact between the device and its heatsink by eliminating air gaps and ensuring that the full surface contact area is available for heat transfer. There is, however, a critical thickness which determines maximum thermal transfer with minimal thermal resistance, and while this will depend on the roughness of the substrates and required spacing, it is generally between 30 and 100 microns.

A thermally conductive heat transfer material will naturally have a lower thermal conductivity than the metallic heat sink material, so keeping the thickness of the film at the interface as low as possible will decrease the thermal resistance and, in turn, lower the operating temperature of the device. While interface materials should be applied at minimum thickness to achieve low thermal resistance, the resulting bond line may also be affected by the substrate's smoothness and spacing, i.e., components and heatsink surfaces at the interface. If a non-curing thermal interface material is applied more thickly due to spacing/materials considerations, the greater the pump-out effect will be.

So, while the general rule is to minimise the thickness of non-curing thermal interface materials, this must not be to the detriment of their stability in use. For example, it is particularly important to ensure that a lower film thickness does not result in air gaps forming in the film between the device and its heatsink as this will increase thermal resistance and the device will not be cooled as efficiently as desired.

Consider using alternatives to non-curing thermal interface pastes such as the new phase change materials that are now coming on to the market, or for those wide operating temperature range applications, a surface-cure silicone thermal paste; both alternatives will minimise pump-out whilst keeping thermal resistance low.

The low phase change temperature of phase change materials allows low thermal resistance over a wide temperature range, ensuring minimal bond line thickness with improved stability and pump-out resistance. And while a surface cure thermal paste doesn't set entirely (thus allowing for easy rework), it is specifically formulated to resist pump-out, particularly for those

applications that are exposed to rapid and frequent changes in temperature.

As well as phase change materials and surface curing silicone thermal pastes, other alternatives to non-curing thermal pastes include thermal gap filler pads, which are available as silicone and non-silicone based sheet materials that can be cut to size and applied by hand. These are highly thermally conductive, but they do have a higher thermal resistance than thermal pastes.

Another approach is to use a room temperature vulcanized (RTV) product, which becomes a flexible rubbery material on curing, combining the properties of silicone gap filler pads with those of a conventional heat transfer paste. RTVs can be used to bond the heatsink to the component while also offering a flexible heat transfer medium. A much less flexible bond can be achieved between the device and its heatsink by using a two-component epoxy resin which cures to a tough solid bond.

A promising new material is the thermal gel which is a silicone-based formulation that offers the low thermal resistance of a non-curing thermal paste minus the latter's pump-out problems. Gels are highly conformable and, even better than the softest of silicone gap filling pads, they impart minimal mechanical forces on delicate components during application and in use.

For certain types and designs of heat generating circuitry, it may be more beneficial to encapsulate the device in a heatsink enclosure using a thermally conductive encapsulation compound. Silicone, polyurethane and epoxy resins provide both heat dissipation and environmental protection all in one.

A variety of thermal management products are available now, but as I hope I've made clear in the foregoing, it's complicated deciding on the right choice of material and/or application technique. I strongly recommend you get some expert advice before you settle on any particular material or method. **PCBDESIGN**



Jade Bridges is the global technical support manager for Electrolube Ltd.

iCD Design Integrity

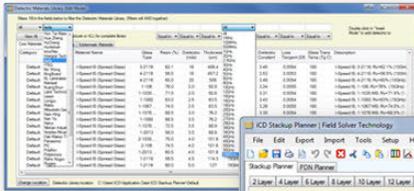
Incorporates the iCD Stackup and PDN Planner software. Offers PCB Designers unprecedented simulation speed, ease of use & accuracy at an affordable price

Termination Planner

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

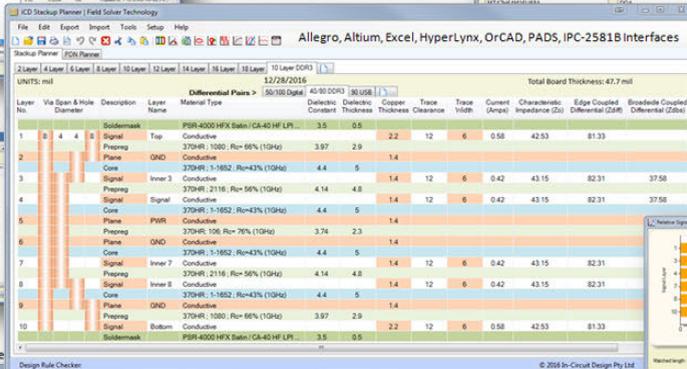
Dielectric Materials Library

30,700 Rigid & Flex Materials to 100GHz



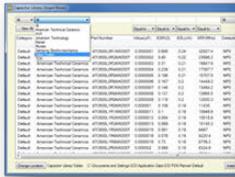
iCD Stackup Planner

Field Solver Accuracy, Characteristic Impedance, Edge & Broadside Coupled Differential Impedance



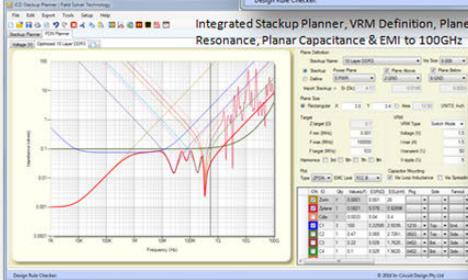
Extensive Capacitor Library

5,650 Decaps Derived from SPICE Models

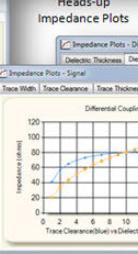


iCD PDN Planner

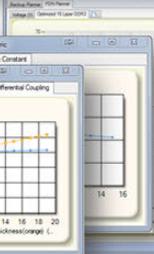
AC Impedance Analysis & Plane Resonance



Heads-up Impedance Plots



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



Matched Delay Optimization

Relative Signal Layer Propagation
Ideal DDRx Trace Delay Matching



iCD Stackup Planner -

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

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

iCD PDN Planner -

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

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

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



MilAero007 Highlights



[Performance Evaluation of Thin-Film Embedded Resistors](#)

Thin-film microstrip circuits have been widely applied in microwave communications, electronic countermeasures (ECM), and aerospace applications, etc. When manufacturing thin-film ICs, it is very important to apply deposited thin-film resistor material to fabricate high-accuracy and highly stable thin-film embedded resistors. Thin-film ICs call for thin-film resistors that meet stringent requirements.

[Prototron Circuits Achieves AS9100 Rev D](#)

Prototron Circuits announced recently that their Tucson, Arizona facility has achieved AS 9100 Rev D qualification.

[Mil/Aero Markets: Defense Budget Analysis—Patience is a Virtue](#)

Patience may be the most necessary watchword for proponents of significant growth in the budget for the U.S. military. The historic increases in defense spending promised by President Trump on the campaign trail won't come immediately, but it is likely that Congress will find ways to provide solid increases to the Pentagon's budget, particularly in 2019 and beyond.

[Eltek Receives an Order for up to \\$3 Million from a Governmental Authority](#)

Eltek Ltd. has received an order from a governmental authority for a project that includes, among other things, manufacturing and procurement in an amount of up to approximately US\$3 million.

[IPC Welcomes Exec Order on Strengthening the U.S. Defense Industrial Base](#)

According to Nextgov, Peter Navarro, Deputy Assistant to the President and Director of the White House Office of Trade and Manufacturing Policy (OTMP), said the order will determine whether there are enough U.S. manufacturers to supply "everything from submarine propeller blades to circuit boards and military-grade semiconductors," and whether there are enough skilled workers to work in the field.

[It's Only Common Sense: Finally, the DoD Gets It!](#)

Last week a friend of mine sent me this excerpt from the Congressional Record. Yes, that congressional record. If this doesn't set your hair on fire, then nothing will.

[IMPACT Interview: Anaya Vardya, American Standard Circuits](#)

I-Connect007's Patty Goldman spoke with Anaya Vardya, president and CEO of American Standard Circuits, a few days after the conclusion of IMPACT, especially wanted his views as a first-timer there.

[Trilogy Circuits: Mil/Aero Demands Technological, Regulatory Expertise](#)

In the 16 years since its founding, Trilogy Circuits has become one of the go-to companies for mil/aero PCB design and manufacturing. I spoke with Charlie Capers, president and founder of the Richardson, Texas firm, about how he keeps ahead of the game. As he explains, succeeding with mil/aero PCB work requires not only mastery of the technological side, but of the regulations and certifications as well.

[Strategies for Compliance with DoD Regulations Including ITAR and DFARS](#)

ITAR is usually the topic when compliance with DoD regulations is discussed. But what about DFARS? This article will examine strategies one can implement to ensure that one is compliant with all DoD regulations, by analyzing internal and external factors in relation to procurement and compliance, and by asking the vital questions: what, how, where and to whom?

[Strategic Technology Office Outlines Vision for "Mosaic Warfare"](#)

STO seeks to turn complexity into a powerful new asymmetric weapon via rapidly composable networks of low-cost sensors, multi-domain command and control nodes, and cooperative manned and unmanned systems.

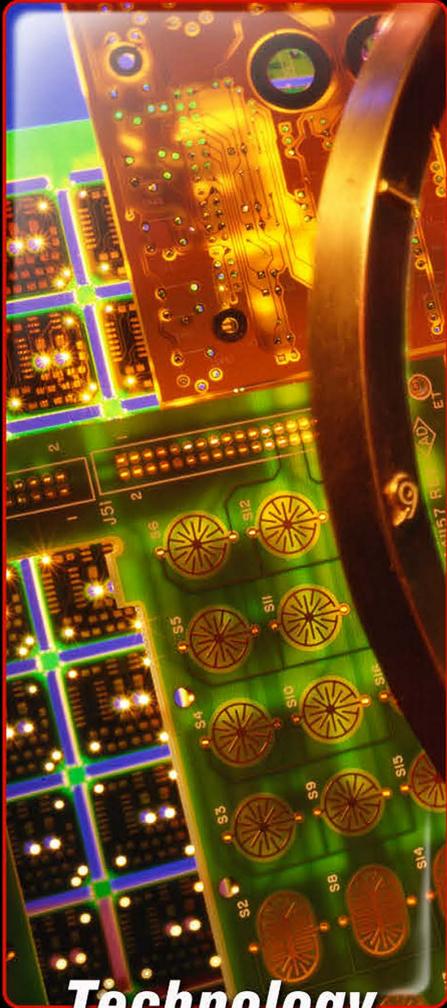
We're Hiring!

prototron.com

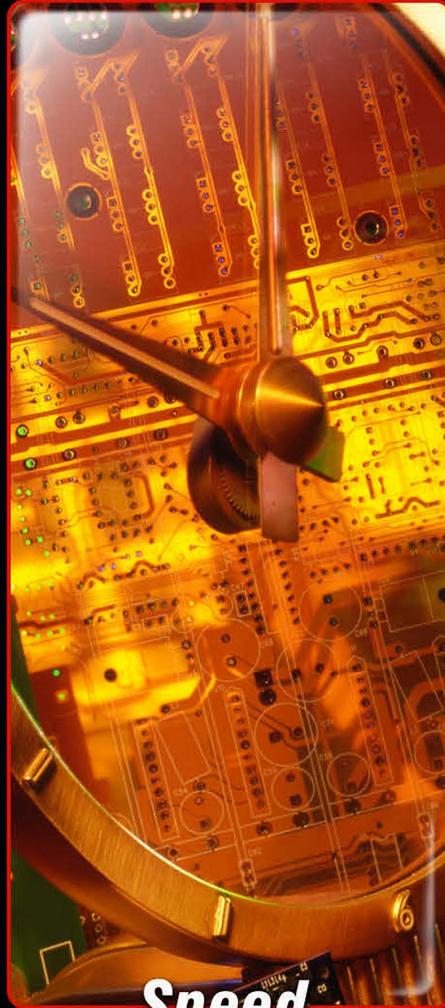
See us in
booth #309



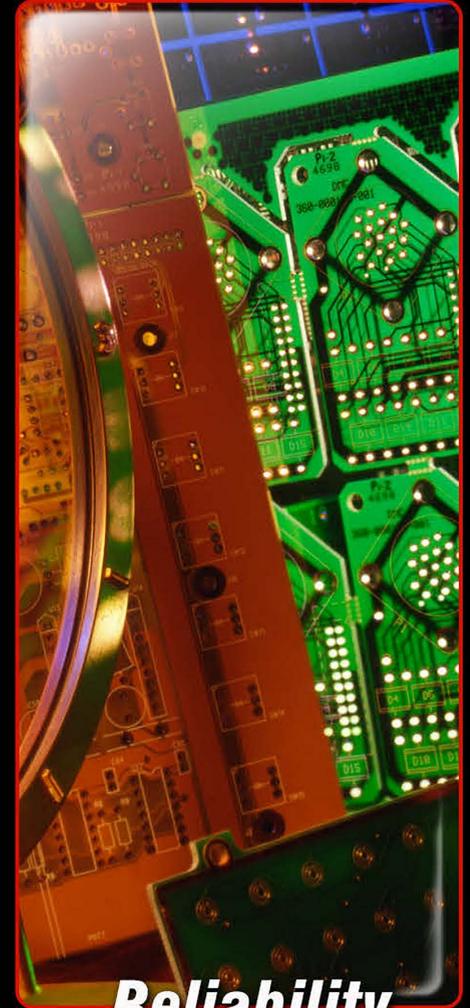
We Take the Time to do it Right
because you don't have the time to do it over.



Technology



Speed



Reliability



Prototron Circuits

High quality quick turn manufacturing, just another reason we are
America's Board Shop

Prototron is Hiring! We are currently seeking candidates to fill positions for both a midwest sales professional and an experienced PCB process engineer. To apply, contact us at 425-823-7000 or email Russ Adams at russa@prototron.com for sales or Kirk Willams at kirkw@prototron.com for engineering.

True Design Efficiency: Think Before You Click

by **Tim Haag**
CONSULTANT

Over the years, I have seen a lot of different circuit board designs. In the service bureaus, I saw many different design technologies, while in companies with captive design departments I saw a lot of different design methodologies. Finally, as a CAD application engineer, I saw both different technologies and different methodologies.

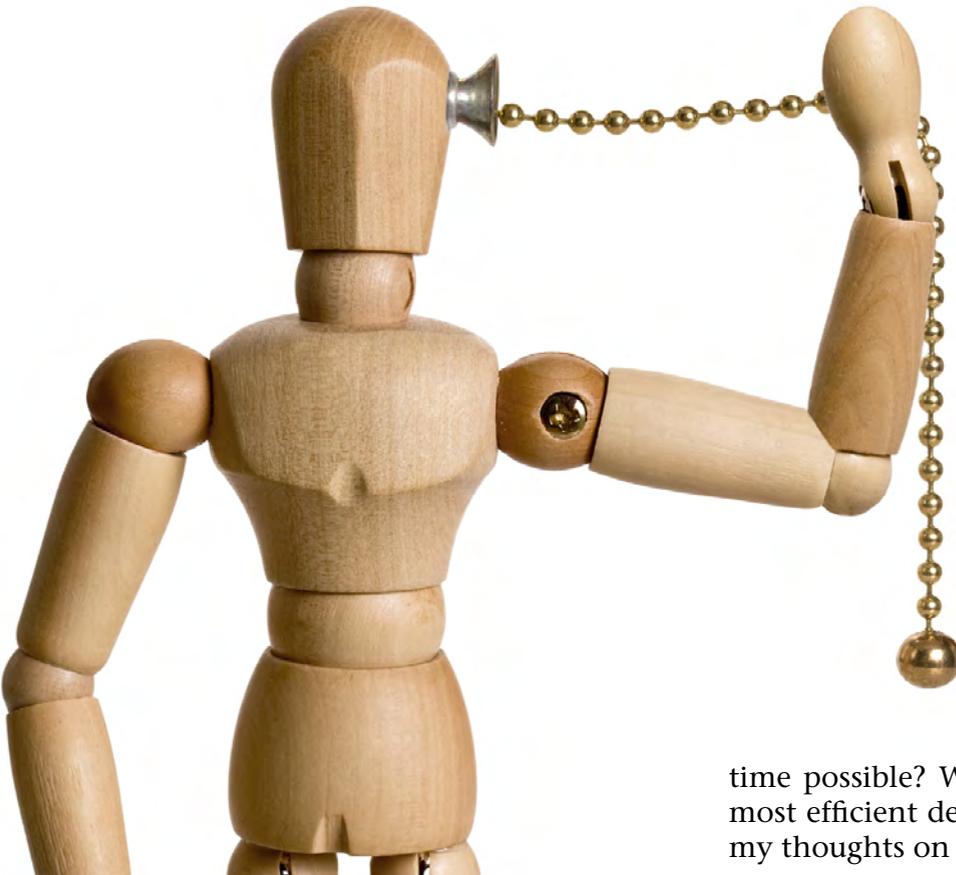
I think that it is safe to say that I have had a pretty good sampling of our industry and I am grateful for the variety that I have been exposed to. The truly interesting thing for me, however, has been in seeing all the different methods that the hundreds of designers that I've worked with have used in the creation of their PCB designs.

We are all unique and that, combined with all the different types of designs out there, adds up to a lot of different ways to successfully complete a PCB design.

In the service bureau, we had multiple customers with different specifications to follow and different libraries to use. Some designers would just plow into a new design and worry about fixing the details later, while others would invest lots of time in research up front to avoid the corrections later in the design cycle. I have seen it happen over and over again; those who plowed into a design would spend a lot of time correcting mistakes later, while those who were overly cautious would take too long in their work.

At the captive shops that I've worked with, where the designers were more involved in the entire design cycle and had better access to the corporate libraries, staff engineers, etc., the story was often the same. Some designers would jump into the deep end of the pool of design without any thought to drowning while others would be so busy lacing up their life preservers of preparation that they would take too long getting out of the shallows and into the depth of their design.

So, what's the best approach here? How can we combine the best of both worlds in order to get the greatest design in the shortest time possible? What does it take to create the most efficient design process? Here are some of my thoughts on the matter:



First of all, be prepared. I know that this approach can be a buzz-kill; after all, we designers really like to get into the fun part of the design as quickly as possible. But I believe that being prepared is something that helps everyone in life, not just the Boy Scouts. When I was flying, we were careful to go through our pre-flight checklists so there weren't any unexpected surprises later. When getting ready for a trip, we make sure that we are completely packed so that five days into a two-week trip we aren't unexpectedly running out of underwear.

“When I was flying, we were careful to go through our pre-flight checklists so there weren't any unexpected surprises later.”

With our designs, I believe that it shouldn't be any different; we should be prepared as much as possible. Do we have all of the information in from the electrical engineer, the mechanical engineer, manufacturing and test? Are we confident with the content of our data or are there holes in it that will surprise us somewhere down the line? Are the libraries in good shape the way they are, or do they need updating or new parts added for our design? Are we in synch with project management so that we understand the schedule and are ready to hit our marks for design reviews and job completion?

Back in the day, I would notice that some designers would plow into a new design without much preparation. Before I had finished organizing the specifications on my design they were already pulling library parts on theirs, and when I was going through the netlist they were already placing components. I would just be starting my critical placement while they were already routing traces, and soon they were ready for review while I was still working away. But sometimes their haste caused problems forc-

ing them to go back and make some changes or corrections in their libraries. Sometimes they would have to rip up hours' worth of hand routing to re-place parts or change whole areas of the board. And sometimes their work was rejected by the engineer during iterative design reviews, and they would have to go back several steps. Meanwhile I was slowly building up steam on my design and going through all the steps in sequential order. Usually there wasn't much that had to be changed in the final review because I had been updating the engineer as I worked so that I knew that I was on the right track. Often I would finish my design ahead of schedule and without much in the way of re-work because of the process that I followed making sure that I completed one step before moving on to the next.

It is embarrassingly obvious that I am pretty conservative in how I approach PCB design, but this approach has worked for me for many years and I have completed a lot of designs with it. But on the flip side, I have found myself getting stuck doing it this way as well. I have had plenty of times staring at the screen wondering how I'm going to proceed next, or having to take a "time-out" in order to clear out the cobwebs in my brain in order to get unstuck in my thinking. And those designers who plunged into the deep end of the pool without the preparation that I was so insistent upon? They were just swimming laps around me while I was treading water in my efforts to stay afloat.

Sometimes an overabundance of caution can kill the creativity needed for brilliance. So, in a complete reversal of what I said earlier, don't be afraid to let yourself go and see where it gets you. There have been plenty of times when designing a board that I needed to get past a logjam, so I jumped ahead and started working on other portions of the board which in turn helped me to get past the area that I was stuck in. I have also had many times where working ahead of myself and playing with placement or routing before I was technically ready yielded a big bonus in how it prepared me for what was to come. I have learned over time to not be afraid to take that jump into the deep end, there can be some very big rewards for it.

So, how do we put this all together?

As I said, be prepared and think before you click. Make sure that your data is complete and ready, that you are communicating with the other relevant design team members, and that you have a complete understanding of what you want to accomplish. Then, don't allow yourself to get stuck by the small stuff. Force yourself to go work on another area or even go and take a break if you need to. Our brains can fool us sometimes by making a mountain out of a mole-hill and sometimes we just need to just push it aside in order to break the log-jam that is holding us up. And finally, don't be afraid to take a giant leap into the great unknown by doing something totally out of the box in order to stimulate your creativity.

As long as your work is backed up, and you were prepared in the first place, take a chance and start some placement or routing that you normally wouldn't do at that point. It might just give you some insight into a new way of doing your design that will lead to a brilliant break-through. And don't be afraid to simply throw your experimental work away and start

over again if you're still stuck. We've got tons of disc space these days for backups, and it is often much wiser to start over again instead of spending hours trying to figure out how to back out of a jam.

After a lot of years of doing board design, I have come to the conclusion that a rigid design process punctuated by occasional moments of throwing caution to the wind gives me the best results. I have found that this both stimulates creativity and gives the protection of staying within established guidelines. It is my opinion that this approach maximizes design efficiency and ultimately gives the best board design results.

What do you think? **PCBDESIGN**



Tim Haag is a consultant based in Portland, Oregon.

Researchers Challenge Status Quo of Battery Commercialization

While more efficient, effective, and economical batteries are critical to the world's clean energy future, the path to bringing such technologies to market over the last 25 years is overwhelmingly littered with failure.

A pair of Northwestern University researchers and a Northwestern-affiliated startup, however, are touting a different way and have pulled inspiration from a seemingly unlikely place: the pharmaceutical world.

In a recent paper published in *MRS Energy & Sustainability*, Vinayak Draid, the Abraham Harris chaired professor of materials science and engineering at Northwestern's McCormick School of Engineering, his graduate student Eve Hanson, and SiNode Systems co-founder and CEO Samir Mayekar propose an updated model of U.S. battery commercialization.

The model is informed by pharma, an industry that faces many similar commer-

cialization challenges yet, unlike energy, boasts a deep history of successful market debuts.

As the head of Chicago-based SiNode Systems, Mayekar confronts the challenges of battery commercialization daily. The startup is commercializing a novel silicon anode technology based on research from the lab of Harold Kung, Walter P. Murphy Professor of Chemical and Biological Engineering, but faces daunting technical and market hurdles.



With more economical and energy-dense batteries holding the potential to solve contemporary energy challenges — such as balancing the intermittency problems of renewables and making possible electric transportation fleets — the researchers believe applying the pharma-inspired development pipeline to batteries can breed rich, transformative technologies.

"We think this is a way for more battery startups to succeed," Hanson said.

Pssst!
Are You Looking
for Someone?



Place your notice in our Help Wanted section.

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

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

To get your ad into the next issue, contact:

Barb Hockaday at barb@iconnect007.com or +1.916.608.0660 (-7 GMT)

I-Connect007
GOOD FOR THE INDUSTRY





Leading PCB Manufacturer Seeks Account Manager

Meiko Electronics, a global leader in PCB interconnect solutions for Tier I and II electronics companies, has expanded its global manufacturing footprint to include two campuses in China and two in Vietnam.

We are looking for a full-time account manager to introduce the company and our outstanding capabilities to new customers, primarily in the automotive space.

Key Responsibilities:

- Work directly with PCB sourcing teams to generate interest in our company
- Manage all customer relations, including scheduling onsite customer meetings with sourcing team decision makers, factory audit and qualification visits resulting in AVL status attainment
- Manage quality/engineering/logistics issues pertaining to key accounts

Qualifications:

- 3 years' professional experience in PCB sales or similar electrical component experience
- Excellent communication and relationship building skills
- Organizational skills, with a strong attention to detail
- Knowledge of Japanese or Mandarin languages a plus

Location:

The ideal candidate will have some initial prospective customers located nearby in the Midwest region and the ability to travel as needed to our Asia-based manufacturing locations.

Competitive compensation and benefits package, including competitive base salary, generous bonus/commission plan, medical/dental/vision and life insurance, matching 401k, PTO.

[apply now](#)



American Standard Circuits

Creative Innovations In Flex, Digital & Microwave Circuits

CAM Operator

American Standard Circuits is seeking a candidate to join its team in the position of CAM operator. Applicants will need experience in using Valor/Genesis (GenFlex) CAD/CAM software with printed circuit board process knowledge to edit electronic data in support of customer and production needs. Other requirements include:

- 5+ years of experience in PCB manufacturing
- Process DRC/DFMs and distinguish valid design and manufacturing concerns
- Modify customer supplied data files and interface with customers and engineers
- Release manufacturing tooling to the production floor
- Prepare NC tooling for machine drilling, routing, imaging, soldermask, silkscreen
- Netlist test, optical inspection
- Work with production on needed changes
- Suggest continual improvements for engineering and processing
- Read, write and communicate in English
- Understand prints' specifications
- Must be U.S. citizen or permanent resident (ITAR)
- High school graduate or equivalent

[apply now](#)



ventec
INTERNATIONAL GROUP
騰輝電子

Ventec Seeking U.S. Product Manager for tec-speed

Want to work for a globally successful and growing company and help drive that success? As a U.S.-based member of the product and sales team, your focus will be on Ventec's signal integrity materials, tec-speed, one of the most comprehensive range of products in high-speed/low-loss PCB material technology for high reliability and high-speed computing and storage applications. Combining your strong technical PCB manufacturing and design knowledge with commercial acumen, you will offer North American customers (OEMs, buyers, designers, reliability engineers and the people that liaise directly with the PCB manufacturers) advice and solutions for optimum performance, quality and cost.

Skills and abilities required:

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

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

Please forward your resume to jpattie@ventec-usa.com and mention "U.S. Sales Manager—tec-speed" in the subject line.

www.ventecclaminates.com

apply now



ZENTECH

Experienced PCB/ Assembly Professionals Needed

Find a rewarding career and become a successful leader with Zentech Manufacturing. With locations in Baltimore, Maryland, and Fredericksburg, Virginia, Zentech is rapidly growing and seeking experienced professionals in all areas:

- Operations leadership
- Engineering
- Manufacturing engineering
- Program management
- Testing
- Quality assurance

Zentech offers an excellent benefits package, including an employer-matched 401(k) program.

Established in 1998, Zentech holds an ultimate set of certifications relating to the manufacture of mission-critical printed circuit card assemblies, including ISO:9001, AS9100, DD2345, ISO 13485, J-STD 001 with space certification, and is ITAR registered. Zentech was also the first in the U.S. to re-certify for IPC 610 trusted source QML status.

apply now

Altium

Application Engineer

The application engineer is the first contact for our customers who have technical questions or issues with our product. We value our customers and wish to provide them with highest quality of technical support.

Key Responsibilities:

- Support customer base through a variety of mediums
- Log, troubleshoot, and provide overall escalation management and technical solutions
- Create various types of topic based content, such as online help, online user guides, video tutorials, knowledge base articles, quick start guides and more
- Distill complex technical information into actionable knowledge that users can understand and apply
- Continually develop and maintain product knowledge

Requirements:

- Understanding of EDA electronic design software, schematic capture and PCB layout software
- Bachelor's degree in electronics engineering or equivalent experience
- Sales engineering and/or support engineering experience
- Circuit simulation and/or signal integrity experience
- Understanding of ECAD/ MCAD market segments
- Understanding of micro controllers, SoC architecture and embedded systems market
- Database experience preferred (i.e., MySQL, PostgreSQL, Microsoft Access, SQL, Server, FileMaker, Oracle, Sybase, dBASE, Clipper, FoxPro) etc.
- Experience with PLM/PDM/MRP/ERP software (Program Lifecycle Management) preferred
- Salesforce experience a plus

Salary based upon experience. Comprehensive benefits package and 401k plan. Openings in USA, UK, and Germany.

For more information, contact Altium.

[apply now](#)

esi

Designed for Brilliance. Engineered for Production.

Embedded Engineer— Portland Oregon

ESI, a pioneer and leading supplier of world-class production laser systems that help microtechnology customers achieve compelling yield and productivity gains, is looking for an embedded engineer.

As a software engineer, you are designing and delivering the embedded software that drives ESI's leading-edge manufacturing systems. The successful candidate will join a multi-disciplinary team focused on developing cutting edge technology in a fast-paced and technically challenging environment. Primary responsibilities will include embedded real-time system development, low-level machine control, system-level troubleshooting, and some supporting application level development.

Desired experience includes:

- Highly proficient in C/C++ programming
- Proficient in working with and programming DSPs or microcontrollers
- Experience designing software for embedded systems with constrained resources
- Knowledge of different embedded runtime environments (Linux, bare metal, RTOS)
- International travel to support ESI system installations (<10%)
- Engineering degree with to 5–7 years of related experience, or equivalent combination of education and experience
- Fundamental engineering knowledge (basic physics, calculus, problem solving)

Interested? Please apply below.

[apply now](#)

CAREER OPPORTUNITIES



Do you have what it takes?

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

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

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

[apply now](#)



Become a Certified IPC Master Instructor

We are growing! EPTAC, a leading provider in the electronics training industry is looking for some great people to join our team. If you love teaching, choosing the classes and times you want to work, and being your own boss, this may be the career for you. We are looking for instructors who have a passion for working with people to develop skills and knowledge. If you have a background in electronics manufacturing and an enthusiasm for education, drop us a line or send us your resume. We would love to chat with you. Opportunities available across U.S. and Canada, especially in our growing markets of California, Florida and New England. Some travel involved. IPC-7711/7721 or IPC-A-620 CIT certification a big plus.

[apply now](#)



Application/Sales Engineers

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

Service Technicians

Positions available for Eastern and Western United States. Service technicians will support our rapidly expanding installed base of Miva Technologie's DLP imaging systems and other systems sold by the company.

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

[apply now](#)



Outside Sales/ Key Account Managers

NCAB Group USA is adding to our existing outside sales team in various U.S. locations:

- Ontario, California
- Itasca, Illinois
- Irving, Texas
- Vancouver, Washington

This is a sales position that requires the ability to convert those cold calls into high-value customer meetings. What we are looking for:

- A “hunter” mentality
- The ability to create solid customer relationships
- A desire to excel and not settle for mediocrity
- 5+ years of experience in the PCB or semiconductor industry
- An excellent ability to present a product and do the “deep dive” during customer visits by asking open ended questions and identifying customer pain points
- The energy to move from prospecting to cold calls to getting the win
- Knowledge of “SPIN” selling
- A college degree
- Willingness to travel, domestically and globally
- U.S. citizens with a valid U.S. passport

Interested? Send your resume.

[apply now](#)

Visit us at www.NCABGroup.com



PCB Process Planner

Accurate Circuit Engineering (ACE) is an ISO 9001:2000 certified manufacturer of high-quality PCB prototypes and low-volume production for companies who demand the highest quality in the shortest time possible. ACE is seeking a skilled individual to join our team as a PCB process planner.

Responsibilities will include:

- Planning job travelers based on job release, customer purchasing order, drawings and data files and file upon completion
- Contacting customer for any discrepancies found in data during planning and CAM stage
- Consulting with director of engineering regarding technical difficulties raised by particular jobs
- Informing production manager of special material requirements and quick-turn scheduling
- Generating job material requirement slip and verify with shear clerk materials availability
- Maintaining and updating customer revisions of specifications, drawings, etc.
- Acting as point of contact for customer technical inquiries

Candidate should have knowledge of PCB specifications and fabrication techniques. They should also possess good communication and interpersonal skills for interfacing with customers. Math and technical skills are a must as well as the ability to use office equipment including computers, printers, scanners, etc.

This position requires 3 years of experience in PCB planning and a high school level or higher education.

[apply now](#)



TECHNICA, U.S.A.

Fulfilling Manufacturing Needs
Throughout the Electronics Industry

Southern California Territory Sales Engineer

Technica, USA, a Western regional manufacturer's representative/distributor, has an open sales position for our Southern California territory. The position will be responsible for selling and servicing our entire product line within the specified territory to the PCB manufacturing industry.

This position requires a highly self-motivated, hands on, confident individual of the highest integrity.

Required Skills:

- BA/BS degree-desired, in a technical area is preferred
- Two years of outside/inside sales or manufacturing experience in the PCB manufacturing environment is desired
- Self-motivated self-starter with the ability to initiate and drive business with little supervision
- Independent worker with a strong commitment to customer satisfaction
- Understanding of consumable sales process
- Ability to organize activities and handle multiple projects simultaneously with effective and timely follow-up
- Ability to solve problems and make decisions for which there are no precedents or guidelines and be resourceful in nature
- Positive attitude while operating under pressure and be an independent problem-solver
- Computer skills in Windows, Outlook, Excel, Word and PowerPoint
- Must have a valid driver's license with good driving record

Please send resume.

[apply now](#)



TECHNICA, U.S.A.

Fulfilling Manufacturing Needs
Throughout the Electronics Industry

Western Regional Equipment Service Technician

Technica, USA, a Western regional manufacturer's representative/distributor has an opening for an equipment service technician covering the Western USA, including but not limited to, California, Oregon, Washington, Utah, Colorado, and Arizona. The position will be responsible for servicing our PCB fabrication equipment product line, including installation, troubleshooting, repair service, rebuild service, etc. This position requires a highly self-motivated, hands on, confident individual of the highest integrity.

Key responsibilities are to install and service equipment, conduct equipment audit, and provide technical service when appropriate to solve problems.

Required Skills:

- 2+ years of experience in a PCB manufacturing environment or similar
- Willingness to travel
- Positive "whatever it takes" attitude while operating under pressure
- Self-motivated self-starter with the ability to initiate action plans
- Ability to work independently with a strong commitment to customer satisfaction
- Excellent communication and interpersonal skills
- Strong ability to use all resources available to find solutions
- Computer skills with ability to write detailed service and equipment reports in Word
- Understanding of electrical schematics
- Able to work in and around equipment, chemical, and environmental conditions within a PCB manufacturing facility

Please send resume.

[apply now](#)

CAREER OPPORTUNITIES



IPC Master Instructor

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

For more information, click below.

[apply now](#)



Field Service Technician

Chemcut is looking for a field service technician who is willing to travel worldwide. Previous experience with industrial machine controls (including PLC systems), mechanical components such as conveyors and pumps, plastic piping and fabrication are desirable.

To learn more about Chemcut and apply for this position, please apply to Mike Burke below, or call 814-272-2800.

[apply now](#)



Experienced PCB Sales Professional

With more than 30 years of experience, Prototron Circuits is an industry leader in the fabrication of high-technology, quick-turn printed circuits boards. Prototron of Redmond, Washington, and Tucson, Arizona are looking for an experienced sales professional to handle their upper Midwest Region. This is a direct position replacing the current salesperson who is retiring after spending ten years with the company establishing this territory.

The right person will be responsible for all sales efforts in this territory including prospecting, lead generation, acquiring new customers, retention, and growth of current customers.

This is an excellent opportunity for the right candidate. Very competitive compensation and benefits package available.

For more information, please contact Russ Adams at 425-823-7000, or email your resume.

[apply now](#)

Process Engineer (Redmond, Washington)

With more than 30 years of experience, Prototron Circuits is an industry leader in the fabrication of high-technology, quick-turn printed circuits boards. We are looking for an experienced PCB process engineer to join the team in our Redmond, Washington facility. Our current customer base is made up of forward-thinking companies that are making products that will change the world, and we need the right person to help us make a difference and bring these products to life. If you are passionate about technology and the future and believe you have the skills to fulfill this position, please contact Kirk Williams at 425-823-7000 or email your resume.

[apply now](#)

CAREER OPPORTUNITIES



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

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

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

[more details](#)



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

SALES ACCOUNT MANAGER

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

Duties include:

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

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

[apply now](#)

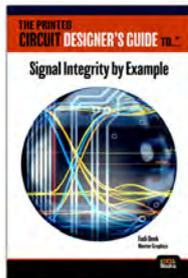
TOP TEN



Recent Highlights from PCBDesign007

1 I-Connect007 Launches 'The Printed Circuit Designer's Guide to... Signal Integrity by Example' Micro eBook

The latest title in this new line of eBooks, *The Printed Circuit Designer's Guide to... Signal Integrity by Example*, is authored by Fadi Deek of Mentor, a Siemens business. This free micro eBook details the importance of eliminating signal integrity challenges. The chapters explore four possible signal integrity problems using an understanding of essential signal integrity principles.



2 Tim's Takaways: It Really Wasn't My Fault

I once received verbal instructions from an engineer who directed me to make a certain change. I didn't think anything of it. Many months later, this same engineer told me that there were troubles with the board and all its successive versions because of the change that I had made. He ended up making it right in the end. But in hindsight, what could I have done to save myself a couple of months of suspense and worry?



3 Altium Expands Cloud-Based Offerings with Acquisition of Upverter

Electronic design software company Altium Limited has acquired Upverter, Inc., the developer of the world's first fully-cloud, fully collaborative electronics design system. Based in Toronto, Canada Upverter's entire team of engineers, including its CEO and co-founder Zak Homuth, will join Altium. This transaction will augment Altium's cloud-based competencies and drive further differentiation and growth for Altium in the market for next-generation electronic CAD software.

4 Enough Blame to Go Around

The idea for this article began a few months ago when *The PCB Design Magazine* conducted a reader survey regarding the topic "Whose Fault is that Bad Board?" After some thought, I submitted my answers. After all, I must have some kind of input after over 25 years of PCB design. But still, whose fault is that bad board? OK, I know what you're thinking: Don't go there. We designers make mistakes too.



5 Orange Co. Designers Council Meeting Focuses on Embedded Passives

On July 19, the Orange County Chapter of the IPC Designer's Council hosted a lunch-and-learn meeting in Tustin to discuss the benefits of embedded passives. Approximately 40 people were in attendance, which was lighter turnout than usual, most likely because of summer vacations. But the crowd was an enthusiastic one, as usual.

6 Stephen V. Chavez Talks Mil/Aero PCB Design

"Steph," as he likes to be called, is also the vice president of his local IPC Designers Council chapter in Phoenix, a CID instructor with EPTAC, and a Designers Council Executive Board member at large.



And each year at IPC APEX EXPO, he's a regular speaker during the Design Forum event. So, for our mil/aero issue, I wanted to get Steph's insight into PCB design at a leading defense contractor.

7 ACDi Hires PCB Design Engineer Priya Sherke

Priya Sherke has spent the past five years with progressive growth at Sanstar Microsystems in Nagpur, India where she was responsible for PCB design, switching mode power supply design, security systems design and technical support for process and field instruments. She is also skilled in testing, and analysis of SMPS, as well as project co-ordination. She holds a bachelor's degree in electronics and telecommunications from Nagpur University and earned a diploma in software testing from SEED Infotech.



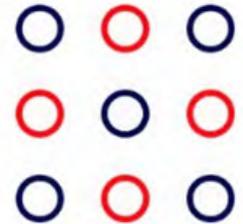
8 Whose Fault is That Bad Board?

Not long ago, I answered Editor Andy Shaughnessy's "Whose Fault is That Bad Board?" survey. When I answered the first question ("If a board fails in the field, whose fault is it, typically?"), I was disappointed that he used radio buttons instead of check boxes. I did not want to blame only the designer for every bad board in the world.



9 Nine Dot Connects Webinar: What Qualifies a Circuit as High Speed?

Technologists at Nine Dot Connects have been asked the following question many times: "What's the frequency in which a design is considered to be high speed?" Would it surprise you to know that a 10 MHz design which could be wire wrapped or placed on a protoboard could constitute high speed? The fact is, frequency is only a part of the story.



10 Beyond Design: Transmission Line Losses

As digital systems evolve and demand for new technology pushes the envelope for smaller and faster systems, transmission line losses, previously considered to be negligible, are becoming a primary design concern. Pragmatic effects such as frequency-dependent losses come into play at clock frequencies above 1 GHz and are of particular concern for fast rise time signals, with long trace lengths, such as multigigabit serial links.



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

Events



For IPC Calendar of Events,
[click here](#).

For the SMTA Calendar of Events,
[click here](#).

For a complete listing, check out
The PCB Design Magazine's
[event calendar](#).

[24th FED Conference](#)

September 15–16, 2017
Bonn, Germany

[SMTA International 2017 Conference and Exhibition](#)

(IPC Fall Committee meetings held in conjunction with SMTAI)
September 17–21, 2017
Rosemont, Illinois, USA

[IPC Fall Standards Development Meetings](#)

September 16–21, 2017
Rosemont, Illinois, USA

[electronicAsia](#)

October 13–16, 2017
Hong Kong

[IPC Flexible Circuits: HDI Forum](#)

October 17–19, 2017
Minneapolis, Minnesota, USA

[TPCA Show 2017](#)

October 25–27, 2017
Taipei, Taiwan

[productronica 2017](#)

(IPC Committee meetings held in conjunction with productronica)
November 14–17, 2017
Munich, Germany

[HKPCA/IPC International Printed Circuit & South China Fair](#)

December 6–8, 2017
Shenzhen, China

[47th NEPCON JAPAN](#)

January 17–19, 2018
Tokyo Big Sight, Japan

[DesignCon 2017](#)

January 30–February 1, 2018
Santa Clara, California, USA

[EIPC 2018 Winter Conference](#)

February 1–2, 2018
Lyon, France

[IPC APEX EXPO 2018 Conference and Exhibition](#)

February 27–March 1, 2018
San Diego, California, USA

[KPCA Show 2018](#)

April 24–26, 2018
Kintex, South Korea

[Medical Electronics Symposium 2018](#)

May 16–18, 2018
Dallas, Texas, USA



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

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

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

EDITORIAL:

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

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

MAGAZINE PRODUCTION CREW:

PRODUCTION MANAGER: **SHELLY STEIN**

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

INNOVATIVE TECHNOLOGY: **BRYSON MATTIES**

COVER: **SHELLY STEIN**

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



The PCB Design Magazine® is published by BR Publishing, Inc., PO Box 50, Seaside, OR 97138
©2017 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

September 2017, Volume 6, Number 9 • The PCB Design Magazine© is published monthly by BR Publishing, Inc.

ADVERTISER INDEX

Accurate Circuit Engineering.....	31	IPC.....	51
Altium.....	9	IPC-2581 Consortium.....	29
American Standard Circuits.....	17	Isola.....	5
Cadence.....	45	Mentor, a Siemens Business.....	25
Candor Industries.....	11	Miraco.....	47
Downstream Technologies....	21, 59	NCAB.....	23
Eagle Electronics.....	43	The PCB List.....	37, 61
Eltek.....	7	Prototron Circuits.....	57
EMA Design Automation.....	13	Rogers Corporation.....	33
General Circuits.....	35	Sunstone Circuits.....	27
I-Connect007 eBooks.....	2, 3	US Circuit.....	53
In-Circuit Design Pty Ltd.....	55	Ventec International Group.....	39

I-Connect007

GOOD FOR THE INDUSTRY



FREE SUBSCRIPTION



myconnect007.com

EDITORIAL CONTACT

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



mediakit.iconnect007.com

SALES CONTACT

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



www.iconnect007.com