FEBRUARY 2020

I-Connect007

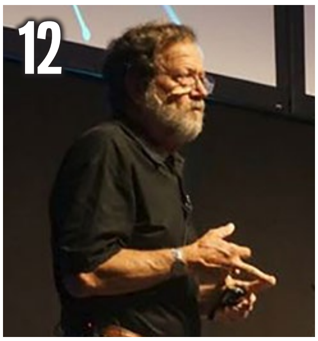
DESIGN007 MAGAZINE

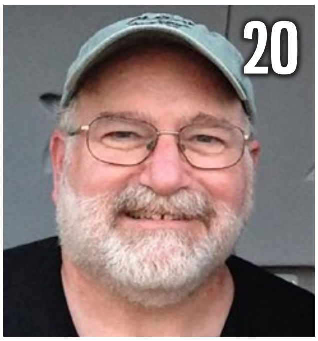
FLEX007
SEE INSIDE!

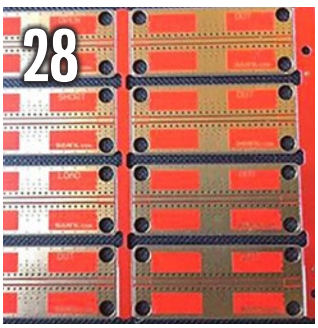
DESIGN007 MAGAZINE

EMI / Power Distribution

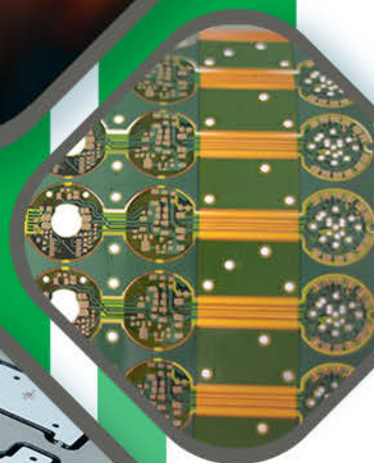
At every trade show and conference lately, designers and design engineers mention issues that they're facing related to EMI and power distribution networks. It's one problem that doesn't seem to be going away anytime soon. This month, we asked our expert contributors to share their thoughts on fighting and even precluding EMI through proper PDN design techniques and more.

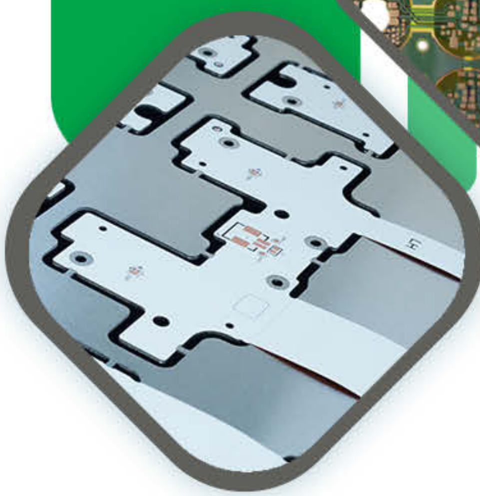






DRIVING TECHNOLOGY TO NEW EXCELLENCE





Multilayer, HDI, Bendable,
Flex/ Rigid Flex,
High Frequency PCBs and more

ABOUT CML

CML are the leaders for printed circuit board (PCB) manufacturing and sourcing solutions. CML's mission "We care about reliable connections," apart from the reliable connections of our PCBs, paramount importance is also placed on quality and the relationship between CML, our customers and suppliers.

CML provides a wide product portfolio and a range of services from consultancy, design, engineering, manufacturing and logistics for all markets - convenient and reliable.

CML has a global network, located across 17 international cities and a local presence with a multilingual and culturally diverse team.

Contact our USA office today

Chris Minard
chris.minard@cml-america.com
+1 650 269 4823

For information on our other offices, please visit our website:
http://www.cml-globalsolutions.com

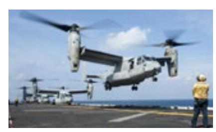
CML
More than a manufacturer

DESIGN007 MAGAZINE

COLUMNS:

SHORTS:

HIGHLIGHTS:



DEPARTMENTS:

IT-170GRA1

IT-170GRA1 Value Proposition

- Halogen Free
- Best in class performance in its space
- Best in class CAF performance
- Very attractive price to performance ratio
- Pass MRT-6 requirements
  - -CAF > 1000 hours
  - -0.65mm pitch equivalent, no thermal issue
- Available with RTF and 2μ copper which provides additional performance on a very solid platform
- Df = 0.0075 @10GHz

Intel Purley Mid-Loss Solution – IT-170GRA1

Futures

Halogen free mid loss, Df < 0.01
Tg 180ºC by DSC, low CTE

Applications

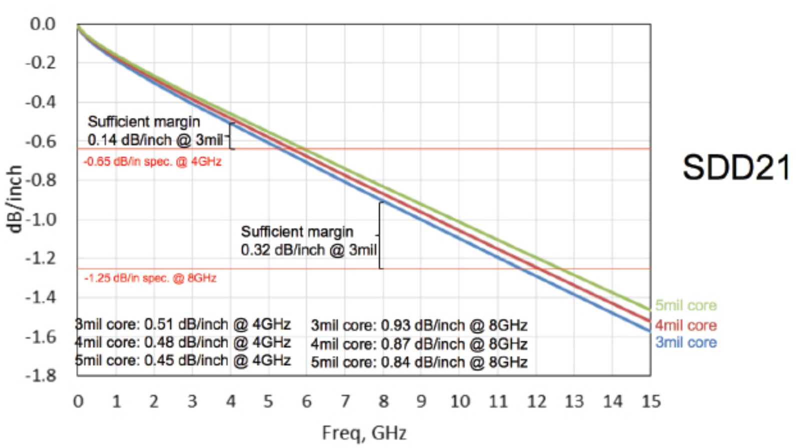
Server/Storage/Switch (10G, 40G)
Backplane, Telecom
Base station

Feature *Purley Platform Mid-Loss Solution – Hi Tg*

General Properties

Items	Methods	IT-170GRA1
Tg (℃)	DSC	180
T-288 (w/ 1 Oz Cu, min)	TMA	60+
Td-5%(℃)	TGA 5% loss	380
CTE (%), 50-260℃	TMA	2.4
Peel strength (lb/inch)	1 oz	7.0
Water absorption	D-24/23	0.1
Dk: 2-10 GHz	Bereskin	3.96 – 3.99
Df: 2-10 GHz	Bereskin	0.0073 - 0.0075

IT-170GRA1 Insertion Loss

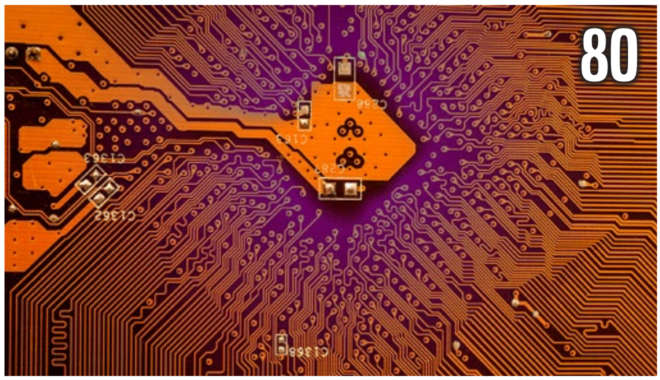


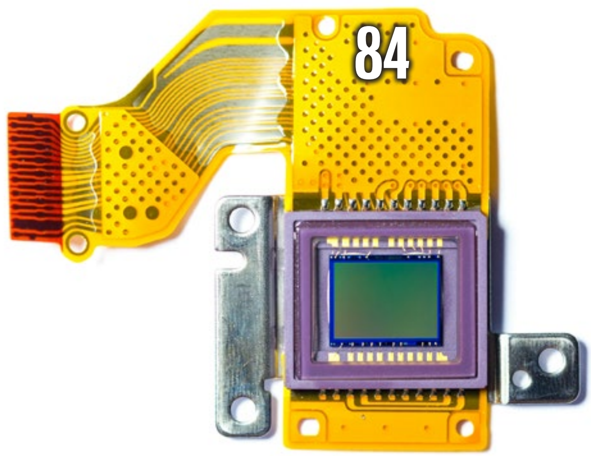
ITEQ

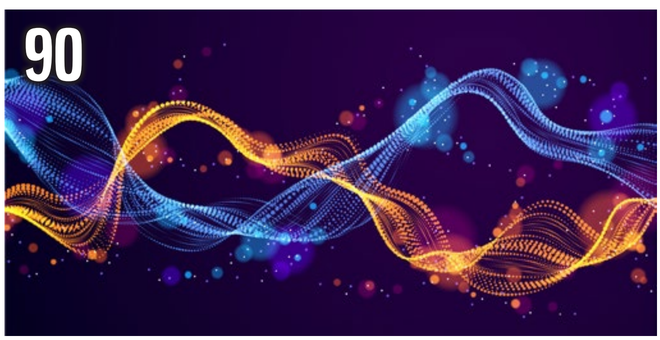
FLEX007

Focusing on Flex

More and more OEMs are also moving into complex multi-board rigid-flex circuits, often for reasons related to space-saving and reliability. This technology can save money and real estate, but it brings with it a variety of caveats.







DesignCon and IPC APEX EXPO Celebrate Milestone Anniversaries

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

I just returned home from attending DesignCon and IPC APEX EXPO. Both of these shows marked milestone anniversaries: DesignCon celebrated its 25th show, and IPC APEX EXPO its 20th.

At DesignCon, the show's mascot, Chiphead, greeted us on the first day, welcoming us to attend a variety of keynotes, panels, and sessions. I spoke with designers, engineers, and EDA tool vendors, and almost all of them said their companies are expanding. The mood was upbeat.

I enjoyed Eric Bogatin's opening-day session "So You Think You Understand What a TDR Measures" in the Chiphead Theater. Eric is a master of audience participation; he constantly worked the crowd, posing questions in different ways to help drive home a point. And if you have a correct answer, you won a candy bar. Eric drew a good crowd.

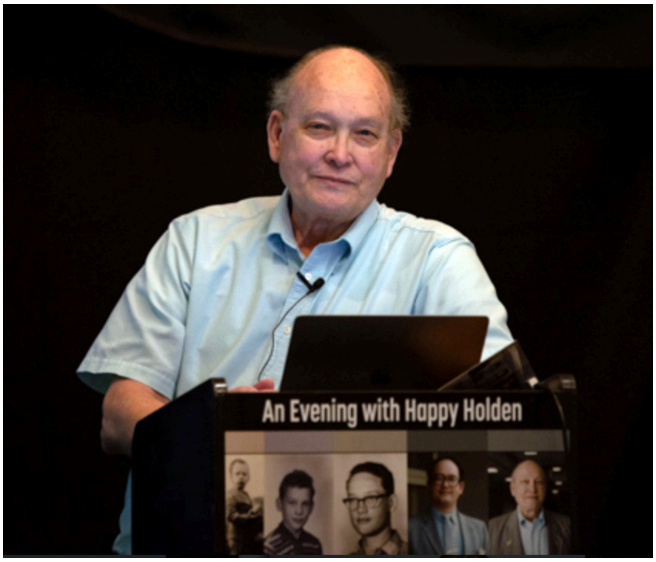
Some analysts were worried that the coronavirus scare would lead to a drop in attendance, but DesignCon seemed as busy as usual.

IPC APEX EXPO 2020 started unofficially on Super Bowl Sunday as I joined Editors Kelly Dack and Pete Starkey for a few hours of sailing around San Diego. Whenever we go sailing and no one drowns, we call it a win. It seems like we just went sailing together a few years ago, but it had been eight years. Time flies!

The show floor was busy at DesignCon 2020.

Monday night, we held a "Night of Happyness" in the Horton Grand Hotel, with Happy Holden sharing funny stories from his 50 years in the business. Guests included a who's who of top technologists and authors in the industry, and many of them had worked with Happy at one time or another.

Editors Kelly Dack, Andy Shaughnessy, and Pete Starkey sailing in San Diego. Kelly Dack entertained everyone during the cruise.



Happy Holden shared anecdotes from his 50 years in the industry.

As the expo opened on Tuesday, the keyword was data—design data, fabrication data, assembly data, and methods for making this data as accessible as possible. IPC-CFX (Connected Factory Exchange) was a big topic of conversation on the show floor.

High schoolers from IPC's STEM Student Outreach Program were everywhere on Thursday, getting a taste of the high-tech world. It was nice to see fresh faces mixing with the grizzled veteran attendees. We certainly need more of them.

We also congratulated our good friend Charlie Capers of Trilogy Circuits, whose company was just acquired by Zentech. And I'd like to recognize our contributors who received honors at the IPC Wednesday Committee Awards Luncheon, including Mike Creeden, Luke Hausherr, Kelly Dack, Steph Chavez, Cherie Litson, Michael Ford, Mike Carano, and Karen McConnell. They volunteer a lot of their free time, so let's give them a big hand.

Speaking of trade shows, we have a scheduling snafu next year, with DesignCon and IPC APEX EXPO taking place the week of January 26–28, 2021. This happened in 2019 as well, and it was bad for both shows. Is it too late for one of these organizations to make a scheduling change?

This Month

At every trade show and conference, designers and design engineers mention issues that they're facing related to EMI and power distribution networks. In this edition, we asked our expert contributors to share their thoughts on fighting and even precluding EMI through proper PDN design techniques and more.

We start with an interview with Lee Ritchey, who explains some of the causes and solutions to EMI issues, and why misinformation is not helping the problem. Next, Eric Bogatin outlines why EMI is such a tricky problem, and why it's likely to continue to be a problem for some time. Istvan Novak discusses how to design PCB test fixtures for improved power integrity. Barry Olney focuses on PDN impedance and its effect on EMI. Tim Haag discusses some of the resources available to designers who are facing EMI challenges.

And freelance designer Alexander Löwer explains the variety of hurdles related to beating EMI, including non-technical aspects, such as the alphabet soup list of national and international regulations. We also bring you columns by our regular contributors Bob Tise and Matt Stevenson, Steph Chavez, Alistair Little, and John Coonrod.

If these events are any barometer, 2020 is going to be a good year to be in this industry. **DESIGN007**

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

IPC Wednesday Committee Awards Luncheon recipients.

Avoiding EMI Problems With Lee Ritchey

Feature Interview by the I-Connect007 Editorial Team

The I-Connect007 team met with design expert Lee Ritchey to pick his brain on EMI (Electromagnetic Interference) problems and what can be done to minimize them. Lee explains the issues are almost always tied to power delivery, as well as the abundant amount of misinformation surrounding this topic.

Barry Matties: Thank you for speaking with us, Lee.

Andy Shaughnessy: We want to discuss finer pitches and features and EMI and EMC because we have talked to people having EMI issues, particularly in Germany on the automotive side. You mentioned that you weren't seeing many EMI problems due to different ways of doing busses and other tricks that helped relieve those issues.

Lee Ritchey: I believe that might have been happening because they have big distributed networks in cars, and if you don't do it right, you turn the whole car into an antenna. To Andy's point, I used to do six to 12 EMI problems a year, but I did only one last year. The source of most EMI is a ripple in the power supply, which is caused by trying to drive these wide data busses for DDR, VME, etc., and the power supply hasn't been designed correctly. When we replace a parallel bus with a differential pair, several things happen all at once. The incidence of wide busses switching from zero to


JOE GRAND
AKA KINGPIN,
ELITE HACKER
TO BE ABLE TO
QUESTION
AND THINK
ON YOUR OWN
REALLY IS THE HACKER
MINDSET
Watch Joe's Story on
Altium's YouTube Channel
Altium.

one goes away, and that's the villain in almost all cases.

Most of what I work on has nothing but differential pairs in it. Big networking products that you would think would be a problem are flying because of inherently quiet differential signaling. To give you an example, everyone has differential twisted pairs in your house. The energy on it is an EMI band, but we don't have any issues, even with unshielded wires. The reason is that the fields cancel each other on the two wires. That does not happen when you have a single-ended circuit.

People having problems have something in their product, such as a parallel bus, with the power delivery system designed incorrectly. That used to be how we made easy money. I'd get a call and say, "Did you follow the app notes?" If the answer was, "Yes," then I knew what their problem was. However, being a good consultant, I won't tell you over the phone because then you think it's free advice. You fly me out at great expense, we fix it, and you're happy. It's hard to get this concept through, but if you follow the app notes, you will likely have an EMI problem. If your problems are typical applications, then you will always have an EMI problem. Almost all of them say to use 0.1-uf and 0.01-uF capacitors in all sorts of different ways. Neither of those capacitors is able to deal with EMI. That's my giveaway. I know what to fix. You don't have the right capacitors in your power system.

Lee Ritchey

Shaughnessy: You're saying if you design the board correctly from the start with an eye toward this, you can design out the problems beforehand.

Ritchey: My clients have not had an EMI problem for 20 years. Once they use the information they get from my class, it isn't a hard problem to solve; it's that the advice given in the application notes is all wrong. Big surprise there, huh?

Shaughnessy: It's a matter of getting the information out. You were saying that you're going to teach a class next June on EMI in Germany.

Ritchey: I have a three-day seminar, and EMI is one of the topics. I also want to add that there are two or three people who have the title of EMI guru, and they have a bunch of rules of thumb, but many of those are wrong. If I know that their advice had been followed, I also know what to fix. There are only a couple of consultants who are well-trained.

Happy Holden: There are a lot more seminars now too. Maybe there's better education and awareness.

Ritchey: I would like to say that, but based on the students in my classes, I don't think that's happening.

Matties: You would have a firsthand point of view to recognize the trend there. When you talk about the students in your class, what do you see?

Ritchey: Universally, the core reason that we have EMI problems is that engineers don't know how to design their power delivery system correctly, and that was true for almost everyone in my class.

Matties: Does it always come down to power delivery?

Ritchey: I had only one failure that was not power delivery out of the 200–300 designs I've fixed. That's the source of the noise.

Matties: In terms of learning how to properly design it, power delivery is extremely complicated to the point where it's elusive to people.

Ritchey: It's not that hard. After a half-day of my two-day seminar on power delivery, when students leave that class, they can immediately get rid of their EMI problems. The problem is we have more misinformation on how to do this than we have good information.

Matties: That's a big problem around the world today, generally.

Ritchey: Yes. When Rick Hartley set out to do this 25 years ago, our task was to get rid of the misinformation. Clearly, we have not succeeded.

Matties: Is there a generational shift? Is this something that the older generation understands better than new designers, or are they relying heavily on the EDA tools to remedy the problem?

Ritchey: Tools don't do any engineering. I have some ways that I like to use when people think tools solve problems. A long time ago, I was with Amdahl when we made the first machine that would compete with IBM. It was Silicon Valley, and we got a huge amount of press. There was a press conference and a Q&A, and a reporter said, "Mr. Amdahl, aren't you afraid you're making computers so powerful that they'll replace thinking?" He responded, "What you don't understand is that what we have here is an exceedingly fast idiot." You must have an engineer. Tools don't solve the problem.

Matties: The only thing that solves the problem is education.

Ritchey: I agree. I still work with the university where I graduated, and I gave the professors there all this information because they don't have it.

Matties: Let's assume that they build a board with an EMI issue. What's the impact of that?

Ritchey: The board can do its job, but if you don't pass the EMI standards for either the CE in Europe or the FCC in the U.S., you can't sell the product.

Matties: In some cases, you don't have to reach that standard, so your board may work, but product may not work very well. Ultimately, it may not perform as well, but if you're trying to reach the standard, you have to solve the issue.

Ritchey: Correct. Years ago, when I was at 3COM, they ignored that and shipped units into Europe that failed. They were caught and fined $10,000 for units shipped. We had shipped about 200 units. Thousands of products are shipped out of Asia that don't comply, and they have stickers on them that say they do. In Hong Kong, there's a place called Computer Market where one stall sold rolls of these stickers. You don't need to even put your product through the test.

Holden: A lot of times, people put it inside a metal box or a Faraday cage.

Ritchey: That's how you contain EMI.

Shaughnessy: It's not pretty, but it works.

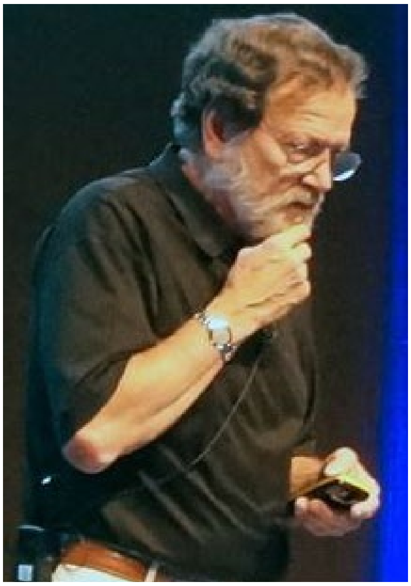
The problem is I see lots of products that fail EMI tests.

Ritchey: The problem is I see lots of products that fail EMI tests. If you have an EMI problem, you need a transmitter. That means you have a source of RF energy and antenna. People build these nifty Faraday cages, and then they pierce them with wires that go off to a mouse, a power supply, or something peripheral, and the connection is the antenna. I have a number of strategies I use to keep noise from getting on those as far as the peripheral.

Holden: On a power supply design network, is a power supply impedance and the need for lower impedance part of the difficulty with the noise?

Ritchey: Yes. The problem is that the places where the noise is coming from are above the frequency where capacitors are useful.

Holden: That's when you get to distributed capacitance with materials and printed circuit fabrication. Of course, those materials have to be designed in. A fabricator can't switch materials because they realize that a 5-mil dielectric between power and ground is not going to be much-distributed capacitance.



Ritchey: Exactly. You're suggesting that fabricators are permitted to change the stackup. If a fabricator did that to me, they would never have my business again.

Holden: They could always recommend or point it out to someone.

Ritchey: No one does my stackups but me.

Matties: How would they know?

Ritchey: They don't know. They can't do it for you. If you look back a couple of decades, you're only worried about impedance and not much else.

Holden: One of the things I did as an engineering manager in fabrication was to hire the best RF engineers we could find to help us with problems.

Ritchey: That's not where the engineering decisions should be made.

Holden: But it helped us discover that they're going to have a problem with prototypes.

Matties: That was a captive facility, though. You have resources at different levels in a captive facility than an independent would.

Holden: All mistakes come back to the company; it's not transferred across legal entities. We could help a new designer out of college by pairing them with somebody that's been designing RF for 35 years and make a great production engineer.

Ritchey: Sure, but that won't work anymore. Why I say that is because we have four things that matter in a stackup, of which only one used to matter. The nature of the glass weave now matters with these high-speed differential pairs, the plane capacitance, and the crosstalk impedance. You can't expect the fabricator to do anything except the impedance part of that; I would not expect that personally. That's not what a fabricator's skills are.

About the history of impedance, I was around Silicon Valley when it started to matter. First of all, my career was in ECL from day one, and I always worried about impedance. When the engineers started to worry about that, they didn't know how to deal with that. They would tell the fabricator, "You have to figure out how to make this design 50 ohms," which is probably what happened to you, Happy. They scrambled around to find a way to get 50 ohms. I probably got a couple of hundred calls from around the world from fab shops, asking, "How do I do this?" They tried to do it, and hats off if they did. It was not their responsibility, but they tried to do it. The accuracy with which I see it done is all over the map. Probably half the boards that come in that are supposed to be 50 ohms aren't even close.

Holden: That's one of the reasons we spent a lot of money characterizing all of our standard

materials much better than anybody else did. We would sample lots of materials for an entire year because to create plus or minus three sigma on the dielectric constant and dissipation factor out to 25 gigahertz, even 35 years ago.

Ritchey: You were lucky; you had a pretty special environment.

Holden: We had standard materials, and we had to show the engineers that the dielectric constant is not going to be 4.4. Maybe once in its life, it's going to change lot to lot. Here are the numbers, plus or minus three sigma, not over frequency, but also over temperature and over humidity as a design file book. These are the standard materials they had to use. They couldn't go out and pick any old material they wanted unless it was thoroughly characterized.

Ritchey: You were lucky enough to work with a good engineering company.

Holden: Yes. Making test equipment, it has to be 10 times better than what it's measuring. I was trained by the best because the test equipment had to be so sensitive, especially the way to deal with noise was sometimes very unique.

Ritchey: I was doing that as a college kid at Tektronix. At that time, Tek and HP were the places to work, and it was good, engineering-wise.

Holden: All of my classmates went to work for Tek because I came from Oregon. I was the only traitor that went south to Hewlett-Packard.

Ritchey: Did you get run out of town?

Holden: They kind of disowned me.

Shaughnessy: As Dan Beeker says, he doesn't even run signal integrity analysis. He said, "Design the board right." Is it that simple?

Ritchey: I agree with that.

Holden: With the semiconductor roadmap, they're shipping 7-nanometer geometries, and they're prototyping 4- and 5-nanometer geometries; these small transistors turn on and off fast. That rise and fall time is what you have to deal with, and because of the shrinking voltages that power these things, they've had to increase the number of ground pins.

Ritchey: Both ground and power.

Holden: The power ground is increasing enormously, and you have to distribute it because a critical length is based on that rise and fall time. They don't want to have to pay for transmission lines, so they use up the entire critical length, and the only way to do that as your transistor shrinks is to shrink your pitch—especially when you have more pins now to put on the package. How they allocate their pins can be an enormous headache.

Once you attach to it, you have an EMI or signal integrity problem, even if you try to do everything right. They've made the job difficult. Other companies understand that somebody who has to use the IC is going to have to wire it on a circuit board, and if they have too many problems to fight, they're not going to buy that component anymore. They'll check to see if somebody else has a component like it or do it the old way and use three devices instead of the new single device. It's all physics.

Ritchey: You're going down the right path. In the latest design John Zasio and I did, we had an IC where the operating voltage was 0.9 volts, and the current was 160 amps. Getting that current into that product without having excessive DC voltage drops turned out to be the hardest job we had to do. One-hundred-sixty amps may not mean anything to you, but let me put it into perspective. Your car requires 60 amps to start it. How big is the cable going from your battery? For what John and I do, our biggest problem is power delivery. It has become the hardest part of the job, getting the high currents into the part. If you don't have enough pins, no matter what you do, you can't make it work.

Holden: If the pitch is too small, the anti-pads on the power ground layers take away all the copper that's required. You have noise issues and voltage drop issues underneath the BGA that will create additional headaches and problems.

Ritchey: This part had 2,000 pins. The planes look a little bit like a screen, there are so many holes. This is still power delivery, but that's not where EMI comes from. The product won't work if you don't get that part right.

Matties: It's all connected. Is there anything else that we should understand about this topic before we move forward?

Ritchey: I don't think it's a complicated problem, personally. The thing is that there's so much misinformation out there that the advice people are given creates problems. Having been doing this for a while, I know what the problem is and that it's simple to solve, but the issue is wading through the misinformation. I would suggest that anyone who is starting out in this area read the book *Principles of Power Integrity for PDN Design: Robust and Cost-Effective Designs for High-Speed Digital Products* written by Larry Smith and Eric Bogatin. I would suggest that you tear up every book that claims to give you good advice on EMI (laughs). Universally, they're all wrong.

Matties: Lee, thank you very much.

Holden: We appreciate it.

Ritchey: Thanks. **DESIGN007**

A Smart Surface for Smart Devices

We've heard it for years: 5G is coming. And yet, while high-speed 5G internet has indeed slowly been rolling out in a smattering of countries across the globe, many barriers remain that have prevented widespread adoption.

Researchers from MIT's Computer Science and Artificial Intelligence Laboratory (CSAIL) wondered if people have had things completely backward this whole time. Rather than focusing on the transmitters and receivers, what if they could amplify the signal by adding antennas to an external surface in the environment itself?

That's the idea behind the CSAIL team's new system RFocus, a software-controlled "smart surface" that uses more than 3,000 antennas to maximize the strength of the signal at the receiver. While the system could serve as another form of the WiFi range extender, the researchers say its most valuable use could be in the network-connected homes and factories of the future.

For example, imagine a warehouse with hundreds of sensors for monitoring machines and inventory. MIT Professor Hari Balakrishnan says that systems for that type of scale would normally be prohibitively expensive and/or power-intensive but could be possible with a low-power interconnected system that uses an approach like RFocus.

"The core goal here was to explore whether we can use elements in the environment and arrange them to direct the signal in a way that we can actually control," says Balakrishnan, senior author on a new paper about RFocus that will be presented next month at the USENIX Symposium on Networked Systems Design and Implementation (NSDI) in Santa Clara, California.

RFocus is a two-dimensional surface composed of thousands of antennas that can each either let the signal through or reflect it. The state of the elements is set by a software controller that the team developed with the goal of maximizing the signal strength at a receiver.

(Source: MIT)

Eric Bogatin Looks at EMI Root Causes and Solutions

Feature Interview by the I-Connect007 Editorial Team

The I-Connect007 team recently spoke with design instructor and author Eric Bogatin about the EMI challenges facing PCB designers today. Eric is a "signal integrity evangelist" with Teledyne LeCroy, as well as an adjunct professor at the University of Colorado Boulder, and technical editor of the *Signal Integrity Journal*. In this interview, Eric explains why EMI is so prevalent and what designers and design engineers can do to avoid EMI from the start.

Andy Shaughnessy: Eric, we wanted to get your thoughts on EMI, a topic that you have spoken and written about quite a bit. What can designers do to avoid EMI?

Eric Bogatin: Here's a little perspective on why I think EMI is still a big issue, will always be a big issue, and, depending on the system, will probably become a bigger issue. In FCC certification and testing, in Part 15, there are Class A and Class B. Class B is consumer electronic products, which is a more sensitive, more stringent test for radiated emissions. Just to give you a number, the test specification says that you put your product down in the chamber, go three meters away, and look at the worst-case electric field-radiated emissions. In the roughly 100 megahertz range, they say the worst case is 100 microvolts per meter, which is the maximum allowable. The important number is that this is within a 120-kilohertz bandwidth detector.

Your product has to radiate less than 100 microvolts per meter, 10 feet away within that 120-kilohertz bandwidth. You can ask, "What if I had a radio source sitting there and broadcasting on some power level. How much power is that? What is the maximum power my little radio station can transmit at and still pass the FCC tests?" Do you know what the maximum radiated power is for that little radio station to pass an FCC test?

Shaughnessy: I have no idea.

Bogatin: The answer is 10 nanowatts of power. This corresponds to a radio station broadcasting in 360 degrees within the 120-kilohertz

band. The total power radiated all around from that radio station is no more than 10 nanowatts, and if more than 10 nanowatts, it will fail the FCC test. And you say, "Wait, it doesn't take much radiated power at all to fail an FCC test. You really have to engineer your product so that it doesn't radiate very much."

You really have to engineer your product so that it doesn't radiate very much.

And that's why one of the fixes for passing a test is adding the spread spectrum clock generator. If you have a clock system, you're radiating at the harmonics, so the spread spectrum clocking just dithers that harmonic frequency range across that 120-kilohertz bandwidth. Because it only takes a tiny amount of radiated energy to fail an FCC test, EMI has been and always will be a problem in many systems.

It's a tough problem to fix, and you have to be a clever signal integrity (SI) engineer, power integrity (PI) engineer, and EMI engineer. You need everybody playing together in order to eliminate that much radiated emissions and pass an FCC test.

Shaughnessy: Is EMI mainly caused by an improperly designed power distribution network, or is that just one possible cause?

Bogatin: There's a lot of noise with PI due to three things. One is the power system. What radiates are currents, and it's in the power delivery system where you have the high currents. That's where you have the potential of having very high currents that can radiate. Then, the way those currents turn into radio emissions is related to where their return paths are, and you have components in the power distribution network that are big whopping components; there's an inductor, as well as the ICs that are on the board, and oftentimes, the way the board is laid out for the power delivery part doesn't keep the power currents and the return currents close enough together, so they have a tendency of radiating. The third piece is that part of the power distribution path is the return path for signals.

It's the discontinuities in the power and ground paths that cause any currents flowing into its path to see the discontinuity, to see a voltage drop, and create patch antennas. And it's because the same conductors that the planes used for power delivery are also used for return path for signals. If their discontinuity is there, the signals can radiate, and the power currents can radiate because of the discontinuity. It doesn't mean all sources of radio emissions are from the PI, but if you fix those PI issues, that will go a long way toward helping reduce the radiated emissions.

Shaughnessy: So, it's a system thing.

Bogatin: It's always a system thing. On the other hand, even if you have fixed those issues and have good SI, it can still radiate because, again, it doesn't take very much radiated power to fail an FCC test.

Matties: How would a designer or engineer determine if this is going to be a problem?

Bogatin: There are a few approaches. There are a lot of known root causes for radiated emissions, and there are a number of these design rule checkers. Mentor has one, and I think Altium might have one, but what Mentor does is look for the known problems in the layout that will cause radiated emissions. They look through your Gerber files, and will highlight potential problems, noting, "Here you have a signal over a gap," or "Here you have a split plane, and you have no path for return current." They will identify them and let you know what areas to check. That's one way ahead of time, but it's difficult because systems are so complicated now. It's challenging to put the whole thing into a 3D full-wave solver and then predict where the radiated emissions are. You can look for trends, but knowing if it's going to pass or not is really hard to do.

You look for the known problems and eliminate those known problems. Sometimes, you can look at the near-field radiation, which is one of the things I show in my classes with a board that has good returns and bad returns. And with just a little sniffer probe near the surface, you can see the bad returns radiate a factor of 10 or 20 times more near-field than the region of the layout that has good returns. Just using a near-field probe will give you an idea of how bad things are. It comes down to looking for the root cause and identifying it, and then fixing the problem.

Shaughnessy: I hear some people say that they don't see EMI issues below 50 or 75 kilohertz. Is there a definite cutoff for EMI?

Bogatin: The FCC started at 30 megahertz, I believe, but there are some low-frequency emissions tests that are a lot lower in frequency. I'm not as familiar with these tests. I've heard that there's a specification for very low-frequency magnetic field radiation or near-field radiation in the hundreds of hertz to kilohertz, but again I'm not sure in the specification. However, if you have the same antenna, the same length of wire, the same current, and you just jiggle the frequency, the far-field radiated emissions drop with frequency. Going to a lower frequency means you're going to have a lower far-field electric field.

Dan Feinberg: Eric, is it a combination of frequency and power, or is it just specifically the frequency? In other words, if the power that the signals are being generated at is much lower, would the frequency be less important? Or is it different specifications, different limits?

Bogatin: Yes, I think so. When I talked about power before, it's the radiated energy, the power in the radiated energy, not in the voltage or current.

Feinberg: Exactly.

Bogatin: So, the current doing the radiating is oscillating back and forth, and the FCC requirement says in this frequency there are five or six different frequency ranges. Within each range, there's a set limit to the maximum far-field electric field strength. If you just take that same current and slosh it around the far-field electric field strength, it's going to be proportional to the current multiplied by the frequency.

In the past few years, there's one area in PI and radiated emissions that I think is really exciting. Everybody uses switch-mode power supplies because they're really efficient. But the problem is you're switching lots of currents around, and current switching is a recipe for radiated emissions. That's why switch-mode power supplies are notorious for radiating and failing FCC tests. I think it was Linear Tech who was bought by Analog Devices, and they came out with a line of this switch-mode power supplies. They're called Silent Supplies, and they have done some clever engineering on the inside of these modules to dramatically reduce the opportunity for radiated emissions.

Shaughnessy: What would you say are some of the biggest mistakes that engineers and designers make as far as EMI? Or some things that designers should do to preclude EMI?

Bogatin: This is where Mentor came up with their design rule checker with a list of design rules. Todd Hubing also has a list of dos and don'ts of how to reduce these problems for reducing radiated emissions by how you do the layout. They may be four or five different things. Number one is don't have signals cross gaps in planes, so if you have a split power plane, for example, then you don't want to have a signal use that split as a return. Another one is when you have a multilayer board, and a signal goes from one layer to another layer, you want to have a return via in proximity to the signal via, so that says you want to use ground as your return and drop the ground via between the two ground planes when the signal switches.

With connectors for coax cables, you want to use a 360-degree connector and make sure the connector attaches to the chassis and not to the ground plane on the card. Also, with unshielded twisted pairs, you want to have something called a magnetic—but it's really a ferrite—choke in the differential pair to reduce the common-mode currents on unshielded twisted pair, every RJ45 connector that's shipped on a router or a switch has a built-in ferrite choke to reduce the common currents. Then, if you want to reduce the common currents, that's where you use shielded twisted pairs, and you connect the shield to the chassis.

All of these are well documented in Mentor's list of design rule checkers. I think Zuken also has a design rule checker for EMI. Todd Hubing's website is also a great source that has good information about design rule checking and kind of a checklist of dos and don'ts for ways to reduce radiated emissions.

Holden: And you teach at the University of Colorado Boulder.

Bogatin: I teach a practical PCB design class. It's a mixed class with seniors and graduate students. And everybody in the class will do three circuit boards in the classroom. I've developed a five-week design cycle, and it's not so much about circuit design as it is implementation. I've talked about it at my AltiumLive keynote the last few years. I've talked about some of the things that I do in the class and some things I've learned about teaching these concepts to students, but I think it applies to engineers in general as well.

Holden: You wrote my SI chapter for *The HDI Handbook* back in 2007. In it, you have your basic diagram that says that all noise comes from one of four or five sources. Do you still adhere to the principle that SI and EMI problems are from one or all these five fundamental cases?

Bogatin: Yes. One of the things I say in my class is that the first step in designing a board is getting connectivity correct, and once you have connectivity correct, then it's all about noise; everything is noise. I'm happy that you have a good memory. At the time, I said there were five families of noise, but now I'd broaden it to six families because I've included high-speed serial links (Figure 1).

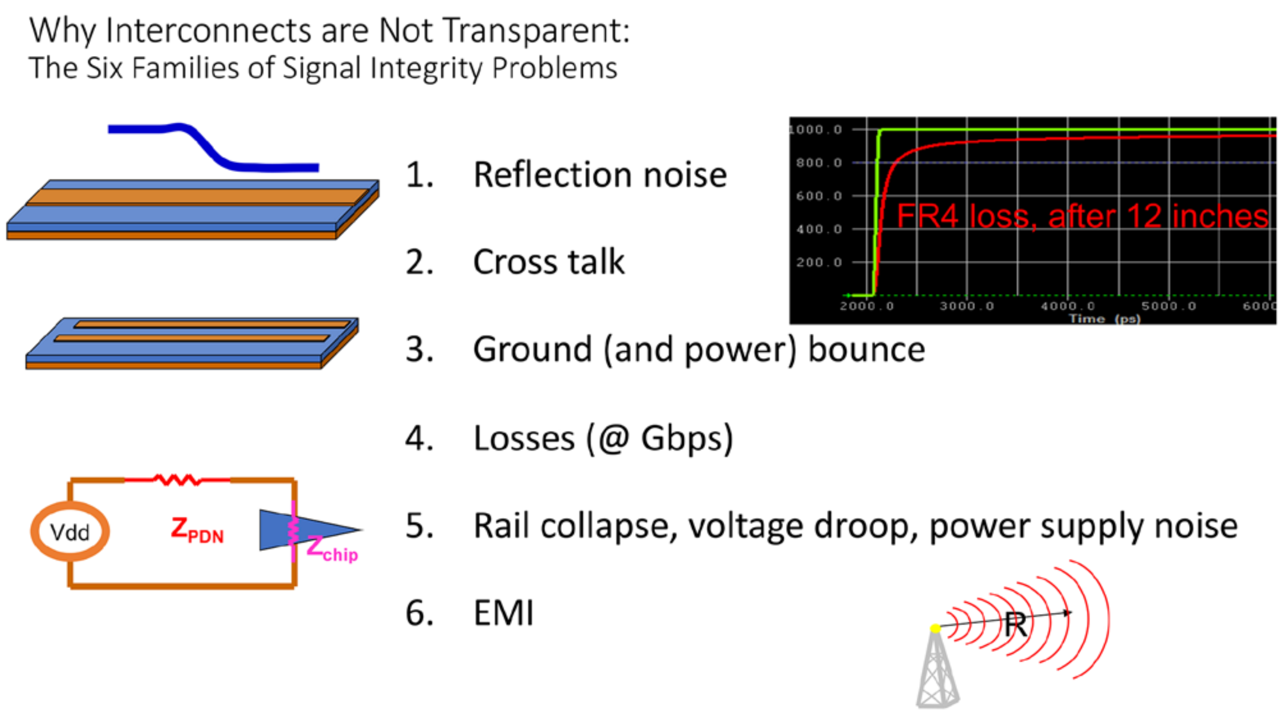


Figure 1: The six families of SI problems. (Source: Eric Bogatin)

Holden: You said you can have the perfect schematic and do all the simulations you want on the schematic. But you have to understand about noise and the physical layout because the physical layout is the thing that generates the noise, not necessarily the schematic. You can't get your arms fully around it until you physically lay it out and build it.

Bogatin: You're right. In my Altium keynote over the last two years, I emphasized that the schematic identifies connectivity, tells you what components you're going to use, and how they're connected together, but the noise lies in the white space. There are also ideal wires in the white space of the schematic, and that's where it's implemented in the layout. The noise is really about the layout. That's where I spent my career as well as that aspect of the electromagnetics of conductors and dielectrics.

Holden: I keep wondering how Maxwell could have figured all of this out so long ago.

Bogatin: Because I've spent a little time in an academic environment, I am constantly asked, "Why is there an SI program in a university? What's new and novel about that? Are you going to discover a fifth Maxwell equation?" It's all just about finding different ways of applying Maxwell in different environments and structures. The challenge, especially with my students, is that you can learn Maxwell's equations and solve the differential equations up to the wazoo with different boundary conditions. However, it still comes down to designing a circuit board and trying to figure out where to route this trace.

Shaughnessy: When you look ahead the next couple of years, especially with these electric vehicles and autonomous vehicles, do you think the EMI problems are going to continue to get worse?

Bogatin: EMI is always going to be an issue with automotive because you have all of those really big transient currents sloshing around there when the motor turns on. I heard one description at a DesignCon keynote that today's cars are data centers on wheels. There is so much electronics in the cars that they're really data centers with computers and multiple microcontrollers, including the autonomous ones, which are moving more and more toward using cameras as sensors. You can have half a dozen or a dozen cameras with HDMI data. The challenges aren't just for EMI but SI and the power delivery as well through these vehicles.

Matties: This has been really great, Eric. We appreciate your time. Thanks so much.

Holden: Thank you, Eric.

Bogatin: Thank you. It's always a pleasure.
DESIGN007

PCB Fixtures for Power Integrity

Quiet Power

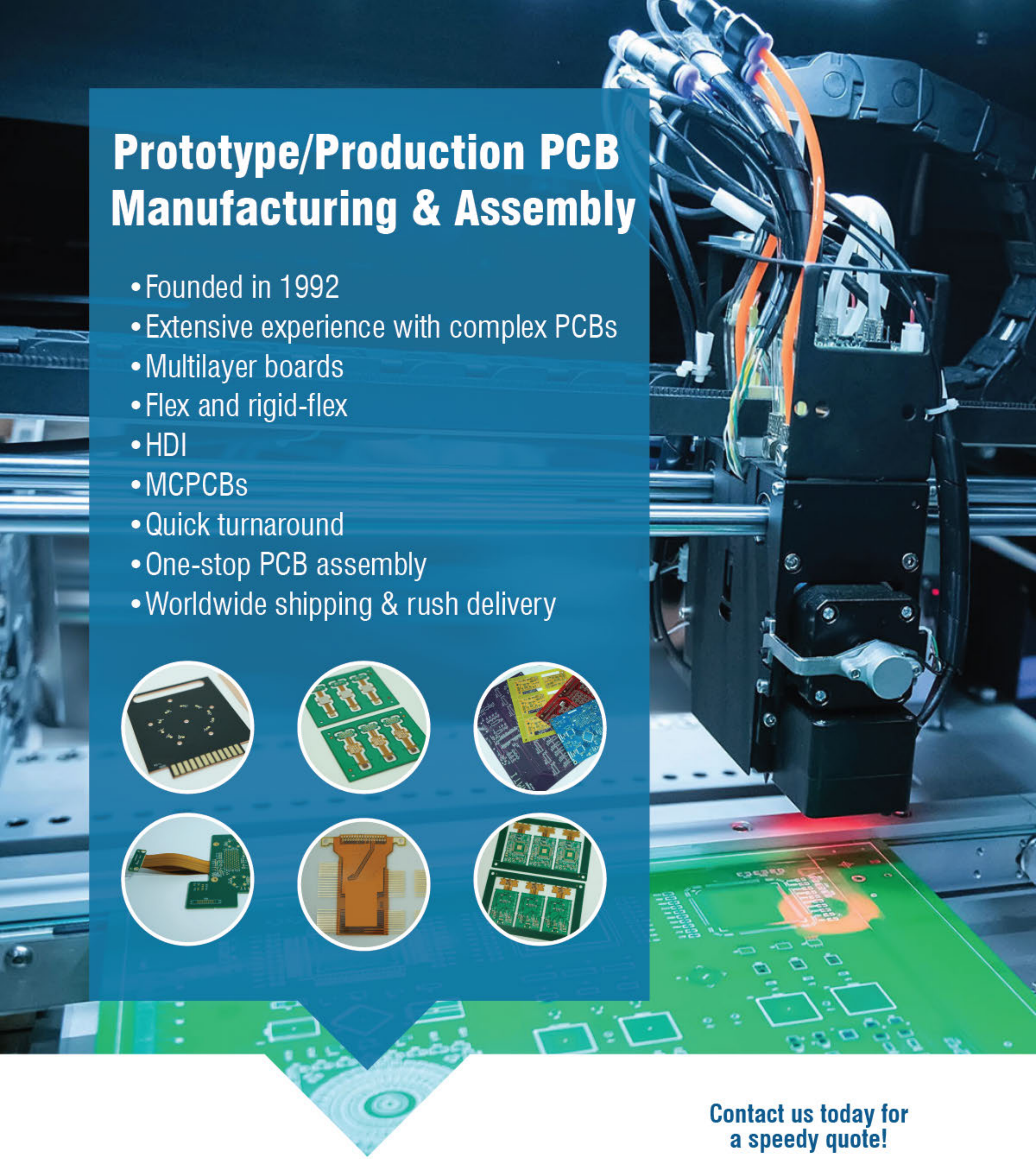
Feature Column by Istvan Novak, SAMTEC

Power integrity components—such as bypass capacitors, inductors, ferrite beads, or other small discrete components—can be characterized in fixtures. There is a wide range of fixtures available, from the professional and very accurate [1] to the home-made and very crude [2]. In between these extremes, you will find various printed circuit board fixtures, such as the decoupling test board kit shown in Figure 1 [3] or the RF experimenter board set shown in Figure 2.

The Picotest boards come with Touchstone files for de-embedding the measured data. I particularly like this kit because it has separate small boards with solder pads specifically for a wide range of surface-mount component sizes, including reverse geometry capacitors and some medium-size bulk capacitors. The range starts with the 0201 size and includes 0204 and 0612 reverse-geometry sizes. Large-size polymer capacitors can be tested on the D-size (7343) fixture; for surface-mount cylindrical capacitors, we get 5-mm and 8-mm sites. To test filter structures, two of the smaller sizes (0402 and 0603) also have a generic T scheme: pads for two components in the series path and for one component in the shunt path. There is a dedicated site for single-body 0402-X2Y filter elements. This test board kit also has a single-piece open-short-load impedance reference board; you can find it in the oversized lower-middle compartment in Figure 1. You can also build fixtures straight out of small coaxial connectors [4].

With all of these fixtures, we need to keep in mind that the current path around the device we characterize does not necessarily match the current path that is created by the layout and stackup in our final application. In the solder-wick fixture, the current path is highly uncertain; the shape of the flexible connections will vary depending on how we achieve the pressure-mount connection. With the fixtures built entirely out of SMA connectors, we have a fixed geometry for the connector pieces, but there are no dedicated pads to solder the parts down, so the actual current path depends on how we solder the DUT between the center pins and outer frame. This means that the extracted mounted inductance values will need to be used with some caution.

Figure 1: Picotest component characterization fixtures.


Prototype/Production PCB
Manufacturing & Assembly
• Founded in 1992
• Extensive experience with complex PCBs
• Multilayer boards
• Flex and rigid-flex
• HDI
• MCPCBs
• Quick turnaround
• One-stop PCB assembly
• Worldwide shipping & rush delivery

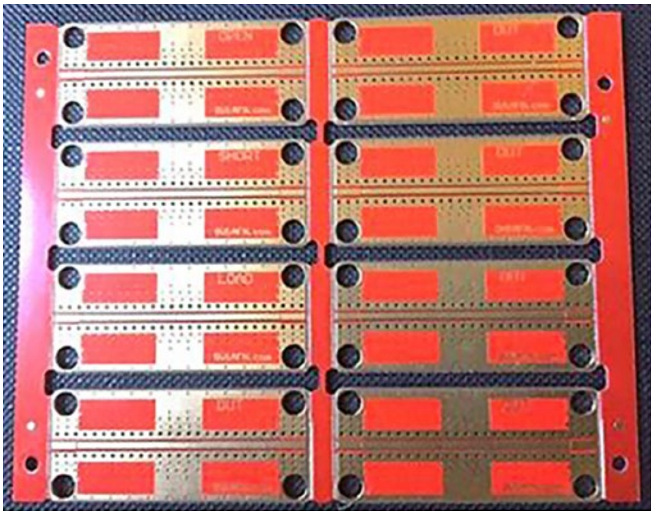


Figure 2: Unassembled panel of RF experimenter boards from SV1AFN [5].

Generic PCB fixtures, such as those shown in Figures 1 and 2, can be created from small, co-planar, 50-ohm traces that have exposed trace and ground next to each other on the same side of the fixture. The DUT can be connected between the trace and the ground shape, allowing us to use the two-port, shunt-through measurement topology. Having a sufficiently large solder-mask-free ground shape next to the trace allows us to accommodate a large number of different case styles and sizes with the same board. Having connectors at the ends of the through trace will allow for quick connections and disconnections, though we could also use permanently attached (soldered) cables. Soldered connections would eliminate the need for separate cables with connectors at both ends, but would make the calibration a little bit more difficult. Figure 2 shows an unassembled panel of eight fixture boards that we can break away. Though the eight boards carry different labels, physically, they are the same.

If we solder SMA female connectors to both ends, the fixtures will conveniently take cables with male connectors. The lines of the fixture are co-planar waveguide (CPW) over ground. The printed circuit material is FR-4, and the board thickness is 0.8 mm. The gold-plated nickel over copper is 35 µm (1 oz), and the line width is 1 mm with 0.254 mm separation to ground. As you can see from the measured TDR response of Figure 3, the coplanar traces are close to 50 ohms and have only a 175-ps delay, which means for a lot of measurements up to 10 MHz, a simple response through calibration is enough.

If we want to start the sweep anywhere below a few times 10 kilohertz, and, at the same time, we also want to measure components that have low impedance at low frequencies, such as low-ESR high-capacitance parts, we run up against the cable-braid loop error [6]. Depending on how we want to reduce the cable-braid loop error, the chosen solution may come with its own limitation at low or high frequencies. For the photo on the right in Figure 4, I used a homemade common-mode choke with an upper bandwidth of approximately 50 MHz. This setup data was collected in the 300 Hz to 30 MHz frequency range with a simple THRU calibration. Professional options for common-mode transformers for power-integrity measurement purposes are also available today [7]. The setup on the left in Figure 4 uses flexible coaxial cables with low braid resistance [8], which eliminates the need for a common-mode transformer, as long as the DUT impedance is not extremely low.

With these fixtures, we also have the option of connecting the DUT in different ways. Mechanically and electrically, we get the most robust and most reliable connection if we solder or firmly clip the part to the fixture. If we want to re-use the fixture and speed up the

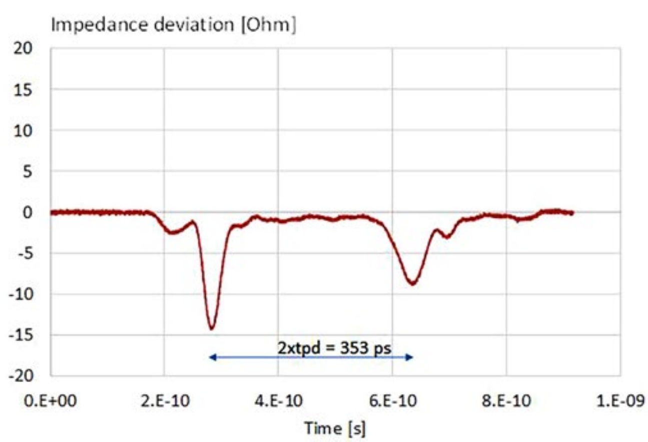


Figure 3: TDR response of a matched-terminated fixture shown in Figure 2.

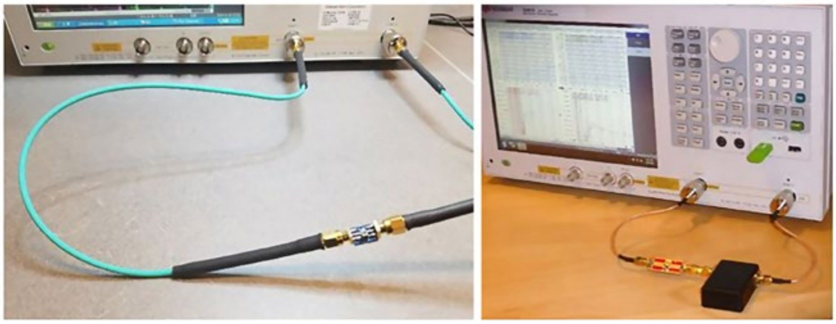


Figure 4: Measurement setup with Keysight E5061B network analyzer. The common-mode toroid transformer on the right photo is encapsulated in a small black plastic box.

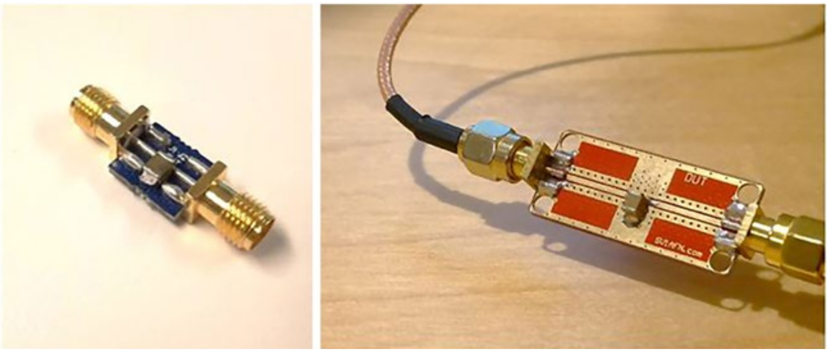


Figure 5: 1210 size MLCC parts soldered on the PCB fixtures (330-μF, 4V part on the left and 47-μF, 16V part on the right).

swapping of components, we can opt to use simple pressure mount; maybe, we can reduce the contact resistance and improve the consistency of our collected data by applying a dot of silver paste under the component terminals. If we decide to solder the component, we can improve the repeatability of the measurement by pushing down the parts on the pads during soldering. Then, we can make sure that the thickness of the solder layer between the component terminal and fixture pad is the possible minimum.

In Figure 5, the two fixtures are shown with 1210-size ceramic capacitors soldered on them. The two setups in Figure 4 use slightly different settings. Though the 330-μF ceramic capacitor has approximately 1 mOhm ESR, the impedance rises as we go toward lower frequencies. We also used cables with low-braid resistance; therefore, we did not need a common-mode transformer. This allowed us to set the upper sweep limit to 100 MHz. Full two-port SOLT calibration was done with a Keysight mechanical calibration standard.

The network analyzer gives us two-port S parameters. From the scattering matrix, we use one of the transfer parameters: S_{21} or S_{12}. They should be the same or very close to minor measurement errors. During the through calibration, the 0 dB level of S_{21} is set when Port 1 and Port 2 are directly joined without a DUT. After calibration, from the measured S_{21} value, we can calculate the Z_{DUT} unknown complex impedance:

$$Z_{DUT} = \left(\frac{50}{2}\right)\frac{S_{21}}{1 - S_{21}}$$

In the next step, we take the imaginary part of the impedance and assume that it comes from capacitance or inductance. If the imaginary part of impedance comes from capacitance or inductance, we can use the following formulas, respectively:

$$Im\{Z_C\} = -1/(wC) \quad Im\{Z_L\} = wL$$

Here, w is the radian frequency, or 2 pF. You can apply both formulas simultaneously over the entire frequency and rearrange them for C and L. They will give the correct (positive) capacitance and inductance values in their respective portion of the frequency range and will give negative results in the frequency ranges where the assumption about the nature of impedance is incorrect.

Figure 6 shows the measurement result from the setups and DUT shown in Figures 4 and 5. We make use of the impedance analysis option of the network analyzer, and we can set up the screen to show four simultaneous traces: impedance magnitude (upper left), effective series resistance, or Rs (upper right);

equivalent series capacitance, or Cs (lower left); and equivalent series inductance, or Ls (lower right). The logarithmic horizontal scale starts at 300 Hz and ends at 30 MHz. For this DUT, the 70-Hz IFBW setting provides a good compromise between fast sweep and low noise floor.

The measurement result shows several familiar details. From 300 Hz to almost 1 MHz, the impedance magnitude slopes downward, indicating capacitive impedance. The trace bottoms out at 850 kHz with a 2-mOhm value at the series resonance frequency. Beyond 850 kHz, the impedance magnitude slopes upward, indicating that we are in the inductive region. There are smaller secondary resonances and inflection points between 1.5 and 3 MHz, beyond which the impedance stays clearly inductive. The effective series resistance plot, which is simply the real part of the measured complex impedance, on the upper right follows a similar pattern. At low frequencies, it runs parallel to the impedance magnitude curve, and their ratio is the dielectric loss tangent [9]. After a broad minimum near the series resonance frequency, the effective resistance also trends upward with a few secondary resonances. On the lower left and right plots, you see the capacitance and inductance extracted from the imaginary part of the measured complex impedance.

With the data shown in Figure 6, we get positive extracted capacitance and negative inductance (that we just ignore) below 850 kHz. Above 850 kHz, we get positive inductance and negative capacitance (which, again, you just ignore). Finally, if we look at the extracted capacitance and inductance curves, we notice that both are sloping downward slightly. The capacitance curve slopes downward due

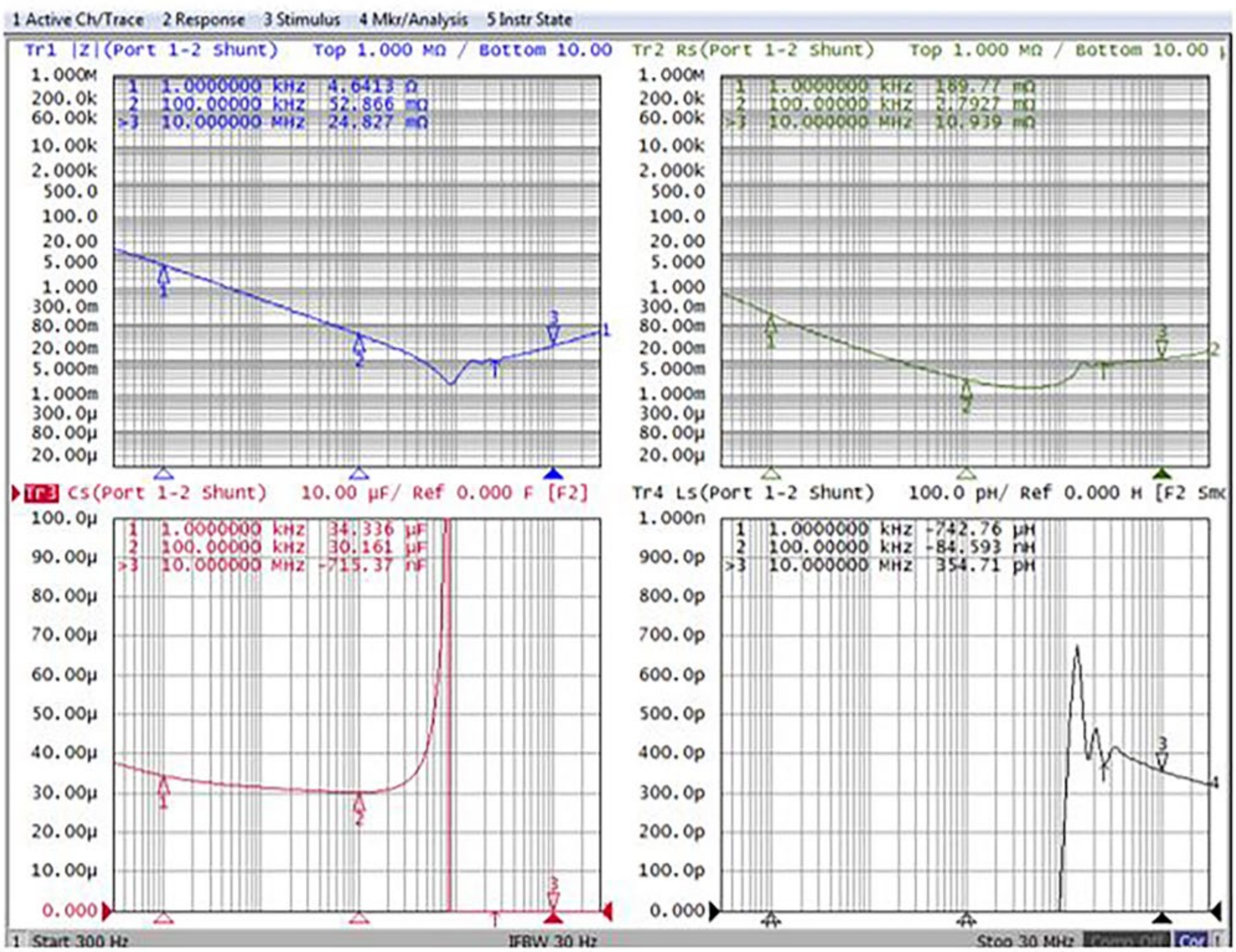


Figure 6: Measurement data of the 47-µF 1210 size ceramic multilayer capacitor taken in the co-planar fixture.

to the dielectric losses; the higher the loss tangent, the more pronounced slope you see. The inductance curve slopes down because of the finite size and thickness of the DUT. At low frequencies, the current path is determined by resistive losses; as a result, the current spreads out utilizing a bigger part of the structure. At high frequencies, the current path is dictated by the path inductance, and inductance is smaller in smaller loops, so if the current has the opportunity to rearrange itself as frequency changes, it will flow in smaller loops as frequency goes up.

Co-planar fixtures are very convenient and universal, but they come with some limitations. Similar to the solder-wick fixture, the co-planar trace represents not only a convenient way to attach a DUT, but it also creates a sneaky path between the two VNA ports, which limits the lowest impedance we can measure this way. Second, being generic, chances are that these fixtures will not match the geometry of our final usage of the component in our design. However, this only has an impact on the inductance; the capacitance and ESR still can be assessed with high confidence. **DESIGN007**

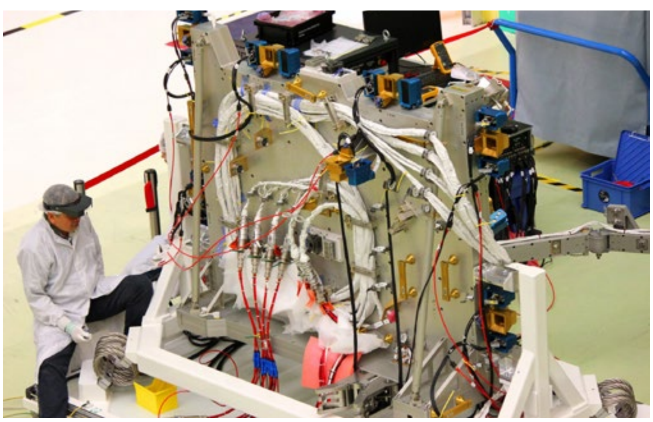
References

1. Keysight SMD component fixture 16034E.
2. I. Novak, "Solder-wick trick characterizes bypass caps," EDN, May 9, 2018.
3. Picotest DTBK01 decoupling test board kit.
4. I. Novak, "Make simple fixtures from SMA connectors," EDN, August 27, 2018.
5. SV1AFN RF experimenter's PCB panel of eight pieces.
6. I. Novak, "Why 2-port low-impedance measurements still matter," *Signal Integrity Journal,* January 16, 2020.
7. Picotest common-mode transformer J2102B.
8. Picotest PDN cable PDNCBLOP5M.
9. I. Novak, "Quiet Power: How to Read the ESR Curve," *The PCB Design Magazine,* November 2012.

Istvan Novak is the principal signal and power integrity engineer at Samtec with over 30 years of experience in high-speed digital, RF, and analog circuit and system design. He is a Life Fellow of the IEEE, author of two books on power integrity, and an instructor of signal and power integrity courses. He also provides a website that focuses on SI and PI techniques. To read past columns or contact Novak, click here.

Bartolomeo Starts Its Journey to the International Space Station

The Bartolomeo research platform, developed by Airbus for the International Space Station (ISS), has been delivered to the Kennedy Space Center in Florida, USA. The move marks a further step toward something never before seen in space. With its planned launch in March, the European-built Bartolomeo is set to become the first commercial research platform to be attached to the ISS.



Bartolomeo is funded by Airbus and will be operated with the support of the European Space Agency (ESA). The platform can host up to 12 different payload slots, also providing them with a power supply and data transmission back to Earth.

With Bartolomeo, Airbus is offering fast and cost-efficient access to research in space, which can also be used by private data service providers. The platform's unique vantage point 400 kilometers above the Earth offers unobstructed views of our planet. Not only does this provide opportunities for Earth observation, but also for carrying out measurements related to environmental and climate research—for example, the concentration of nitrogen oxide or CO_2 in the Earth's atmosphere.

Bartolomeo will now be subject to further inspections at the Kennedy Space Center before being integrated into a Dragon space transporter. The launch is currently scheduled for March 2, 2020.

(Source: Airbus)

The New Printed Circuit Engineering Association

The Digital Layout

by Stephen V. Chavez, MIT, CID+, PCEA

In this month's column, we'll take a first look at the new Printed Circuit Engineering Association (PCEA), including an overview of membership and why we established the organization. We'll also look at the mission of the PCEA, and how it will unfold to the industry.

Membership

The PCEA is an international network of engineers, designers, and specialists related to printed circuit development. In essence, we are a mixed group of individuals that cover the entire product development cycle. There are no limitations or restrictions as to who can become a member. Membership is free and open to all those interested in gaining and sharing their knowledge with others. We will serve the industry as a non-profit organization.

Why We Established the PCEA

There are several well-organized groups that cater to the electronics industry. These organizations tend to feature a specific segment of this industry by hosting highly focused, major conferences for their followers. These conferences tend to be one or two times a year at different locations.

Each of these conferences specifically focuses on a skill segment, such as design, assembly, manufacturing, etc. With technology changing at a rapid pace, it becomes very difficult and expensive for professionals to attend the plethora of major conferences conducted in the course of a year.

In an attempt to address the changing demands, some organizations, in addition to major conferences, have set up local chapters to address our rapidly changing industry. They have found it difficult to fully address the needs of each industry discipline required to produce a workable PCB product. The main theme was to understand each other's challenges and how to assist each other. Unfortunately, printed circuit engineers were often not understood by other industry segments and these organizations.

Printed circuit engineers are technologists who must understand the needs and requirements of all the disciplines that build the final product. They must have a comprehensive understanding of electrical engineering, design, mechanical engineering, printed board materials, fabrication/assembly processes and limitations, quality assurance testing, and fully populated test requirements at all levels of

PCEA
Printed Circuit Engineering Association

the product development process. In addition to these disciplines, printed circuit engineers must have an understanding of compliance and field service requirements of each product. The PCEA is structured to meet the challenges we face today and take on the challenges that are coming for printed circuit engineers.

The PCEA's Mission

The PCEA's mission is to promote printed circuit engineering as a profession and encourage, facilitate, and promote the exchange of information with the integration of new design concepts through communication, education, seminars, workshops, and professional certification through a network of local, regional, and international PCEA-affiliated consortiums. We're seeking to provide a constructive, professional environment working with all other industry organizations.

We're seeking to provide a constructive, professional environment working with all other industry organizations.

Since our inception, we have been achieving great industry acceptance. We have received endorsements from professionals across the electronics industry. In the coming months, we will expand the number of affiliates while realizing a steady growth of our membership. If you're interested in joining or starting an affiliated consortium in your area, please feel free to participate and join the many professionals who are the backbone of our industry. You can join us by visiting our website (pce-a.org) and contacting us for more information.

Action Items

Here are our Q1 2020 action items.

1. Continuing to develop our website (pce-a.org)
2. Elect an executive board to care for the oversight of the association
3. Expand the content of our current communication vehicles
4. Assist in the formation of new, regional affiliate groups
5. Establish a working relationship with other associations, such as IPC, SMTA, and IEEE

Professional Development and Events

Here are some up-and-coming industry events to look out for in 2020. I hope you have the opportunity to attend one or more.

- March 24–25: PCB2Day—SMT Assembly Boot Camp (Austin, Texas)
- March 26–27: PCB2Day—Design Essentials for PCB Engineers (Austin, Texas)
- April 27–30: Zuken Innovation World Americas 2020 (Coronado, California)
- June 22–25: Realize LIVE 2020 (Las Vegas, Nevada)
- October 7–9: AltiumLive 2020 (San Diego, California)
- September 8–11: PCB West (Santa Clara, California)
- November 11: PCB Carolina (Raleigh, North Carolina)

As always, if you have any local or regional industry event coming up in your area and would like to announce it, please feel free to submit the details to be listed in an upcoming column. For 2020 CID and CID+ certification schedules and locations, contact EPTAC to check current dates and availability (dates and locations are subject to change). DESIGN007

Stephen Chavez, MIT, CID+, is a Master Instructor of PCB Design for EPTAC, an SME in PCB design for a major aerospace corporation, and is a member of the Printed Circuit Engineering Association (PCEA).

To read past columns or contact Chavez, click here.

The Impact of PDN Impedance on EMI

Beyond Design

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

In last month's column, I looked at why interconnect impedance is so important to the performance of our system. I also mentioned that impedance is defined in both the time and the frequency domains. In the time domain, we need to consider the impact of the interconnect impedance on the propagation of electromagnetic energy. However, in the frequency domain, the AC impedance needs to be kept below an acceptable level to prevent excessive radiation.

Decoupling capacitors are generally spread throughout the power distribution network (PDN). In the past, we thought that it was only necessary to add a few capacitors to each IC power pin to stabilize the power at that device. That worked fine at low frequencies. Thinking in the time domain, you can say that decoupling capacitors store and supply charge to the loads until the power supply can respond. However, in the frequency domain, decoupling capacitors also lower the impedance at different frequencies to help meet the AC impedance target. Thus, there are two distinct functions of capacitors that work in unison but in different domains. In this month's column, I will look at impedance in the frequency domain and its impact on EMI.

The PDN acts like an on-board ecosystem (Figure 1). If one small part of the PDN were to change, then the entire network performance may be impacted. This is particularly noticeable when mounting, loop, and spreading inductance are added to the profile of the capacitors. Inductance has its greatest influence in the high-frequency band, where odd harmon-

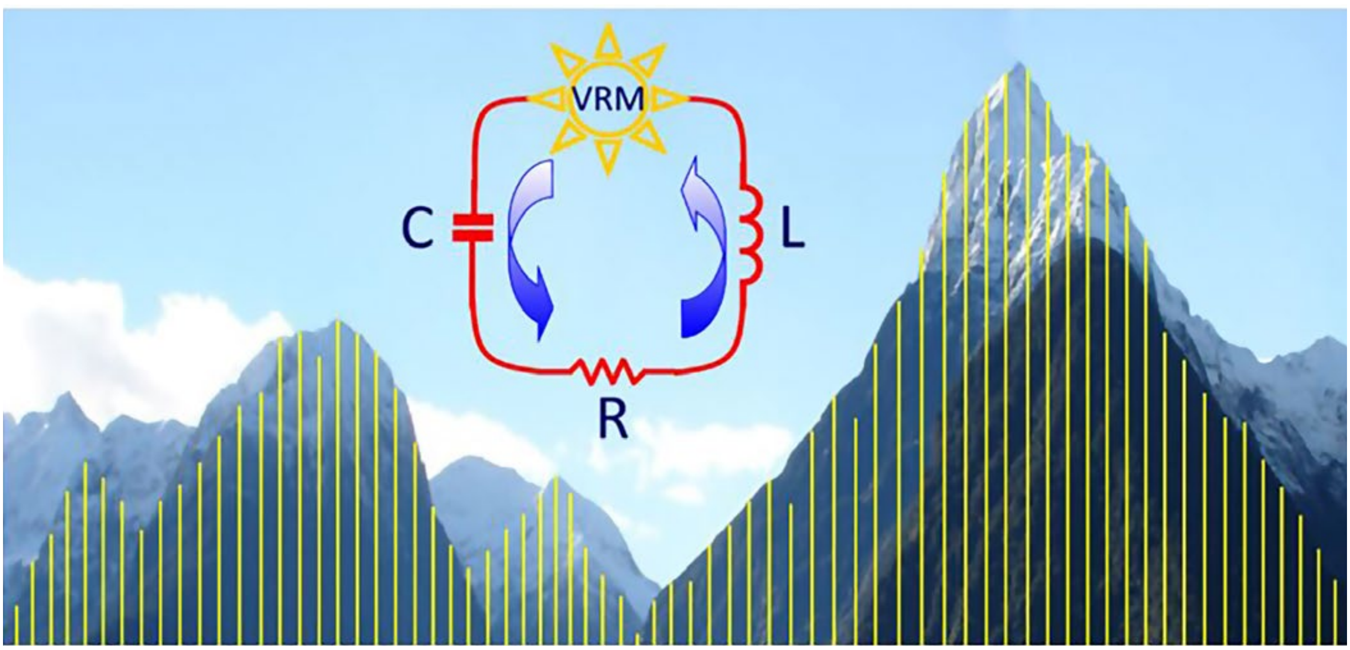


Figure 1: The PDN acts like an on-board ecosystem.

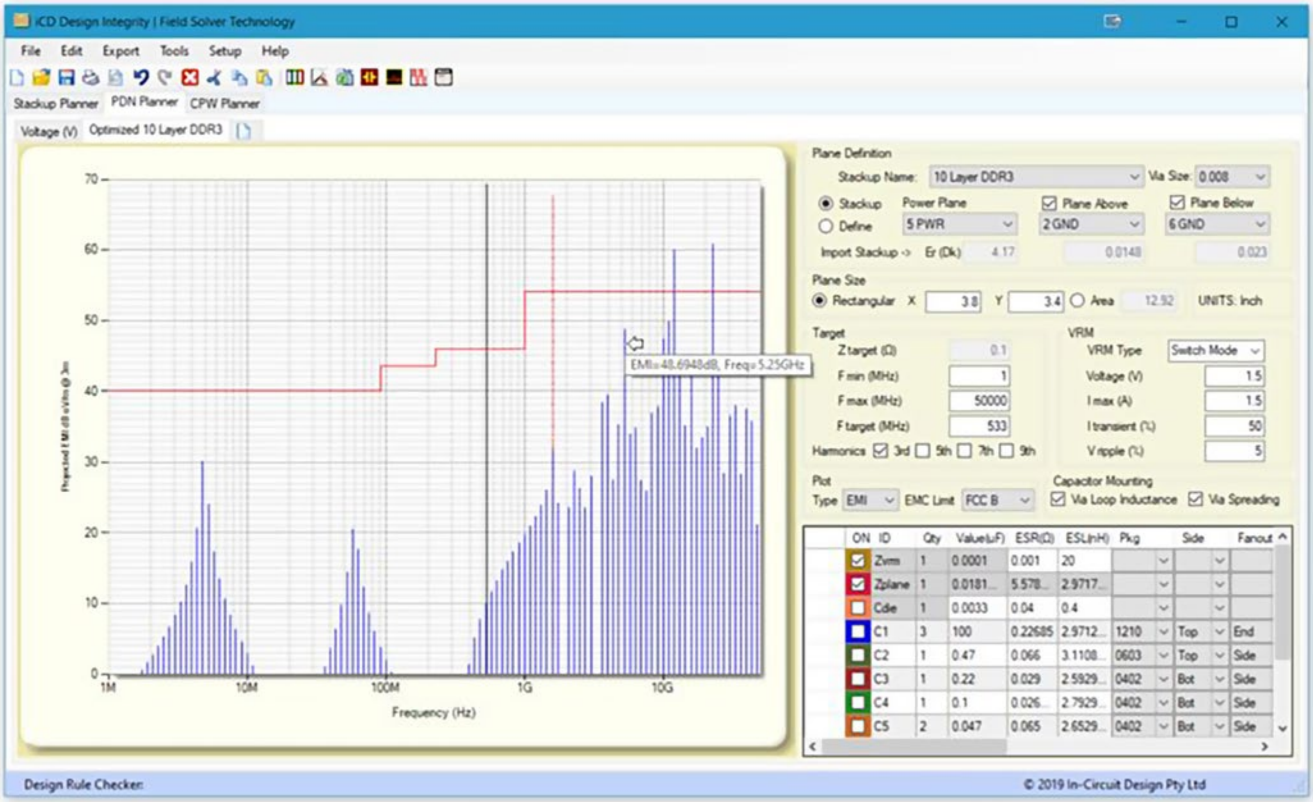


Figure 2: Odd harmonics of the fundamental tend to radiate at high frequencies (iCD PDN Planner).

ics of the fundamental tend to radiate if impedance peaks appear at the harmonics (Figure 2). Inductance pushes the impedance up and shifts the resonant frequency of capacitors down. The addition of more capacitors to the PDN lowers the impedance at particular frequencies, while the combined parallel resistance determines the minimum impedance at the resonant frequency (dips).

The PDN has four major functions:

1. Provides a low-impedance, high-current supply path to the ICs.
2. Reduces noise on the power supply at the IC die.
3. Minimizes ground bounce and common-mode potential between the IC die and the return path.
4. Reduces electromagnetic radiation from the board fringing fields.

The typical PDN topology (Figure 3) consists of numerous connections from the voltage regulator module (VRM) to the capacitors, vias, wide traces, copper pours, power and ground planes, solder balls, and interconnects on the IC silicon itself. The PDN must provide a constant supply voltage within a tolerance of 5% at the power pins of each IC. This voltage must be stable from DC up to the maximum bandwidth, which is typically five times the fundament frequency. At the same time, these connections that provide the power are also tasked with providing the return signal path. This is where high-current, simultaneous switching noise from the power supply can interfere with the signal return currents.

When the return current flows through the impedance of a cavity between two planes, it generates voltage. Although quite small, the accumulated noise from simultaneous switching devices can become significant. Unfortunately, as core voltages drop, noise margins become tighter. 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 (e.g., between copper pours over planes). Other signal vias passing through

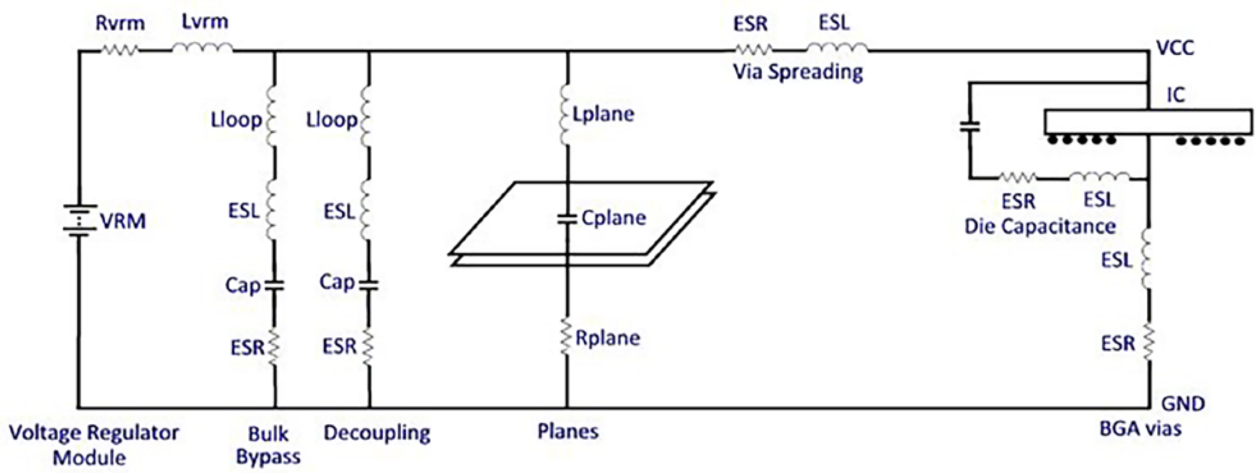


Figure 3: Typical PDN topology.

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.

If the plane cavity is not dampened, then electromagnetic fields can radiate from the board (as in Figure 2). However, when optimized, the PDN can mitigate many potential EMI issues and help prevent EMC certification test failures. How do we effectively dampen the plane cavity? With the continuous trends to smaller feature sizes and faster signal speeds, planar capacitor laminate, and embedded capacitor materials (ECM) are becoming a cost-effective solution for improved power integrity. This technology provides an effective approach for decoupling high-performance ICs whilst also reducing electromagnetic emissions.

Embedded capacitance technology has a very thin dielectric material (0.24–2.0 mils) sandwiched between two copper planes that produce distributive decoupling capacitance and takes the place of conventional discrete decoupling capacitors over 1 GHz (Figure 4). Unfortunately, standard decoupling capacitors have little effect over 1 GHz, and the only way to reduce the impedance of the PDN above this frequency is to use ECM or alternatively on-die capacitance. These ultra-thin laminates replace the conventional power and ground planes and should be positioned in the stackup close to and directly below/above the IC. This reduces the loop inductance dramatically. With a capacitance density of up to 20 nF per square

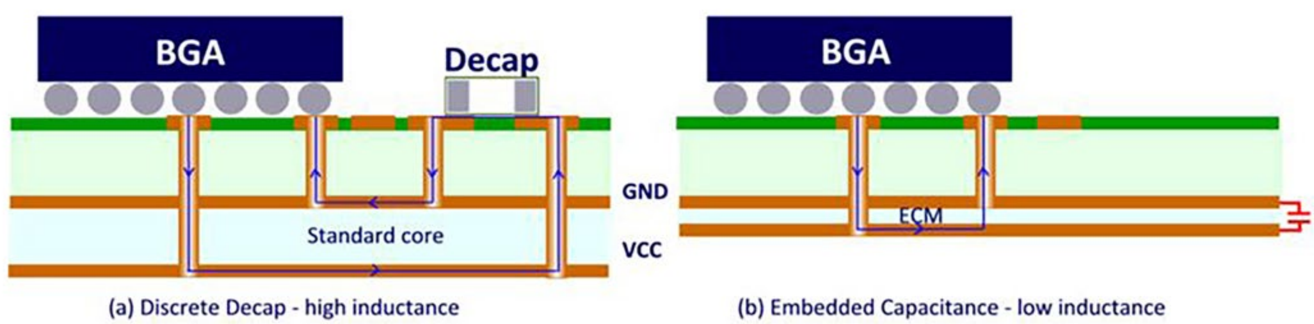


Figure 4: Embedded capacitance dramatically reduces inductance.

Manufacturer	Material	Description	Thickness (mils)
3M	ECM	Embedded capacitance materials	0.24, 0.47, 0.55
DuPont	Interra HK4,11	Ultra-thin laminate	0.5, 1.0
Integral Technology	Zeta Bond	High-Tg, epoxy-based adhesive film	1.0, 1.5, 2.0
Integral Technology	Zeta Lam SE	Low-CTE, C-stage dielectric with a high Tg	1.0
Integral Technology	Zeta Cap	High-performance polymer-coated copper	1.0
Oak-Matsui Technology	FaradFlex	Planar capacitor	0.31, 0.47, 0.63, 0.94
Samina	ZBC-1000	Buried capacitance, high-performance decoupling	1.0
Samina	ZBC-2000	Buried capacitance, high-performance decoupling	2.0

Table 1: ECM (available in the iCD Dielectric Materials Library).

inch and extremely low inductance, 3M ECM is the highest capacitance density material on the market. Table 1 lists available ECMs. ECMs have excellent stability of dielectric constant (Dk) and dissipation loss (Df) up to 15 GHz.

These specialized ECMs are precision-fabricated laminates with extremely thin dielectrics. However, if your budget is limited, then a planar capacitor can be constructed with two build-up layers so that they are close to the top/bottom of the stackup. Alternatively, use a thin 2-mil laminate as a plane pair on the inner layers. These are not as good as ECM but much better than a typical thicker laminate. Please check the minimum thickness with your fabrication shop first.

The goal of designing a high-performance PDN is to reduce the impedance peaks below the target impedance level and push the peak frequency components above the bandwidth of the signals.

To achieve this, one needs to reduce 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 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 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 have little impact on 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. Thus, 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 mils) 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

Planar capacitance can reduce EMI by minimizing the loop area, power supply ripple, and plane resonance. It is also up to three times better at heat transfer compared to a standard core. System price performance can also be improved by reducing the discrete capacitor count and the size of the PCB and by improving signal and power integrity. The best feature is that it is truly a drop-in replacement for standard core materials. If you need a quick fix for the PDN, look no further!

Key Points

- In the time domain, decoupling capacitors store and supply charge on demand to the loads
- In the frequency domain, decoupling capacitors also lower the impedance at different frequencies to help meet the AC impedance target
- Inductance pushes the AC impedance up and shifts the resonant frequency of capacitors down
- High-current simultaneous switching noise from the power supply can interfere with the signal return currents
- As core voltages drop, noise margins become tighter
- The 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
- This cavity noise propagates as standing waves spreading across the entire plane pair
- If the plane cavity is not dampened, then electromagnetic fields can radiate from the board
- When optimized, the PDN can mitigate many potential EMI issues and help prevent EMC certification test failures
- Planar capacitor laminate or ECM is becoming a cost-effective solution for improved power integrity
- ECM takes the place of conventional discrete decoupling capacitors over 1 GHz
- These ultra-thin laminates replace the conventional power and ground planes and should be positioned in the stackup close to IC
- ECMs can have a capacitance density of up to 20 nF per square inch and extremely low inductance **DESIGN007**

Further Reading

B. Olney, "Beyond Design: Plane Crazy, Part 1," *The PCB Design Magazine,* December 2015.

B. Olney, "Beyond Design: Plane Crazy, Part 2," *The PCB Design Magazine,* January 2015.

B. Olney, "Beyond Design: PDN Planning and Capacitor Selection, Part 1," *The PCB Design Magazine,* December 2013.

B. Olney, "Beyond Design: PDN Planning and Capacitor Selection, Part 2," *The PCB Design Magazine,* January 2014.

B. Olney, "Beyond Design: Decoupling Capacitor Placement," *The PCB Design Magazine,* January 2016.

B. Olney, "Beyond Design: Plane Cavity Resonance," *The PCB Design Magazine,* September 2017.

B. Olney, "Beyond Design: Ground Bounce," *Design007 Magazine,* January 2018.

B. Olney, "Beyond Design: The 10 Fundamental Rules of HSD, Part 3," *Design007 Magazine,* November 2018.

B. Olney, "Beyond Design: Materials Selection for SERDES Design," *The PCB Design Magazine,* September 2013.

Oak-Mitsui Technologies, "Improved Power Delivery and Reduce Noise with Embedded Capacitance."

I. Novak, *Signal Integrity Journal.*

E. Bogatin, *Signal and Power Integrity: Simplified,* Prentice Hall, 2008.

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

Clearing Up the Buzz

Tim's Takeaways

Feature Column by Tim Haag, FIRST PAGE SAGE

My first "real" job in the world of electronics was working at a Radio Shack store back in the late '70s. It was a step up from flipping burgers, but it didn't last long. However, there was one notable aspect of that job; I was there during the time that Radio Shack introduced its first personal computer—the TRS-80. Although it is practically unimaginable now, in those days, there wasn't much in the way of personal computing available for the general consumer. "Pong" and a few of its related home video games that connected to a television set had been out for a handful of years, but that was nothing compared to the millions of gaming options available today. And as far as personal computers go, in the late '70s, there were only a few options to pick from, including Apple, Commodore, and Tandy, or more commonly known as Radio Shack. After its introduction, the TRS-80 enjoyed a short period of time where it was the bestselling computer on the market with the largest selection of software.

I would like to say that I leveraged my short time at Radio Shack to the fullest by immersing myself in the world of computers and electronics. It would be great to include on my resumé how my experience with the TRS-80 gave me the computing and, ultimately, the PCB design foundation that I would base my career on, but it would all be a lie. The truth is that I was a kid who had just graduated from high school who didn't have a clue to what the future held and was more interested in the "toy" qualities of the computer than in its educational possibilities. Because of that, I regret to admit that I spent all of my time on the TRS-80 playing games. Hey, if you find a subject like that tough to read, just imagine how hard it was for me to write. As they say, the truth hurts.

However, there was something very interesting about playing games on that computer. Unlike today's systems, the first TRS-80s were so simple that they didn't come

Photograph by Rama, Wikimedia Commons

with a built-in speaker; therefore, the games didn't have any sound effects. To remedy this, I would grab a standard, portable AM radio off the shelf and set it up next to the computer while playing a game. The TRS-80 had a big problem, though—it radiated a lot of electromagnetic interference (EMI). The effect was so pronounced that it would produce a buzzing over the radio's speaker that would vary in intensity and volume, depending on how active the computer was. For a game, this worked out better than you might think, and it is perhaps the only time in my life that I can think of where I used an EMI problem to my advantage. According to Wikipedia, the TRS-80 was actually discontinued primarily due to stricter FCC regulations on radio-frequency interference to other devices. As it turns out, I wasn't the only one using a radio to produce sound effects out of that computer.

As it turns out, I wasn't the only one using a radio to produce sound effects out of that computer.

I have no idea what the circuit board or boards for the TRS-80 looked like, or with what level of technology they were designed, but from the amount of EMI that was created, there is probably a lot that they didn't have. I would guess that the boards probably had as few layers as possible to reduce their production expenses. This would be at the cost of adequate ground planes for clean signal return paths and shielding. I would also assume that the routing on those boards may have been optimized for production instead of signal integrity, or perhaps there wasn't any optimization at all, and the traces looped all over. Those would be my first guesses, but there are many other design details to consider, as well. These would range from the type of components that were on the board, how they were used, and how the casing was constructed. Wouldn't it be fun to open up one of those old computers now just to see what made it tick, or rather, buzz?

PCB layout has changed a lot over the years. Back in those days, the signal speeds weren't anywhere close to what we are dealing with today, and most of us had no idea how many design errors and EMI nightmares we were creating by today's standards. Like the boards in the TRS-80, many of the PCBs I designed in those days would probably spray interference around like a yard sprinkler on a summer day if they were to be evaluated by today's standards. Things are different now, and most designs that I am familiar with go through strict design control for EMI by the design team and the manufacturer. In addition, there is a lot of help available for designers to better understand EMI problems and how to avoid them in their designs. Some of this help includes the following:

Tools

The PCB design tools available to the user today are packed with features and functionality, and more is being added every day. In addition, circuit simulators and signal analysis tools are now more powerful, as well as being more affordable than ever. All of this has resulted in these analysis tools increasingly being integrated into the PCB design tools so that everyone has easy access to them.

Information

When I first started designing, any additional information that you wanted on a design subject would only be available in print, and I just didn't get to the library that much. Today, you can easily search for information online and find scores of publications written on any given topic. You can also find various user groups online that are often eager to help and share their knowledge.

Education

The different venues for increasing your knowledge of design subjects continue to increase regularly. From the traditional education provided by schools and universities, to online courses, you can add to your education as needed. Additionally, there are scores of seminars and workshops available, and design trade shows will also offer many different classes hosted by industry professionals.

Networking

From co-workers and peers to online industry professionals, there's a lot more professional and social networking available today that you can go to for help and advice. Designers aren't nearly as isolated as they once were.

The industry is poised now better than it ever has been before to develop and create the next generation designs needed for our advancing technologies. Yet we still tend to bump into problems that stem from interference. As we've seen, there are plenty of different ways to deal with EMI problems in our designs, but what about other sources of interference? I'm sorry to say that for all of our efforts to reduce EMI in our designs, we aren't always that great about dealing with workplace interference (WPI). Isn't it ironic that for all the effort we can put into a PCB design to ensure its electromagnetic compatibility, the design could still be undermined by WPI?

Communication, or lack thereof, seems to be one of the greatest causes of WPI in design teams. Some members of the team may not get informed in a timely manner when their input is needed or required. In other cases, team members are unsure of who to report to, or who gets the handoff of the next phase of the project. And no matter how much this topic gets covered in different seminars, workshops, and trade show classes, there is still a communication gap between some design teams and their manufacturers. In engineering teams, time is wasted, and mistakes are made because clear lines of communication have not been defined. If you want to avoid this kind of WPI in your workplace, here are a couple of ideas that may help you to clear up the interference.

- **Have a defined workflow process in place:** Your team members won't know who does what, what the previous step was, where to go next, or how to escalate a problem without a plan that they can refer to.

- **Identify a chain of command:** Someone has to be the boss to make the decisions and field the important problems. If that person is you, then roll up your sleeves and get to work. Whether it's on the court, at a school, or in the workplace, every team needs a leader.

- **Design reviews:** To ensure that the project is ready for each succeeding phase of the design, put into place regular design reviews. These reviews should be part of your workflow process, and key members of the design team should be involved.

- **Keep the team engaged:** Growth is an important part of success, but it can also isolate team members if you aren't careful. Make sure to keep everyone engaged so that they understand their importance to the overall goal. In other words, don't shy away from the occasional taco Tuesdays and casual Fridays. These are all good ideas to keep your team's morale at a high level.

Whether it's on a PCB or in the office, interference is never a good thing. Our engineering teams don't need the annoying disruption that lack of communication brings to work, and I no longer have an AM radio on my desk to broadcast EMI-generated pseudo game sounds through. The more that we can clean up the buzz, the better off we'll be.

Until next time, my friends, keep on designing. DESIGN007

Tim Haag writes technical, thought-leadership content for First Page Sage on his longtime career as a PCB designer and EDA technologist. To read past columns or contact Haag, click here.


Alexander Löwer

EMI Challenges: A Contract Designer's Take

Feature Interview by Barry Matties and Andy Shaughnessy
I-CONNECT007

Andy Shaughnessy and Barry Matties recently spoke with Alexander Löwer, a freelance design engineer based in Germany, specializing mostly in EMC optimization. A self-described "project firefighter" called in to fix the direst of design situations, Alex discusses some of the EMC issues he sees and why properly designed differential pairs and power distribution networks and can help preclude EMI problems later.

Andy Shaughnessy: Alexander, start off with a little background about yourself. How did you get into electronics?

Alexander Löwer: My physics teacher got me interested in electronics back in high school. I went to university and studied classical electrical engineering. After graduation, I started to work in a small company, mostly with ASICs. I found that instead of working on something that is the size of a thumbnail, it's much more satisfying to work with something larger, so I went into PCBs.

The company I had worked for was not much into PCBs, which meant I had to find something else. A friend of mine was self-employed as a freelance engineer. He introduced me to the concept of freelancing, and I ended up where I am now, as a one-man fire brigade.

It usually goes like this. A customer comes to me and says, "Alex, something isn't working." That's about all the information I get, paired with a date when "it" has to work. Speaking of firefighting, they tend to contact me once something has already gone wrong. Then, I have to find a solution to their problems that is feasible for the company. It heavily depends on the options left in the project and the state of the product. Sometimes, it's too late, or the customer has to make a tough decision, but I usually come up with a solution that satisfies them.

Shaughnessy: You're an electrical engineer by education. What's your specialty?

Löwer: My favorite area is EMC optimization. Of course, I work in other areas as well, especially as companies are having a hard time finding engineers. My other sweet spots are schematic and layout optimization and the

early design stages, where you can make good and bad decisions.

Shaughnessy: You'd rather have the customer come to you early in the design before the EMC problems arise.

Löwer: Yes. In that stage, it's possible to avoid problems. Unfortunately, most of the customers come to me when decisions are already made, meaning after the first EMC test results come in, and their customer or product manager says, "No."

I got into EMC when I was with a customer. It was one of the typical "we don't have enough engineers for the amount of work" jobs. I didn't have any clue about EMC at that point. The project had already run for more than a year. Their customer said, "We're canceling this project if you don't fulfill our EMC requirements within the next three months."

I had the unfortunate luck of being the only one who could make PowerPoint presentations. The chief engineer of my customers' customer visited and demanded to see how we were planning to improve the EMC performance of the product. All my colleagues in the company were more experienced engineers, but their presentations were strongly influenced by the local dialect. It was hard for non-native German speakers to understand them.

Again, they were all great engineers, but they did not grow up with PowerPoint. I was the youngest, so I was the one who had to give the presentation, but I made a deal. It was clear that their customer would assume that whoever did the presentation was responsible for the EMC part and would be blamed if it didn't work out in the end, so I agreed to give the presentation under some concessions. I requested the local EMC chamber for the next couple of months, an additional engineer for general support, and a measurement engineer to help me because I had no clue how to measure stuff. My customer agreed.

Over the next four months, we had our own EMC chamber and started experimenting a lot. We learned by doing. Looking back, some ideas were stupid, but others were great. We delivered it on time—one day before the deadline ended. That's how I started working on EMC.

Shaughnessy: Are your customers primarily in Europe?

Löwer: They're mostly in Germany.

Shaughnessy: Are they primarily automotive?

Löwer: About one-third of my customers are, but half of the workload is automotive.

Shaughnessy: We don't seem to have as many EMC problems now, do we? I don't hear as many horror stories.

Löwer: The problems have changed in appearance. Engineers have grown accustomed to handling EMC issues. It is normal to solve EMC issues in more complex projects. Therefore, engineers recognize EMC or EMI problems earlier in the project. They try to handle these issues, but there are a lot of things you can do—some better, some worse. Most people only think in differential pairs and split power planes, but there are more possibilities depending on the issues, starting with good layout practices and not ending in more elaborate power filtering. I see that many engineers have an understanding of EMC but lack experience in handling it.

However, the issues start earlier. There are a lot of non-technical issues that need to be addressed, as well. Look at the number of regulations. In electrical mobility and electric cars, you have a lot of EMC norms and regulations to fulfill. Each region of the world has its own regulatory bodies with specific rules. For example, the U.S., Canadian, European—UN ECE—market and Chinese market all have slightly different EMC regulations in place that you need to comply with.

There are other challenges, too. Chinese regulations, for example, are similar to the European ones, but when you only comply to the European norms, you will not be within the Chinese norms. It is not complicated to comply with them as well, except for the issue of reading Chinese, and most European EMC engineers don't understand Chinese. You have similar situations in consumer products and industrial applications. The number of norms and regulations is quite large, so the first EMC issue I see is, "Which norms do we have to fulfill?"

Shaughnessy: And in America, you have to deal with FCC regulations.

Löwer: Yes, but they are pretty straightforward. There are a lot of them, but they don't contain any surprises. You can read them for free online, which makes it a lot easier. Everything is open-access, and you can look into it if you like. This is not the case in all regions. I don't see that EMC regulations or problems are getting fewer and fewer, but they're changing.

I have customers in the automotive industry who have lots of issues in their cars. Each car gets more and more electronics, and the trend is not stopping. We already see more issues in interference between different systems, such as interoperability between components in the same system becoming more and more EMC-related. Therefore, it gets more and more work.

Shaughnessy: Which segment of boards has the worst EMC problems right now: automotive or consumer?

Löwer: The worst I've seen personally is in automotive. You have a lot of systems that are tightly packed. What I see is that the general regulations, such as FCC, are met, but the cars interfere with themselves, so you get interference in radio reception, for example. In the U.S. market, you typically have long-ranged AM radio bands in use. That's quite a challenge for switch-mode power supplies. The AM bands are pretty much the same frequency range as the higher-order harmonics of these power supplies.

You also have a lot of power supplies in your car. Each device usually has several DC/DC converters integrated, and they are not the only source of interference. Displays, for example, are also known to interfere. Many devices are in or near the dashboard; if you now have an antenna close by, you get all this interference in radio or mobile phone reception or 4G communication.

Shaughnessy: Sometimes, you still have to use some kind of shield to block the radiation.

Löwer: Sometimes, you are forced to do so, but that is usually a sign for bad planning; it's a last-ditch effort. If you try to shield something, then something must have gone wrong in an earlier stage. The best solution is to either prevent emitting anything from the start or to emit into non-critical frequency ranges.

The best solution is to either prevent emitting anything from the start or to emit into non-critical frequency ranges.

Therefore, it is important to plan the operating frequencies of power supplies, electric motors, and inverters. That's one of the examples of pre-planning you can do in early project stages. Later on in the project, operating frequencies are fixed. You can only work on the mitigation of interference. That reduces possibilities and can possibly increase costs. As I said earlier, I am often called into the project once it's too late.

Shaughnessy: They have already made design decisions.

Löwer: There are different design decisions. Sometimes, it has to do more with the manufacturing or cost optimization, or issues were not seen beforehand. Other times, the designer wants to have a specific form or shape. For example, today, nobody wants to have antennas sticking out of the car, but the best way to build an antenna is by putting it on the highest place of your car. However, designers don't want it, so from my point of view, it's a mistake, but is it? In the end, you want to have a beautiful car, so you have to find solutions the designer likes.

Matties: When you deal with the customer, do they come at you with specific design parameters, or do they say, "This is the functionality," and expect you to utilize whatever?

Löwer: It depends on the customer. In automotive, you get a pretty precise definition of what they want and what they have. My favorite example of this is a 30-page description of how a button has to be designed, including the feel, touch, and how much pressure you have to apply to it to activate it. Everything was defined to the smallest bit. On the other end of the spectrum are some of my customers who come from industrial applications. They say, "We have this and want this. Build it." That's about it.

Matties: Do you use HDI?

Löwer: Yes, but I've learned from Happy Holden's AltiumLive class that I'm doing it wrong. There's a limited amount of freedom. Happy told us we have to do it in a certain way. Then, we reference him and tell the customer to do it that way. There are times where I'm pretty sure he could do it, but it's not as easy for me.

Matties: When Happy talked about the cost of your board going down with HDI, if it hasn't, then you're doing it wrong or your fabricator is…

Löwer: Ripping you off.

Matties: Is that the experience you've had? Have you seen a reduction in cost by using HDI?

Löwer: No, but Happy has a lot of data to prove it. I will have to adjust my approach to get cost reductions.

Matties: Approach may be an issue, though, as you were saying.

Löwer: Yes. The idea is to use metrics on available design data to proof cost reduction and performance improvements. My customers don't have data for this, or they haven't developed a way to use this data. Mostly, HDI is used the following way. The PCB designer talks to the mechanical engineer or vice versa, and together, they conclude that there's not enough space for the PCB, so they go for microvias. That's the typical approach that I see.

Matties: Do they also use landless vias?

Löwer: I've never seen a company using landless vias.

Matties: It makes a lot of sense.

Löwer: It does, especially if you see that the reliability is better.

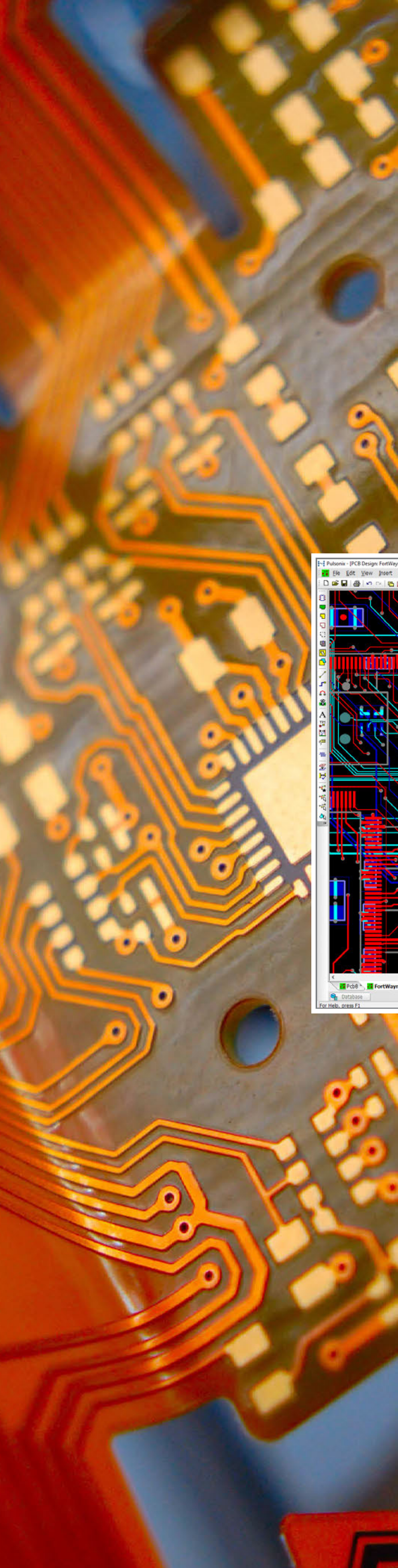
Matties: Is this something that you would introduce to your clients to consider using?

Löwer: I try to.

Matties: What's the obstacle for something like landless vias?

Löwer: It's a combination of, "We have never done it like this. We have always done this way, and this way worked," and "We don't want to have any risk." Happy had some data that showed that landless vias are better in every way and more reliable, so why don't we use them? It's not a risk assessment; we're used to not using them.

Also, I've noted that most suppliers are not keen on landless vias. I have the impression that manufacturers are not sold on landless vias yet. Only very few even consider the option on request. Until now, I did not propose them to my customers, and my customers didn't request them. I have to admit that I was not a strong proponent of suggesting one use landless vias, but especially the fact that



they are more reliable is a strong argument to reconsider. I will present that to my customers in the future.

Matties: You also gain real estate with landless vias. The other thing that Happy started with was what he called the measured performance. I thought that was interesting because he said for designers, salaries should be based on performance. The better designers do at their jobs, the less the boards will cost, and then you should pay them more. One key measure he mentioned was density. Is that a measure that you look at?

Löwer: Not until today. That's something I will try to get from customers and my boards. I will go through my PCB designs and see if those measures went up. I hope to get more insight into his data because I am curious about these relations and metrics on my PCB designs.

Shaughnessy: Most companies don't, apparently.

Löwer: Sometimes, there are rough estimates for the area of big ICs, so for density. I've seen a company that tried to define necessary space or real estate for each component into the libraries. They used old designs to get this information. But during pre-planning, they failed to go for it and get the benefits out of this information. They did not go to the extent Happy did.

Matties: Happy also asked if people in the audience used thermal modeling tools, and everybody said, "No." He talked about the variation that you get on a board from side to side. What did you think about that point that he made?

Löwer: That represented how he and other measurement people thought, coming from HP. I've seen thermal measurement or thermal simulation in products if the customer or engineers knew that there would be issues. There was one product in a vacuum that had these issues known early in the project. In a vacuum, you don't have normal convection, so thermal is a big issue there. They have done a lot of thermal simulations. Roughly speaking, the simulations were shown to be correct after tests.

I had another customer who was designing their own ASICs, which were getting too hot. They had done some simulations to find the best cooling solution and die build-up. If issues are known early on, there will be simulations, but I usually don't see it being done.

Matties: But if it has an impact on noise, or signal integrity, that's an important consideration when designing a board at that level.

Löwer: It has to be done. But the way most boards are designed, it's easier to test it than to make a big simulation. It's a matter of cost and effort. For the simulation, you have to have the tool, which costs money, the expert for the tool, and you need time. However, the reality is that time, tools, and the workforce are scarce in the industry. Tools and people who know tools well usually are rarely available.

Shaughnessy: Until recently, for the thermal management tools, you had to have a Ph.D. or a master's degree to understand it. A designer with no degree might have a hard time using most of the thermal management tools.

Löwer: Also, detailed thermal simulations are not in the electrical domain. Most thermal simulation tools I've seen are introduced into the realm of mechanical engineers. You need to have a mechanical engineer who is doing more work for this. Very few electrical engineers can use MCAD tools.

Electrical engineers like simplified models with "thermal resistances" and "thermal voltages." This way, we can use our known electrical simulation tools. The results are only rough approximations, but so far, this method has been quite successful. You get a rough estimate on what will happen, and you use prototype measurements to verify your model. That's far away from detailed thermal simulations using ANSYS tools.

Shaughnessy: It's cheaper to build it and model it. Prototype it and see what happens.

Löwer: That's what I think. If you do not have very specific requirements, it's easier that way. For detailed simulations, you have to provide good models and detailed data on electrical power consumption. Most engineers will tell you, "We have a good estimate on how much power we will have from this ASIC, IC, etc.," but it's an educated guess—especially when using large FPGA or microcontrollers because you use average data for initial system design. You can run with the old models without too much trouble—as long as you stay in the model's limitations. And with the first prototype and a more evolved software, given the appropriate time, you have the chance to reduce the component costs for power supplies and cooling solutions, or you make a cost reduction in the next product evolution step.

Shaughnessy: So, it's not to a critical point yet. They don't need to use the thermal management tools because what they're doing works, more or less. You build a model and do a prototype.

Löwer: Yes. It's faster and cheaper—as long as you do not run into unforeseen issues and surprises. As mentioned, it depends on how well you know your product, and whether or not you can see thermal issues beforehand. Usually, companies know their product range and try to avoid the—from their point of view—overhead of extensive simulations as long as possible.

However, this changes as soon as there are good reasons for simulations. Signal integrity is a good example of this change. There, you need simulations for high-speed systems. That was not an issue until people started building DDR memories or using ultra-high-speed SerDes with multiple lines. When the first serial I/O systems from Xilinx and Altera came on, then people understood. But until then, LVDS was the worst that could happen, and everybody made sure that they had to be careful and tiptoe around it, but it's okay and stable. They had no need to do the simulations.

If high-speed requirements reach a point where we have to take thermal issues into consideration, this will change as well. But aside from vacuum applications or high-power densities, I have not seen thermal simulations on a regular basis.

Matties: The other thing that Happy mentioned was a design for a robot or automated assembly. Is that a consideration that you consciously make when you're designing a board? When someone's coming in, do you go to the assembler, and do you collaborate in advance?

Löwer: It depends on the customer. Remember, I come into a company and hope that they have something like a design guide that I can read. That's not always the case, or it is not as detailed as I want it to be. Often, I have to talk to the engineers and ask what they are doing to get a feel for their way of designing. Design philosophies in different companies are quite different, and you can feel this as an engineer.

If you are in a bigger company, especially in automotive, you have pretty strict design guides, and they take care of all kinds of production issues. If you're doing 10 million devices for VW, for example, then you have to make sure that you know how to do those. They have special review boards where multiple people sit in a room, checking that each resistor is in the right alignment and that, for example, the soldering process is optimized for production. Automated production is an issue, depending on the company. On the other hand, companies that do 100 pieces a year don't care. It's much more work for them to optimize for automated production.

Matties: You have a unique perspective because you service a lot of companies. What advice would you give to an organization that's looking to contract with a designer for their work?

Löwer: Ask them for their ideas and listen to them—especially if it's a designer who has experience in different companies and parts of the industry. Usually, they can give you new and different perspectives and ideas on how to handle problems, issues, systems, and even processes.

Shaughnessy: Then, it's not just tribal knowledge because a lot of companies have their own way of doing it.

Löwer: Exactly. Many companies have their own way of designing. I am not talking about the respective process, but more like the style of the engineers, and they want to stick to it.

One of the saddest things I've seen was when I worked at a company for close to a year, and they had some new engineers coming in who were fresh from university. They were interested and curious to see a new world, but the first thing the older engineers made sure of was that they didn't make any stress. They did not do any crazy stuff. The young engineers learned a lot from them, but they did not make any trouble.

Shaughnessy: Recent grads are dying to try things out that they just learned, and they want to see if they can push the envelope.

Löwer: Yes. But depending on the company, it's hard to push that envelope—not because management is stopping you, but because the engineers are stuck in a mindset. That happens a lot. I don't know whether it's a typical German thing. In the olden days, when somebody learned a job or skill, such as carpentry, they had to go to other cities to learn from other masters so as not to be stuck in one master's ideas and ways of thinking. I wish that were true for students today.

Too often, I see that the students go to one company to earn some money during their studies, write their thesis there, and then after their degree, they go to a company for their first and last job. The company takes them because they know them, so they will get stuck in one kind of mindset. That kind of engineers will not change until they are forced to, and that is not good for the company.

Matties: Alex, I appreciate your time. Thank you.

Löwer: Thank you. DESIGN007

PCB007 Highlights

CES 2020: Here We Are Again ▶

Dan Feinberg shares highlights from probably the largest and most fascinating technology event globally—CES—which he has been attending and writing about CES for over 20 years. The progress that has taken place is immense, and the rate of change has also accelerated.

Punching Out! Preparing for Life Post-transaction ▶

Congratulations! You have punched out! Now what? Tom Kastner details how being prepared for life after a transaction is a good idea not only to help set up a smooth sale but also to give you the motivation to get through the deal process.

Rogers Highlighted High-speed Laminates and Next-gen Thin Materials at DesignCon 2020 ▶

Rogers Corporation exhibited at DesignCon in Santa Clara, California, highlighting some of its high-performance circuit materials used in multilayer structures which include a family of thin laminates, bonding materials, and sheeted copper foil options.

IPC Statement on U.S.-China Trade Deal ▶

In a statement recently, Chris Mitchell, IPC's VP of global government affairs, said, "The electronics manufacturing industry welcomes the U.S.-China 'phase one' trade deal signed recently in Washington, and the pathway it offers to the resolution of broader issues. The deal leaves many issues unaddressed, including cyber and structural economic reforms, and a broad range of increased tariffs that are still in place."

The Plating Forum: New Developments in ENIG ▶

ENIG has been around the printed circuit industry for more than 25 years. George Milad provides an update and explains how, although the occurrence of corrosion was recognized, a better understanding of the defect has led to a series of improvements over time.

Fresh Thinking on the Logistics of Laminate Distribution ▶

Mark Goodwin, COO of Europe and the Americas for Ventec International Group, sits down with Barry Matties to explain his approach to supply chain management, efficient distribution, and maintaining definitive product identity at every stage.

Logistics Are Frank Lorentz's Passion ▶

Recently, Barry Matties had the chance to visit Ventec International Group's German facility in Kirchheimbolanden (KIBO), where he met Frank Lorentz, Ventec's general manager for the location. Logistics are clearly Frank's passion. He lives and breathes it. If you spend any amount of time with him, that is abundantly clear. This interview with Frank was conducted after a tour of the KIBO site.

Ventec Builds Global Inventory of High-speed, Low-loss, High-frequency Materials ▶

In response to high demand and laminate material supply issues affecting some producers of low transmission loss and high-frequency laminates and prepregs, Ventec International Group Co. Ltd. (6672 TT) has implemented material inventory and supply chain measures with favorable order requirements.

You Can't Afford Not to Consider ISO 9001

Connect the Dots

by Bob Tise and Matt Stevenson, SUNSTONE CIRCUITS

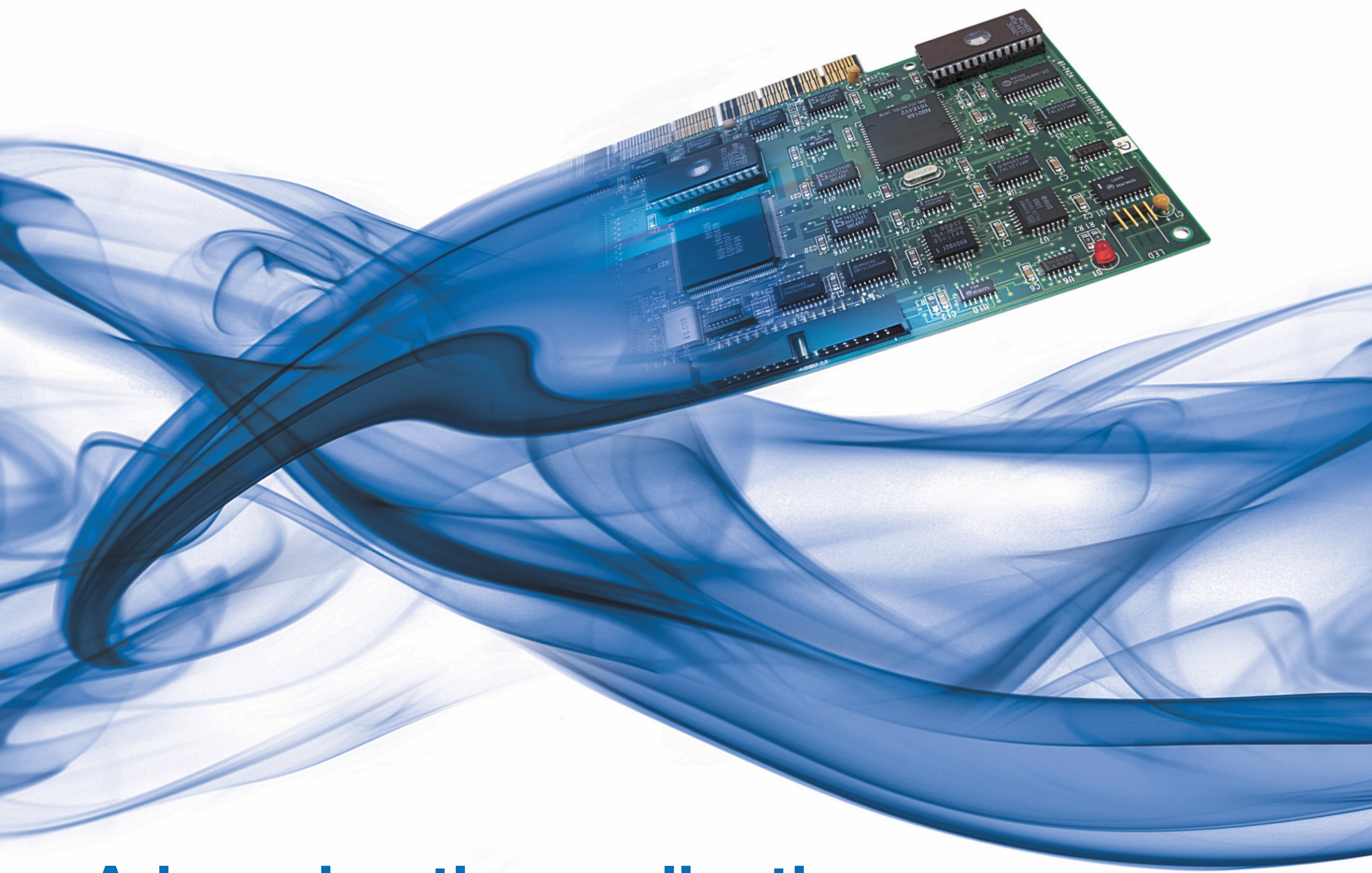
"Produce quality or die" may sound harsh, but for manufacturers in the electronics industry, it is true. The ability to consistently produce a quality product profitably is the baseline for business success; if you can't do it cost-effectively, then you can't innovate, develop new products, or open new markets. No one knows this story better than Nancy Viter, VP of operations at Sunstone Circuits.

Nancy was a key player in Sunstone's recent implementation of a certified quality management system (QMS) and a yearlong certification process. The International Organization for Standardization (ISO) 9000 family and ISO 9001 certifications are built on standards achieved with a QMS that addresses far more than "just quality." Done correctly, we learned that this standard provides a foundation for a robust business management system. It was a journey worth taking, and one we encourage you to consider. Fair warning: ISO adoption can be challenging at times.

Roadblocks can materialize quickly from the top of the organization chart as well as on the production floor. Lack of management commitment can stop a QMS initiative before it starts. Our company owners and officers indicated ISO was a top priority on day one. They made themselves available to employees, answered questions, and kept everyone engaged.

Absent guidance saying otherwise, employees may equate improved efficiency with a reduced workforce (loss of jobs) or elevated job standards (more "red tape"). Once all employees were comfortable that this endeavor would bring more business and address their pain points and make their jobs easier, it began to set the stage for acceptance and participation.

Moving forward with ISO certification is a big step for most businesses, especially for or-



Sunstone Circuits' Sample Quality Policy

The entire Sunstone Circuits team is committed to providing the highest overall experience for our customers. Through our dedicated focus on our employees, community, and environment, we are able to thoroughly understand our customers' needs and strive to profitably meet or exceed those expectations every time. We back up these comments with goals of greater than a 99% production satisfaction rating, 99% on-time delivery, and around-the-clock, daily customer support attained through continuous improvement and our "customer first" culture. Working toward these common goals is critical to ensure the effectiveness of our quality policy.

Figure 1: Review your quality policy and make sure it represents your organization.

ganizations that are already successful, growing, and profitable. An overwhelming menu of options exists for how to get certified, with offers for everything from procedural manuals to quality management software to on-site consultants. Our first question was, "Where do we start?"

Our journey began with an internal focus. Leadership decided what we wanted from ISO certification. We were not interested in just a badge for our website. To make the investment worthwhile, our goal was to create a system that would drive continuous improvement, grow the business, and improve profitability.

To start, we reviewed our quality policy. Did it represent what we truly believed? Once the team all agreed that these were words we lived by, we built our system to support every aspect (Figure 1).

We then considered how much we were going to invest in the process. What is the cost of quality? We set up a spreadsheet to compare "good spend" (activities like project planning, internal audits, 5S methodology, calibration, preventative maintenance programs, process improvement, training, management review, risk assessment, etc.) vs. "bad spend" (rebuilds, repair, equipment breakdown, overbuild, order holds, RMAs, etc.). Our goal was to spend less overall on quality activities and move the scale toward more proactive spend and less reactive spend. We then charted the progress on a monthly basis.

The costs associated with items like scrap and rework were not a surprise, but the deeper we peered into our own manufacturing process, the more opportunity for improvement we found in the "hidden" costs of quality. Some of the biggest opportunities are found in the places and people that we depend on most.

ISO 9001:2015 focuses a great deal on organizational knowledge, meaning information that is not documented; this is also known as tribal knowledge in most companies. For example, a customer called and asked our CSR to place a reorder for them. Our new employee was happy to help, and the order was placed in our ERP system, built, and delivered in 24 hours.

Unfortunately, the wrong revision was built due to changes made on the previous build and not properly documented to ensure the future builds would be correct. A senior associate would have known what questions to ask to ensure the proper order was placed. Can you relate?

Part of the ISO process involves ferreting out "stuff employees just know" and making it available to and useful for the broader organization, as well as a process for follow up and correction to address customer complaints to ensure it doesn't happen again to anyone else. This specific case required the documentation, training, and instruction that ISO standards provide.

We had to admit that there were many of these types of improvement opportunities that went well beyond the manufacturing floor. Hiring an outside consultant was critical to our success. We have a lot of experienced people on our team who have played a huge role in establishing Sunstone's reputation for delivering quality. A fresh perspective on our operation would ensure objectivity as we evaluated and improved our processes.

We reached out to the American Society for Quality (ASQ) for help choosing a consultant

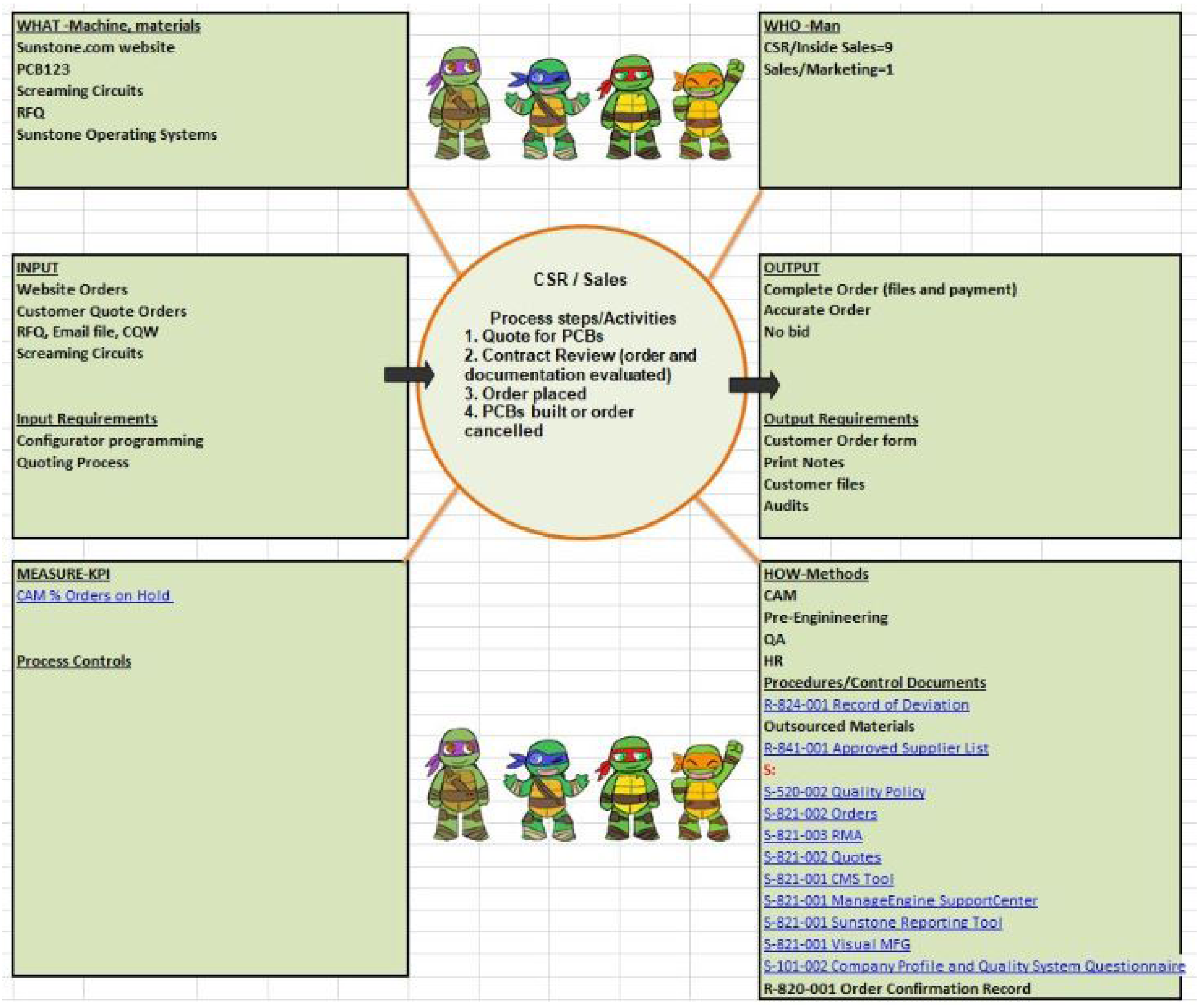


Figure 2: Have fun with your implementation! Our ISO-certified sales and ordering process with help from turtles (our process map), who also happen to be ninjas. While all processes must contain certain requirements, we allowed room for creativity on each process owner's map.

who could perform a gap analysis, facilitate the implementation process, help create the internal auditing program, and provide ongoing support. Most importantly, we found the right person for our organization—someone who fit with our management style and corporate culture.

Further, we assigned an internal coordinator. Consultants cannot do it alone. We believe you need a key person (or persons) on your team who is passionate about QMS, respected throughout the organization, and able to effectively communicate with each business unit and individual personnel as needed.

Your internal champion(s) will serve as a full-time facilitator during both the QMS implementation and the ISO certification process. Employees may view an outside consultant as a threat, and it is up to the internal champion to allay these fears, foster a positive attitude, and sustain the overall effort (Figure 2).

Once you've done these actions, celebrate your success! Have lunches for reaching milestones, and when the final audit week is completed, and you have reached your goal of certification, do something special to mark the occasion. Our team proudly wore "ISO strong" apparel to celebrate the accomplishment.

Figure 3: Don't let obstacles become permanent roadblocks.

The results have exceeded our expectations in terms of increased customer satisfaction, additional sales opportunities, and more efficient, happier employees. We are now seeing the fruits of our labor dropping to the bottom line allowing us to entertain additional certifications to satisfy customer requirements.

Remember to focus on continuous improvement and never stop the PDCA cycle: Plan, do, check, act! Obstacles do not have to become permanent roadblocks. ISO adoption is a challenge, but with management commitment, sound process, and good communication, ISO certification is both achievable and sustainable for any size organization (Figure 3). **DESIGN007**

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

Bob Tise

Matt Stevenson

OMRON Donates $10,000 to Support Girls in STEM

OMRON will donate $10,000 to the Robotics Education & Competition (REC) Foundation in support of its "Girl Powered" initiative, which provides teachers and mentors resources for co-ed or all-girl competitive robotics programs. The contribution from OMRON Foundation Inc. was boosted by matching funds generated from social sharing by attendees and followers of the 2020 Consumer Electronics Show (CES) in Las Vegas.

OMRON conducted fundraising efforts during CES 2020 to raise awareness and support girls in STEM. "OMRON supports the development of the next generation of leaders in science, technology, engineering, and math," said Nigel Blakeway, managing executive officer of OMRON Corporation. "We believe in the mission of the REC Foundation, and CES presents an outstanding opportunity for us to engage others in the cause."

The Robotics Education & Competition (REC) Foundation engages students in hands-on, affordable, and sustainable robotics engineering programs. The Foundation manages competitions for elementary school through college students, with more than 200,000 participating students from 57 countries.

OMRON committed dollars to the non-profit at the start of CES, and every tweet, retweet, or Instagram post during CES that included #OMRONforSTEM and #OMRONCES20, triggered an additional one-dollar donation, up to the maximum of $10,000.

"Studies show that boys and girls perform similarly in STEM, but girls are less likely to consider careers in STEM," said Kate Cramer, an automation engineer from OMRON Automation Americas. "Together with the REC Foundation, we want to inspire girls to embrace STEM learning and a potential career by developing their passion for engineering and robotics early. Ultimately, they may become my colleagues."

(Source: OMRON)

Encapsulation Resins: What Could Go Wrong?

Sensible Design

by Alistair Little, ELECTROLUBE

In last month's column, we looked at the pitfalls of potential contaminants and best mixing practices for two-part encapsulation resins. This month, I'll address some of the pain points with mixing resin packs and air bubbles, including what can go wrong and why. I will also examine some of the key differences between conformal coatings, encapsulation resins, and potting compounds to help designers make decisions that are more informed, ultimately increasing the reliability and lifetime of your electronic circuitry.

Extending lifetime and improving the performance of devices is essential to a brand's reputation; no one appreciates a product recall, or worse, a product failure. Encapsulation resins are an excellent way of protecting electronic circuitry; however, there are some excellent conformal coating alternatives on the market too. I will elaborate further to help dispel any confusion.

For instance, we have created the 2K coatings range, which behaves like a resin with the application ease of a coating. This has been a great success in applications, such as automotive, where protection in harsh environments is critical. I will explore coatings vs. resins, as well as resins vs. potting compounds, using my usual five-point Q&A format. Let's take a closer look at five critical factors affecting encapsulation resins.

1. What are the consequences of having air bubbles trapped in encapsulation resins, and does this impair performance?

Air bubbles can have a number of effects on the performance of the cured resin. Depending on the number and distribution of the bubbles, the thickness of the polymer layer applied will be decreased. The level of protection will also be reduced, particularly against chemical attack.


APCT
Passion | Commitment | Trust
Does Offshore Manufacturing
Make You Want To Run & Hide?
Don't Bury Your Head In The Sand – Look To The Experts at APCT Global
APCT GLOBAL
Contact APCT Global Today
203.284.1215 | APCT.com
Global Printed Circuit Board Solutions
APCT.com
APCT Santa Clara HQ
408.727.6442
APCT Anaheim
714.921.0860
APCT Orange County
714.993.0270
APCT Wallingford
203.269.3311
APCT Global
203.284.1215

If there are air bubbles next to components, wiring, or tracks, then particularly when high voltages are applied, corona can build up inside these voids and ultimately destroy the components or wiring. Voids also act as a weak point for thermal and physical shock, which can lead to the resin cracking in service.

Bubbles in optically clear resins not only look unattractive but will also distort the light passing through the resin either by the changes in the refractive index or by diffusion of the light. Bubbles on the surface of the resin may break over time, with dirt and detritus collecting in the resulting crater.

2. What are some key differences between resins and coatings?

The most noticeable differences are the methods of application, such as aerosol, conformal coating spray equipment, manual spray gun, and brushing for coatings. This is compared to mixing and dispensing equipment and resin packs for resins, thickness of application (< 100 microns for conformal coatings, < 500 microns for thick coatings, and > 500 microns for resins), and approval ratings (coatings are generally approved to UL746, whereas for resins, it heavily depends on the application).

Because of the coating thickness, coatings occupy less space and have a low increase in overall weight compared to resins. There are both coatings and resins based on epoxy, polyurethane, and silicone chemistries, but there are also acrylate, acrylic, and parylene coatings that do not have a direct resin equivalent. Ninety-nine percent of resins are 100% solid systems, meaning they have low, or no, VOCs released during curing, while many coatings are solvented.

However, it is important to mention that there are two-component (2K) and UV-curable acrylate systems with 100% solids available. These 2K coatings, in particular, can be applied up to 500 microns thick without cracking during thermal shock testing and enable a greater degree of component lead coverage to be achieved, resulting in improved performance during thermal shock, powered salt-spray testing, MFG testing, and condensation testing (i.e., traditionally gruelling test regimes that are commonly used during automotive qualification campaigns). The 2K series is also VOC-free, solvent-free, and fast curing.

3. Why would I choose a resin instead of a coating?

The choice between a resin or a coating is usually down to application specifics. If the unit involved is to be subject to long-term immersion in various chemicals, continuous thermal, and/or physical shock cycling, then a resin is generally preferred. Additionally, if there are a large number of big components on a PCB, it is generally better to use a resin to encapsulate these than to coat them. Further consideration should be given if the unit is to be used in a situation where it is not easily accessible or if a long continuous service life is required, in which case a resin would be recommended to provide the extra protection and durability needed.

4. Are encapsulation resins and potting compounds basically identical, or are there differences?

There are no real differences between the two in terms of the resins used, but there is a difference in what the resin is required to do. An encapsulation resin will totally cover the PCB and the components and can act as the protective support structure, while a potting compound is used to fill a housing or enclosure containing the PCB and components.

An encapsulation resin will adhere to the PCB and the components, and its outer faces will act as the primary barrier to protect the unit. However, a potting compound has to adhere not just to the PCB and components, but also the housing. In this case, the differences in CTE (coefficient of thermal expansion) between all the materials used can become a critical factor, as the resin will be subjected to different rates of expansion and contraction due to the materials concerned, as well as its own CTE. This can put a resin under extreme stress and, over time, lead to failure.

5. What is the most effective method for mixing a resin pack, what can potentially go wrong (and why)?

First, remove the resin pack from the outer packaging. In the case of polyurethane and silicone resins, do not remove from the foil outer pack until ready to use. Lay the pack out onto a flat surface and remove the centre clip. Use the clip to push the resin from one half of the pack to the other, then pick the pack up in both hands and mix in a circular motion for a couple of minutes.

Place the pack back on the flat surface and use the clip to push the resin from the corners into the centre. Pick the pack up in both hands and continue mixing for a further minute. Repeat pushing the resin from the corners into the centre of the pack, and then continue to mix for another minute to ensure that the material is fully mixed and is a uniform colour.

Use the clip to push the mixed resin to the side of the pack with the angled seam. Roll the pack film up so that the pack fits into the hand. Cut the corner of the pack off and then tilt the pack to dispense the resin, applying slight pressure as required to maintain the flow. If the material from the corners of the pack is not pushed into the centre of the pack, then unmixed material can be dispensed.

If the resin is not mixed sufficiently, it may not cure, or have a patchy cure. In the case of filled resin systems, some sedimentation might have taken place over time, so it may take a little more mixing to ensure that the fillers are correctly distributed throughout the resin. With the optically clear resins, when first mixed, the resin will appear hazy. This is perfectly normal, and the haziness will disappear as the material cures.

Conclusion

I hope the points covered this month have been helpful. Look for my next column, where I'll cover more issues on encapsulation resins to get the best protection for your circuitry requirements with the least amount of aggravation. In the meantime, please contact me if you have any questions. DESIGN007

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

Intelligence-aided Manufacturing From Ucamco

During IPC APEX EXPO 2020, I-Connect007 technical editor, Pete Starkey sat down with Luc Maesen, director of Ucamco. Luc explains how AI has been applied in the automation of the CAM/CAD process, resulting in the company's iamcam system, which enables intelligence-aided manufacturing designed to lower cost while achieving fewer errors and faster delivery.

Click on the image to view this video.

PCB Materials for High-power RF Applications

Lightning Speed Laminates

by John Coonrod, ROGERS CORPORATION

Most commercial PCB-based applications that use high power are typically associated with cellular base station technology; however, there are other applications. There are also several things to consider when working with high-power RF applications. This column will concentrate on PCB-based power amplifiers used in base station applications, but the basic concepts discussed here will apply to other high-power applications.

Most high-power RF applications will have issues with thermal management, and there are a few fundamental relationships to consider. One relationship is the loss-heat relationship. A circuit with higher loss will cause higher heat to be generated when high RF power is applied. Another issue is frequency heat, which causes more heat to be generated at higher frequencies. Additionally, any dielectric material that is subjected to an increase in heat will have a change in Dk (dielectric constant), and that is the thermal coefficient of dielectric constant (TCDk). There will also be a change in insertion loss with a change in temperature. These changes in Dk due to TCDk can impact the RF circuit performance and could be problematic for the application.

There are multiple materials and PCB properties that should be addressed for the loss-heat relationship. Sometimes, when a designer chooses a low-loss material for a PCB application, they will only consider the dissipation factor (Df, or loss tangent). The Df is related to the dielectric losses of the material; however, a circuit will have other losses. The overall loss of a circuit, as it relates to RF performance, is

insertion loss. Insertion loss is made up of four other losses and is a summation of dielectric loss, conductor loss, radiation loss, and leakage loss.

A circuit using a very low loss material with a Df of 0.002 and very smooth copper will have relatively low insertion loss. The same circuit using the same low-loss material, but using a high-profile electrodeposited (ED) copper instead of smooth copper will lead to an increase in insertion loss.

Insertion loss is made up of four other losses and is a summation of dielectric loss, conductor loss, radiation loss, and leakage loss.

Copper surface roughness will have an impact on the conductor loss of a circuit. To be clear, the surface roughness that is a concern for losses is the copper surface roughness at the copper-substrate interface as the laminate is made. Additionally, if the circuit substrate is thin, the copper planes will be closer together, and the copper surface roughness will have a larger impact on insertion loss compared to a circuit using a thicker laminate.

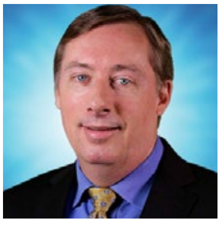
For high-power RF applications where thermal management is typically an issue, choosing a laminate with low Df and smooth copper can be advantageous. Additionally, choosing a laminate with high thermal conductivity is generally a smart thing to do as well. The high thermal conductivity will assist in moving the heat more effectively out of the circuit and into the heat sink.

The frequency-heat relationship basically causes more heat to be generated when there is an increase in frequency, with the assumption of the same RF power being applied at both frequencies. As an example taken from some thermal management experiments done at Rogers, we found that a microstrip transmission line with an applied RF power of 80 watts at 3.6 GHz had a heat rise of ≈50°C. When that same circuit was tested with 80 watts applied at 6.1 GHz, the heat rise was ≈80°C.

There are several reasons for having an increase in temperature with an increase in frequency. One reason is the Df of a material will certainly increase with an increased frequency, which will cause more dielectric losses and will ultimately cause an increase in insertion loss and heat. Another issue is the fact that conductor losses naturally increase with an increase in frequency. Some of the conductor loss increase is due to a thinning of skin depth as frequency increases. Additionally, with increased frequency, the fields will condense, and there will be more power density in a given area of the circuit, which will also increase the heating effects.

Lastly, TCDk, which has been mentioned several times in this column previously because it is a material property that is often overlooked, is basically how much the Dk will change with a change in temperature. In the case of power amplifier circuits, they often have ¼ wavelength matching networks, and these networks are sensitive to Dk fluctuations. When the Dk changes greatly, the ¼ wavelength matching will shift, and the power amplifier can vary in efficiency, which is very undesirable.

In summary, when selecting high-frequency materials for high-power RF applications, the material should have low Df, relatively smooth copper, high thermal conductivity, and a low TCDk. There are many tradeoffs when considering these material properties, along with the final end-use application requirements. Therefore, it is always wise for the designer to contact their material supplier when choosing materials for high-power RF applications. **DESIGN007**



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

MilAero007 Highlights

Defense Speak Interpreted: What in the World Is MINSEC?

The Defense program designated MINSEC (Microelectronics Innovation for National Security and Economic Competitiveness) is probably one that you have never heard of but will likely gather more headlines in the future. Dennis Fritz explains.

AT&S Now in the Top League of Aircraft Suppliers

With the NADCAP accreditation, the Austrian high-tech company has been recognized as a supplier of the highest quality standards in aircraft construction. The accreditation is now opening new doors in the aerospace sector.

The Big Picture: Globalization—Tariffs and Alternate Sources

The new year is upon us, so Mehul Davé started thinking about the main challenges his company and customers are facing as we enter 2020: tariffs and finding alternate sources for PCBs. Mehul shares his thoughts.

NASA's Newest Astronauts Ready for Space Station, Moon, and Mars Missions

NASA welcomed 11 new astronauts to its ranks, increasing the number of those eligible for spaceflight assignments that will expand humanity's horizons in space for generations to come. The new astronauts successfully completed more than two years of required basic training and are the first to graduate since the agency announced its Artemis program.

A New Tool for 'Weighing' Unseen Planets

A new instrument funded by NASA and the National Science Foundation called NEID (pronounced "NOO-id"; sounds like "fluid") will help scientists measure the masses of planets outside our solar system—exoplanets—by observing the gravitational pull, they exert on their parent stars. That information can help reveal a planet's composition, one critical aspect in determining its potential habitability.

The World of PCBs: Anything but Boring

Andy Shaughnessy had the chance to catch up with Megan Teta, CID+, product manager of design and education services at Insulectro. Megan explains why she's excited to become more involved in training and why the world of PCBs is anything but boring.

I-007e Micro Webinars Releases Part 2 in 'Coatings Uncoated!' Series

The second part of the popular webinar series, Coatings Uncoated, is now available to view. Author of The Printed Circuit Assembler's Guide to Conformal Coatings for Harsh Environments and topic expert Phil Kinner from Electrolube shares highly focused educational information on conformal coating and encapsulation.

The Government Circuit: IPC Marks Progress in Government Advocacy in 2019

As we turn the calendar from 2019 to 2020, it is a natural time to take stock of the past year and look ahead to the year to come. Here are a few of the top stories of 2019 from Chris Mitchell's perspective as a government relations professional working to support the electronics manufacturing industry through public policy advocacy.

Zentech Manufacturing Acquires Trilogy Circuits

Zentech accelerates effort and strategy to build leading electronics manufacturing services (EMS) provider for defense, aerospace, medical, and other high-reliability industries.

Flex Is Heating Up

What the Flex?

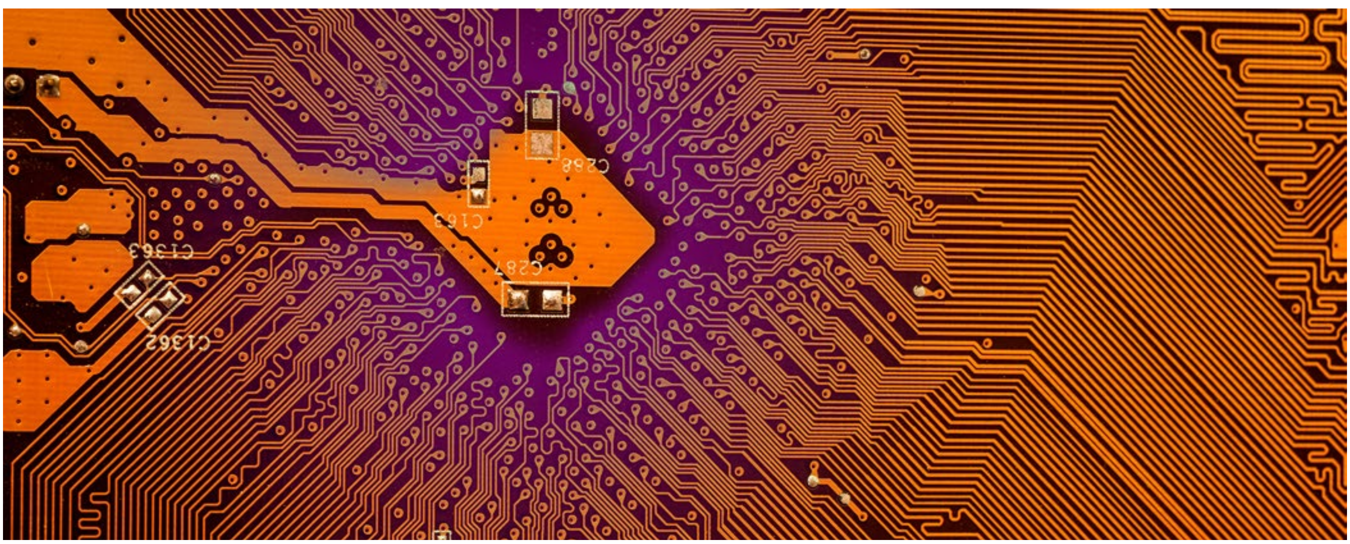
by Andy Shaughnessy, I-CONNECT007

It has been a busy few weeks for I-Connect007, and if the first few trade shows of this year are any indication, it's going to be a good year for our industry and for flex designers and manufacturers.

This year, the calendar was our friend, with DesignCon and IPC APEX EXPO falling one week apart in Silicon Valley and San Diego, respectively. After DesignCon was over, I stayed on the West Coast until IPC APEX EXPO began. When it's freezing in Atlanta, there's no sense in coming back home for three days.

DesignCon and IPC APEX EXPO both had good attendance numbers, despite fears that the coronavirus outbreak would take its toll. Very few attendees wore masks at either show, but a few Americans, who live primarily in China, told me it would be summer before they returned home.

On the show floor, I was struck by the number of exhibitors who were using virtual reality in one form or another to demo their latest solutions. Some companies used the "old-school" headset, while others offered 360° views of their machines via a tablet, with no headset required. This is a great idea; anything that we can do to interest young people in this industry is a positive move.



One thing I noticed at both shows was that the flex exhibitors were busier than ever, with attendees checking out the latest flex and rigid-flex technology. Formerly "boutique" applications, such as metal-backed flex with aluminum added as a heat sink in non-flexing sections, are now gaining mainstream acceptance.

Further, the divide between printed electronic circuits and standard flexible circuits continues to shrink. I learned about a Texas company, NovaCentrix, that has made great strides in wearable technology. NovaCentrix, the inventor of photonic curing, has figured out a way to solder parts onto textiles, including regular clothing. I need to find out more about this; it seems almost impossible, given the heat required to melt solder. (As Burt Rutan pointed out in his keynote at IPC APEX EXPO, for a truly great idea, at least 50% of your team should believe the objective is impossible. This one fits the bill.)

Rigid-flex was in abundance at both shows. This combo once made up just a few percents of overall flex circuits, but it's now nearly ubiquitous and continues to gain in market share, especially among handheld devices. More and more OEMs are also moving into complex multi-board rigid-flex circuits, often for reasons related to space-saving and reliability. This month, we bring you a feature article, "Multi-board Etching: Managing Rigid-flex Designs and Conductivity" by Hemant Shah of Cadence Design Systems. We also have a great column by our regular contributor Joe Fjelstad, "Power and Thermal Management: Dealing With the Heat."

The economy is humming along, and our industry is still expanding—slowly but surely. So far, 2020 is looking like a great year for flex and the whole industry. Let's hope that we can get the coronavirus under control. **FLEX007**

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

Gift Will Allow MIT Researchers to Use Artificial Intelligence in a Biomedical Device

Researchers in the MIT Department of Civil and Environmental Engineering (CEE) have received a gift to advance their work on a device designed to position living cells for growing human organs using acoustic waves. The acoustofluidic device design with deep learning is being supported by Natick, Massachusetts-based MathWorks, a leading developer of mathematical computing software.

"One of the fundamental problems in growing cells is how to move and position them without damage," says John R. Williams, a professor in CEE. "The devices we've designed are like acoustic tweezers."

Inspired by the complex and beautiful patterns in the sand made by waves, the researchers' approach is to use sound waves controlled by machine learning to design complex cell patterns. The engineers developed a computer simulator to create a variety of device designs, which were then fed to an AI platform to understand the relationship between device design and cell positions.

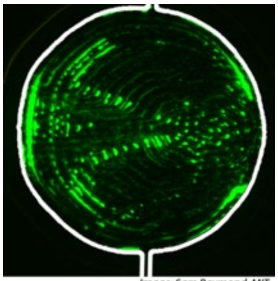
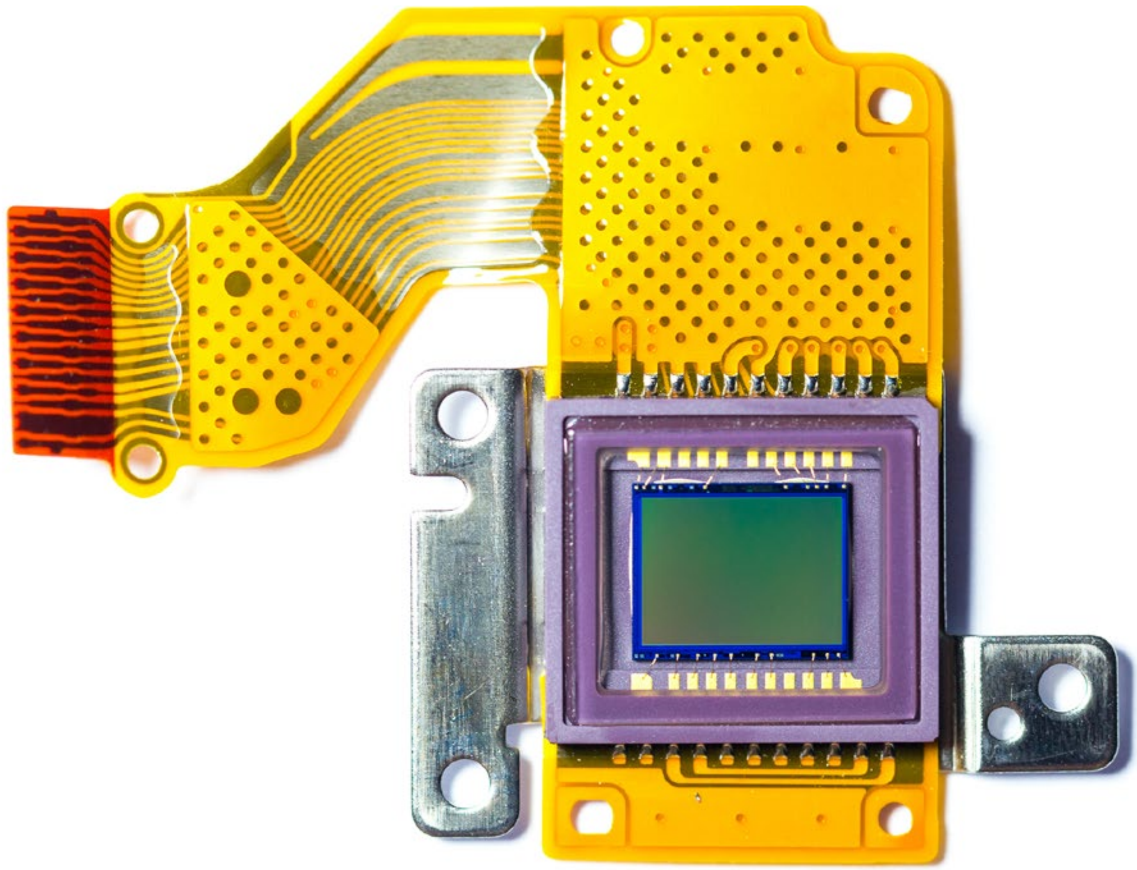


Image: Sam Raymond, MIT

"Our hope is that, in time, this AI platform will create devices that we couldn't have imagined with traditional approaches," says Sam Raymond, who recently completed his doctorate working with Williams on this project. Raymond's thesis title "Combining Numerical Simulation and Machine Learning" explored the application of machine learning in computational engineering.

"MathWorks and MIT have a 30-year long relationship that centers on advancing innovations in engineering and science," says P.J. Boardman, director of MathWorks. "We are pleased to support Dr. Williams and his team as they use new methodologies in simulation and deep learning to realize significant scientific breakthroughs."

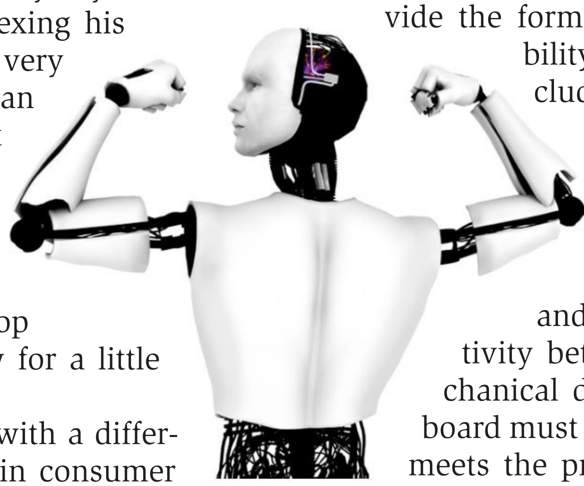
(Source: MIT)



Multi-board Etching: Managing Rigid-Flex Designs and Conductivity

FLEX007 Feature by Hemant Shah
CADENCE DESIGN SYSTEMS

When I was a very young boy, my dad would delight and tease me by flexing his muscles. He did this with a very special flourish. Rather than simply flexing, he would put his thumb in his mouth, blow, and flex, which gave the appearance that huffing and puffing on his thumb made his bicep pop out. This was a great show for a little boy.



In PCB design, we work with a different type of flex. Advances in consumer and industrial electronic devices have allowed manufacturers to produce smaller devices that have more functionality.

Those devices use rigid-flex designs that provide the form factors needed for portability and—many times—include multiple interconnected boards. With all the interconnections between the boards, PCB designers pay equal attention to signal paths and to the electrical connectivity between boards. On the mechanical design side of things, each board must fit within an enclosure that meets the product specifications. As a

result, every multi-board design combines electronic design automation (EDA) with mechanical engineering design (MCAD).

While thermal conductive layering under floors may differ in size to your boards, considerations are similar.

The Sum Is Greater Than the Whole

Good troubleshooting techniques involve considering a system as individual parts rather than as a whole. The same techniques apply to your work with multi-board PCB designs. Each board consists of a single unit that has its own lifecycle.

Some product designs may use a single PCB design for multiple functions or for multiple devices. Others may interconnect multiple PCB designs to produce a complete, fully functional system. Depending on the product design, a single PCB design may have multiple uses. No matter the method, the multi-design represents an overall approach to system design.

Multi-board designs feature rigid-flex PCB technologies. Combining flexible circuits and rigid circuits extends the benefits of individual technologies. While rigid PCBs carry most of the components, flexible PCBs serve as interconnecting pieces between the rigid boards. In addition, the combination of rigid and flex reduces complexity and cost, saves weight, and improves reliability.

Let's Be Flexible Here

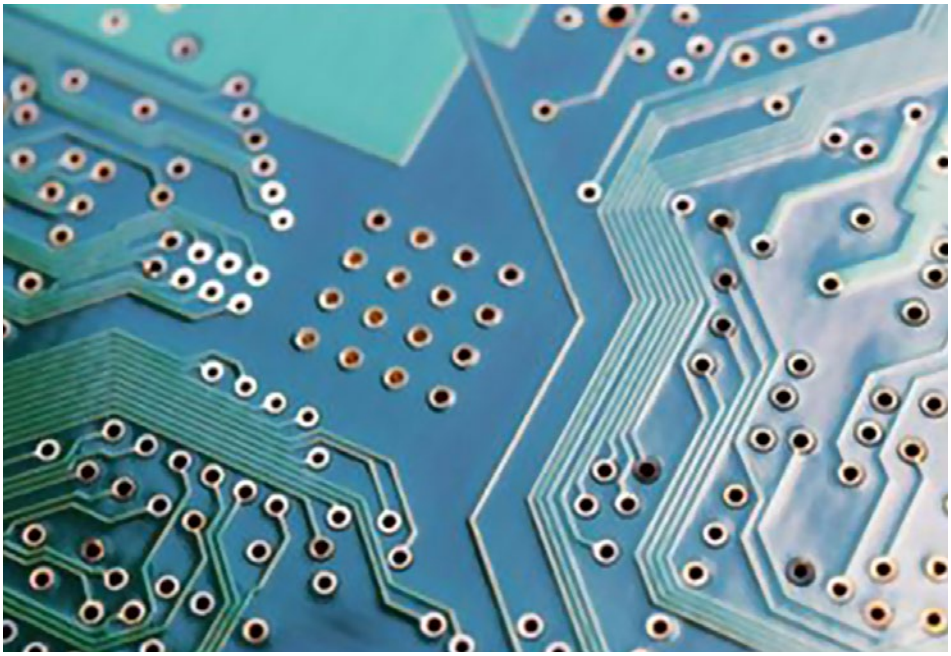
While some applications use static flexible circuits that have minimal movement, other products use dynamic, flexible circuits that move frequently. The two types of flexible circuits use different types of materials and require different methods for construction. For example, dynamic, flexible circuits require a much longer flexing lifespan and require elongated grains in the copper.

As we consider the construction of flexible circuits, we begin with a stackup of flexible substrate material and copper. A combination of adhesives, heat, and pressure laminates the materials together.

The substrate consists of a polyimide, or a strong, flexibly thermosetting polymer. PCB manufacturers use different types of polyimides based on application requirements. Given the need for flexibility, the circuits rely on rolled and annealed copper along with acrylic adhesives that combine softness with higher coefficients of thermal expansion.

The different types of flex circuits include the Type 1 single layer with one conductive layer sandwiched between two layers of polyimide, the Type 2 double layer that has two conductive layers, and the Type 3 multilayer flexible PCB that uses three or more conductive layers. The rigid part of a rigid-flex circuit uses standard PCB design and manufacturing processes.

Along with prepreg, the board typically consists of an FR-4 substrate, conductive copper layers, the solder mask, and identifying information. Type 4 multilayer rigid-flex boards have three or more conductive layers. Because most designs feature rigid sections that have a different set of layers than the flexible sections, multilayer rigid-flex designs do not have a consistent set of layers across the entire design. The circuit consists of the inner layers of multiple circuits attached together with adhesive.



Etched in Memory Forever

A multi-board circuit combines the attributes of rigid and flexible circuits. Given these attributes, the circuit offers the advantage of allowing circuits and components to fit into limited spaces. In addition, the use of rigid-flex technologies reduces plug and connector components. Because multi-board circuits have applications across a wide range of industries and applications, the circuits often feature dense, fine line trace width and spacing. The circuits also have complex layout options that meet the design requirements of specialized applications.

Satisfying those requirements requires routing software that provides contour routing for curved, flexible circuits. Contour routing eliminates any imprecise spacing and allows design teams to specify uniform spacing for traces.

The unique attributes of multi-board circuits also require adjustments in traditional manufacturing processes. While rigid-flex PCBs follow most of the traditional PCB design processes in terms of the use of design documentation, the printing of outer and inner layers, the substrate, and the removal of copper, the manufacturing processes can become more complex. For example, etching the conductive layer of a flexible circuit can cause the polyimide core to shrink and may force manufacturers to use materials that compensate for the shrinkage.

Traditional etching practices also offer challenges for the production of the fine lines and line width tolerances seen with multi-board designs. The need for precision requires greater attention to etch rates, the reduction of pools, and higher etching efficiency. With the growth in multi-board applications and the use of rigid-flex technologies, manufacturers have incorporated features into standard mechanical PCB operations that accommodate etching of traces that have spacing as low as four mils. As processes and technologies emerge, PCB manufacturers have begun using other technologies such as laser machines for small flexible circuit etching.

Laser technologies offer the precise positional accuracy and depth-of-cut calibration needed for etching complex designs and or working with unique flexible polyimide materials. A 20 µm diameter focused laser beam easily handles smaller traces and spacing requirements. Laser systems use hatch and controlled delamination processes for removing copper from double-sided designs. While laser etching systems place less stress on flexible boards, the adhesive layer of a flexible circuit absorbs the laser energy. **FLEX007**

Hemant Shah is group director of product management at Cadence Design Systems. This article appeared originally as a Cadence Design Systems blog item.

Flex007 Highlights

DuPont Introduces New Pyralux and Riston Products ▶

DuPont Interconnect Solutions has announced that it is introducing several new products across its DuPont™ Pyralux® and Riston® product families to address a diverse set of needs for both the manufacture and performance of advanced electronics devices.

DuPont Interconnect Solutions Expands Insulectro's Role in North America ▶

DuPont Interconnect Solutions announced recently that it is expanding Insulectro's role in North America by adding Canada to its sales territory for DuPont™ Pyralux® flexible circuit materials.

Flexible Thinking: Additive Manufacturing of PCBs ▶

We are seeing increasing interest in technologies that will allow one to make electronic substrates in near real-time using additive processing techniques and 3D printers. It is a true game-changer in product development. The surge in interest in additive manufacturing technologies shown in recent times—as indicated by the significant increase in published articles and press releases—suggests that the electronic interconnection manufacturing industry could be on the verge of a manufacturing renaissance.

Toyochem Develops Highly Flexible EMI Shielding Film For High-speed FPCs ▶

Toyochem Co., Ltd., a member of the Toyo Ink Group, has launched a highly flexible electromagnetic interference (EMI) shielding film, the LIOELM™ TSS510-HF, designed to meet the demanding performance requirements of 5G flexible printed circuits (FPCs).

EPTE Newsletter: Electroless Plating for Flexible Circuits ▶

Electroless plating is a popular method in manufacturing printed circuits or flexible circuits. The electroless plating concept is not new; it is used to protect conductors against oxidation. However, it is not the default process for printed circuit manufacturing. Dominique Numakura explains.

DuPont and SCHMID Announce Partnership For New PCB Plating Applications ▶

DuPont Electronics & Imaging and SCHMID Group have announced that they have entered into a nonexclusive joint development agreement to explore new PCB plating applications to bring advanced innovations to their global customers. The new solutions which will include equipment and chemistry are expected to be launched in late 2020.

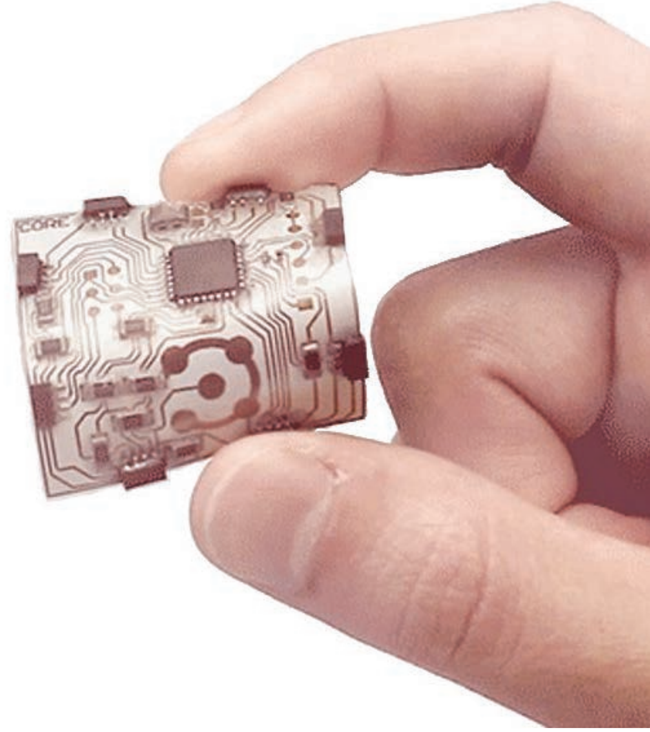
SEMI-FlexTech Launches Six New Projects to Accelerate Flexible Hybrid Electronics Innovation ▶

SEMI-FlexTech has announced the launch of six projects to accelerate sensor and sensor system innovations for new applications in industries including healthcare, automotive, industrial and defense.

Call For Papers Printed Electronics Europe: Flexible Structural Hybrid ▶

IDTechEx is now accepting applications to present in May at Printed Electronics Europe 2020 in Berlin, Germany. Tell us about your new concepts, technologies, materials and applications. Successful submissions will be speaking alongside global experts flying in from all over the world.

DUPONT®



PYRALUX® FLEXIBLE CIRCUIT MATERIALS

PYRALUX® AP
high reliability flex & multilayer circuitry

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

PYRALUX® HT
for high temp, high performance flex

PYRALUX®TK
Teflon®/Kapton®

PYRALUX® LF
industry standard for high reliability

PYRALUX® FR
flame-retardant for products requiring UL rating

PYRALUX® APR
reliable resistor, temperature tolerant

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

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

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

PROUDLY DISTRIBUTED BY
INSULECTRO
20362 WINDROW DRIVE • LAKE FOREST, CA 92630 • 949.587.3200 • INSULECTRO.COM

Power and Thermal Management: Dealing With the Heat

Flexible Thinking

by Joe Fjelstad, VERDANT ELECTRONICS

Without power, electronics are useless. With power, miracles happen. Managing that power is critical in both design and operation in terms of heat generation and energy conservation, especially for battery-powered devices. Moreover, often in electronic products, designers find themselves providing power to an electronic module or system at multiple different voltages and currents. This brings to fore one of the things that is often underappreciated until after the fact, and that is power's omnipresent by-product—heat.

Heat is a reality in the operation of every electronic device, no matter the size. The amount of heat produced by some devices, such as digital watches and small calculators, may be nearly immeasurable, but where there is resistance, there is also heat generated. However, for larger and more powerful electronic devices and systems, heat is much more apparent; anyone who has used a modern laptop on their lap for any length of time has likely felt the accompanying discomfort.

With the industry's unceasing effort to increase product performance by shrinking transistors and circuits, an unfortunate increase in thermal energy densities on ICs has resulted. Today, the matter of elevated temperatures on the chip is an increasingly important issue for chip, package, and system designers for a very important reason: there's an inverse relationship between long-term reliability of electronics (specifically transistors) and higher temperatures.

The simple fact is that as the chip gets hotter and/or spends more time at an elevated temperature, the reliability of the IC die, and the product in which it is used, tends to worsen. This is due in large part to the shrinking material gap for the diffusion of metals to an inevitable short and failure as transistors shrink with each new node, which is now on the cusp of 4–5 nanometers. Earlier transistor nodes could be expected to last for decades or even centuries; today, the expectations can be measured in a few years or even months. Armed with this knowledge, technologists have directed significantly more attention toward the thermal management of electronic systems. Once an afterthought, management of the thermal effects is increasingly moving up in the design process.

When it comes to the task of cooling, two, staged modes of thermal transfer are used: primary thermal transfer modes and secondary thermal transfer modes. The primary modes are generally based on conduction, the first of which is direct thermal transfer normally through a solid conductive material as a metal since metals are generally good thermal conductors. However, they can vary widely in terms of their thermal conductivity.

To interface with the device requiring heat removal and the feature that affects that removal, a thermal interface material (TIM) may be used. These specialty materials bridge the gap between the two surfaces completely to assure there are no "hot spots" or points where the heat generated by the device would otherwise be excessive. Moreover, they often serve to mitigate the difference in rates of thermal expansion between the heat source and the heat removal device.

Figure 1: A high-performance CPU cooler. (Source: Cooler Master)

The method of heat energy transfer to the local environment for most electronics is by convection. (Thermal engineers frequently use the axiom, "At the end of the day, the heat all goes back to air.") An example of a conventional release structure would be a finned metal which acts as a radiator. This can be done in concert with a heat transfer accelerator, such as a heat pipe, a sealed system with microchannels and a fluid.

When one end of the heat pipe is placed on a hot surface (e.g., an IC chip), there is the evaporation of the internal liquid at the interface with the chip and cooling and condensation at the distal end. A heat pipe offers much better thermal performance than solid metal and can be very low-profile (~0.5 mm), self-contained hollow metal device filled with a liquid that cycles from liquid to gas and condenses back to liquid to remove more heat from the device.

If you have ever looked at a recent computer motherboard, you will see a finned metal device that looks like it was designed more for a modern Formula 1 racing car than a computer motherboard (Figure 1). This high-performance CPU cooler has a horizontal vapor chamber, eight heat pipes, and can reportedly dissipate up to 250 watts from its microprocessor-sized footprint—think cooking stove heat densities.

While voltages and operating currents have been dropping steadily, due to shrinking transistor sizes, watt densities have been increasing dramatically over the last decade because of the huge increase in transistor counts on some ICs that in some applications exceed one billion. One billion transistors times billions of on-off switches, and even a minuscule amount of energy per cycle, adds up.

Summary

In summary, electronics continue to find applications in ever-increasing areas of daily life. Power management by design, both before and during use, is going to be critical. When making decisions, product designers need to consider many relevant and important issues. Power management will likely only increase in importance. In the future, it will be more necessary to think about managing the thermal aspects of power usage upfront rather than after the fact. Also, microprocessor cooling structures will resemble cooling systems used on high-performance racing cars. FLEX007

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

Thinfilm to Energize Innovation in the Wearables and Sensor Markets With Ultrathin, Flexible, Safe Batteries

Thin Film Electronics ASA ("Thinfilm") has updated its corporate strategy focused on the design, development, and production of innovative battery solutions targeting existing market demand with differentiated solutions to power wearable devices and connected sensors. This decision follows extensive evaluation of multiple markets to determine the most commercially compelling use of the company's patented process technology innovations and state-of-the-art, production-scale roll-to-roll flexible electronics factory located in San Jose, California.

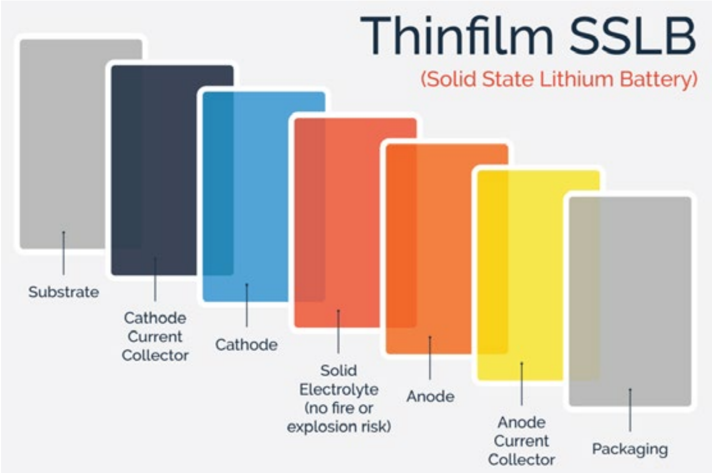
Thinfilm battery solutions incorporate an innovative solid electrolyte material that enables thinner, stackable cells that can endure more charging cycles and deliver more power at sub-freezing temperatures, compared to commonly used battery technologies. Because the solid electrolyte cannot catch fire or explode, Thinfilm SSLBs can also improve the safety profile for wearable and medical applications.

Initially pioneered by Oak Ridge National Laboratory in the 1990s, SSLB technology is primarily used in embedded electronics applications including real-time clock and SRAM backup.

By leveraging its core capabilities in materials and manufacturing innovation, the Company believes it can produce compelling energy storage products that provide greater battery life and improved reliability, with the form-factor flexibility to create unique battery shapes enabling sleek, comfortable end products. The Company will initially focus on key portions of the wearables and sensor markets, particularly the rapidly growing connected and wearable medical sensing market, in which continuous glucose monitoring alone is forecasted to double in volume to over 100 million units by 2023, according to IDTechEx. Beyond wearable medical sensing, Thinfilm has identified a number of additional growing applications in existing markets that are expected to provide meaningful opportunities for additional growth.

To accelerate the development of ultrathin battery technology in the San Jose factory, Thinfilm has entered into a partnership with a leading process technology development company.

(Source: Thin Film Electronics ASA)



Editor Picks from PCBDesign007 and Flex007

1 Lady Gaga's Keyboardist Designs a Circular Piano ▶

Who says a modern piano has to be a certain shape? Not Brockett Parsons, Lady Gaga's keyboardist. He worked with Dave Starkey and Chuck Johnson to create PianoArc—a custom-designed 360-degree circular keyboard and a company with the same name. I spoke with Brockett and Dave at AltiumLive to learn more about the design efforts that went into this fascinating instrument.

2 Beyond Design: The Big Bang—Lumped Element to Distributed System ▶

The simplistic approach to analyzing electronic circuits is to use the lumped element model. However, in reality, that is not the case. Barry Olney discusses the difference between the lumped element model and the distributed system.

3 Dana on Data: Creating IP-protected PCB Design Rules ▶


Dana Korf

One of the primary reasons that data packages aren't compatible is the fabricator/assembler does not provide a complete set of design rules out of concern of giving away their intellectual property (IP). Dana Korf explores the design rule development hierarchy as well as what should be included in an IP-protected design rule document.

4 The Shaughnessy Report: New Year's Resolution—Get Involved ▶

It's 2020, and it's time to hit the ground running as we approach DesignCon and IPC APEX EXPO. If you're not already networking with other designers or volunteering in our industry organizations, there's no better time to start. In the January issue of *Design007 Magazine*, we give a special shout-out to the volunteers who donate their spare time to improving the PCB design community.

5 The Digital Layout: A New Beginning

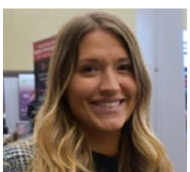

Stephen Chavez

In this column, Stephen Chavez shares a letter from the Legacy Officers and Board Members of the IPC Designers Council, introduces the formation of the Printed Circuit Engineering Association (PCEA), and invites readers to consider future professional development and event opportunities.

6 Elementary, Mr. Watson: Rebuilding Trust When Things Go Wrong

When you look at the long list of steps involved in designing a PCB, it can be somewhat overwhelming and sometimes pretty easy to miss something. Be assured when things go wrong, they go very wrong. In his debut column, John Watson shares some tips for rebuilding trust when things go wrong.

7 What You Need to Know: The High-tech Job Market


Taylor Rousse

Andy Shaughnessy met with Taylor Rousse, an engineering recruiter for Aerotek, a high-tech staffing company, at PCB Carolina to discuss the demands she sees in different industry segments and how it varies region to region. Taylor also offers advice for designers or electrical engineers in the job market, including tips on writing that perfect résumé and the return of the counter-offer.

8 Connect the Dots: A Penny for Your Thoughts on Copper


Bob Tise

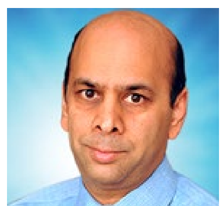
Copper is the primary metal for standard PCBs. And while standard PCB capabilities depend on what materials are used and how they are constructed, copper is the go-to choice. Bob Tise explains some of copper's applications, advantages, and challenges.

9 Flex Talk: The Challenge of Change


Tara Dunn

I recently kicked off a presentation on flex and rigid-flex by asking for a show of hands of those who had never worked with flex materials or considered themselves to be just learning how to design with flex.

10 Standard of Excellence: Five Ways to Stay on Track With Your PCB Vendors


Anaya Vardya

One of the most important aspects of the customer-vendor partnership is to keep the vendor aware of their standing with you. Anaya Vardya shares five ways to ensure that you and your PCB vendors are on the same page when it comes to how both of you are doing as working partners.

Career Opportunities

Talent for Hire

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

- **Director of Operations, PCB and CEM**
  Grew previous operations to over $100 Million.

- **Technical Field Applications Engineer**
  Over 15 years of experience working in the U.S. with offshore companies.

- **Two Highly Experienced Front End Engineers**
  Over 25 years of experience. Will work remotely.

- **Four Experienced Sales Professionals for Both PCB Fabrication and Assembly**
  All four exhibit successful growth records.

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

207-649-0879

or emailing him at:

danbbeaulieu@aol.com

more details

Sr. PCB Designer–Allegro

Freedom CAD is a premier PCB design service bureau with a talented team of 30+ dedicated designers providing complex layouts for our enviable list of high-tech customers. Tired of the commute? This is a work-from-home, full-time position with an opportunity for overtime at time and a half.

Key Qualifications

- EXPERT knowledge of Allegro 16.6/17.2
- Passionate about your PCB design career
- Skilled at HDI technology
- Extensive experience with high-speed digital, RF and flex and rigid-flex designs
- Experienced with signal integrity design constraints encompassing differential pairs, impedance control, high speed, EMI, and ESD
- Experience using SKILL script automation such as dalTools
- Excellent team player that can lead projects and mentor others
- Self-motivated, with ability to work from home with minimal supervision
- Strong communication, interpersonal, analytical, and problem solving skills
- Other design tool knowledge is considered a plus (Altium, PADS, Xpedition)

Primary Responsibilities

- Design project leader
- Lead highly complex layouts while ensuring quality, efficiency and manufacturability
- Handle multiple tasks and provide work leadership to other designers through the distribution, coordination, and management of the assigned work load
- Ability to create from engineering inputs: board mechanical profiles, board fabrication stack-ups, detailed board fabrication drawings and packages, assembly drawings, assembly notes, etc.

apply now

Career Opportunities

CAM Engineer

Eagle Electronics Inc.

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

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

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

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

apply now

West Software Application Engineer

orbotech® A KLA Company

This position reports directly to the Orbotech West software support manager and works with customers to support Orbotech's pre-production software products. Acts as a focal point for technical issues, manages product implementation projects, provides customer training, and supports the sales process. Advanced knowledge of Frontline PCB products, including InCam, InPlan, InStack, InSight, Genesis, and Genflex. Ability to travel and manage time to maximize results. Requires both written and oral technical communication skills. Skilled in the use of scripting languages, including C-Shell, Perl, or Python. Knowledge of relational databases and HTML/XML highly desirable. Knowledge of PCB manufacturing processes. Familiar with the processes used in front-end engineering departments at PCB fabrication sites. Requires use of project management skills to organize and complete projects that involve the implementation of sophisticated software tools used in printed circuit fabrication facilities.

An expected average of 35%+ travel. College degree or equivalent technical education, in addition to a minimum of five-plus years of related experience. Experience supporting sales and sales activities is a plus. U.S. citizen with the ability to work and travel within the U.S., Canada, and internationally.

apply now

Career Opportunities

ventec INTERNATIONAL GROUP 騰輝電子

OEM Sales Manager

Chicago/Home-Office-Based

Want to advance your career by joining a globally successful and growing world-class CCL manufacturer and help drive that success? We are seeking to hire an OEM sales manager to grow and manage key customer accounts with OEM's and Tier 1 manufacturers in the USA, focusing on Ventec's core market segments: mil/aero, automotive, and medical, offering a full range of high-reliability materials, including polyimide, IMS, and thermal management products.

Skills and abilities required for the role:

- Non-negotiable: Drive and tenacity!

Required:

- 7 to 10 years' experience in the PCB industry in engineering and/or manufacturing
- Detail-oriented approach to tasks
- Ability to manage tasks and set goals independently as well as part of a team
- Knowledge of MS office products

Full product training will be provided.

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

Please forward your resume to jpattie@ventec-usa.com and mention "Technical Sales Engineer—Chicago" in the subject line.

apply now

FREEDOM CAD SERVICES

Sr. PCB Designer—Mentor Xpedition

Freedom CAD is a premier PCB design service bureau with a talented team of 30+ dedicated designers providing complex layouts for our enviable list of high-tech customers. Tired of the commute? This is a work-from-home, full-time position with an opportunity for additional compensation for overtime work at time and a half.

Key Qualifications

- EXPERT knowledge of Xpedition VX 2.x
- Passionate about your PCB design career
- Skilled at HDI technology
- Extensive experience with high-speed digital, RF, and flex and rigid-flex designs
- Experienced with signal integrity design constraints encompassing differential pairs, impedance control, high speed, EMI, and ESD
- Excellent team player who can lead projects and mentor others
- Self-motivated with the ability to work from home with minimal supervision
- Strong communication, interpersonal, analytical, and problem-solving skills
- Other design tool knowledge is considered a plus (Altium, Allegro, PADS)

Primary Responsibilities

- Design project leader
- Lead highly complex layouts while ensuring quality, efficiency, and manufacturability
- Handle multiple tasks and provide work leadership to other designers through the distribution, coordination, and management of the assigned workload
- Ability to create from engineering inputs, board mechanical profiles, board fabrication stackups, detailed board fabrication drawings and packages, assembly drawings, assembly notes, etc.

apply now

Career Opportunities

ROGERS CORPORATION

Advanced Connectivity Solutions

Senior Development Engineer

Rogers Corporation is seeking a senior development engineer accountable for the development of more complex products and processes, the establishment of sound technical bases for these developments, and effective interaction with technology, process, and platform innovation; operations; sales and marketing; and process engineering personnel to commercialize these developments.

Essential Functions:

- Design and conduct experiments and interpret the results
- Report on projects in both written and verbal formats at all levels of the organization
- Perform technical troubleshooting of new products and processes; act as new product/ concept incubator for new technologies and platforms, identifying opportunities for improvement and incorporation design for manufacturing requirements resulting in a viable, scalable product
- Provide ongoing process and manufacturing support to newly launched products as applicable
- Provide support in terms of analytical equipment maintenance, methods development, material analysis, and documentation of new process or products
- Manage capital projects for the purchase and installation of new process or support equipment; train employees in new processes

Required Education and Experience:

Ph.D., Ch.E., M.E., or material science, or B.S. or higher in a technical discipline with accomplishment in product development and project management.

Rogers Corporation provides equal employment opportunities to minorities, females, veterans, and disabled individuals as well as other protected groups.

apply now

GO Gardien

Gardien Is Hiring!

The Gardien Group, a leading solutions provider in the PCB industry, is looking to fill multiple openings in their China, Japan, Taiwan, and United States service centers.

We are looking for electrical engineers, operations managers, machine operators, and sales executives. Prior experience in the PCB industry is beneficial but not essential. Training will be provided along with excellent growth opportunities, a benefits package, and periodic bonuses.

Our global teams are from diverse cultures and work cohesively as a tight-knit unit. With performance and initiative, there are plenty of opportunities for professional growth.

Gardien is an equal opportunity employer. Employment decisions are made without any regard to race, color, religion, national or ethnic origin, gender, sexual orientation, age, disability, or other characteristics.

Interested candidates, please contact us with your resume and a cover letter. Kindly note that only shortlisted candidate will be contacted.

Apply at careers@gardien.com.

apply now

Career Opportunities

Career Opportunities

Career Opportunities

INDIUM CORPORATION®

Multiple Positions Available

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

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

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

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

www.indium.com/jobs

apply now

Manncorp™

SMT Field Technician

Huntingdon Valley, PA

Manncorp, a leader in the electronics assembly industry, is looking for an additional SMT Field Technician to join our existing East Coast team and install and support our wide array of SMT equipment.

Duties and Responsibilities:

- Manage on-site equipment installation and customer training
- Provide post-installation service and support, including troubleshooting and diagnosing technical problems by phone, email, or on-site visit
- Assist with demonstrations of equipment to potential customers
- Build and maintain positive relationships with customers
- Participate in the ongoing development and improvement of both our machines and the customer experience we offer

Requirements and Qualifications:

- Prior experience with SMT equipment, or equivalent technical degree
- Proven strong mechanical and electrical troubleshooting skills
- Proficiency in reading and verifying electrical, pneumatic, and mechanical schematics/drawings
- Travel and overnight stays
- Ability to arrange and schedule service trips

We Offer:

- Health and dental insurance
- Retirement fund matching
- Continuing training as the industry develops

apply now

Career Opportunities

Career Opportunities

BLACKFOX
Premier Training & Certification

IPC Master Instructor

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

For more information, click below.

apply now

Events Calendar

Embedded World
February 25–27, 2020
Nuremberg, Germany

IEEE Aerospace Conference
March 7–14, 2020
Yellowstone, Montana, USA

DATE—Design Automation & Test
March 9–13, 2020
Grenoble, France

Microwave & RF
March 18–19, 2020
Paris, France

Electronica & Productronica China
March 18–20, 2020
Shanghai, China

LOPEC Exhibition and Conference
March 24–26, 2020
Munich, Germany

SMTA Atlanta
April 16, 2020
Peachtree Corners, Georgia, USA

KPCA and KIEP Show
April 22–24, 2020
Kintex, Korea

Additional Event Calendars

Coming Soon to *Design007 Magazine*

March 2020: Design for Profitability

The PCB designer's footprint is all over the electronics development process. How can designers create more profitability for their final products? In this issue, we look into the real cost of the design.

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

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

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

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

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

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

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

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

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

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

TECHNOLOGY SPECIALIST: **BRYSON MATTIES**

COVER: **SHELLY STEIN**

COVER IMAGE: **ADOBE STOCK © MRCMOS**

DESIGN007
MAGAZINE

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

February 2020, Volume 9, Number 2
DESIGN007 MAGAZINE is published monthly,
by BR Publishing, Inc.

ADVERTISER INDEX