I-Connect007

JUNE 2020

DESIGN007 MAGAZINE

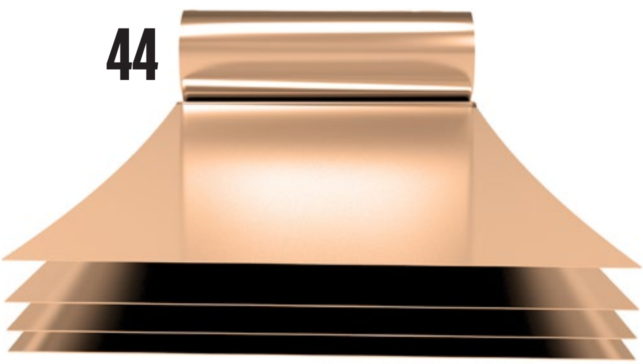
SURVIVING THE APP NOTE CHALLENGE


Is your PCB documentation process outdated?
Today's automated documentation solutions can eliminate post-processing errors and speed up time to market.
THE PRINTED CIRCUIT DESIGNER'S GUIDE TO...™
Documentation
Assessing Your PCB Documentation Process
Mark Gallant
DownStream Technologies
I-007e Books
DOWNLOAD NOW
I-007e Books

DESIGN007 MAGAZINE

Surviving the App Note Challenge

There's a split in the PCB design community, and—as is often the case—the divide centers on data: app notes. Should designers trust app notes or assume that they're incorrect? Are app notes kissing cousins to datasheets, or are they overly optimistic marketing collateral, with results you're not likely to see in real-world operation? What should a young designer or engineer do when facing an unfamiliar app note?

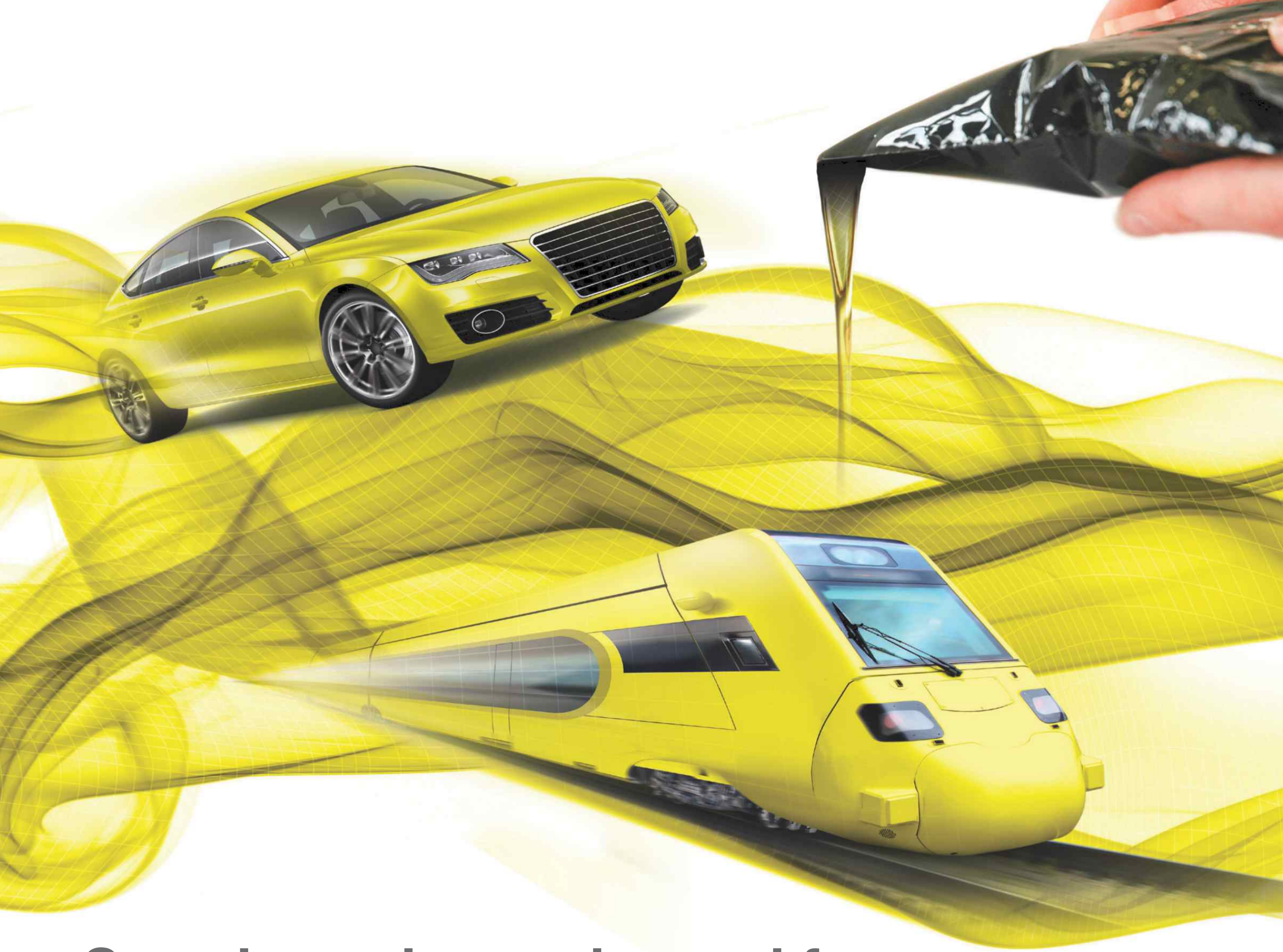


FEATURES:

FEATURE COLUMNS:


We are dedicated
to excellence through
innovation,
technology
and most importantly,
service.
COMMERCIAL • MILITARY • MEDICAL • BAREBOARD
U.S.C.
U.S. CIRCUIT
www.uscircuit.com
ISO
9001 CERTIFIED
DEPARTMENT OF DEFENSE
UNITED STATES OF AMERICA
ITAR
Registered International Traffic in Arms Regulations

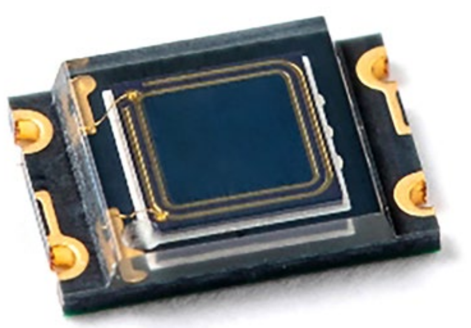
DESIGN007 MAGAZINE

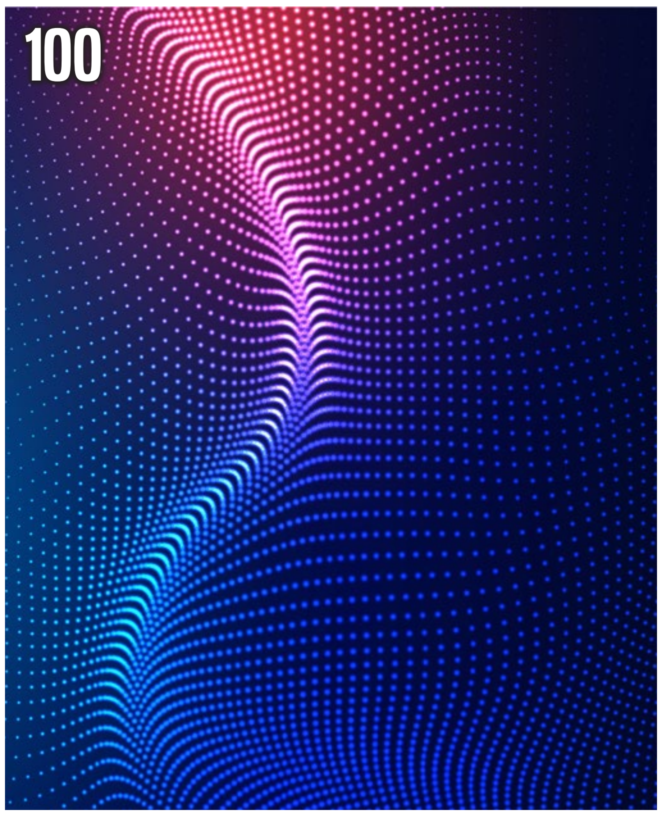


FLEX007

This Month in Flex

Designing and manufacturing rigid PCBs can be challenging. But flexible circuit technologists face a plethora of issues that their rigid counterparts never dreamed of. One advantage for the flexible circuit designer: the three-dimensional nature of flex allows for innovations that are impossible for rigid boards. Is flex really the new frontier?






LEAVING THE OTHERS
IN OUR DUST BY INVESTING
IN SPEED & TECHNOLOGY

App Notes: To Trust or Not to Trust?

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

There's a split in the PCB design community, and—as is often the case—the divide centers on data, specifically app notes.

It's a lot like a boxing match. In one corner, we have the PCB designers and design engineers who have been taught to follow app notes provided by the manufacturers of components and laminates. Many of these technologists work for companies that stress the importance of following these app notes. As one senior designer told me, "What else are we supposed to use?"

And in the other corner, we have veteran design instructors like Lee Ritchey, Rick Hartley, Dan Beeker, and quite a few more. Their position is that app notes should not be trusted unless you know that they're accurate. Designers and engineers don't automatically trust any other data, so they shouldn't trust app notes either just because an IC manufacturer says they're accurate.

So far, neither side has managed to land a knockout blow, but there has been plenty of trash talk. There seems to be a lot of confusion regarding the role of app notes. Are app notes kissing cousins to datasheets (which most designers trust), or are they overly optimistic marketing collateral with results you're not likely to see in real-world operation?

Apparently, the accuracy of app notes was not much of a problem 25–30 years ago when board speeds were much lower. IC makers cranked them out with each part, and if they weren't exactly accurate, it didn't matter. There was a lot of leeway back then, and a designer or design engineer could follow a bad app note and still wind up with a perfectly functioning PCB.

But now, there's almost no leeway, and an app note that's a little off can have major consequences for a high-speed design. People who write app notes make mistakes, just like the

rest of us. As Clive "Max" Maxfield points out this month, the engineers who write app notes are not "infallible creators."

The one topic that kept popping up was trust. Have you had good luck with app notes from that IC maker in the past? Do you know the track record of that particular app note writer?

Experience may be the best tool in your toolbox when facing an unfamiliar app note. Designers and design engineers with decades of experience can often tell right away whether a point in an app note is too good to be true, or only likely under laboratory conditions. But how do we train the new designers and design engineers to handle new app notes?

For this month's issue, we asked a variety of experts to share their thoughts on app notes, datasheets, and when to trust this data. Rick Hartley and Dan Beeker start off by explaining why app notes should never be trusted unless they've been verified, and they present some great horror stories to back up their position. Geof Lipman of component distributor Octopart discusses parts data in general and why the company doesn't provide customers with app notes.

Martyn Gaudion explains why app notes often provide timely information but should still be treated a "best endeavors." Max Maxfield posits that the old Soviet détente maxim "trust but verify" also holds true for app notes. John Coonrod explores the difficulties that laminate companies face in crafting app notes for high-speed materials. Kelly Dack discusses the pros and cons of app notes, and why—like much of PCB design—their proper usage is a matter of making the right trade-offs.

We have an interview with Pallav Aggarwal, a designer based in Bangalore, India, who discusses his growth from a hobbyist to a degreed engineer. We also bring you columns from regular contributors Barry Olney, Patrick Crawford, Istvan Novak, Jade Bridges, Joe Fjelstad, Dominique Numakura, and Bob Tise and Matt Stevenson.

Next month, we're shining the spotlight on you and your home office. We want to see your workspace! Click here to take our workspace survey and upload a photo of the desk, coffee table, or kitchen countertop where you make the design magic happen.

See you next month! **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.

Design007 Survey: Show Us Your Workspaces!

Since the COVID-19 lockdown began, more and more PCB designers and design engineers have moved into home offices. Some designers adapted to telecommuting right away, and have no plans to ever work in a company location again, but others say they can't wait to get back into the company office when the smoke clears. What's your opinion on working remotely?

In this survey, we ask our designer and design engineer readers to share their thoughts on working from a home office: the dogs, the crying kids, everything. Best of all, we want to see photos of your workspaces! Take a photo of your office, whether it's a dedicated office, a dining room table, or even your company office, and upload it into the survey. Just remember: If you upload a photo, it may appear in *Design007 Magazine* or on the Design007 website.

Thanks for taking the time to fill out this survey. Surveys like this help keep on track covering the issues that matter to you most. To take this survey, click here.

What is the Proper Role of App Notes?

Feature Interview by the I-Connect007 Editorial Team

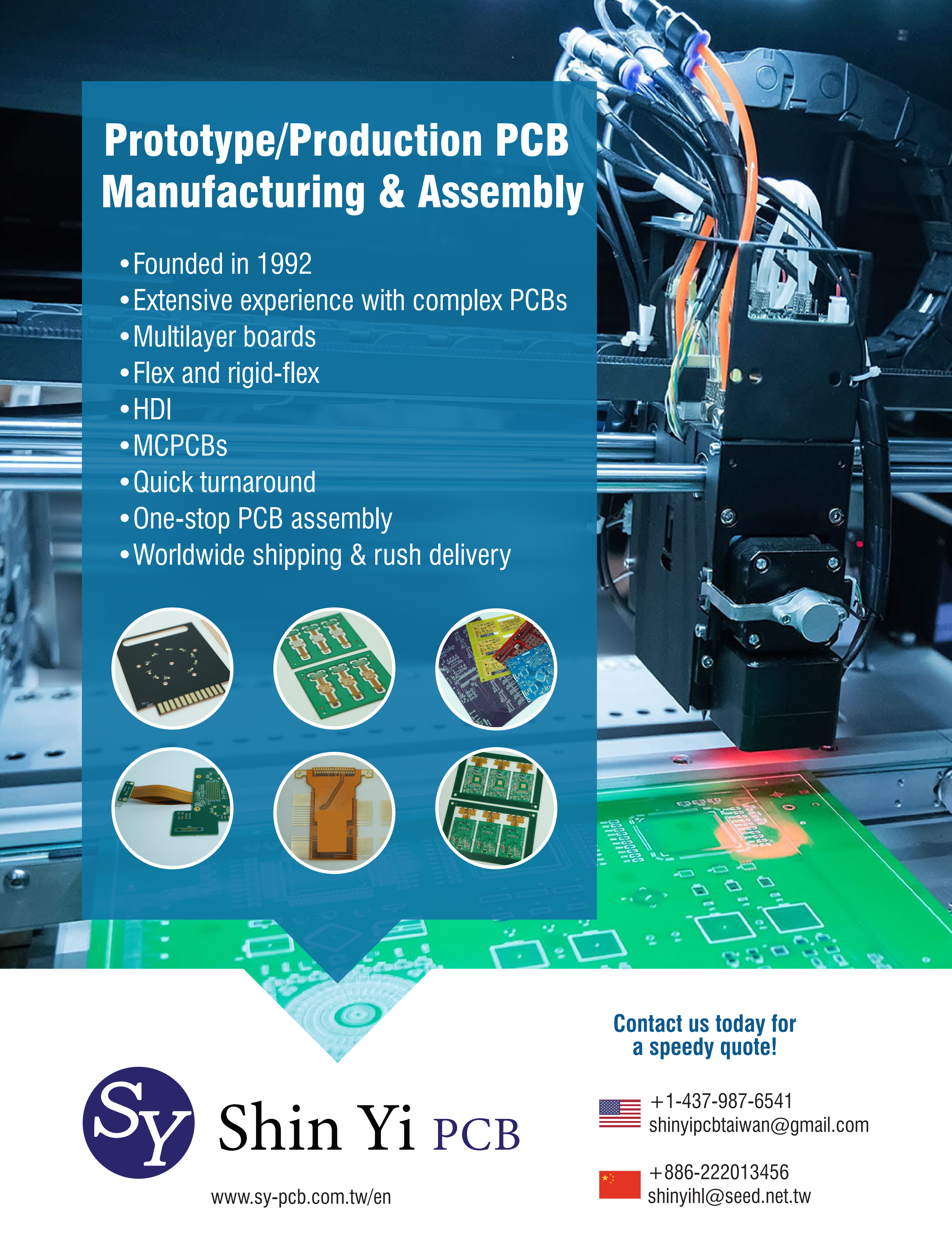
If you're a PCB designer or design engineer, you're probably familiar with app notes—documents produced by the chip makers and other suppliers that are supposed to provide design teams with vital information for creating better circuit boards. But do app notes do what they're supposed to do?

There's a disconnect in the design segment. Some designers say they follow app notes every time, but a growing number of designers approach app notes with a wary eye. Many of the leading voices in the design community have been advising designers not to rely on app notes without running the numbers first. We decided to ask PCB design instructors Rick Hartley and Dan Beeker to help us shine a light on this issue.

Andy Shaughnessy: A couple of years ago, while Rick was doing a class, somebody in the class stood up and said, "We're required to follow the app notes." Rick, you almost had a heart attack.

Rick Hartley: I start all my classes with a comment I heard Lee Ritchey make 27 or 28 years ago, that app notes produced by IC manufacturers should be assumed wrong until proven right. A few people questioned him and said, "They're not all wrong." Lee responded, "I didn't say that. I'm telling you that enough of them are wrong that you should not assume they're right; you should assume the other way and prove they're right if you believe they are."

When Lee made the statement, I giggled, and he asked, "Do you disagree with me?" I said, "No. I've believed that for some time, and I'm glad to see there are two of us." And then we both laughed. Dan Beeker is the one who made me understand. I had always had heartburn with app notes, but I never thought about why until I attended one of his classes. Somebody asked Dan, "What's wrong with them?" because he made a similar comment in the class that app notes aren't always good advice. He said, "App engineers all understand circuit theory. They all have a circuit background and can talk about schematics all day long, and they're right 99% of the time. What


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
Contact us today for a speedy quote!
Sy
Shin Yi PCB
www.sy-pcb.com.tw/en
+1-437-987-6541
shinyipcbtaiwan@gmail.com
+886-222013456
shinyihl@seed.net.tw

Rick Hartley

they don't understand is PCB layout." Would you like to comment on that, Dan?

Dan Beeker: Yes. These guys have the greatest intentions. But even if they've been involved with board layout, it's for a test board or evaluation board. If they are going to sell it, the highest level of certification is at the commercial level. Typically, they are not involved at that point. A team does the CAD work, and that's the way the board comes out. If anything, they start with an app note that somebody else had written in the past that's sort of close to what they need. And this is not their primary job; this is the second, third, or tenth job on their list, but they get hammered if they don't write app notes, so they take one that exists. They do their best to copy and edit it so that it matches the product or application that they're working on, and then that goes out the door. There's not a lot of detailed peer review because if you've been in those types of design reviews, everybody sits in a room, you don't look at it until you get there, and then you all agree that it's a good thing and go out the door. The app note goes out.

Even to the level where I was engaged in a corporate-level evaluation team, we went through all the app notes that we could and looked for the ones that were probably okay, not very good, and were dangerous, and we made the lists and presented it to our corporate marketing people. Their response was that they would rather have bad app notes than no app notes, and they wouldn't take even the ones that were blatantly incorrect off of the website or from the information sources. I've been involved in that too. I was one of those circuit guys who did a few application notes, and I was all smiles without a clue about what was going to end up happening.

Hartley: About a year ago, I was at a company doing some training, and a guy brought me an app note and said, "Based on what you've said in the first day and a half, I get the feeling this app note is way out in left field." It had a feedback line in a switch-mode power supply where they had suggested isolating the ground for it away from the other grounds. I'm sure Dan's cringing right now because it's problem number one. On top of that, with the example they showed in the app note, they split up the ground signal and its return. At least you have to put them together to contain the fields. Then, they routed the trace and feedback line with a different ground. I see stuff like this all the time.

Beeker: And you'll see it on datasheets where there will be an analog and a digital ground, especially on voltage regulators. When I confronted the IC designers, they admitted that it's there for test, and they didn't expect people to see that. Even in their own design, there are control signals that go from the digital side to the analog side that depend on the two grounds being connected at the PCB. Otherwise, even in their control of the switching power supply, the control signals would then have to go find the magic isolation point, whether that was a star connection, inductor, or a ferrite, which people love for some reason. Then, they play back to the die.

Hartley: Besides grounding problems, and we both know there are a million grounding problems in app notes, most of them have that wrong. What are some of the other common errors that you've seen?

Beeker: They sometimes have an overly abundant desire for isolation in the power supply. They love inductors and ferrites, and they end up in the application notes; it's aimed at products, but they are coming from a board where they were going to do an evaluation for silicon. They have to put all these special hooks in there to isolate each pin so they can do things to it, but they don't remember which things were there for testing, and which things were there to make it work. They throw all that stuff in there, increasing the bill of materials cost and making the design far more complex than it needs to be, costing extra layer pairs and often not working.

Hartley: And costing extra, and with extra components, too.

Beeker: All of that makes it much more difficult to do a good job. The other thing is the arbitrary insistence on length matching. I see a lot of that, and it makes me crazy. They pick an arbitrary number that is admittedly arbitrary by the memory vendors, for example. In our case, we will have an application note, telling you how to design your memory interface. They will verbatim copy from their favorite datasheet for the memory company they're working with and throw it in there with no consideration about what's required because it's a state machine and you're trying to meet the timing requirements. It's a lack of understanding.

Barry Matties: How do you know when and when not to follow an app note? Do you have to do all of this due diligence for every app note?

Hartley: No. You have to be educated enough as an engineer to know what's good and what isn't good. When I went to that company in

Dan Beeker

2003, the one with the serious problems due to excessive length matching of memory lines, I already had almost 45 years of experience under my belt and was able to recognize the problem very quickly.

Matties: When you think of the young engineers who don't have that, are coming out, and want to have a career in layout, what do they do?

Hartley: That's the challenging part. I hate to say this, but it comes down to new engineers. If no one has told them that random layout advice doesn't work, then they will have to find out the hard way. That's the bottom line. New engineers are going to have to work by trial and error and figure out what works and what doesn't. Dan hit the answer. Start reading and attend conferences.

Beeker: This isn't taught in school, and they don't want to change what they're teaching. When I was first learning, I was confused. I'd go from Rick's class to somebody else's class, and with as much passion, they'd be saying something that did not connect with what Rick was saying. To figure out what was true and wasn't took quite a bit of time. In the near term, they have to go look online for some of the people who do understand this. Right now, that's me and Rick and Lee.

You have to look for things that don't make sense. You can, like Rick said, always count on the app note circuit being correct. They have probably given you a functional start, but it's when it starts to get into layout. The fundamental thing is, "Are they telling you to design transmission lines where the returns are the same copper that the energy comes from?" Those are the ones where you start to say, "I'm not going to pay any more attention to their layout advice because it doesn't have any basis in science."

You have to look for things that don't make sense.

Matties: When you think you know it, there's more to learn.

Beeker: The big problem is that the "consumers" of engineers—the companies who hire them—are not complaining; they aren't going back to universities, saying, "We need to change the skillset that your graduates are coming into the field with." And then, to make it worse, they are not willing to send their engineers to training, but they're willing to pay for 3–5 iterations of a circuit board each time with their fingers crossed and eyes closed.

Matties: But you talked about the complaining. Sometimes, they don't even know to complain because maybe they're not doing multiple iterations. Instead, they may be buying a $150 board when, with the proper approach, it could have been a $30 board.

Beeker: And that's if they're lucky enough and it works, which is a very rare exception. The boards almost always fail tests, and they have to race to get new boards designed, try to get enough parts—which might often be scarce because they're developing early articles—beg for time in the chamber, and still not be sure they're going to pass. My automotive customers will admit that it's three or four times, and they do not budget for either the cost or the time, but they still will not send their engineers to train. They're very happy to let me teach them for free.

Shaughnessy: This sounds like what we heard in a recent survey about cost-aware design. Many of the senior designer respondents said that cost is not something that they worry about until they get it almost to assembly, and then they try to get the cheapest components and save money that way. A lot of respondents said cost is a manufacturing thing, not a design problem.

Beeker: All my customers would draw their designs with a pencil and a piece of paper if they could get away with it. The other challenge is the young engineers, even if they do understand it, because they were paying attention both in their fields classes and their circuit classes and figured out the two are connected somehow. They get into these companies where there are old farts like me who are dead in the circuit land, and they aren't strong enough in their confidence to fight them about it; they get forced into submission to follow these worthless design guidelines that caused their designs to fail. Then, their confidence gets reduced even more when originally they probably knew this was not the right thing to do.

When I first started to understand this, I still wasn't strong enough in my belief in fields. Even though Ralph Morrison kept telling me over and over again, "It's all about the space." I would go back to my teammates, and they would want to argue about it and start talking about return current instead of transmission lines. I couldn't win. After 15 years of working with Rick and Ralph, I'll fight to the death to prove that science is real.

Shaughnessy: Has anyone ever talked to these IC companies and said, "Why don't you get your act in gear and have some designers do some input on these things?" They must know that people are out here saying that their app notes are bullshit. Why do they keep putting out bad info?

Beeker: They're looking at circuits, and these aren't circuits. This is about high-speed analog. Their IP people have rules that were based on things that seemed to work 20 years ago and are now cast in concrete. It doesn't help that the simulation tools will back them up because a simulator will tell you exactly what you want to see. It only looks at the three main elements: RLC.

Hartley: The bottom line to all of this, and we've said it a couple of times without saying it, is there are two schools of thought about how circuits work: there's circuit theory and field theory. When I was in school, I was taught circuit theory, but so was Dan and every EE. I was also taught field theory, and I went to one of my other profs and said, "I'm having trouble with field theory." He said, "Don't worry about it. Do well enough to get through the class. What you need to understand is the circuit theory." He couldn't have given me worse advice because the truth is circuit theory is how things work on paper. Field theory is how things work in the real world. It's the physics behind how things function, and almost all engineers learn circuit theory. They learn voltage and current, but they don't learn enough about field theory.

Matties: There are a lot of people working at these app notes along the way. Is there any community feedback to improve an app note beyond the OEM or the provider within the design community?

Beeker: There's a little bit, but not very much because most of the people, like Rick and me, were disgusted and ignored it. Or, like Rick says, "Use it to start your campfire." They don't understand it well enough to push back.

Nolan Johnson: What do you see as the role of companies that are out there right now selling parts and library services and trying to implement models based off of app notes?

Beeker: Most of the time, that's pushed off on somebody else's desk. They don't take a role. What I keep telling my customers is that, if they can't go to a conference, hire one of these people to come and teach their team. The cost of one board's re-spin will more than pay for that class, and then you probably will reduce your cycles from three or four or five down to one or two max. If that isn't a good return on your investment, then you aren't paying attention.

Happy Holden: Where do new engineers go to get the real numbers on the laminates they may be working with or specifying? The 10 megahertz number from the IPC slash sheet is not going to be sufficient for the margins, nor do they mention temperature and humidity effects on laminates.

Hartley: Exactly. You and I both know to go to a datasheet for material and look at dielectric constant; they'll give you a number. What they need to give you is a graph that shows how it changes across temperature, humidity, and frequency, and they need to show you all the characteristics, such as the conductivity of the material.

Beeker: A nanosecond edge is a gigahertz, so they're way out of line. Remember, 60 picoseconds is a common signal now, and DDR is 12.6 gigahertz.

Holden: My only experience with app notes was early on in the HDI rollout. I received calls from people designing reference boards for app notes for the IC companies, and they were asking me about HDI design rules, etc. I asked them why they were using HDI and what material, and they said they designed this board on a specific material and were going to use blind vias because they're low inductance. I said, "What kind of chip is this?" They gave it to me, and I said, "That chip is probably going to be used on FR-4 with through-holes. Why are you using these expensive ones?" They explained, "It gives us the best graphs and charts on our app note specs." I responded, "Yes, but blind vias with low impedance is not the way it's going to be used. It's going to be used on FR-4 with through-holes. They're not going to see those curves that you're presenting." Is this common? It's like McDonald's and Burger King commercials showing you a big juicy hamburger, but it's not big and juicy like that.

Beeker: That's true of ours. I've had a couple of unhappy conversations with some of our app note people who were specifying material for something that didn't need anything like that. On top of it, it costs them business because if their customers go look at the system costs and find out that this material is 5–10 times more expensive than what they're used to, they will find another device. Sometimes, you lose business you didn't know you had to lose. That's what's so pitiful about app notes is they are important, but nobody puts that kind of importance on them.

Holden: I don't think these people recommended that. They were simply using it to get a better response from their device because of the parasitics.

Beeker: They probably described the materials used in their board in their application note, but if they didn't, then they're cheating. The bottom line is if you're trying to show the real capability of the integrated circuit, which is important, and then you will do something like that. You'll put it in a perfect world so that you can see what is happening at the pins. If you can't afford to make it that good, you're going to have reduced performance, it gives you the possibility, but maybe not a practical solution.

Matties: I'm still stuck on that story you were talking about earlier, Rick, of the board and the designer following the app notes who thought he was optimizing and drove the cost way up. Is there a peer review process for somebody who doesn't have all the experience that's commonplace?

Hartley: Yes. Every engineer in that company reviewed that design, but none of them realized it was going to be a problem.

Matties: Maybe they have to go to a design bureau or a service that is independent of the world that they live in.

Beeker: And they're not going to go outside the company. They're proud of these designs. I often get customers requesting help with their EMC issues. They'll send me a snapshot of a two-inch square on the board and say, "What's wrong with this?" They don't want to share the design because it's secret, but there's nothing secret here; they're guarding a dead cow. They don't want to pay for anybody to do any outside analysis. The money's not there. They'd rather pay to have it done 14 times than pay to have it done right the first time and save themselves $1 million on it.

They'd rather pay to have it done 14 times than pay to have it done right the first time and save themselves $1 million on it.

Johnson: Lest we paint the wrong picture, are there some semiconductor companies that do an excellent job or a consistently good job? What is good about app notes out there?

Beeker: I don't have any experience except with my stuff and the ultra-conservative stuff from memory vendors and SMPS application notes. The circuit design is probably okay, and the schematic itself is probably solid. There are still going to be things you need to understand to make it better for your application, but you have a functional starting point.

Matties: You're making a good case, though. If the app note is good, and it's not over-materialized or doesn't require the best of the materials, the vendor is likely to sell more parts.

Beeker: The engineers don't want to admit that they're wrong. The most humbling moment in my entire career was five minutes into Rick's signal integrity class when I figured out that every design that I had ever done had worked by accident, not on purpose. I thought, "I hope my boss doesn't figure this out."

Matties: How do we tell the people at the top that incorrect app notes matter and they need to change their approach?

How often in your programs do you have to re-spin the circuit board to pass the EMC? And would you be willing to have your teams find solutions for this?

Beeker: A question to ask them is, "How often in your programs do you have to re-spin the circuit board to pass the EMC? And would you be willing to have your teams find solutions for this?" They have to think about the numbers. Some companies don't do new product development anymore because they can't afford to go through the multiple iterations; others are out of business now because they missed their market window and exceeded their engineering budget while trying to get the boards to pass EMC, but people sit and blindly re-spin the board using the same rules that didn't work the first time to do it again and hope it's going to pass. In the long run, the reliability of the product in the field is reduced, and that has to be in the hundreds of millions—if not billions of dollars—when you look across the entire industry.

Matties: And if you look at an in-the-field product failure, that's a cost of design.

Beeker: But they finally lock upon a combination of band-aids and voodoo that allows them to pass EMC, and then a lot of times, that's how some of these guidelines were created. "It worked on this one. We're going to do it again, and it worked because it was only good on that particular bad architecture." Nobody is willing to start over on that second board, even if it blatantly doesn't work. Most of the time, when I do EMC analysis for my customers, I look at the board stack, and I'm done. Until they fix that, there's nothing else I can do to make it better.

Hartley: Exactly.

Beeker: Sometimes, for my analysis, it's five minutes, and I'm done. Then, it's, "How do I write this to tell them how ugly their baby is and not make them hate me?"

Matties: But there's also a cost of lost opportunity because now your designer is busy redoing what they could've done right the first time instead of working on the new opportunity.

Beeker: Right, instead of working on the new program. That's absolutely key. That's why I say it could be billions of dollars of lost revenue. It's a cascading impact.

Shaughnessy: The app notes probably didn't matter much until the 1990s when the IC speeds got faster, did they? You could be wrong, and the thing would still work.

Beeker: There was a time when everything was wrong.

Hartley: Before the '90s, all circuits were a lumped length. The only advice the application engineer had to give was schematic advice. If they got that right, then everybody was a happy camper. Then, along came distributed elements. Once, speeds were fast enough that things were no longer lumped, and all of a sudden, they didn't understand why things worked the way they did, and they started testing and trying to come up with concepts. But they got most of them wrong.

Beeker: When we first started adding multiple features onto a device to go from a microprocessor to a microcontroller, they started adding clock outputs and the A/D converters. They started seeing problems with the A/D converters on the accuracy or bit count, so they would always lean on the IP people designing A/D converters. They started becoming paranoid about what the system wanted to be, and nobody looked at the root cause: The circuit board design was crap. They found that if they put a ground pin close to the A/D converter, it worked better. These design guidelines and pin-outs of the devices evolved into band-aids that became the design philosophy because nobody, until recently, found the root cause, which was a poor distribution of energy in the circuit board or not designing the spaces.

You go in, and with everything that's supposed to be ground, something is ground to only one. Then, you design the circuit boards so that the energy flow from the power supply to the ICs, which are multi-port power distribution systems. They turn on and off transmission lines. When they turn on a transmission line, they need energy to come from the power supply through the switch net into the new transmission line until the switch is turned off, and then the energy flow stops. But this is plumbing.

Hartley: I loved Lee Hill's comment that he made years ago. "When I'm doing a review of a circuit with a problem, if I see more than one ground, I know there's money to be made."

Beeker: Look at where you're moving the energy on the board. If you have a pipe in your basement and you want to get water upstairs, and if you don't put a pipe between the basement and the upstairs, eventually, you'll get water upstairs. That's exactly what happens on a lot of these circuit boards is the field gets there eventually, but it has to fill up everything between here and there.

Shaughnessy: This has been great. We appreciate your time and thoughts on this.

Beeker: You've unleashed the dragon.

Hartley: Thank you very much.

Matties: Stay healthy, both of you. Take care. **DESIGN007**

App Notes: What Are They Good For?


Geof Lipman

Feature Interview by the I-Connect007 Editorial Team

There are several schools of thought regarding IC manufacturers' app notes. Some engineering departments expect their PCB designers and design engineers to follow app notes exactly. But the top PCB design instructors often warn their classes to assume app notes are wrong until they've been proven right.

We wanted to find out more about this, so we spoke with Geof Lipman, director of operations for part data at Octopart.com, a component data platform acquired by Altium in 2015. Geof shared his thoughts on app notes, datasheets, and the need for due diligence when utilizing any unfamiliar data.

Andy Shaughnessy: Geof, why don't you start off by telling us about your duties as director of part data at Octopart, and we'll go from there.

Geof Lipman: I was brought on because of my background in electrical engineering and manufacturing. And as a member of the Altium organization, we focus on the needs of electrical engineers. I help to guide our acquisition of part data, the types of part data that we take or get, and who we get it from, such as which manufacturers and distributors. We work in concert with our business development department so that we can do this in ways that don't lose us money.

And when I say types of part data, app notes are one type. Our technical specifications would be things like mechanical dimensions and electrical characteristics, including input voltage, output voltage, and supply voltage—the long laundry list of specs that we see. The other types of part data would be datasheets, CAD models, and cross-references. I help sort out the most valuable thing for Octopart users.

Shaughnessy: I understand that you don't hand out app notes with each part. What is the reasoning behind that?

Lipman: Our prime mover for that would be usefulness for our user base. There are many different types of parts. People tend to focus on things like integrated circuits because they're the "glamorous" parts, but most parts that get sold are resistors and capacitors, which are "boring." But nevertheless, people need to specify them, and they need to know how they perform and what they cost. First, we focus on getting the data that every part has. Every part costs money, every part has supply chain information, and almost every single part has a datasheet and specifications.

Something like 70% of the parts that get used in the electronics industry can be mounted to a circuit board—in which case, having ECAD models for the various ECAD suites is a helpful thing for people to have. Sometimes, you can't get the part that you want, so you need to get something that's adjacent to it. Having that cross-reference information is useful to people. We go down through each of these assets. Which are the assets that will help the greatest number of users in the greatest number of cases that we can imagine? You'll see on Octopart that most of our parts have datasheets.

Barry Matties: We see some of the industry's top design instructors telling their classes, "Don't trust app notes. Assume they're wrong until they're proven right." What do you think about that?

Lipman: They might have been being dramatic. I'm not sure. And if you're doing aerospace or medical, life-critical information is needed, then you can't simply trust the documentation. Many datasheets will specify how to test the part, so an engineer will set up a test and verify that they see the same thing on the parts that they have on their bench with the data on the datasheet. It's a very rigorous matter to select parts. Somebody that's in an IoT or a hobbyist space probably wouldn't do that. App notes offer a similar type of information.

In most cases, the datasheet is like a service-level agreement between the company and the users of their parts. They try very hard to make these datasheets a guarantee of performance and specifically state what they will not guarantee. At the other end of the spectrum, you'll see many manufacturers that do "demonstration" designs. They'll design a product, such as a smartphone or an IoT application, which would be called a reference design. A reference design might come with board files. It would probably come with a schematic and then a lot of text about testing and the design. It's a demo of how you would build the entire product.

In between these reference designs and the datasheet is where app notes would live because they describe not an entire product, but rather a single problem or design challenge. Let's say you have a problem with an unstable startup for a microcontroller design because you have poor power supply sequencing. An app note may say, "Check out our power supply monitor. This will solve your power supply sequencing problems for these complex new designs." App notes are that thing in the middle ground dealing with individual problems as opposed to full designs or individual parts.

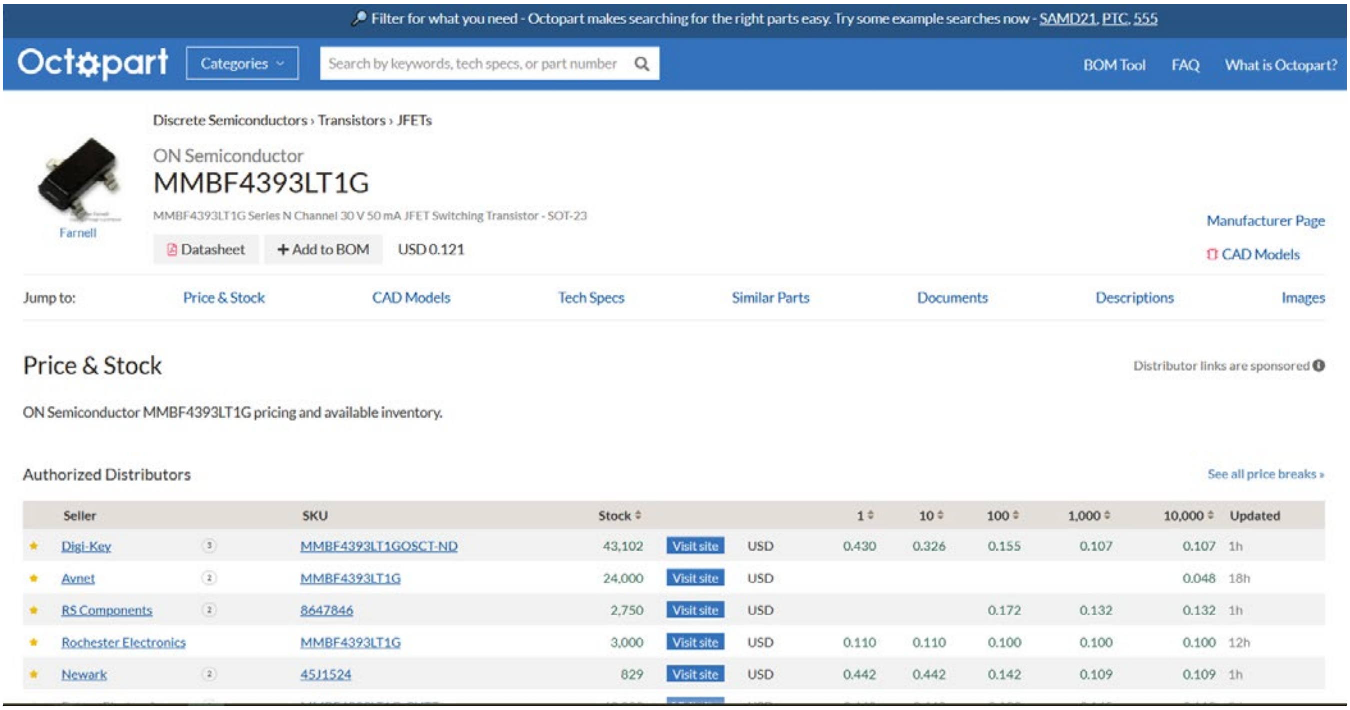
Matties: Does the industry understand the distinction? It seems like there's some confusion about app notes and datasheets.

Lipman: It's remarkable the level of standardization that exists in our industry, considering that it's almost completely market based. That being said, there is still a lot of variation from datasheet to datasheet and app note to app note. I've seen datasheets that are Microsoft Word documents that have probably been typed by interns with almost no component knowledge. I've also seen datasheets that are 1,000 pages long. Those are very different documents, but they share an intent—to accurately describe a single part and its properties.

And app notes have technical descriptions and properties but are marketing materials as well. That's another important distinction between app notes and reference designs versus datasheets. The datasheet is sort of a guarantee or a representation of things that should be true. But app notes are more like, "Check out what you could do with our part." Different companies will use different quality engineers and have different communication priorities during the building of an app note.

There is also a personality-driven aspect to app notes where people will know the writer. It will depend on the person who is writing the app note as to the quality that you get from it.

And you have to be aware of this when you read an app note. Especially for things like thermal issues, performance can vary a lot depending on design choices. If you plan on designing a $3 product and the app note uses an aluminum circuit board to improve cooling, you might want to think twice about paying



Octoparts provides customers with a range of data for each part.

attention to the app note because the aluminum PCB is very expensive. There's no question that some app notes are better than others, but for the most part, if you're going with one of the larger manufacturers, you're probably going to get high-quality app notes.

Matties: Are there any collective or crowd-sourced ratings of app notes through your organization?

Lipman: I'm totally writing that down (laughs)! That's a spectacular idea, and I'm sure one that our product team has discussed in the past, but we have not discussed it recently. We use "trust signals" as much as we can on the site, and user ratings could certainly find a place in that context.

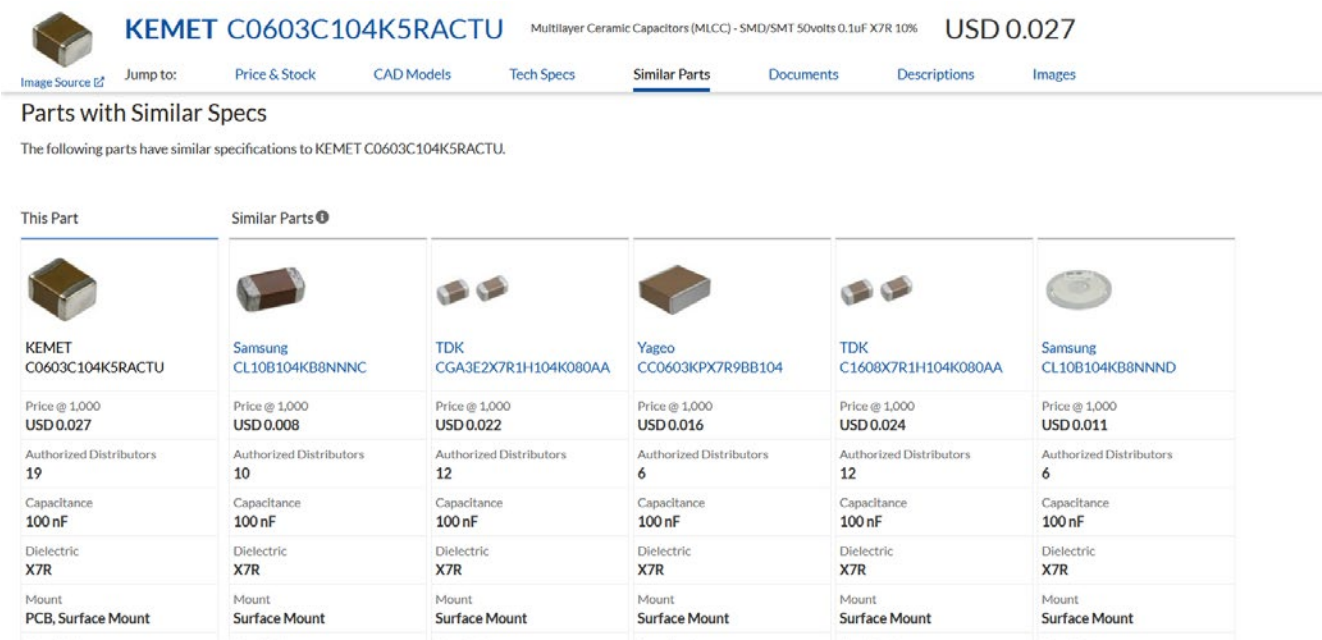
Our priorities for this year include getting more data for app notes and similar types of assets on the website. Collections of app notes are often called cookbooks. In the case of some companies, they put out a newsletter with app notes every month. Those types of documents would also lend themselves to rating systems.

Matties: It seems like there's a big need for it, especially as we see younger engineers coming in and engineers with 30–40 years of experience retire "tribal" knowledge," which is hard to pass on.

Lipman: Very true. I've dealt with a couple of young companies with less experienced engineers, and they lean heavily on app notes and reference designs. It's a classic move to simply copy the design in the app note, and then make small adjustments. It's possible to copy a reference design or use an Arduino or Raspberry PI that's probably not optimized for your application, so you can get something running right away. The younger crowd probably leans on that stuff more.

Shaughnessy: You're in charge of acquiring it and prioritizing and presenting the data to the users. How do you get hold of your data, and what format is it when you present it?

Lipman: It's making sausage, so it's not pretty. We get data from many sources. We have partnerships with a number of distributors and manufacturers who send us data in tabular form. We built tools in Octopart that ingest that data and map that data into the Octopart syntactical system so that it's compatible and nor-



KEMET C0603C104K5RACTU Multilayer Ceramic Capacitors (MLCC) - SMD/SMT 50volts 0.1uF X7R 10% USD 0.027

Image Source Jump to: Price & Stock CAD Models Tech Specs Similar Parts Documents Descriptions Images

Parts with Similar Specs

The following parts have similar specifications to KEMET C0603C104K5RACTU.

This Part	Similar Parts				
KEMET C0603C104K5RACTU	Samsung CL10B104KB8NNNC	TDK CGA3E2X7R1H104K080AA	Yageo CC0603KPX7R9BB104	TDK C1608X7R1H104K080AA	Samsung CL10B104KB8NNND
Price @ 1,000 USD 0.027	Price @ 1,000 USD 0.008	Price @ 1,000 USD 0.022	Price @ 1,000 USD 0.016	Price @ 1,000 USD 0.024	Price @ 1,000 USD 0.011
Authorized Distributors 19	Authorized Distributors 10	Authorized Distributors 12	Authorized Distributors 6	Authorized Distributors 12	Authorized Distributors 6
Capacitance 100 nF	Capacitance 100 nF	Capacitance 100 nF	Capacitance 100 nF	Capacitance 100 nF	Capacitance 100 nF
Dielectric X7R	Dielectric X7R	Dielectric X7R	Dielectric X7R	Dielectric X7R	Dielectric X7R
Mount PCB, Surface Mount	Mount Surface Mount	Mount Surface Mount	Mount Surface Mount	Mount Surface Mount	Mount Surface Mount

There are hundreds of thousands of components available, and many have similar specifications, making correct data even more critical.

malized. Then, that information is displayed on the website on the part detail page.

Shaughnessy: The app notes are more like possible applications, whereas the datasheet is the so-called "real" data.

Lipman: Yes, but there's an educational part to app notes too. A good example might be switch-mode controllers for power supplies. A switch-mode controller is often a very flexible device that a designer can use to realize a variety of power supply topologies, such as buck, boost, flyback, SEPIC, etc. The datasheet would typically give you detailed information on the part and maybe a single or two possible typical supply realizations. But the app notes are where you could go to get examples of specific supplies that discuss the higher-level issues. Often, you can add additional external components to supply design to adapt it for your specific purpose, like a higher voltage, inverted polarities, isolated or non-isolated, etc. The app notes are one of the places that I would look to for this kind of information.

Shaughnessy: If people shouldn't automatically trust app notes and they have to verify them, this would take you an impossible amount of time, right?

Lipman: It would definitely take time, but professional ethics would be the reason for that. If I'm an aerospace designer and I'm putting something on the SpaceX Dragon that people are going to be traveling in and lives could be at stake, and I specify a charger IC based on what was represented in the datasheet or app notes but didn't test it, that would be a failure of professional ethics on my part. For instance, NASA doesn't allow you to use parts that aren't tested under the conditions that they deem to be important.

Matties: But if you're saying the datasheet is a guarantee, there should be no reason to test it.

Lipman: It's a difficult line to tread. When I say it's a guarantee, I mean it's trying to be a guarantee, or it is in many ways like a guarantee. But from a professional engineering standpoint, depending on what the situation is and how important something is, someone is going to be left holding the bag if something goes wrong.

Matties: It's still buyer beware.

Lipman: Exactly. There's a chain of liability or responsibility that, depending on the industry and application, could be very high stakes or not high stakes at all. If your art project goes wrong and your LED displays the wrong colors, that's not terrible, but if the same thing happens with a LIDAR in orbit, you could endanger the mission and crew. It depends on the lens that the user is using to view a datasheet, an app note, or a reference video.

Matties: Most companies would build a library of acceptable components based on the application, and that's what you verify, correct?

Lipman: For many companies, that is how they do it. Something for aerospace or medical is going to have different requirements than a consumer product as to how demanding each individual component is. Various applications have different requirements, and companies have to track that.

Johnson: Geof, what I'm getting as I'm listening through to this is app notes are of as high quality as the engineer. And there's a need there to have a source for this sort of technical data that can be trusted and verified, right off the bat. Where does this come together?

Lipman: It's a challenge from the professional ethics side of things. In the same way that you trust that GM and Ford have done their due diligence, I would trust that Microchip or Texas Instruments had done their due diligence before writing an app note. There could still be mistakes, but engineers have traditionally had faith in particular manufacturers. They think, "Analog Devices wouldn't put an idiot in charge of their I/Q modulators."

Matties: With mutual funds, they have a rating for the fund manager because even though it's a trusted brokerage firm, there are still rising stars and under-performers within that spectrum. An EE would see a particular writer's name on the app note and say, "That's gold."

Lipman: This conversation has revolved, in some ways, around the idea of trust signals for users. When we can, and as much as possible, we try to expose trust signals to the user on our website. An example of that would be the five-star rating concept, which is a trust signal that would enable engineers to place smarter bets, as opposed to guaranteeing that things are going to be good and stops them from having to do their due diligence. A five-star rating is a better bet for you, and you're more likely to get through the vetting process—whatever your vetting process happens to be—unscathed with a five-star product.

Shaughnessy: Do you have any advice that you would give a young EE or designer who has to work with an unfamiliar component as far as looking at the app notes and datasheets?

Lipman: App notes are a wonderful thing. It's the kind of thing where you have to become familiar with what's available, and different companies will have very useful stuff or not. Some white papers have amazing technical revelations in them, but others are straight-up marketing, and that's the same case with app notes.

Read the app notes, but make sure that you understand what is and what is not being represented. I've learned tons by looking at and comparing parts on websites like Octopart. You can learn a lot by seeing what the differences are in their technical specifications. To go deeper on that, you would use the datasheet, app notes, etc. You want to start by making a good bet, having a good probability of success, and then doing your due diligence.

Shaughnessy: We appreciate your insight, Geof. Thanks for breaking this down.

Matties: Yes, this has been very helpful. Thank you.

Lipman: Thanks for the opportunity. I love to talk about this stuff. **DESIGN007**

Application Notes: Advice for Authors

The Pulse

Feature Column by Martyn Gaudion, POLAR INSTRUMENTS

Application notes are the key to shedding light on new topics or new products and software tools in an easily digestible form. As both a consumer and an author of more application notes than I care to remember, Andy Shaughnessy posed an interesting question on whether designers should rigorously follow application notes or verify the content.

Types of Application Notes

I tend to categorize app notes into the following areas:

- How to set up equipment or software
- How to apply a product or software tool to get the best results
- Background notes, which involve explanations of the engineering or science behind a particular task or problem

The Approach

As always, the approach I recommend depends on the nature of the note. Notes based on how to use a piece of equipment or software do require following to the letter, and personal "interpretation" can lead to disappointment when a system doesn't set up or function as expected. Customers sometimes call to say a setup hasn't worked as expected, so applications engineers will remote in to follow what is happening only to see the end-user miss several key steps "because they always miss those out." For notes aimed at how best to ensure a design meets a specific need, it is always worth looking at several sources as there are often multiple correct ways to solve certain challenges.

Delivery of Content

Historically, application notes were just that: a series of written notes to help explain to a

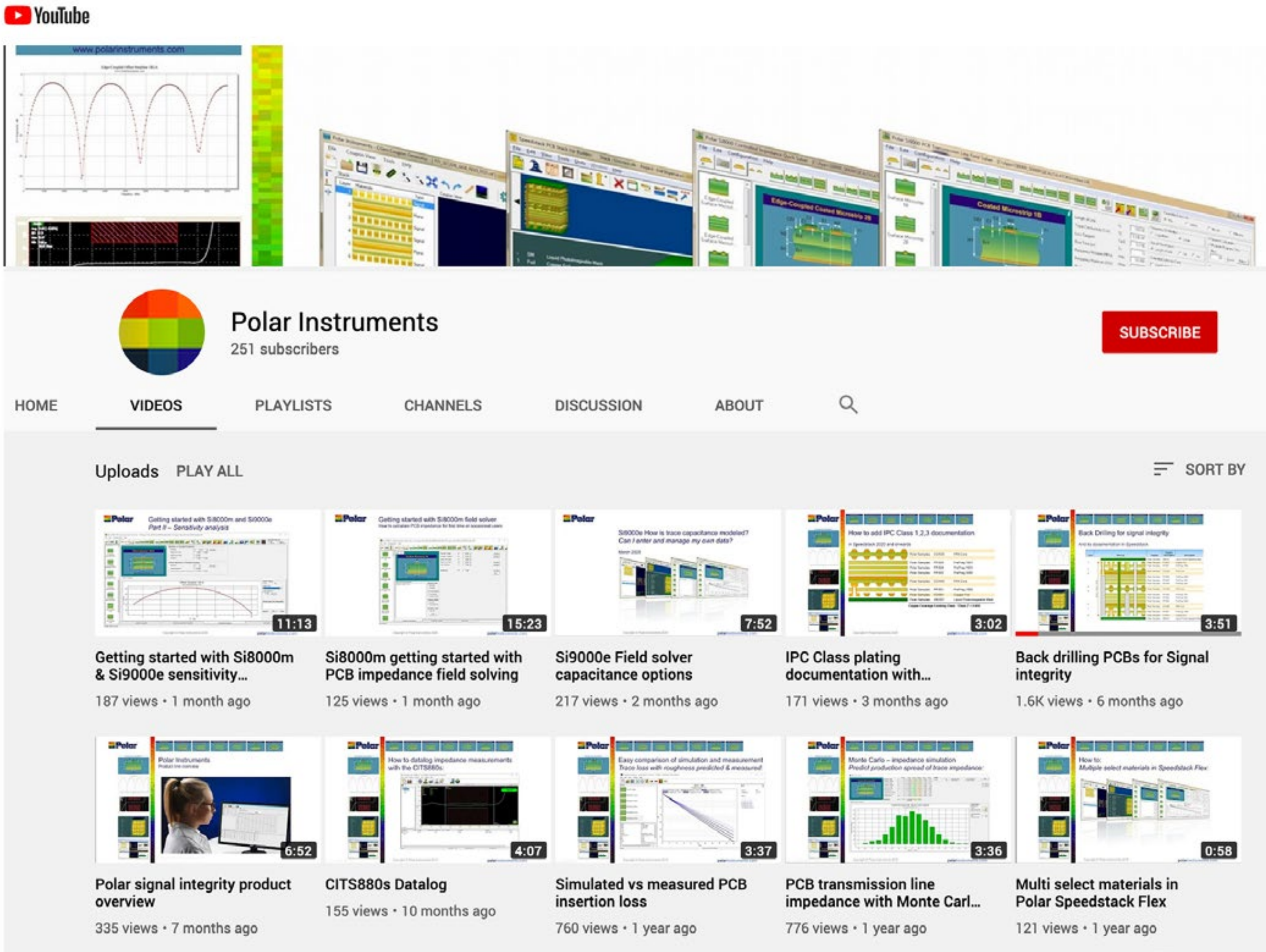


Figure 1: Notes delivered through YouTube or streamed videos.

designer or fabricator how to approach a new and perhaps unfamiliar situation. And written notes are still great for this; they're easily printed out and annotated by the reader at leisure. But increasingly, PowerPoint presentations and video clips can be used to bring application notes to life (Figure 1).

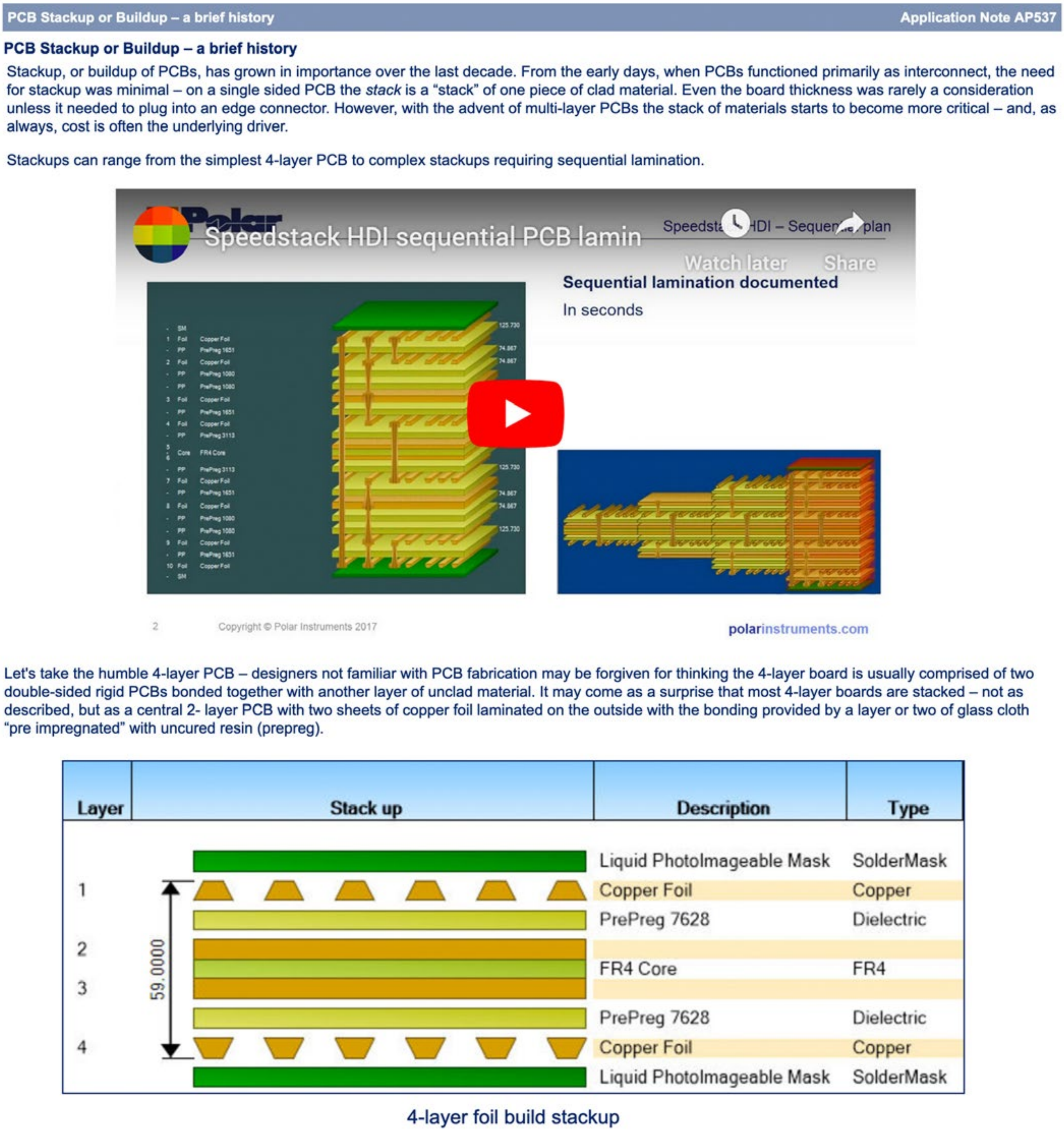
Each has its place, and some content suits one delivery method more than another. Some schools of thought note that different people prefer audio or video or written content for their personal learning style, so having content in several media is often preferable. However, for the content manager, this can be a nightmare to keep updated.

Another alternative for designers who may prefer to mix video content with written learning is to embed appropriate video within web pages (Figure 2).

It is worth remembering that some readers may have a block on access to services like YouTube in a work situation, so offering the video stream from an mp4 file on the writer's website gives an alternate source for content in these situations.

Timely

Some notes based on engineering principles stand the test of time, so it is good for the writer to make them application-agnostic—the background information depicted is timeless, with improving technology sometimes more depth needs to be added—but if the original note is well-written, then the new findings or


PCB Stackup or Buildup – a brief history Application Note AP537

PCB Stackup or Buildup – a brief history

Stackup, or buildup of PCBs, has grown in importance over the last decade. From the early days, when PCBs functioned primarily as interconnect, the need for stackup was minimal – on a single sided PCB the *stack* is a "stack" of one piece of clad material. Even the board thickness was rarely a consideration unless it needed to plug into an edge connector. However, with the advent of multi-layer PCBs the stack of materials starts to become more critical – and, as always, cost is often the underlying driver.

Stackups can range from the simplest 4-layer PCB to complex stackups requiring sequential lamination.

Let's take the humble 4-layer PCB – designers not familiar with PCB fabrication may be forgiven for thinking the 4-layer board is usually comprised of two double-sided rigid PCBs bonded together with another layer of unclad material. It may come as a surprise that most 4-layer boards are stacked – not as described, but as a central 2- layer PCB with two sheets of copper foil laminated on the outside with the bonding provided by a layer or two of glass cloth "pre impregnated" with uncured resin (prepreg).

Layer	Stack up	Description	Type
		Liquid PhotoImageable Mask	SolderMask
1	59.0000	Copper Foil	Copper
		PrePreg 7628	Dielectric
2			
		FR4 Core	FR4
3			
		PrePreg 7628	Dielectric
4		Copper Foil	Copper
		Liquid PhotoImageable Mask	SolderMask

4-layer foil build stackup

Figure 2: Combined written and video note.

enhanced detail can be added in a supplementary note. A typical example of a timeless note on controlled (characteristic) impedance is shown in Figure 3.

When readers want a background to add insertion loss knowledge to impedance control knowledge, in my opinion, the best approach is to link the existing note to a new note which expands on the next level of knowledge required (Figure 4).

Both the above notes stand in their own right, but read together, they broaden the background knowledge and give the reader new to the subject of PCB transmission lines

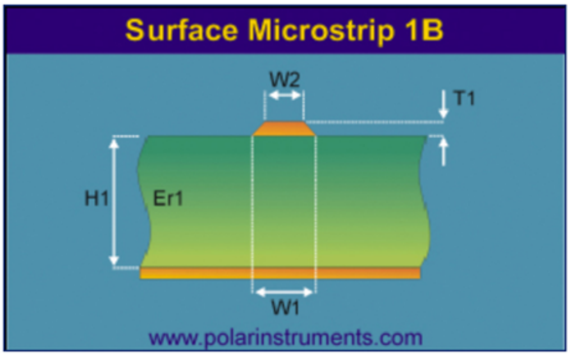
What is controlled impedance? | Application Note AP120

As PCB signal switching speeds increase today's PCB designer needs to understand and control the impedance of PCB traces. With the short signal transition times and high clock rates of modern digital circuitry, PCB traces need to be considered not as simple connections but as transmission lines. This simple introduction is aimed at lines running from a few hundred MHz to around 2 to 3GHz where reflections are the main concern. For higher frequencies 3GHz and upwards both this introduction – and AP8196 What is insertion loss? adds a higher frequency perspective to this easy to understand background information.

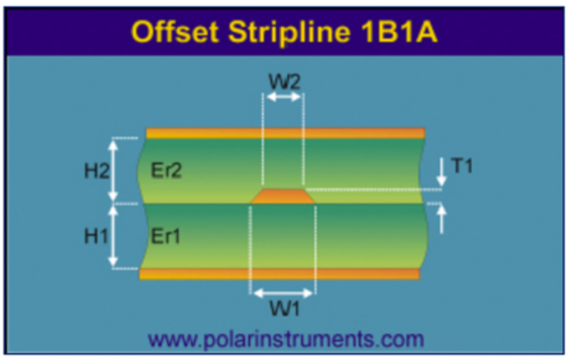
What is controlled impedance?

Probably the most common example of a controlled impedance component is the downlead (or feeder) connecting a receiving aerial to a wireless or television set. Aerial feeder leads usually take the form of "flat twin" cable (commonly supplied with VHF broadcast receivers) or low-loss coaxial cable. In both cases the impedance of the feeder is controlled by the physical dimensions and material of the cable.

PCB trace configurations



You can think of PCB traces as short cables, precisely constructed, connecting the devices mounted on the board, where the PCB trace, like the coax inner conductor, carries the signal and is insulated from its return path (in this case a ground plane) by the board laminate. This is shown in cross section in the microstrip configuration, left.



The dimensions for trace width W1 and W2, thickness T1 and laminate height H1 and H2 and the dielectric constants Er1 and Er2 must be strictly controlled. Solder resist on the surface reduces the impedance slightly so the more predictable stripline configuration shown left is often used.

Figure 3: Timeless application note.

enough information to be able to ask further questions to expand their knowledge of the subject.

Product-related application notes can age as the product is enhanced over the years. Smart product development can minimize the UI changes, so the supporting literature does not age so fast. Often, customers prefer to retain a familiar UI with controls maintained in their positions to maximize the use of muscle memory when engineers use the tools. This is especially the case for infrequently used products. I am sure you are all familiar with web conferencing tools that keep "improving" and moving all the controls around the desktop leading to frustration for occasional users.

Verified Application Notes?

Application notes should always be treated as "best endeavors;" they are not the same as peer-reviewed published papers. Even with peer review, it can be difficult in complex areas to get two engineers to agree on the same solution to an engineering problem.

However, with technical information, it is

What is insertion loss? **Application Note AP8196**

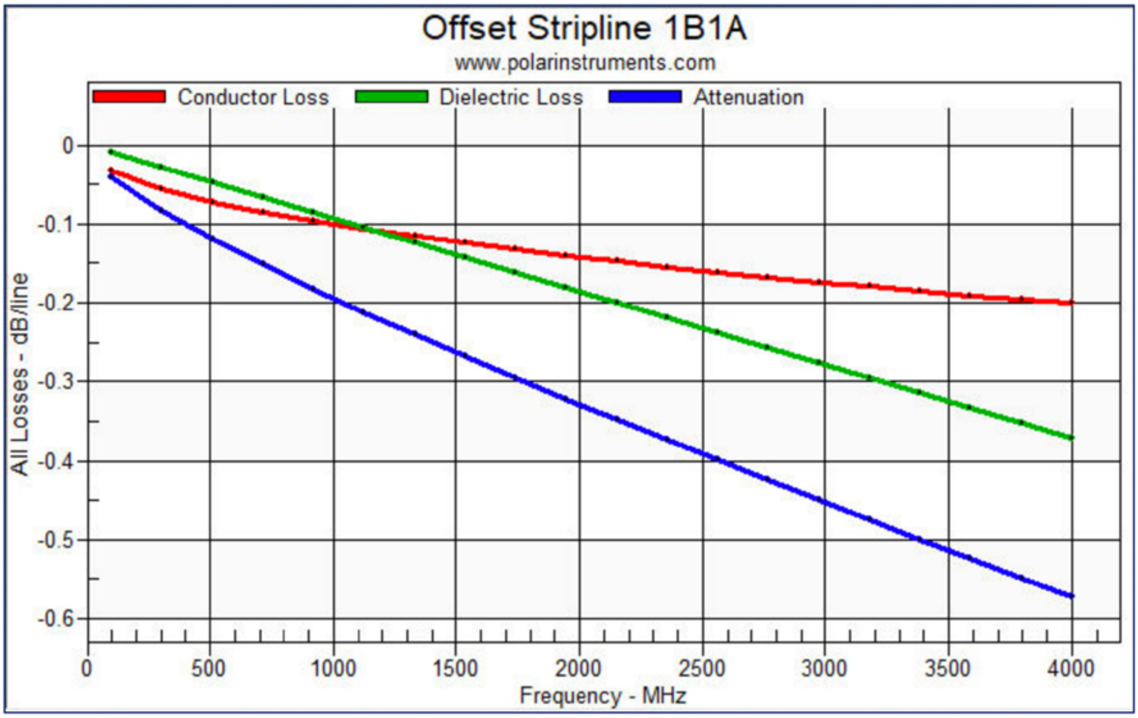
Insertion loss

Application note AP120 What is controlled impedance? discusses the importance of considering traces on high speed PCBs as transmission lines of known – and controlled – characteristic impedance. This application note is an introduction to the cause and effect of insertion loss which becomes predominant as you push upwards from 3GHz.

Typically, *impedance control* will be necessary on PCBs running in the 100MHz to 3GHz frequency range, where signal integrity will be significantly affected by signal reflections. Above these frequencies *signal loss* will be an increasing (and ultimately predominant) factor in signal integrity.

Energy is lost from a signal as heat in the conductor (i.e. the PCB trace and ground planes) resistance (it's a function of the material and plating used) and as heat in the dielectric caused by the inherent material properties of the base material iteslf. Total loss is therefore the sum of conductor loss and dielectric loss. The dielectric loss mechanism is similar to that in a microwave oven; heat is generated in the food (where it is desirable) and in a PCB dielectric – where such loss is undesired or at least needs to be understood and managed.

At low frequencies, conductor loss exceeds dielectric loss; dielectric loss, however,increases at a higher rate than conductor loss and will overtake it as a proportion of total loss (at around 1.1GHz in the chart below.)



Conductor loss and dielectric loss

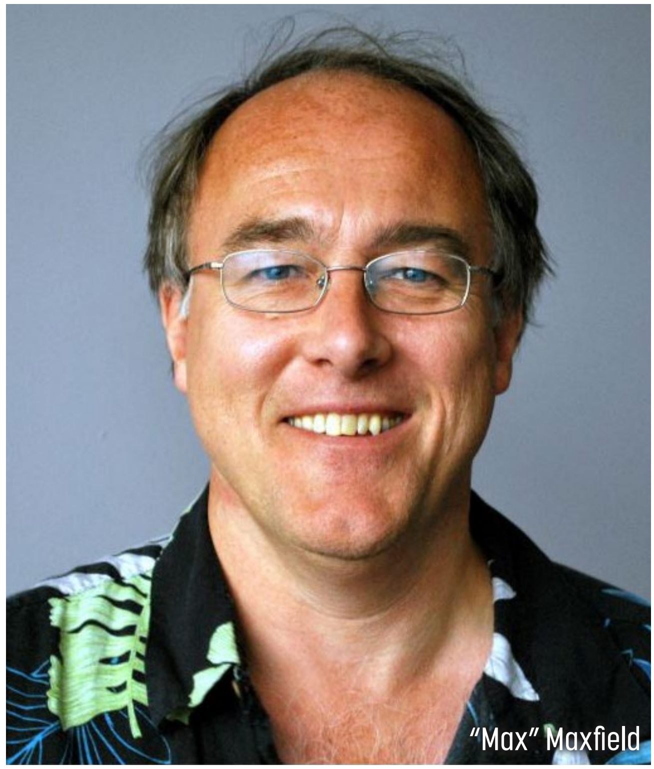
Figure 4: Add on to timeless note on impedance expanding on insertion loss.

also worth speaking with the product specialist and applications engineer at your supplier, as they are often in the best place to guide if a technical note has been superseded; perhaps they can also offer some useful secrets from insiders, which are off the record. But a designer should use their own training mixed with common sense, research, and theory combined with the information in the application note to gather information in a timely way to make the best possible design choices.

If, while reading this column, you have had an idea for a useful signal integrity or stackup related application note, please get in contact. **DESIGN007**

Martyn Gaudion is managing director of Polar Instruments Ltd. To read past columns or contact Gaudion, click here.

With App Notes, Trust But Verify


"Max" Maxfield

Feature Interview by Andy Shaughnessy
I-CONNECT007

For this month's issue on app notes, I checked in with Clive "Max" Maxfield. The author of *Bebop to the Boolean Boogie* and countless technical articles and blog posts, he has been involved with all manner of app notes and datasheets during his long career. Max shares his thoughts on data. As he points out, "Remember that these documents are not written by an 'infallible creator.'"

Andy Shaughnessy: Max, you've been involved with electronics design in a variety of different segments: ICs, PCBs, programmables, etc. Give us an overview of how you view the role of app notes vs. datasheets for a PCB designer or EE.

Clive "Max" Maxfield: First of all, you can have datasheets and app notes for all sorts of things, including subsystems, systems, products, machines, etc. From my perspective, I usually think of these in terms of electronic components—either discrete components like resistors, capacitors, diodes, transistors, or more complex components like integrated circuits, from op-amps all the way up to microcontrollers and the like.

In this context of datasheets vs. app notes, they serve two different purposes. A datasheet summarizes the characteristics and performance of a component. By comparison, an app note briefly describes how a component is to be configured and used for a specific example application.

Shaughnessy: Have you ever written any datasheets or app notes?

Maxfield: I haven't written any datasheets, but I have written a couple of application notes.

Shaughnessy: We've heard from some PCB designers and EEs who work for companies that demand that they follow app notes, but some well-known design instructors advise designers that IC makers' app notes should be assumed wrong until proven right. What's your take on this?

Maxfield: I think app notes can be really useful when you are first starting to work with a

new component or with a known component in a new or "new to you" configuration or application. Having said this, I think everything you read, including datasheets and application notes, should be assumed to be lacking or misleading or indeed incorrect/wrong just in case.

Shaughnessy: It looks to me as if IC manufacturers' datasheets are more grounded in facts, while app notes are sometimes overly optimistic. Some designers say app notes are used more like marketing materials. Is that a fairly accurate description?

First of all, you can have datasheets and app notes for all sorts of things, including subsystems, systems, products, machines, etc.

Maxfield: It depends on who wrote the application note and what their purpose was. In some cases, it's true that the marketing team instigates the writing of an application note, in which case they may serve a marketing role. In other cases, the writing of the app note was triggered by the engineering department that created the document in an honest attempt to show how that component may be used.

Shaughnessy: I've been wondering why there isn't something like a Yelp for app notes and datasheets. Is it worthwhile to let an IC manufacturer know that their app notes are inaccurate? It seems like they would want to know either way.

Maxfield: I think that would be awesome, so long as the people writing the reviews were verified as real users and not paid or company sycophants. But I'm not aware of anything like that out there; on the other hand, I don't design things professionally anymore.

Shaughnessy: Have you found any app notes or datasheets that are dead-on accurate? I've heard that app notes for memory devices seem to be a little more accurate because they're more conservatively written.

Maxfield: Sure, but only the ones for simple components. In the case of more complex components, how could you tell if it's dead-on accurate? It may look dead-on accurate, but how would you know if it wasn't? On the other hand, I've seen lots of app notes that left something to be desired because they left important information out; the desired information was included, but not in an easy-to-access way; they presented some of the information in a "fuzzy" way that was subject to misinterpretation; or they were blatantly and obviously incorrect regarding one or more details.

Shaughnessy: What advice would you give to a new EE or PCB designer as far as dealing with app notes and datasheets?

Maxfield: First, read Elizabeth Simon's series of articles about how to use datasheets. Second, after you've made yourself familiar with the document—datasheet or app note—go to an experienced engineer and ask him or her to go through it with you line-by-line so you can learn how they look at it and read it and understand it.

Third, follow the maxim "Trust, but verify." Remember that these documents aren't written by an infallible creator—they are written by people who may have been delegated to the task and have no interest in doing it. The writers may not know the device as well as you might hope, or they may know it, but they might not be very good at explaining things. They may also simply forget to capture some vital piece of information or accidentally write a "3" when they meant to write an "8."

Shaughnessy: I appreciate your time, Max. Thanks for your help.

Maxfield: Thank you, Andy. DESIGN007

Do You Really Need That Ferrite Bead in the PDN?

Quiet Power

by Istvan Novak, SAMTEC

People who do power distribution design know that it is not easy. I would argue that power distribution design is even more challenging today than signal integrity. Not downplaying the technical challenges of high-end signal-integrity designs, my point is that signal integrity today can be considered a rather mature discipline, where system designers, circuit designers, and PCB designers alike can rely on a number of standards and other forms of help and support.

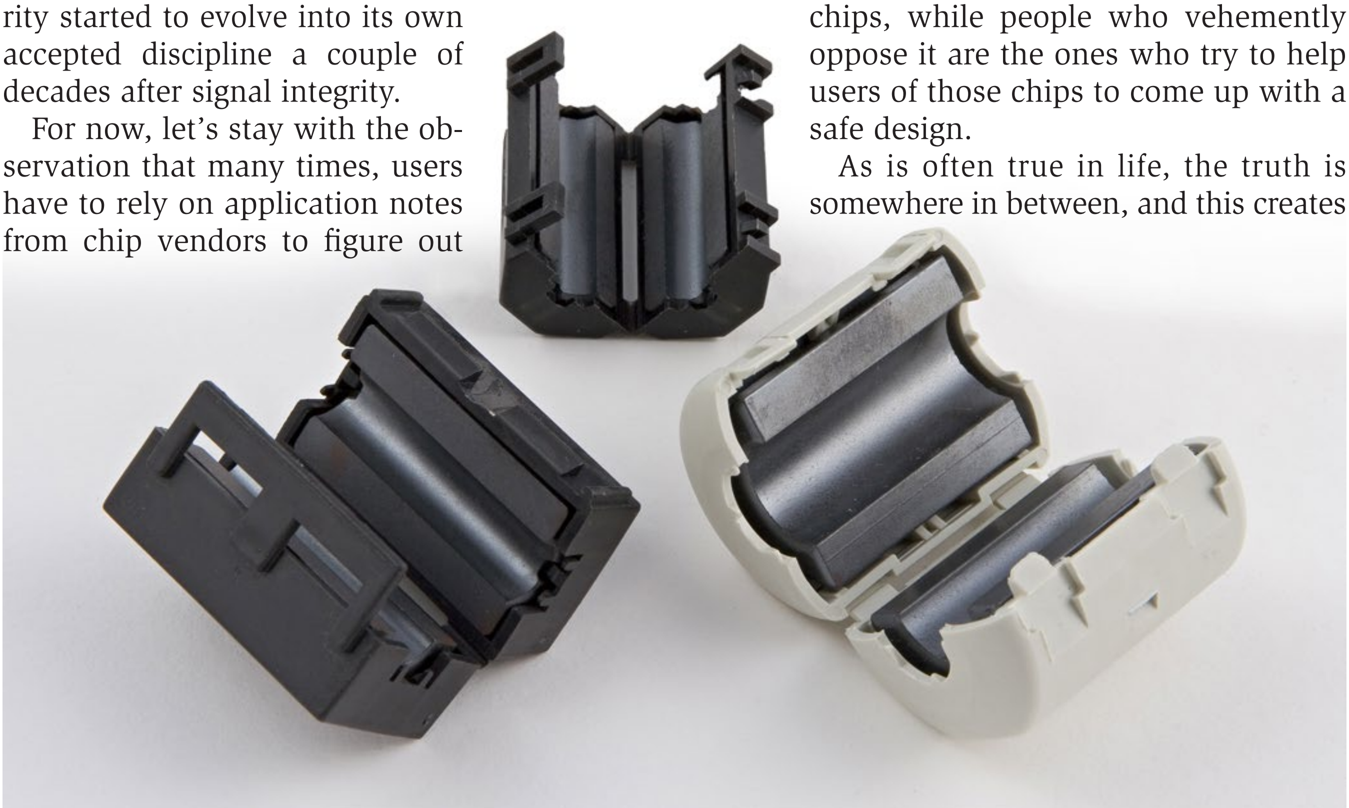
Power distribution design, on the other hand, is much less mature as a discipline, and the industry has not figured out yet what is the best way to help designers and users. This has multiple reasons, not just the fact that power integrity started to evolve into its own accepted discipline a couple of decades after signal integrity.

For now, let's stay with the observation that many times, users have to rely on application notes from chip vendors to figure out how to design the power distribution network (PDN) for the active device. Within this still vast area of application notes, I'll focus on just one question that greatly divides even the experts: Is it okay, necessary, or harmful to use ferrite beads in the PDN?

There are two distinctly different camps: people in the first camp will categorically denounce the use of ferrite beads in PDN, while people in the second camp will insist that the PDN they came up with need the ferrite beads.

I start with a generic observation. I admit it may be a little bit of a generalization, but anyway, here it goes. It appears to me that people who suggest using ferrite beads tend to be writing application notes for their company's chips, while people who vehemently oppose it are the ones who try to help users of those chips to come up with a safe design.

As is often true in life, the truth is somewhere in between, and this creates



a third camp of people—mainly the practicing system and circuit designers—who may be torn apart between the opposing opinions, suggestions, and recommendations. To add insult to injury, designers working on fast-paced projects may often lack the time, resources, and information to come up with the correct answer themselves.

Before we start the illustrations, we need to nail down a few things about the nomenclature. Over the years and decades, as the complexity of our electrical systems have grown, people have been using different names for the same thing. For example, the switching regulators of any kind—whether it converted AC to DC, DC to DC, or DC to AC—were all called switching-mode power supplies decades ago. Later, the regulators converting DC to DC were called DC-DC converters, and in recent years, we started to refer to those circuits and devices as a voltage regulator module (VRM).

For the purposes of this column, and to differentiate between the fundamentally two kinds of ways that we can try to keep the power noise within limits, I will call the passive part of the PDN that has only parallel bypass capacitors "parallel PDN," and I will refer to a circuit that has an intentional series element (an inductor, a ferrite bead, a resistor, or a combination of these) for the purposes of blocking the propagation of noise a "PDN filter."

Note that on PCBs, the plane shapes, patches, traces, vias—which are part of the current-carrying path—will also have series resistance and inductance, and most designs consider them as side effects that have to be minimized. On the other hand, these parasitic series elements can also be part of intentionally created filters around them [1].

These two different kinds of circuits can be combined to create various power distribution trees. Figure 1 shows a small part of the block schematics of a larger system PDN that we may have today, illustrating these two kinds of passive circuits. Here we assume that the PDN filter is placed on the output of a voltage regulator (for instance, to further suppress the switching ripple), but we can also use PDN filters on the input side of a noisy DC-DC converter to block noise from spilling out to the upstream distribution network [2].

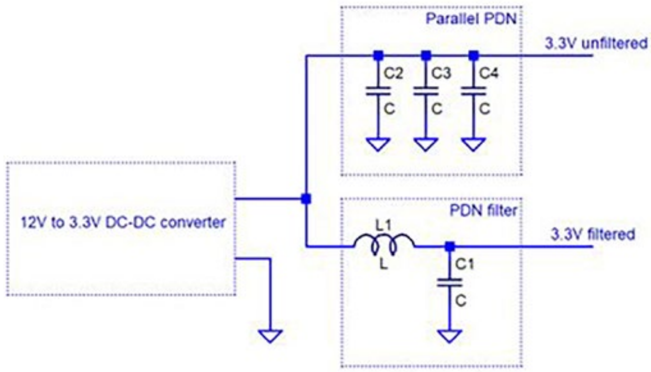


Figure 1: Block schematics of a system PDN illustrating the definitions of parallel PDN and PDN filter.

These block schematics are highly simplified: three capacitors are shown in the parallel PDN path, but it can be a mix of any number of same-valued and/or different-valued capacitors. Similarly, the PDN filter can be more complex, having an entire parallel PDN on its output composed of multiple capacitors, and the series path can be more complex—for instance, having series and parallel resistors around the inductive component.

To illustrate the situation and to help explain what should go into the design decision considerations, take a look at the PDN filter circuit suggested for an encapsulated oscillator circuit (Figure 2).

The intended use of this filter was feeding an oscillator circuit with a few milliampere current consumption. If we choose to use filter components by Murata, we can select, for instance, a BLM21PG221SN1 0805-size ferrite bead for L1, a GRM219R60J226ME47 22uF 0805-size X5R 6.3V capacitor for C1, and a

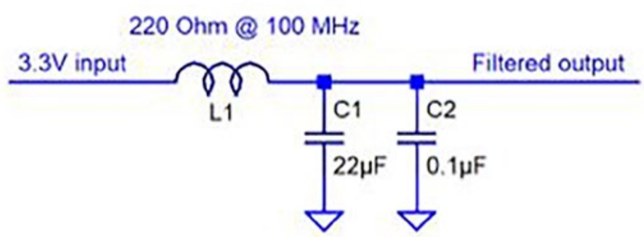


Figure 2: Suggested schematics of a PDN filter with a series ferrite bead.

GRM155R61H104ME14 0.1uF 0402-size X5R 50V for C2.

These components have detailed simulation models [3], including temperature and DC bias dependence, so that we can use a circuit simulator to predict what happens, or we can purchase the components, build the filter, and measure it. If we have the time, we should do both. But before we do either simulation or measurement, we need to decide what to look at.

For filters connecting a high-power noisy rail to a low-current sensitive consumer, the proper metric is the input-to-output voltage ratio [4]. In simulations, we can achieve this by using a voltage source for excitation and simulate the output voltage. In measurements, since having a source with zero internal impedance is not feasible, we need to use an instrument with one source and two inputs to measure complex voltage ratios. These instruments are commonly called frequency response analyzers, and they allow us to measure the ratio of input and output voltages of the filter.

When we build and measure and simulate this filter's input-to-output voltage transfer function, we get the plots shown in Figure 3.

There are two notable features of these plots. First, on the left plot, we see two lines—one for each condition of DC bias. The blue line shows what happens without input power when the input voltage is zero, and the red line shows the transfer function with 3.3V DC applied to the filter. The DC voltage across the capacitors and the load current consumed by the oscillator may change the characteristics of the filter components.

In general, we should not worry about the blue line because when there is no input DC voltage, the circuit does not work. However, the difference between the blue and red lines is a reminder that unless we take the biasing conditions of the DC voltage and DC current into account, we may get the wrong answer. For instance, with this filter at 200 kHz, we get only 20 dB attenuation when the circuit is powered up. Without the DC bias, we would see attenuation of 30 dB. The simulated response with 3.3V DC bias is shown on the right.

The second notable feature, which is equally troubling, is the peaking of the transfer function at 57 kHz. The peak value is 14 dB, which translates to a five-times voltage amplification of the noise. Instead of making the noise voltage smaller, this filter makes it five times bigger at 57 kHz. We may argue that the purpose of this filter is to suppress the output ripple of a DC-DC converter, and these days, the typical switching frequency is above 200 kHz. This is true, but wide-band noise of the converter and the noise generated by the loads connected to the unfiltered rail still could produce significant energy at lower frequencies, including 57 kHz. The peaking of the filter behaves like a high-Q band-pass filter, and if we have

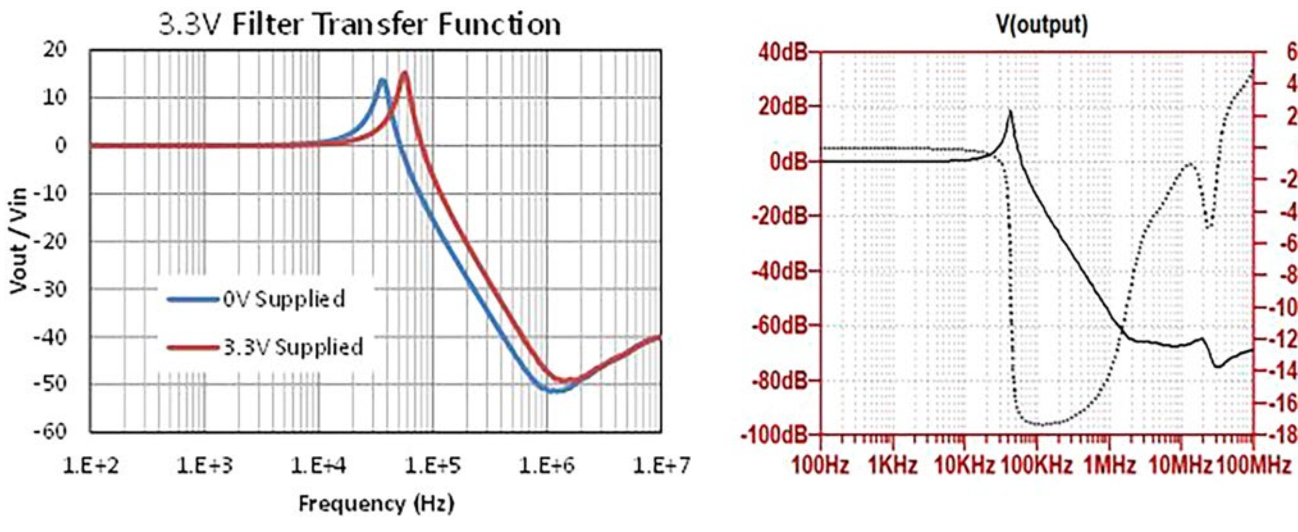


Figure 3: Measured (L) and simulated (R) voltage transfer function of the filter circuit in Figure 2.

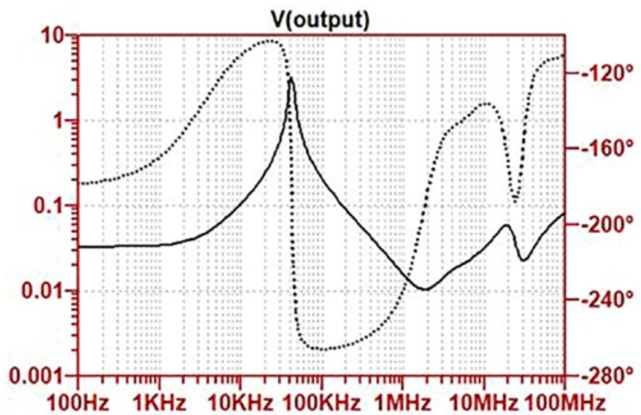


Figure 4: Impedance of the filter looking back into its output.

a noisy unfiltered rail, we can watch on an oscilloscope the output of the filter producing a sine-wave looking ripple.

Also, when we look at the impedance presented by the filter to the load (Figure 4), it has a significant peak at the same frequency where the transfer function had its peak.

Whether this impedance peak poses a risk and potentially might create a problem heavily depends on the nature of the load. If the load is "quiet" in the frequency range of the peak, the peak in the filter's output impedance creates no problem. For instance, if the load is an oscillator with no digital logic (say a synthesizer) in the same package, this impedance peak would create no issue. But the peaking of the transfer function is still a risk of getting noise on the oscillator supply pin, which will create a higher-than-expected jitter on the output.

If we conclude that the peaking in the transfer function and/or in the output impedance is a risk we want to eliminate, the solution is to add sufficient damping to the circuit. We can do it in multiple ways. We can simply add a larger capacitor with higher ESR on the output, like an electrolytic or tantalum capacitor, or we can add a discrete resistor in series to the 22uF ceramic capacitor. We can also try the add a parallel resistor across the inductive element. Finally, for filters that need to handle only very little DC current, we can add a series resistor.

If we want to totally eliminate the peaking in this particular filter without modifying the series path, we need to increase the total capacitance, so adding a separate lossy capacitor would be the simple way to go. Although in LTSpice, ESR and ESL of a capacitor can be added as attributes of the capacitor symbol, and for the sake of clarity, those elements are shown in the following figures as separate components. Figure 5 shows the SPICE deck and the result for the transfer function when we add a 220 µF tantalum capacitor with 0.1 Ohm ESR. On the left, the schematics in LTSPICE has a voltage source connected to the input. With flat unity source voltage, the output gives us directly the transfer function. The plot on the right shows the magnitude of the transfer function with a solid line and the phase with a dashed line.

Figure 6 shows the SPICE deck and result for the output impedance. To simulate the imped-

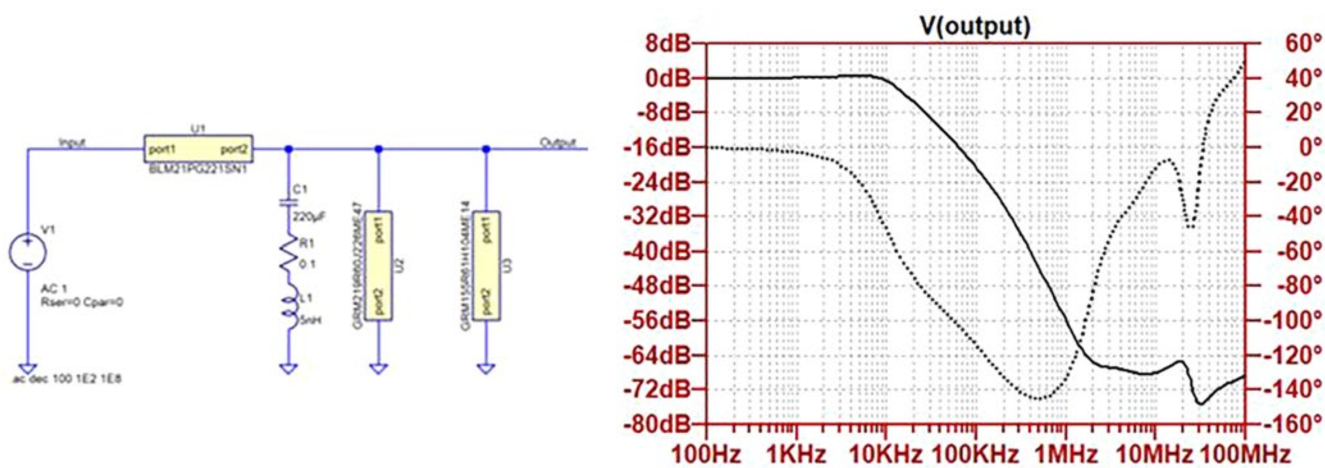


Figure 5: Voltage transfer function with a lossy capacitor added.

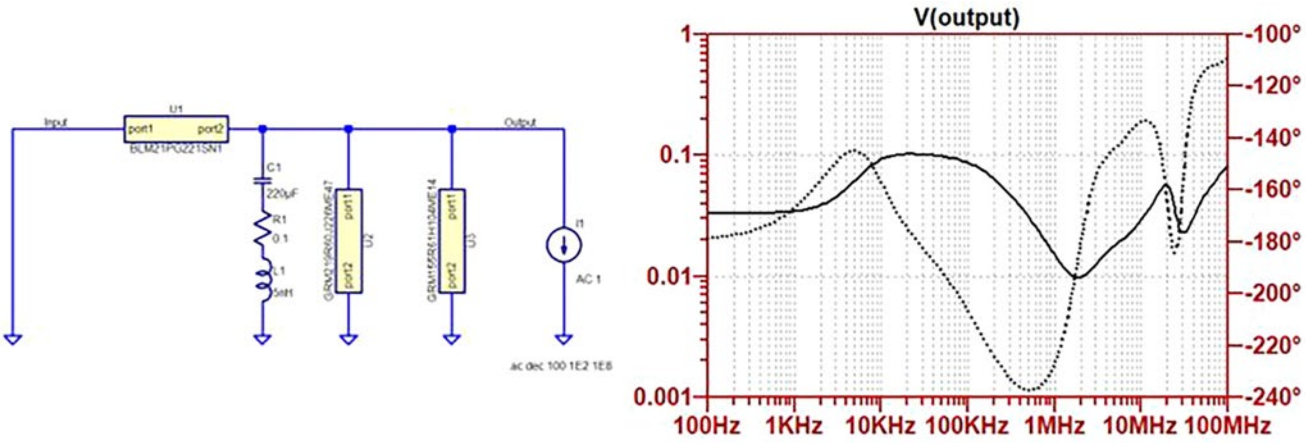


Figure 6: Output impedance with a lossy capacitor added.

ance shown by the filter at its output, we short the filter input and connect a one ampere AC current source across the output. With 1A current, the voltage will be equal to the output impedance. In general, using a 1A swept-frequency AC current source is a convenient way to simulate impedance for any linear network.

Once we see the performance of the filter suggested in the application note and understand the potential risk generated by the peaks in the transfer function and output impedance, we understand what is behind the claims of people who say that ferrite beads should never be used in PDN filters and should not follow the advice of application notes. We can ask why those peaky filters are suggested in the first place. Were those circuits not tested on a real board? Most likely, they were, but we need to see the motivation and the constraints behind those efforts: People creating the application notes for their chips have no way of knowing the many different possible applications their users will come up with. The testing is done in an evaluation board on the bench. This immediately removes most of the external noise and leaves only the noise that is created by the circuit being tested. If the self-generated noise is acceptably low, the circuit will properly function, though we could argue that in a low-noise environment, we would not need such a filter in the first place.

We can also see that if we design the filter properly, it should work fine, and it gives such isolation benefits that would be much harder to achieve without a series inductive element. In this short column, I have not looked at the other benefit of using ferrite beads as opposed to regular inductors: Ferrite beads have an AC series loss that increases with frequency sharper than what we get from regular power inductors, and it further helps blocking the propagation of high-frequency noise. If you are interested in the full process how to design PDN filters with no peaking, you can view my presentation on “How to Design Good PDN Filters” [4]. **DESIGN007**

References

1. Alex Waizman, et al., “Novel Isolation Scheme for Mitigating PDN Coupling,” DesignCon 2018.

2. C. Zhang, “Analysis and Design of Input Filter for DC-DC Circuit,” Texas Instruments Application Report, SNVA801, November 2017.

3. Murata SimSurfing tool.

4. I. Novak, “How to Design Good PDN Filters,” DesignCon 2019.

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.

High-Frequency Circuit Material App Notes

Lightning Speed Laminates

Feature Column by John Coonrod, ROGERS CORPORATION

High-frequency circuit materials are used in a variety of diverse applications. Due to this diversity, it is difficult to write an application (app) note for a specific high-frequency circuit material.

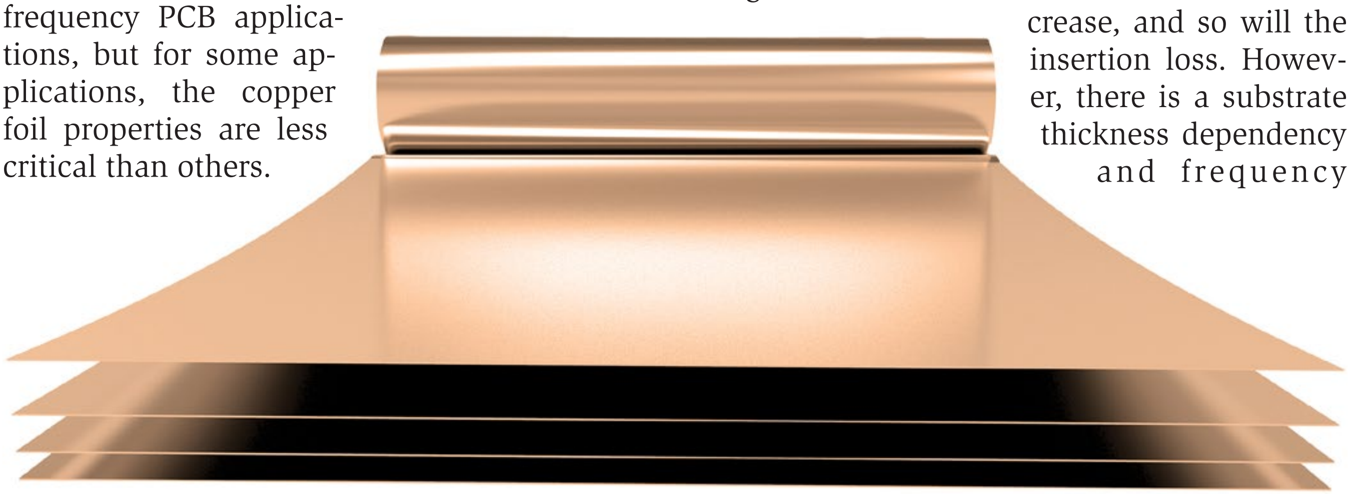
Combine that with the fact that one designer can use a different circuit design technique than another designer for the same circuit application, and that could make one high-frequency material a better choice than another based on design technique. Rogers Corporation has app notes, but not as many as one may assume, and for a good reason.

The app notes for high-frequency circuit materials are typically related to a large-scale topic. For example, Rogers has an app note on the copper foils used in high-frequency PCB applications. This app note covers the different manufacturing processes to create a copper foil, different types of copper foils, and properties of the copper foil as they relate to high-frequency performance and measurement values of the critical properties.

Copper foil is obviously critical to high-frequency PCB applications, but for some applications, the copper foil properties are less critical than others. The same circuit application may be very sensitive to copper properties at high-frequency, but at lower frequencies, the copper properties may be much less critical. Unfortunately, the topic of critical copper properties for different RF PCB applications is a large subtopic and cannot be addressed in a relatively short app note.

To expand on the copper comments, the copper surface roughness of a copper foil can be very influential for high-frequency applications. To be more specific, the copper surface roughness at the substrate-copper interface can have an impact on insertion loss and phase response, which is related to dielectric constant (Dk).

Basically, if the copper surface is rougher, the propagating wave will slow down, and the circuit will behave as though it has a higher Dk. This is true regardless of the Dk property of the substrate. Additionally, a rougher copper surface will cause increased conductor loss, which is a component of insertion loss. As copper roughness increases, the conductor loss will increase, and so will the insertion loss. However, there is a substrate thickness dependency and frequency



dependency for the effects that roughened copper has on high-frequency performance.

If the circuit application is operating at a low frequency, the copper surface may not have any influence on the RF performance. However, when frequency increases to the point of the skin depth having the same dimension as the copper surface roughness, the surface roughness can certainly impact insertion loss and phase response. As the frequency continues to increase, which causes a thinner skin depth, the roughened surface of the copper will have an increased influence on RF performance.

Continuing with more detail on the copper foil surface roughness, the substrate thickness can exaggerate the RF effects of copper surface roughness. Here's a simple analogy: When the copper planes are close together, which is the case for a circuit using a thin substrate, the copper surface roughness effects will be more significant. As opposed to using a thick substrate and the copper planes being farther apart, the copper surface roughness will have less impact on the RF performance of the circuit.

When the copper planes are close together, which is the case for a circuit using a thin substrate, the copper surface roughness effects will be more significant.

Another example of a high-frequency circuit material app note is related to space applications. The circuit material testing requirements for space are well-defined through agencies like NASA and other similar organizations. Rogers has an app note which gives a summary of specific high-frequency circuit materials tested per NASA requirements.

There are typically many different circuit applications for one space system, and each circuit application may have different material property needs; however, the overriding specification may be the NASA data, for example. Given the list of high-frequency circuit materials that will meet the NASA requirements, some materials may be good for power amplifier applications, and other materials may be good for antenna radiating elements. Typically, the designer will need to perform a trade-off analysis with the list of materials that are acceptable to the NASA requirements and consider the different properties of the materials for their space application.

Another app note that Rogers offers is related to laminates which have resistive foil embedded in the laminate. The resistive foil laminates can be used for a wide range of PCB applications, and the app note cannot cover all aspects of these applications.

The app note can, however, cover the critical topics associated with these types of laminates, and the topics may or may not be applicable to a specific application. For this app note, the main issues related to resistive foil laminates are discussed. In general, these issues are a normal variation of the visual appearance of the resistive layer and the resistor tolerance which can be achieved by using these laminates to make planar resistors.

While app notes are a very good tool for the designer to use, the app notes which a high-frequency circuit material supplier will issue are typically not like an app note that a component supplier would issue.

The app note for a material supplier will usually address the different nuances of the material, which may or may not be critical to a particular PCB application. It is the designer who will need to understand how the app note applies to their design, but it is always suggested that the designer consults with the material supplier if there is any confusion on the app notes or related datasheets. **DESIGN007**

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

APEX EXPO IPC 2021

TECHNICALLY SPEAKING: IT'S THE PLACE TO connect

Jarrod Webb
Manufacturing Engineer, Lockheed Martin

Meetings & Courses:
January 23 – 28

Conference & Exhibition:
January 26 – 28

SAN DIEGO CONVENTION CENTER | CA

2021 Call for Participation

Join us at **IPC APEX EXPO 2021** as a presenter or instructor and share your latest work with the electronics industry. The time to plan your participation is now!

IPC APEX EXPO is the electronics industry's premier technical conferences and exhibition, drawing an international audience to experience the latest advancements in design, materials, assembly, processes, inspection, and equipment. Areas of interest are diverse and ever-changing to keep up with the pace of innovation.

Presenting at **IPC APEX EXPO** provides visibility for you and your organization. The technical conference proceedings reach an audience of thousands and are regarded as the best place to follow current technologies and best practices. Your material will be seen by key managers and executives from all segments of the worldwide electronics industry.

To recognize exceptional achievement, an award will be presented for "Best Paper."

Technical Conference

Provide an abstract of approximately 300 words that summarizes technical work, covering case histories, research and discoveries. Authors of papers selected for the conference will each receive a detailed speaker manual to assist them with numerous items, including the formatting of their papers and presentation slides. Speakers receive a complimentary technical conference pass for the day they present and special pricing for additional conference options.

Conference Paper Timeline

Abstracts due **June 20, 2020**

For more information on TOPICS, SPEAKER BENEFITS and SPEAKER RECOGNITION AWARDS, please visit www.ipcapexexpo.org/cfp.

Any questions or special requests, contact **BrookSandy@ipc.org**.

IPC APEX EXPO is in San Diego in January 2021, and we plan to proceed as scheduled.

Clearing Up App Note Confusion

Feature Interview by the I-Connect007 Editorial Team

The I-Connect007 editorial team recently spoke with Kelly Dack about some of the challenges designers face when dealing with app notes and datasheets. Kelly helps clear up some of this data confusion, and he offers advice to younger PCB designers and design engineers who aren't sure whether to trust this data.

Barry Matties: Kelly, we often have design experts standing up in front of large crowds of designers and telling them that app notes and datasheets are not reliable. Can you tell us what an app note is from your point of view? What is the difference between an app note and a datasheet?

Kelly Dack: The term "app note" is short for "application note." It documents how the part performs in a given application. On the other hand, a datasheet typically documents the physical properties of a part: the size and shape of its available package, its pin assignments, and quite often, a recommended footprint pattern for soldering the part onto the PCB assembly. Manufacturers publish datasheets for all parts, but not all parts are supplied with app notes.

Andy Shaughnessy: Why do some industry layout experts consider app notes unreliable?

Dack: While I don't disagree with that perspective from some of the expert designers of the world, I am aware of the manufacturer's responsibility to define their product and the performance of their products responsibly in these app notes. I want to be able to trust them, but I've read enough horror stories told by our industry experts to know that changing the circumstances—working a complex part or chipset into a layout which varies greatly from the app note's surface topology or materials stackup—can lead to some gruesome results. There are so many stories regarding app notes and even datasheets being unreliable that I can see how it can be confusing to a new PCB engineer.

Shaughnessy: How should an inexperienced PCB engineer move forward if they cannot trust an app note?

Dack: I like to say that a PCB layout designer needs the creativity to see six different ways to approach a design challenge. An app note usually provides only one way. And without experience, reliance on an app note is the only way for some designers to move forward. An app note serves as a starting point for the design. As a designer, I recognize that I am not always enough of a subject-matter expert to challenge the validity of an app note for a complex IC as it relates to a given set of manufacturer-controlled circumstances. In cases where I find myself in over my head, it is important for me to stay well connected with the experts. I attend trade shows where I can touch, listen, and discuss design considerations for components. If I do not understand the information in a datasheet or app note, I will first make an effort to contact the supplier to discuss the design criteria with an applications engineer. For alternative and more in-depth perspectives, I read magazines like *Design007 Magazine*.

I tell a story in my CID design certification classes regarding some design app notes for LEDs. I designed a lot of LED boards in my career as a designer for a gaming company. At the time, I worked with engineers who were constantly upgrading our light boards with the latest LEDs to provide stunning backlighting for our machines. The form factors for most of these PCBs had already been determined months or years prior. Many were narrow PCB strips only 1.5 inches wide with driver chips, discrete parts, and circuitry filling up the secondary side. The primary side was reserved for banks of bright LEDs. Over several years, LED technology exploded, and I recall a time in which some of these bright LEDs began getting really hot. They came with special app notes describing how to lay out patterns in the copper topology to serve as a heat sink to help cool the parts.

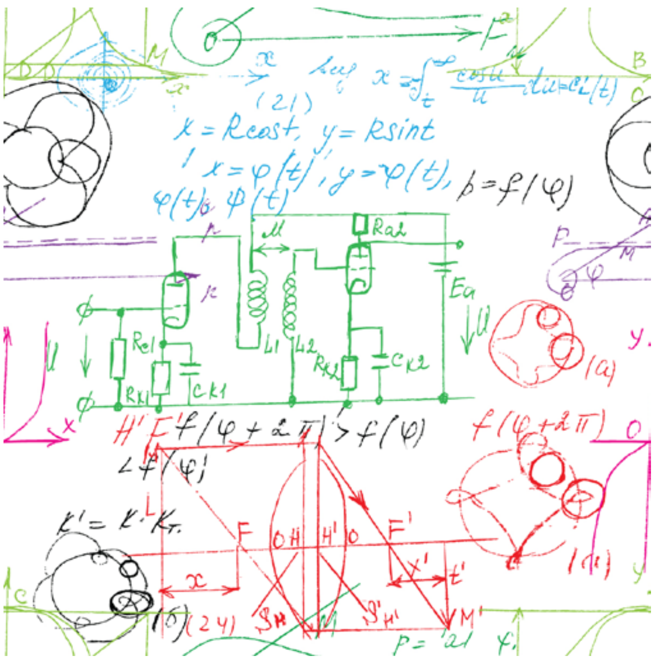
Now, I didn't doubt these app notes—especially after I'd already burned my fingers touching a few operating samples of these parts. I knew they needed heat sinking to prevent PCB surface damage or their own soldered power contacts from reflowing. But you know what? Though I never distrusted these app notes, I never had a chance to follow them either. The app notes for these LEDs provided a single, lab-proven solution to keep these little buggers from burning up. The notes recommended that all the designer had to do was to surround the LED with three square inches of copper attached to the center contact of the LED, solder it on, and everything would be okay. They even provided a lab photo of what the etched copper should look like. With its exposed ENIG finish, it was so large I could almost see my own full-face reflection on its surface, saying, "OMG! How would I fit this much copper onto the limited surface area of my design?"

I called a meeting with our industrial engineers who had selected these parts. I included our mechanical engineering team that had already been incorporating these parts into their mechanical layouts before our electronics engineering group had even heard they were coming. To make a long story short, I'll just say that I wouldn't have formally proposed that these engineering teams widen the 1.5-inch PCB strips to 3.5 inches to allow for heat sinking unless I fully trusted those app notes. But frankly, I was told the mechanical designs were frozen and would have to find another way. Ironically, it was mechanical designers who had disregarded these app notes and specified them to be incorporated onto these skinny PCBs. Now what?

I relate this story to drive home the point that PCB design should not always involve ignoring app notes or manufacturer's recommendations. Good design may involve considering the app notes first to understand the context. But an app note usually provides only one solution. I think we designers need to be able to come up with more ways to proceed. Explaining to your engineering stakeholders that an app note is the only way to move forward may not end well.

Shaughnessy: Did you end up trusting the app note or disregarding it?

Dack: By applying previous design experience, I was able to apply the thermal dissipation requirements addressed in the app note and apply its veracity in a required, alternate form factor. I'd previously worked with other manufacturers' LEDs who recommended the use of thermally conductive vias underneath the part, which were to be interconnected with thermal planes on the secondary side of the PCB. This technique dispersed the required surface area for cooling to both sides of the PCB. To make room for the secondary side planes, I had to add internal routing layers. To provide solid return paths for the internal signal layers, I had to add inner layer planes.



By the time this design was completed, it looked nothing like the simple but expansive single layer solution illustrated in the app note. But by leveraging experience gained by using techniques for dissipating heat from other types of components, I was able to merge the requirement with the given mechanical constraints. Granted, this was not an example of good engineering, but it is an example of using design experience to quickly consider all of the design constraints and come up with a viable solution for an awkward hand that was dealt.

Matties: My understanding is that a component may have an app note, and there's also a datasheet. I'm trying to understand, and help our readers understand, the difference between an app note and a datasheet because they're almost used synonymously in conversation.

Dack: It can be confusing, but it shouldn't be. My take, as I alluded to earlier, is that suppliers publish datasheets for most components. But only complex parts or ICs with critical applications require accompanying app notes. The app note describes how the electronic aspects of a part is supposed to perform as applied to a given set of criteria. Our advanced PCB design educators make a good point that in PCB design, we cannot always stick with the manufacturers' given layout criteria to achieve performance. We must often work the component into a PCB environment that does not exactly match the optimized two-layer, four-square-inch surface area that the manufacturer had based all of their testing on. Without design experience or engineering ability to run complex analyses or time-consuming DOE and testing, new designers can get into a lot of trouble taking things for granted and applying the component to a design scenario that does not exactly match the app note.

Datasheets for components, on the other hand, have been knocked as unreliable too. Datasheets are supposed to document the physical and mechanical attributes of an electronic component, including package size and critical dimensions, pin function and assignment, polarity, and component footprint pattern for assembly. But all too often, datasheets from reputable suppliers are missing critical information. I have seen datasheets for components improperly mirrored or missing critical alignment dimensions. Recommended footprint patterns for smaller parts often do not specify or take in to account the type of solder that will be used or reference the stencil thickness for optimum solder deposition. This has led to misdirected finger-pointing between design and manufacturing stakeholders. One relies on the datasheet for recommended geometry, and another must account for those recommendations and then select an appropriate solder paste, stencil thickness, and oven profile to get each component to process successfully with all others.

When I build a component footprint to fit onto a circuit board, I'm faced with the options of relying on the component supplier's specifically recommended footprint or ignoring it and creating the footprint geometry based upon the IPC-7351 land pattern calculator.

The way I see it, there are holistic design and manufacturing benefits to be gained by putting aside the supplier's recommended footprint specs and opting for the IPC-7351 solution. Unique supplier parts are not consistent with manufacturing criteria required by other supplier's parts. Why not use a consistent component land calculation for all component parts? This philosophy usually rests well with our manufacturing assembly stakeholders. I encourage all new designers to check in with their assembly manufacturing stakeholders while in the process of creating footprint land pattern geometries as it affects the outcome of their target solder fillet conditions.

Matties: We've heard from a few veteran designers that an app note is often nothing more than the sales pitch for the device. Is that how you see it?

Dack: I can relate to that. An app note might be like the operating criteria for a four-wheel-drive vehicle. While the salesperson on the commercial shows the vehicle going up a steep canyon at a 45-degree angle, the app note says, "Don't do this or you could die." That's the app note. But the physical specification datasheet is in another book that lists the physical properties—its engine size, horsepower, and turning radius. Data specs are legal and factual in context but do not infer performance.

Matties: The datasheet is sort of the mechanical or physical specification, and the app note is where you start crossing over into the functional spec.

Dack: Yes, from a component standpoint. There are a lot of different contexts here. Even PCB laminate materials have app notes. Teflon materials and polyamide materials are going to have performance specifications and app notes or application engineers who will show their best utilization for performance. I wouldn't doubt the reliability of those app notes or specifications. But I would definitely seek wise council from an applications engineer before incorporating them into a design.

Matties: Right. I understand that the app notes for a component will say that it performs at this level on this certain material. However, it may also perform with a lower-grade material as well but not give you quite the same performance. What triggers in your mind, as a designer, the need to question this or not? Because you can't question every single app note that comes through, can you? You don't have the time.

Dack: Correct. I am not a scientist. I'm a designer with lots of layout experience who works with a customer who may have a need to design a complex part into a legacy product design. Sometimes, the customer is a scientist or an engineer who will specify what they want. Other times, they're trying to spec a component into a design realm that is totally outside the configuration it was intended for. In this case, an app note can be a helpful base starting point to at least put some context into the required design intent.

Matties: I appreciate that. That helps clarify the issue for me.

Andy Shaughnessy: Kelly, what would you tell a group of young designers you were mentoring regarding datasheets, app notes, footprints, and how to consider all of this data?

Dack: I mentor younger designers by encouraging them to engage with industry experts. Attend industry trade shows and seek IPC designer certification. There is a new wave of educational resources coming out of the IPC educational resource committee willing to help fresh engineers. There is also new, dynamic energy building up within the Printed Circuit Engineering Association (PCEA), who are ready to connect them with active, experienced stakeholders in the printed circuit industry. I tell them to read articles and interviews with Rick Hartley, Lee Ritchey, and Eric Bogatin. If they are already laying out a board, I would advise them to expand their knowledge base by reading layout articles and taking classes from Susie Web. Of course, I'll pitch a few of my articles on considering manufacturability. And there is a whole library of eBooks offered by I-Connect007 for free.

PCB design and engineering are all about trade-offs. Just because you have the latest, greatest component doesn't mean you will have a successful design until it has been proven to perform electronically and be manufacturable. You have to learn to rob Peter to pay Paul in design and engineering. There are lots and lots of things that have to come together, and you will only recognize the most important ones by gaining experience on your own or by gleaning it from others. That's what I tell them.

Many folks in our design community are 55 years old or older. We have so much experience because we've seen almost everything before. It is usually easy for us to see six different solutions based on the given circumstances of cost, reliability, environment, availability of materials, and timeframe. We can put all these things together with experience very quickly. This is the benefit of experience that the industry is going to be losing very fast.

Matties: And circling back to your original point, it takes experience to know when to question an app note, a datasheet, or any data, for that matter. I appreciate your time answering these questions. Thank you, Kelly.

Dack: I've been glad to. I hope I've helped! **DESIGN007**

Split Planes: Reprise

Beyond Design

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

A high-speed digital signal crossing a split in the reference plane impacts at least three aspects of design integrity: signal quality, crosstalk, and EMI. The problem is the impedance discontinuity in the return signal path crossing the split. Any discontinuity in impedance reflects energy back toward the source, particularly the higher-frequency components of the signal. At high frequencies, the return current follows the path of least inductance, which is directly above and/or below the signal trace, but that path is broken by the split. In this case, the return current has to find an alternative path back to the source, creating a larger loop area. In this month's column, I will review the two common solutions to this issue, plus introduce a third optimal solution for high-speed design.

1. The Pass-Through Gap

Low-frequency circuits in the audio range can often benefit from splitting the ground planes, whereas digital circuits cannot be effectively isolated from analog circuits using this method. Many years ago, wiring the ground connections to one common point was a great way to eliminate noise in a guitar amplifier, particularly value-based amplifiers. But as frequency increases, the parasitic capacitive and inductive components dominate, and distance or electromagnetic shielding is the only solution.

The key to a successful mixed analog/digital design is functional partitioning, understanding the current return path, and routing control and management—not carving up the ground planes. Analog and digital grounds should be connected together through a low impedance path. It is always better to have just one single reference (ground) plane for a system.

When both analog and digital devices are used on the same PCB, it is usually necessary to partition (not split) the ground plane. The components should be grouped by functionality and positioned such that no digital signals will cross over the analog ground, and no analog signals will cross over the digital ground. Precise partitioning will minimize the trace lengths, improve signal quality, minimize the coupling, and reduce radiated emissions and susceptibility. This is traditionally done by using keep-out zones, whereby no trace can cross through the keep-out area. But this also creates issues in that data and control signals need to go into and out of these sensitive areas.

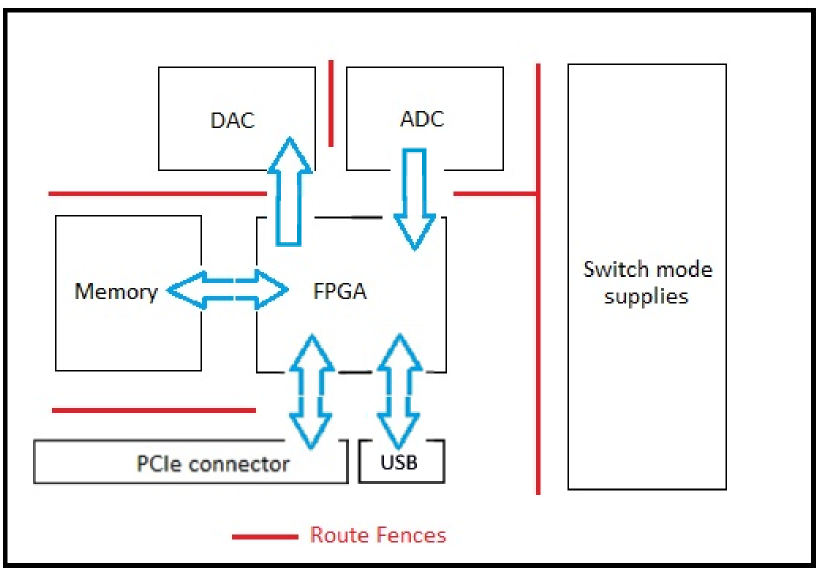


Figure 1: Route fences used to control routing and isolation.

Particular care needs to be taken with oscillators and switch-mode power supplies that may generate high-frequency electromagnetic fields. If space permits, keep these circuits at least 250 mils from any critical signals to avoid parasitic coupling.

Route fences (a thin elongated keep-out), rather than regional keep-out areas, are a useful tool to control the routing. The ground plane should not be split; rather, a pass-through gap is left in the fences so that data and control signals can enter and leave that area, as seen in Figure 1. At high frequencies, return current follows directly under the signal trace on the reference plane, and as such, will follow the trace through the pass-through gap.

2. Bridging Planes of Different Potential

These days, it is quite common to have multiple power supplies on a board. Instead of allocating one or two power supplies per plane, it is best to use the dual-stripline layers to provide mixed-signal/power pours and reduce the layer count. This also prevents the overlapping of electromagnetic fields, which creates crosstalk.

This is particularly critical with dual asymmetric stripline configurations where one or two signal layers are sandwiched non-uniformly between two planes, as in Figure 2. It is important to have a clearly defined return current path and to know exactly where the return current will flow. The question is not, "Which plane does the return current flow on?" but, "How is the current distributed on each plane?" Also, if a return path discontinuity exists, then the current tends to divert increasing the loop area, inductance, and delay.

A via that provides the connection between signal layers, referenced to planes of different DC potential, creates return path discontinuities. In other words, the return current has to pass between the planes to close the current loop, which increases the inductance, affecting the signal quality. This return current can also excite the parallel plate resonance mode, causing significant electromagnetic radiation from the fringing fields.

If the reference planes are at the same DC potential, then they can be directly connected by stitching vias near the signal via transition to provide shorter paths for the return current.

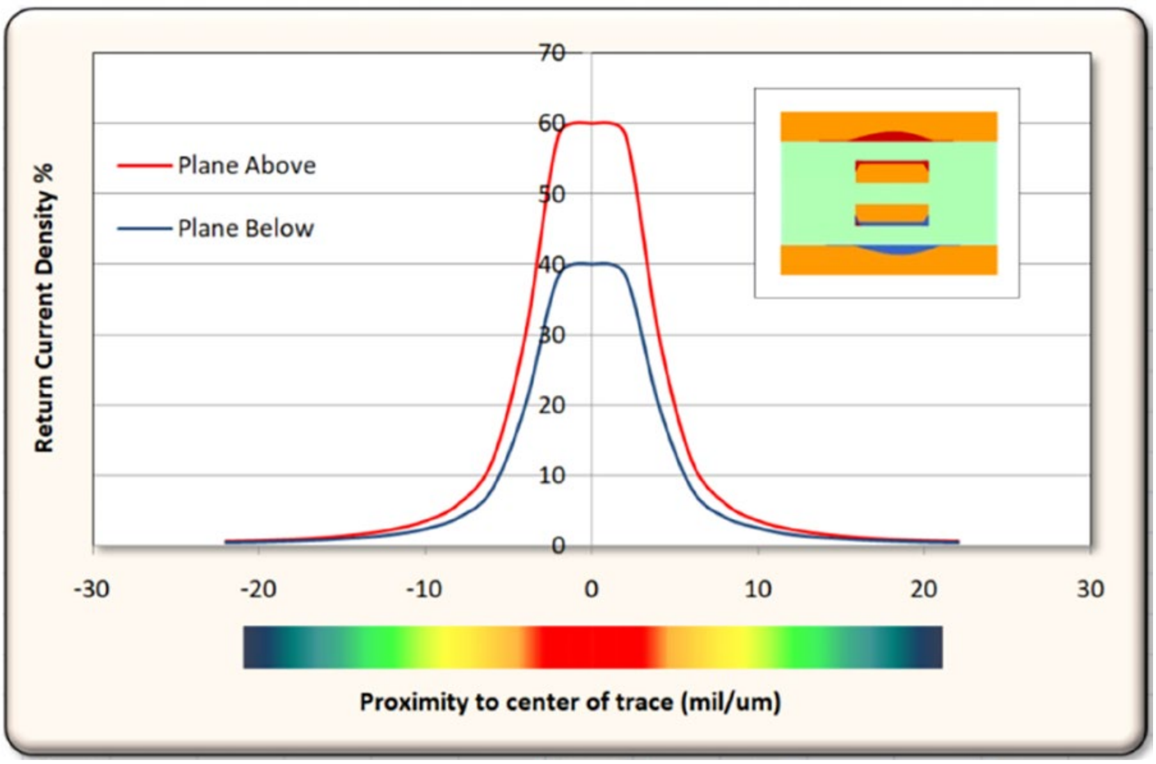


Figure 2: Return-path current density for a dual asymmetric stripline.

Figure 3: Eliminating the transfer of noise in the return path of split power planes (right).

However, if the planes are at different DC potential, then a bypass capacitor can be connected across the planes at these points, as in Figure 3 (left). Unfortunately, this can pass AC noise between the power supplies. Two bypass capacitors, configured as in Figure 3 (right), is a much better solution, as this eliminates the transfer of power supply noise from one supply to another. Although this does add an additional loop area, it also provides additional decoupling to the planes, reducing power distribution network impedance.

3. Planar Capacitance

The aforementioned solutions work for most designs, but are they the optimal solution for a high-speed signal to cross a split plane? A slot in the plane under two or more nearby signal traces will usually cause return currents to overlap (shared return path), resulting in some degree of crosstalk. And since dense designs can have 20 or more power regions, spanning just five or so plane pairs, then most complex designs can't possibly be routed without crossing a split. This issue arises in nearly every intricate design. Adding 100 nF ceramic capacitors to span the split consumes real estate and creates a larger loop area, which, in turn, creates emissions to some extent.

The secret here is to design the stackup in such a way as to make use of planar capacitance, instead of bypass capacitors, by making sure each VDD/VCC region is closely coupled to a continuous ground plane. A plane pair is basically a large capacitor that becomes more efficient as the dielectric becomes thinner. Signal layers must also have an adjacent solid ground return plane to properly steer the electromagnetic wave. By using a combination of three plane layers as in Figure 4, the planar capacitance will provide an alternative return path to transverse the gap. This does not increase the plane count because the next signal layer in the stackup can also use the additional ground plane. Since the

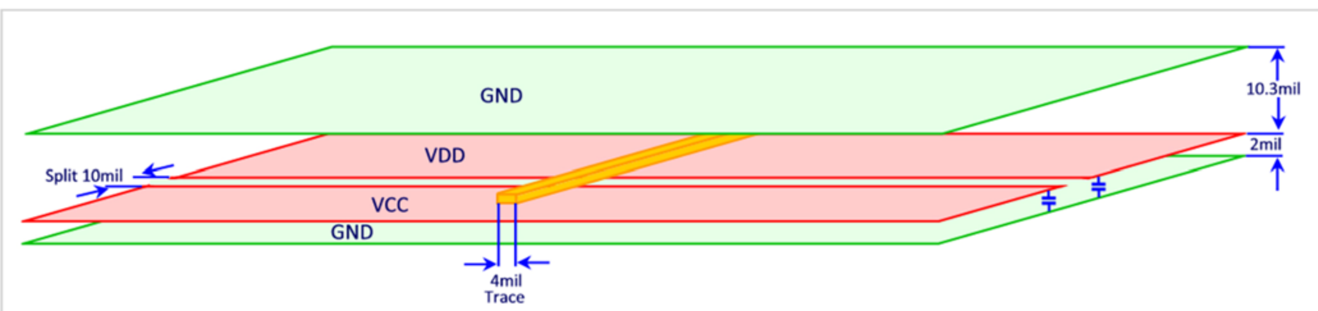


Figure 4: 3D view of the split plane stripline configuration.

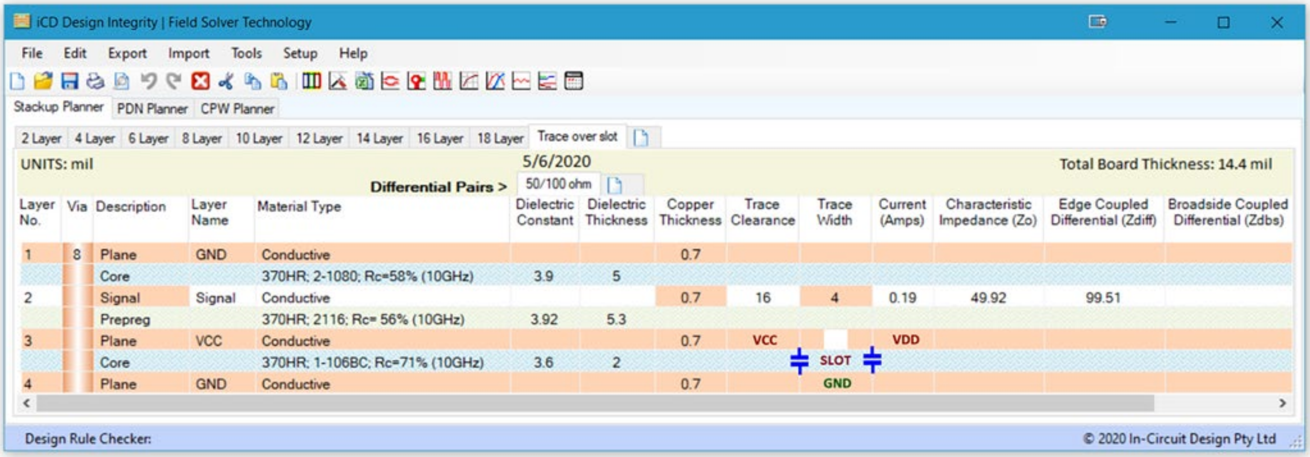


Figure 5: Thin dielectric creates more planar capacitance (iCD Stackup Planner).

skin effect forces current to flow in the outer periphery of the plane at high frequencies, it virtually creates two independent ground planes on the top and bottom surfaces with no return path crosstalk, although it is actually one physical sheet of copper.

However, this methodology is also frequency dependant and determined by geometry. As the frequency increases, more power is injected into the plane cavity propagating along the slot. Above 10 GHz, the slot mode radiation begins to couple excessive electromagnetic energy into the adjacent traces.

Properly designed power distribution networks have the power planes tightly coupled to their respective ground planes, as in Figure 5, to provide planar capacitance. Of course, none of this is valid if the two planes are far apart. Stackup data can be transferred to the iCD PDN Planner to demonstrate the impact of planar capacitance on plane resonance for different configurations.

If very thin dielectrics (2–3 mils) are used between the power and ground planes, split planes can have minimal impact on the quality of signal transmission if designed correctly. If the split plane is covered with solid ground planes both above and below in a stripline configuration, radiation will not be an issue. And keep in mind that this also applies to differential pairs traversing a split plane. The return current flows in the reference planes(s)—not the opposite signal trace, particularly if the pair is not very tightly coupled.

One would typically use a minimum gap of 20 mils to isolate planes of dissimilar potential. However, with today's modern fabrication etching techniques, a 10-mil gap is quite acceptable for low voltage applications (but please check with your fab shop first).

Ground planes should never be split with the exception of audio frequency circuits, special RF/microwave applications, and for high-voltage isolation. Functional partitioning and controlled routing is the key to a successful mixed-signal design, while utilizing planar capacitance is an excellent solution for high-speed digital designs up to 10 GHz.

Key Points

- A split in a ground plane creates an impedance discontinuity in the return signal path.
- Low-frequency circuits in the audio range can often benefit from splitting the ground planes.
- The key to a successful mixed analog/digital design is functional partitioning, understanding the current return path, and routing control and management.
- Route fences are a useful tool to control the routing.
- The ground plane should not be split; rather, a pass-through gap is left in the fences so that data and control signals can enter and leave that area.
- Dual-stripline layers can provide mixed-signal/power pours and reduce the layer count.

- If a return path discontinuity exists, then the current tends to divert increasing the loop area, inductance, and delay.
- A via that provides the connection between signal layers, referenced to planes of different DC potential, creates return path discontinuities.
- If the reference planes are at the same DC potential, then they can be directly connected by stitching vias near the signal via transition.
- If the planes are at different DC potential, then a bypass capacitor can be connected across the planes.
- Two bypass capacitors are a much better solution, as this eliminates the transfer of power supply noise from one supply to another.
- A slot in the plane under two or more nearby signal traces will cause return currents to overlap, resulting in some degree of crosstalk.
- The secret for high-speed design is to construct the stackup in such a way as to make use of planar capacitance by making sure each VDD/VCC region is closely coupled to a continuous ground plane.
- If thin dielectrics are used between the power and ground planes, split planes can have minimal impact on the quality of signal transmission. DESIGN007

Further Reading

- B. Olney, "Beyond Design: Stackup Planning, Part 5," *Design007 Magazine,* July 2019.
- B. Olney, "Beyond Design: The Proximity Effect," *Design007 Magazine,* March 2019.
- B. Olney, "Beyond Design: Split Planes in Multilayer PCBs," *The PCB Design Magazine,* March 2015.
- B. Olney, "Beyond Design: The Plain Truth About Plane Jumpers," *The PCB Design Magazine,* November 2012.
- H. W. Johnson & M. Graham, *High-Speed Digital Design: A Handbook of Black Magic,* Prentice Hall, 1993.
- Yuriy Shlepnev, et al. "Stripline With One Solid and One Split Reference Plane: SI List."

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.

National Instruments to Acquire OptimalPlus

National Instruments Corporation, the provider of a software-defined platform that accelerates the development and performance of automated test and automated measurement systems, announced it has entered into a definitive agreement to acquire OptimalPlus Ltd., a global leader in data analytics software for the semiconductor, automotive and electronics industries.

The acquisition will expand NI's enterprise software capabilities to provide customers with business-critical insights through advanced product analytics across their product development flow and supply chain.

NI and OptimalPlus serve highly complementary positions in the semiconductor, automotive and electronics industries. NI test systems are used in semiconductor manufacturing with OptimalPlus serving as a leading supplier of semiconductor manufacturing data analytics.

Similarly, the NI automotive and electronics production test offerings are complementary to OptimalPlus' growing automotive and electronics analytics business. Combining the strength of NI's software-centric approach with OptimalPlus' enterprise-level analytics software is expected to dramatically increase the value of test and manufacturing data, enabling product insights that will improve quality, efficiency and time to market for both NI and OptimalPlus customers.

"The addition of OptimalPlus' data analytics capabilities will enable us to accelerate our growth strategy by increasing enterprise-level value for shared customers in the semiconductor and automotive industries," said Eric Starkloff, NI President and CEO.

(Source: Business Wire)

An Expectation of Collaboration

The Digital Layout
by Kelly Dack, CIT, CID+, PCEA

Introduction

This month, I share updates on the PCEA website and reports on some inspiring communication on collaboration taking place between the PCEA and a well-known industry organization that appears to have a high potential for synergy for all engaged.

I'm always happy to mention that PCEA Chairman Steph Chavez has prepared a message as well, where he provides an overview of his take on adapting to the new normal by doing more with less.

Lastly, we are not able to provide any reports on local chapter activities due to continued social distancing requirements. However, we will share our most updated list of professional development opportunities and events, which we hope you will find useful. As always, stay tuned for more updates.

PCEA Updates

Words ending in "ation" can affect our mission. Stay positive!

Throughout the week, I lap up a lot of evening news. Like you, I'm getting tired of watching so many negative news reports about "frustration in the nation regarding the presentation of some information regarding an observation that it is a violation to meet together in an organization without six feet of separation until after there is an indication of a successful vaccination declaration." In effect, I've become over-sensitized to words ending in "ation." They cause me to itch. But a brief conversation I had with PCEA Chairman Steph Chavez quickly desensitized me to these "ation" words.

The PCEA and SMTA Discuss Ways to Collaborate

Steph briefed me on an exciting meeting held between himself, PCEA Vice-Chairman Mike Creeden, and Global Executive Director of the Surface Mount Technology Association (SMTA) Tanya Martin. The three of them spoke for over an hour primarily about the consideration for collaboration between the SMTA and the PCEA.

Mike Creeden

Wow, a realization! Steph employed some "ation" words in a positive conversation, which offered me some inspiration. I then decided that the focus of this month's column would be collaboration. Steph, Mike, and Tanya also discussed future goals and how the two organizations might work together beyond the COVID-19 pandemic.

The SMTA is headquartered in Eden Prairie, Minnesota. It is volunteer-driven and relies on its members for leadership. The SMTA membership is filled with many electronics industry corporations and individuals who have just about everything to do with the processes, materials, and services essential for sustaining the electronics industry. The PCEA was formed to organize, collaborate, educate, and inspire an already sizeable, established community of PCB designers who sought to include and encourage fellowship with anyone involved in printed circuit engineering—stakeholders and representatives of manufacturing, materials, and processes.

Before the formation of the PCEA, these professionals had been basically functioning on their own at local levels for years. They were moving into the new decade, largely unrecognized and underrepresented in the electronics industry. Local functions often involved local leadership soliciting experts from the outside PCB manufacturing and engineering communities to educate and inform the design based chapters about new design software, materials, and manufacturing processes.

Steph Chavez commented that the three spent over an hour discussing their goals for the near future. "It has been pretty much a one-way street," Steph said of the flow of awareness of process and materials. "People in our chapters have been engaging and learning about our engineering stakeholders for decades to design better." But he also pointed out, "Still, today—due to limited representation—there are a very few stakeholders in our industry who understand what a PCB designer does or how a designer fits into the engineering and manufacturing puzzle."

Steph makes a good point. When communicating with most anyone other than PCB engineers, designers continue to get blank stares returned when defending their design response to management during a project. It can be vexing for designers who attempt to explain their reasoning for stackup and line width for a controlled-impedance PCB stackup solution to HR and management during a job interview. Management is included in the list of PCB project stakeholders, yet they appear to not understand what a PCB designer does and fail to see the value of the knowledge.

Tanya Martin acknowledged SMTA's desire to engage more with the design community energy coming out of the PCEA. "The SMTA is pleased to support PCEA as we've long recognized the importance of more collaboration in areas that affect PCB design, manufacturability, cost drivers, etc." The goals of electronics industry stakeholder awareness between the two associations appear to fit well, according to Tanya. "Working together will expand SMTA's breadth of technical topics as well as provide the PCEA with an established platform to share tribal knowledge and education, creating additional member benefits for both organizations."

Tanya Martin

It appears that in just a short period of time, new fruit from blossoms of collaboration between the SMTA and the PCEA will be ripe for harvest, and the electronics industry's supply chain of PCB engineering knowledge and awareness will certainly be strengthened. Tanya summarized the meeting well: "SMTA has outstanding representation at the chapter level that will pair well with PCEA's chapters to drive the mutual mission of technical education, dissemination of knowledge, and industry connection, which will build upon the grassroots efforts that both associations were founded upon."

PCEA Website Launch

Good web authors are hard to find. Our PCEA executive staff has been working with teams from a commercial web authoring business to provide the printed circuit engineering communities with top-rate, rich content of which we can all be proud. Unfortunately, the commercial authoring service we contracted with was not cutting the mustard.

Our meetings to move forward with them on our content started looking like a bad PCB design review: What are all those extra holes for, how will my differential pairs couple when spaced an inch apart, and what are all those splits doing in the reference planes? Design reviews can be like that for any type of design. The good news is that the input data our team provided was complete and clearly defined. The bad news is that there appeared to be a lack of comprehension and education on the part of the web designer with regard to the input details, resulting in poor layout.

What can we learn from this? Other design communities—web design and others—are not immune to a lack of collaboration, education, or inspiration in their roles. Ironically, while the PCEA is specifying a website to foster these attributes within the electromechanical industries by creating and leveraging a prime website, the web authoring industry is failing to "meet spec" due to lack of the same attributes. It is a vicious circle, which I exaggerate slightly here for emphasis.

Figure 1: The PCEA's new website is up and running.

Visit our new website at pce-a.org, which launched on June 1 (Figure 1). We want our website to help our members in the best way possible.

Message From the Chairman

by Stephen Chavez, MIT, CID+

Stephen Chavez

We're in a world of quarantine, isolation, and social distancing. Many are finding it mentally and emotionally challenging to deal with this new normal these days. This COVID-19 pandemic has definitely changed our "normal" working day and environment. This is taking its toll on all of us one way or another. Days have drifted into weeks, and weeks have drifted into months as we continue to maintain our isolation or social distancing.

In this "new normal," with most finding ourselves working remotely from home, the separation of work life and home life has morphed together, which can be a double-edged sword. In many cases, we find ourselves doing more with less, as our new normal working day seems to have simply gotten longer. Those 9–10-hour working days before COVID-19 are now 12–16-hour working days, simply due to all of the additional virtual meetings or teleconferences taking place.

As I talk to many friends and colleagues, all agreed that I am not the only one feeling as if we're participating in a three-ring circus due to all these virtual meetings, working on designs and presentations, and having a family life. Without even realizing it, by the time we walk away from our computers at the end of our "working day," we worked much longer than 9:00 a.m. to 5:00 p.m. In some cases, we've even lost track of whether it's a weekday, Saturday, or Sunday; they've morphed together.

On the flip side, we may find that we are adapting, evolving, and collaborating at a much higher level. Because we're isolated and with less "walk-up" interruptions or that simple "5–10-minute water cooler" chat that is actually more like 20 minutes, we're able to focus longer and get more done with less of those distractions.

Due to COVID-19, most—if not all—in-person expos, conferences, and seminars have gone virtual now. This has led to many doing more collaboration with less in-person time. For many of us whose business travel has come to a stop, "in-transit" time or "return back to the office and settle back in time" has been replaced by online opportunities, such as webinars and global virtual events related to this industry. Again, we are finding ways to adapt, collaborate, and do more with less.

Our PCEA Executive Board members are good examples of those doing more with less. Our day jobs have intensified over these last few months as COVID-19 has changed the industry, and the required effort that is needed to solidify the foundation of PCEA has been a challenge. Nothing ever goes as well as you'd like it to, and we have had our share of hurdles to get past. Murphy's Law always seems to make its presence, yet we adapt and overcome. We have crazy days where we feel as if we are in a double-dutch jump rope contest while we are juggling multiple tasks all at the same time.

In the end, what we are doing and creating is awesome. We have started collaborating and strategically planning with other great industry associations, such as the SMTA, for the better of the industry as a whole. We are identifying ways to fill the educational gap in our respective associations, as we see the PCEA and the SMTA being a "one-two" punch combination for success in our industry. Mike Creeden and I will collaborate as we represent the PCEA in an SMTA webinar scheduled for July 9. The PCEA will also have a presence at SMTA International, being held later this year. We'll be offering an educational track at this event. Stay tuned for more details to come.

We launched our official website as of June 1. The website had been one of our hurdles, as we had planned and expected it to have been online by mid-May. With lots of hard work, collaborative effort, and passionate determination, we adapted and overcame. We'll follow the website launch with our virtual grand opening scheduled for July 14 at 11:00 a.m. PST, featuring a presentation from Rick Hartley on the topic "PDN Tips: Successful Power Distribution."

Now that our official website is active and online, and with our upcoming virtual grand opening, you'll start seeing a lot more industry activity come from the PCEA—both virtual and hopefully in person in the near future. Our focus is slowing shifting from our PCEA structure and foundation, to collaboration, education, and inspiration through industry content and activities.

As always, I highly encourage you all to get involved! Join PCEA by visiting our website (pce-a.org) and registering as a member to become part of the PCEA collective. You can always reach out to me (stephen.chavez.pcea@gmail.com) or Kelly Dack (kelly.dack.pcea@gmail.com) to get more information, as well.

Professional Development and Events

It has been our custom to highlight all up-and-coming industry events to look out for in

2020. We will continue this; however, with the challenges brought upon our industry by the COVID-19 outbreak, we can only remain hopeful that these events will not be affected. If you have an interest in any of these events, please search and contact the event coordinators directly for the latest event status.

- June 9-10: **PCB2Day—SMT Assembly Boot Camp** (Austin, Texas)—***Postponed***
- June 11-12: **PCB2Day—Design Essentials for PCB Engineers** (Austin, Texas)—***Postponed***
- June 14-20: **IPC SummerCom 2020** (Raleigh, North Carolina)—***Canceled***
- June 22-25: **Realize LIVE 2020** (Las Vegas, Nevada)—***Virtual***
- July 7-8: **CadenceLive 2020** (Santa Clara, California)
- July 14 (11:00 a.m. PST): **PCEA's Grand Opening Webinar**—***Virtual***
- September 8-11: **PCB West** (Santa Clara, California)
- September 16-17: **Del Mar Electronics & Manufacturing Show** (San Diego, California)
- October 7-9: **AltiumLive 2020** (San Diego, California)
- November 11: **PCB Carolina** (Raleigh, North Carolina)
- May 11-13, 2021: **IPC High-Reliability Forum 2021** (Baltimore, Maryland)

Spread the word. If you have a significant electronics industry event that you would like to announce, please send me the details at kelly.dack.pcea@gmail.com, and we will consider adding it to the list.

Conclusion

How are we to collaborate within our organizations, companies, offices, and homes while staying socially distanced and staying at home? We have to do the best we can with what we have. Just watch; the world seems to be adapting before our eyes and ears. Advancements in conference trends are astronomical.

Within our local PCEA community, it is a good time to plan for the future and reach out. This pandemic is not going to last forever. It is prudent for all of us to keep looking forward to ways we can hit the ground running when things get back to "normal."

Eat, drink, and be merry; collaborate, educate, and inspire. Soon, some of us may go back to the office. As always, have hope for the future, and plant your happy seeds now. DESIGN007

Kelly Dack, CIT, CID+, is the communication officer for the Printed Circuit Engineering Association (PCEA). To read past columns or contact Dack, click here.

Thermal Management: Keeping Cool Starts From Within

Sensible Design

by Jade Bridges, ELECTROLUBE

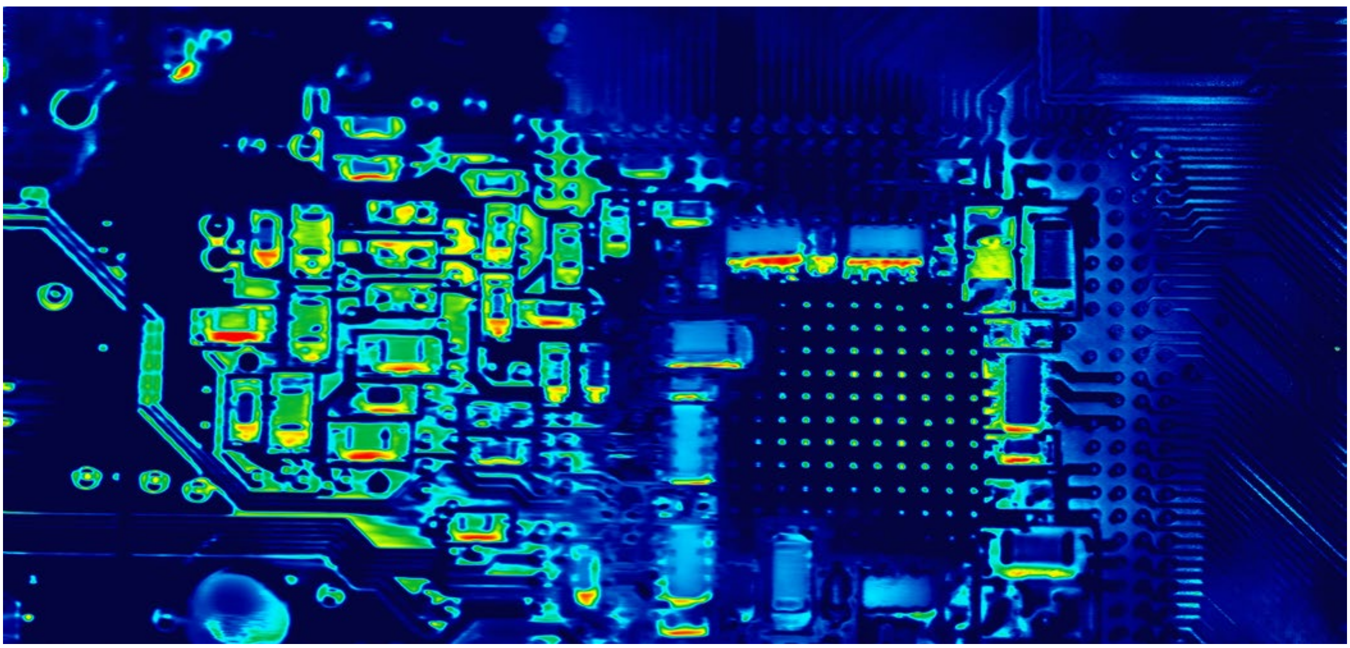
Introduction

Thermal management is a critical component of the design phase for any electronics device. Factoring in thermal management earlier on in the design process will lead to a more reliable and cost-effective end product. This month, Jade Bridges takes a fresh look at popular subjects within the field of thermal management. For instance, how do you know for sure if your thermal management process has been a success? Jade also explores what occurs when devices overheat, as well as the benefits of thermal gap fillers and how to best avoid pump-out. Thermal management plays a significant role in protecting electronic circuitry, so let's explore some of these key areas in more detail and discuss practices that ensure better thermal management.

1. How do I know if my thermal management is a success?

There's so much to consider when selecting a thermal management material, and getting it wrong could ultimately compromise the reliability of an electronic assembly and shorten its life expectancy. No pressure! When you think you have it right, how can you be entirely sure that your thermal management process has succeeded? Critically, the product would have been tested under real-world conditions to see how it holds up under the heat, allowing engineers to alter the design before it goes to market and giving designers the opportunity to make any necessary changes to preserve the product's functional integrity and operational reliability.

Evaluating thermal performance in all phases of the design cycle will also confirm any issues



being brought to light early on and help to prevent a costly system-level teardown. The criteria for evaluating the success of your thermal management process is actually fairly simple and involves three factors to consider. If you can say “check, check, and check” to the following, then you can reassuringly consider your thermal management process as a success:

1. Improved efficiency of heat transfer.
2. Reduced thermal resistance.
3. Observe a lower temperature around the heat-generating component/device.

2. What is the worst that can happen if a device overheats?

Increasing miniaturization in electronics means that heat dissipation problems are becoming increasingly important. More effective thermal management will often lead to enhanced reliability and life expectancy of devices. Insufficient thermal management will quite simply lead to overheating, which may be caused by different factors at play as to why an electronic component becomes subjected to excessive levels of heat. For instance, consumer electronic devices, such as portable laptops and smartphones, are becoming more prone to overheating. This is because the physical dimensions of these devices are becoming smaller and more compact. To be specific, as the demand for smaller devices increases and becomes even more challenging, manufacturers of electronic components need to pack far more into even smaller areas.

With overheating, failure of the component is typical. If we consider a heat-producing electronic component in isolation, then during operation, its temperature will rise until the heat produced within the device becomes equal to the heat lost to the surroundings, and the device has reached equilibrium. The rate of loss of heat from a hot object is governed approximately by Newton’s law of cooling, which states that the rate of loss of heat is proportional to the temperature difference between the body and the surroundings.

As the temperature of the component rises, the heat loss increases. When the heat loss per second equates to the heat produced per second within the component, the device will have achieved its equilibrium temperature. This temperature may be high enough to significantly shorten the life of the component or even cause the device to fail.

Here, the worst outcome is if an entire device overheats. This will consequently affect neighboring materials, and it must be considered how these materials will react if the temperatures reach the maximum possible level. Thermal management products are the ideal solution for this scenario. A similar approach can be applied to a complete circuit or device which incorporates heat producing individual components.

3. What exactly is a gap filler?

A gap filler is a material that is designed to be used at higher thicknesses than a standard thermal interface material. It could be used on a gap of 300 µm up to a few mm. A gap filler will provide a good thermal conductivity but also maintain its stability at these higher thicknesses, ensuring good heat transfer throughout the life of the product.

Thermal gap fillers are widely used for mobile and touch screen applications; however, some products are extremely adaptable and can be used in a multitude of applications from PCB assembly and housing electronic components discretely to automotive electronics, including HEV and NEV batteries, power electronics, LEDs, and fiberoptic telecoms equipment. Gap fillers are typically soft and compliant for low-stress applications and are easy to dispense due to their low viscosity. They also offer high thermal conductivity, and their low modulus elastomer prevents pump-out, which conveniently leads to my next point.

4. How do I avoid pump-out, and what are the potential consequences of this phenomenon?

In general, most devices will go through some form of thermal cycle, even if it’s as simple as switching the device on and off. When changes in temperature occur, all materials in the unit will expand or contract to a certain degree, depending on the temperature the device reaches in operation and, ultimately, the temperatures that the individual components reach. The coefficient of thermal expansion will vary from component to component, so contraction and expansion can happen at different rates with adverse effects such as pump-out occurring as a result.

To overcome this issue, a product that is stable during the thermal shock cycle is required. Pump-out can occur when a device, such as an IGBT, is subject to temperature changes, resulting in relative motion between the conductor and its heatsink between which a non-curing thermal paste has been applied. This motion can cause such pastes to be squeezed or pumped out from the interface gap, reducing the thermal transfer performance.

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

In addition to these temperature considerations, the materials that are being used may affect the interface material, particularly regarding the spacing between the device and its heatsink (otherwise known as the bond line thickness). While interface materials should be applied at a minimum thickness to achieve low thermal resistance, the resulting bond line may also be affected by the substrate's "smoothness" and spacing (i.e., components and heatsink surfaces at the interface). If a non-curing thermal interface material is applied more thickly due to spacing/materials considerations, the pump-out effect will be greater.

5. What are the potential problems associated with thermal shock and thermal management material performance?

As described previously, a thermal management product at an interface could be affected by the pump-out effect. This will result in the product moving and possibly allowing air to creep in between the interface. By having an uneven surface of thermal management material, hot spots can form, and the dissipation of heat will be affected. If a cured product is used, the flexibility of this material must be able to withstand the changes in the coefficient of expansion of all the different materials involved. The cured TIM should not crack or lose adhesion to the interface substrates during the thermal shock cycle.

Phase change materials are highly suited to the thermal management of electronic assemblies for a number of reasons. They offer efficient thermal transfer, along with enhanced performance with thermal shock cycles and greater thermal protection where temperature spikes can occur due to their ability to store and release thermal energy (latent heat) during the phase change process. Providing an alternative to traditional thermal greases, once heated above their phase change temperature, the new phase change materials become highly thixotropic liquids that perform equally as well as, but often superior to, a thermal grease. The consistency and performance of these new materials avoids possible application and migration problems that can be associated with thermal greases.

Help is available for thermal management material selection and/or application technique, so always seek expert advice. We thrive on challenges, so if you have a particular thermal management issue or question, please reach out. No matter how great the challenge may seem, we always aim to achieve a streamlined production process and efficient heat transfer both in the initial application and final use. **DESIGN007**

Jade Bridges is global technical support manager at Electrolube. To read past columns from Electrolube, click here. Download your free copy of Electrolube's book, *The Printed Circuit Assembler's Guide to... Conformal Coatings for Harsh Environments*, and watch the micro webinar series "Coatings Uncoated!"

PCB007 Highlights

IPC's Dr. John Mitchell: Leadership in Times of Crisis ▶

On May 13, Dr. John Mitchell, IPC president and CEO, spoke with Barry Matties and Nolan Johnson in another installment in our series of industry updates. In this interview, Mitchell discussed the challenges of leadership in crisis situations.

Online Instruction and Distance Learning ▶

Online courses have become increasingly available and popular. For this to be effective, specific requirements must be met for courses taken or produced over the internet in order to provide the user with a positive experience. Read Happy Holden's updated article from his series on 25 essential skills for engineers.

3D Additive Electronics Manufacturing: Are We Nearing an Inflection Point? ▶

Recently, Dan Feinberg was invited to attend a detailed and broadly informative webinar by nScrypt titled "The Strength of 3D-Printed Electronics," which covered the status and advances in the use of 3D printing for electronic device design and manufacture. nScrypt is an Orlando-based company founded in 2002 that focuses on 3D printing. Here's what Dan learned from each of the speakers.

Punching Out! Pivot Like MJ ▶

Most successful businesses have pivoted at least four or five times during their history. The COVID-19 pandemic has forced many companies to close and many others to find other business models. You might not want to stop making boards and start delivering for Amazon, but Tom Kastner explains how it is good to keep looking for new opportunities.

Sunstone Circuits Pledges Proceeds From Orders Will Be Donated to Feeding America ▶

Sunstone Circuits—a PCB solutions provider for prototypes, medium-volume, and production quantities—will donate proceeds from PCB sales to help families in need across the United States.

North American PCB Industry Sales Up 4.3% in April ▶

IPC announced the April 2020 findings from its North American PCB Statistical Program. The book-to-bill ratio stands at 1.19.

The PCB Norsemen: Can Better Guidelines on Cosmetic Failures 'Save' Functioning PCBs? ▶

Every year, fully functional PCBs are scrapped due to cosmetic "failures" that are not approved. Is this right, or do we need to make an even more precise set of rules on how to handle this? Jan Pedersen shares his thoughts on the issue.

Language of Electronics: Key Considerations for Your Next Direct Imaging System ▶

In this column, Orbotech West DI expert, Rick Jackson, provides a guide detailing which issues should be considered when choosing a new DI system. As your PCB manufacturing company considers a new DI system, here are some key considerations to keep in mind.

Survey: Lessons Learned in Unparalleled Times ▶

Parents often share nuggets of wisdom with their children in hopes that they won't repeat past mistakes. But what do our electronics industry leaders want to share about lessons learned during the COVID-19 crisis?

Pallav Aggarwal: From Hobbyist to PCB Designer and Embedded Engineer

Interview by Andy Shaughnessy
I-CONNECT007

I recently discovered a great blog by Pallav Aggarwal, a PCB and embedded systems designer based in Bangalore, India. On his blog, he discusses his journey from a young hobbyist to a degreed engineer who designs PCBs and embedded systems for companies all over the world.

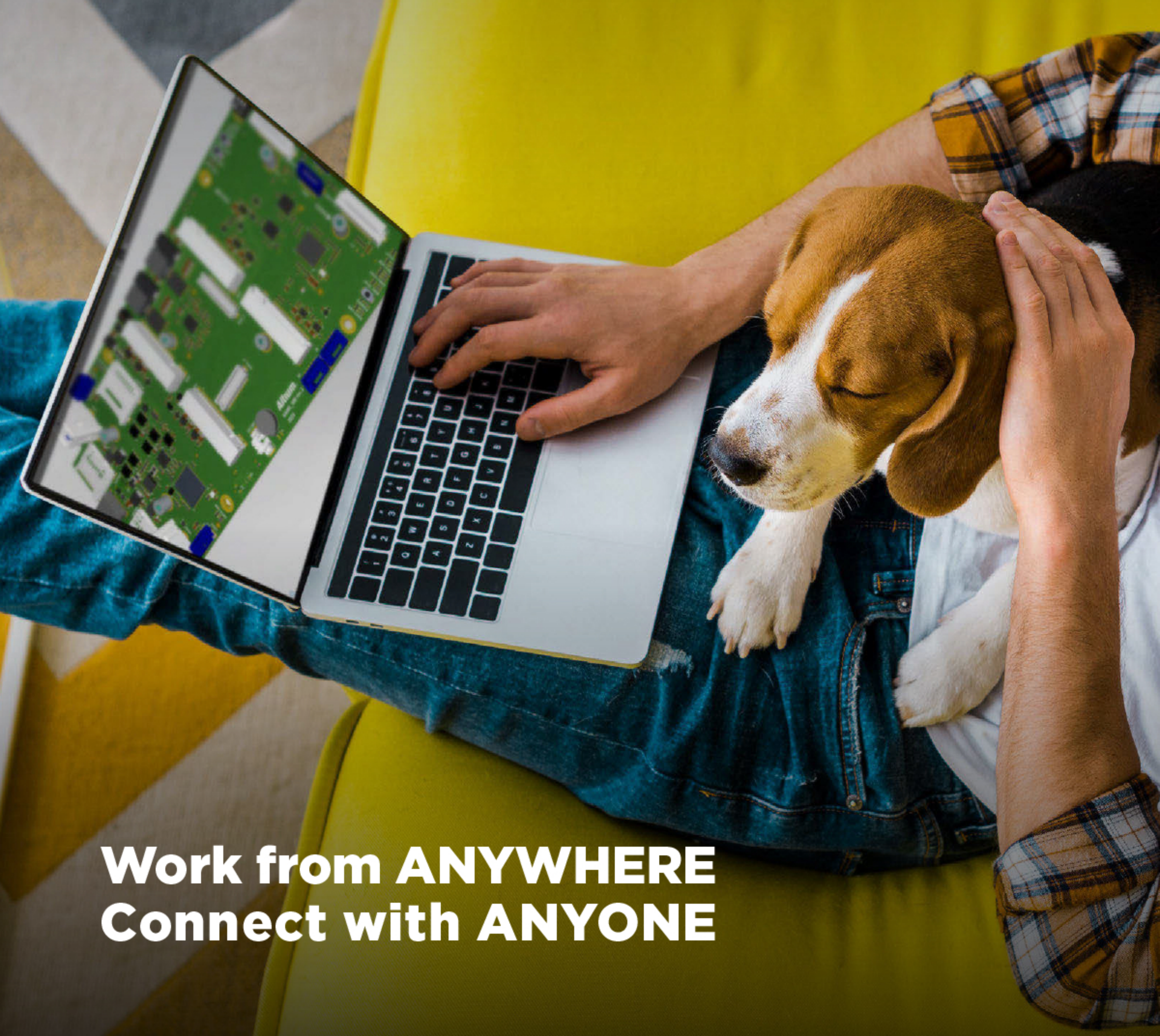
On his site, Pallav explains how he endured a two-hour ride each way to college, with part of that trip in a three-wheeled vehicle. Are you that invested in continuing your training and education? In this email interview, Pallav discusses his journey from hobbyist to engineer.

Andy Shaughnessy: Pallav, give us a quick background about yourself, and tell us how you got involved in electronics.

Pallav Aggarwal: I have been an electronics hobbyist for two decades, and now I work as an independent embedded design consultant. I help various companies in India and abroad design embedded products. One of my associations is with a company called M2MLogger in India, where I have helped them develop various IoT products. Before working as a consultant, I worked in various engineering roles in different national and multinational organizations for 15 years.

Around 1998, I was studying for IITs—the most prestigious universities for engineering education in India. I was going through some of the mock tests. While doing so, I came across a one-page electronic circuit in one of the science refresher magazines, and I fell in love with it instantly. I started visiting various electronic component shops nearby to see what these components were, what they looked like, how to buy them, etc. That was when I started exploring electronics in a full-fledged manner.

I began visiting a nearby library to access electronics magazines to go through the published circuits. I built an RF transmitter with a home-etched PCB, and I was able to receive audio on an FM radio receiver one meter away. I was doing these types of experiments every day and night. I was very lucky to have found what I really love to do early in life. When you work in a field you love, you are happy every day and never tired or frustrated. That's a huge thing to achieve.


Work from ANYWHERE
Connect with ANYONE

Shaughnessy: How did you think to take batteries apart and harvest the zinc for resale? What a great idea.

Aggarwal: I used to explore all of the scrap in my home. Seeing me dismantling batteries and cells, my grandfather told me that batteries contained zinc, or jasta in Hindi—an expensive metal used for making corrosion-resistant pipes.

I used to open AA and D batteries and extract the zinc foils. One day, I melted those foils and wound up with a big zinc log, which I then sold in the market. Even then, I think my inner salesman was active! With the zinc money, I bought more electronics—mainly chemicals for my experiments.

Shaughnessy: On your blog, you mentioned that you once rode a bus two hours each way to the Tico Institute of Embedded Technology. You must have been really serious about your training and education. Do you attend conferences, trade shows, or webinars?

Aggarwal: I did, but those were pre-COVID-19 conferences, of course. My parents didn't have much money, and that was the most affordable way to reach the Tico Institute, which was 60 kilometers away from my residence. I used to take an auto—a three-wheeled vehicle—from my house to the train station, a train to Delhi, and then a bus to the training center. A one-way journey would take two hours or more, but I never minded hard work. I think it is in my blood from my father, as well as my grandfather, who was a farmer.

I was crazy about electronics, so I wanted to do whatever it takes to learn. I feel so happy now that I listened to my inner voice and stuck with my passion. I attended many seminars conducted by various semiconductor manufacturing companies in India, such as Texas Instruments and Freescale (now NXP), as well as other exhibitions.

In addition to the seminars and exhibitions, I also visited all of the electronics shops and markets in the city. I have explored them hundreds of times. In Delhi, we have one of India's largest electronic component markets—the Lajpat Rai Market. I understood that I needed to go out and meet people, understand what products the world is making, and improve my own knowledge. I think every engineer should visit these exhibitions and seminars; they help you improve your perspective for whatever you build, and you learn so much from others.

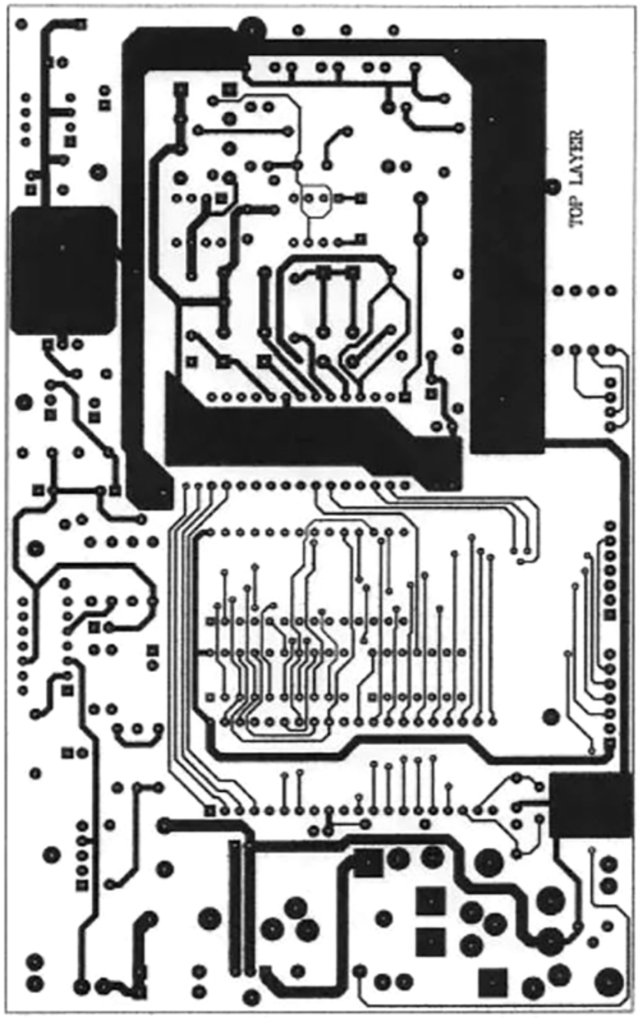


Figure 1: One of Pallav's early designs—a power board laid out with Protel 2.7.

I had poor communication skills back then. I remember being scared to answer the phone at home. English was my second language at school, so talking to people was also a way of forcing myself to speak to people and improve my communication skills. Now, I am like a child when I see new hardware. If I enter your office and see a PCB, I will ask if I can take a look at it.

Shaughnessy: I understand that you learned PCB design by working, almost for free, using Protel. Tell us how you got started in design work.

Aggarwal: When I was first learning about electronics, the PCB was always a part of the circuit. I became curious and started to learn about PCBs, including how they're manufactured and which software tools were used to design them. I somehow arranged to get a copy of Protel 2.7 and started learning design techniques by using the help file. The shortcuts in the help file gave me a quick idea of how to use it. I found it very interesting, and I started making some PCBs to see what I could do.

I visited the Lajpat Rai Market and saw one shop that sold PCBs, such as inverters, chargers, and LED boards, etc. I asked the shopkeeper if I could buy one small PCB. He asked why I wanted a PCB, and I told him that I was learning PCB design and planned to try to design this PCB. The shopkeeper gave me a PCB for free and said, "If you design it correctly and give me the design, I will pay you for it." I was very happy. I came home, designed the PCB, and I think I was paid 40 Indian rupees (about $0.50) for my first PCB design job.

This episode gave me an idea about how to capitalize on my skill. I worked hard and improved my PCB design skills, and then I decided to contact various companies in my city and ask if they had any design work. I explored the phone directory to find all the PCB manufacturers, and I called them all. I told them I was a PCB designer, and soon I was working for 17 different companies. I was getting 10–15 Indian rupees per square inch for my design work. It was very little, but I was on a mission to learn—not to earn. The money was just a bonus.

Shaughnessy: You're involved with embedded system design, AI, and all kinds of Arduino projects. What is your favorite part of your job or your sweet spot?

Aggarwal: I help companies build embedded products, which includes hardware and firmware/software. For many, I help improve the company's existing designs for size, power consumption, cost, reliability, etc. I do both hardware design and firmware as part of the complete embedded design. I regularly review technical specs, hardware designs/schematics, and overall design architecture.

I have gained many years of embedded firmware (MCUs/MPUs) design experience as well, so I love architecting new products, finding issues in a client's specs and designs, and helping them improve based on my field experience. The best part of my job is determining the client's new project requirements and challenges and helping them with different architectures and strategies in the design. That's the most challenging part as well.

Shaughnessy: Do you primarily design for companies that export products or for the domestic market in India?

Aggarwal: I have clients in India, Africa, Saudi Arabia, and in other parts of the world as well. My clientele is a mix: Some manufacture for domestic needs and others are exporters. I am now exploring opportunities in the U.S. and Europe. I want to associate with a few good companies and help them build successful products.

Shaughnessy: How has the COVID-19 outbreak changed the way you do your job and the electronics industry in India?

Aggarwal: COVID-19 is terrible. Everyone is at home. Just before India started the lockdown, I had traveled to Dubai for an exhibition and a few meetings. I can see that business is moving very slowly. New projects are on hold until the situation improves. Many companies have taken this as an opportunity to face their shortcomings and improve what they've always wanted to improve but never had the time.

Likewise, while I have a smaller load of commercial projects, I am building some software IP that could be reused in projects in the future. I have started developing some open-source embedded projects that can help the embedded community at large. One is a low-

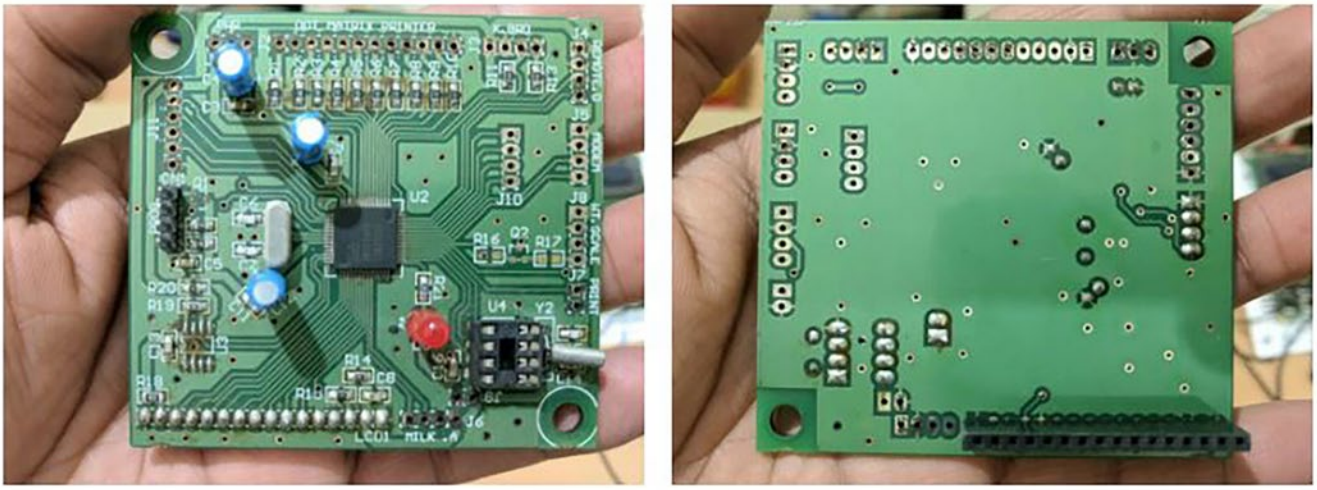


Figure 2: Pallav works on embedded systems designs as a hobby.

power measurement device that could be used for measuring microampere or even lower currents. There are no good solutions available in the market if you want to measure the power consumption of your low-power embedded/IoT device.

India is a market where personal relationships are more powerful, and face-to-face meetings are preferred, but COVID-19 has changed this fundamentally. People are more digital now, so digital meetings and online demos are more acceptable now in India. Post-COVID, businesses will be more acceptable to remote work than before.

Shaughnessy: What advice would you give to someone just starting out designing PCBs or embedded systems?

Aggarwal: My first advice is to do what you love. If you love football, don't bother about hardware, but if you love electronics, go all out and think long term. Young engineers are looking for quick results, quick money, and quick fame. There are no shortcuts. Be ready for many days, weeks, and months of debugging and reworking.

Believe me, the happiness of working on what you love is 100 times more satisfying than any money or any fame. It's also easier to perform well if you're doing what you love. Work hard every single day and have patience. It takes time to build knowledge and experience, and sometimes we lose patience and start thinking that we are less intelligent or this career is not for us. Don't get into that mode. You have to tell yourself that you can do it.

Don't worry about failures, either. Review what you have achieved in a year, not in an hour. Improve your knowledge every day. When I first started, I didn't have a computer or internet at home, and Google and YouTube were not yet born. Today, your educational possibilities are huge.

Remember that PCBs are central to electronics. Start with the basics. Learn how PCBs are manufactured and become familiar with different terminologies. Always ask, "Why is it this way, and not that way?" This curiosity will help you gain in-depth knowledge of the subject.

For engineers exploring embedded systems, I have the same advice. Please understand that embedded systems encompass hardware and firmware—not just firmware, as many might think. Hardware knowledge helps in a big way if you want to be a truly embedded design engineer.

Shaughnessy: Is there anything else you'd like to add?

Aggarwal: Around 2016–2017, while heading an engineering team in India for a Swiss-based system-on-module manufacturing company, I had the opportunity to travel and meet hundreds of customers. I discovered that many engineers lack knowledge and proper direction, so I started to write articles that I thought might help some of these engineers.

Over a period of time, I realized that I should have my own website to have everything organized well and available in one place. My long-term goal is to build a very good and practically applicable knowledge base for embedded systems engineers and maybe write a book someday. I hope to help them in their journey and give back to society in my own way.

Today, I am what I am because so many people gave me direction and guidance. Now, it's my turn. Thank you for the opportunity to share my journey with you. I wish all readers a healthy and happy life.

Shaughnessy: Thank you, Pallav. I appreciate it. This is a really good story.

Aggarwal: Thank you. DESIGN007

Ergotron's New Thermal Imaging Cart Supports a Healthier, Safer Return to Work

Ergotron, a global movement company that builds environments that help people thrive, announced the launch of its Thermal Imaging Cart with Onboard Power designed to quickly detect a temperature in spaces where groups of people gather.

As employers slowly start to bring their teams back to work, businesses gradually begin to reopen, and educators weigh their options for the upcoming school year, safeguarding public health is a top priority. Thermal imaging technology brings body temperature measurement wherever it's needed to support efforts to maintain healthy communities.

Ergotron's Thermal Imaging Cart provides total flexibility. The cart's open architecture allows for customers to mount their thermal imaging camera of choice facing forward or backward depending on the application or to accommodate privacy considerations. The easy-to-clean mobile cart is compact and fits in small spaces so it can travel to wherever it's needed. Optional accessories like a sani-wipes holder make disinfection simple and the onboard LiFe battery technology provides a safe and reliable charge.

"We developed the Thermal Imaging Cart based on a need we recognized in our customers and in the marketplace," said Megan Nightingale, general manager, industrial, education and new markets at Ergotron. "As we navigate this new world we're in, we've seen a huge demand for thermal imaging solutions. Our cart fits a variety of thermal imaging cameras and we're excited to introduce this solution to support employers, businesses and educators as they work to keep their populations healthy."

In addition to the Thermal Imaging Cart with Onboard Power, a Thermal Imaging Cart with Hot Swap Power is also available to order. Hot swap power, with Ergotron's innovative LifeKinnex Technology, allows for uninterrupted workflows. These carts can be ordered from resellers globally.

(Source: Business Wire)

An Update on IPC and IPC Design

Design Circuit

by Patrick Crawford, IPC—ASSOCIATION CONNECTING ELECTRONICS INDUSTRIES

While the world seemingly stands still, IPC's technical standards committees are teeming with activity. I do not track by-the-numbers attendance for all my committee meetings, but I have noticed a sharp uptick in the number of individuals who have attended these virtual meetings in the past couple of months.

In fact, most of the IPC technical liaisons have observed the same thing and have expressed something to the effect of, "Everyone is so willing to voice their opinions and donate their time." One of my colleagues told us that their committee members have even pleaded for more work. This is a bit of a shock—while we are always proud and appreciative of how our committees contribute their time and talents to produce our standards—this level of engagement is phenomenal.

As this involvement has increased, productivity has naturally increased. In the past few months, the timeline for design standards has accelerated. For example, the IPC 1-14 DFX Subcommittee has made great strides in finishing a working draft of the IPC-2231A DFX Guidelines. I am very proud to announce that the IPC 2-16 DPMX Subcommittee has just finished dispositioning a Final Draft for Industry Review (FDIR) for the IPC-2581C "Generic Requirements for Printed Board Assembly Products Manufacturing Description Data and

Transfer Methodology" standard. I will be writing about these committees and their achievements more extensively in future columns.

Considering the world in which we find ourselves, and after a couple of months' worth of hard work, the question arises: Why is there such a high level of engagement? We have observed a few reasons.

First, in a time when many professionals in the electronics industry are surrounded by uncertainty, any opportunity for them to focus on something that is well-understood and productive is welcome. The next reason is a logistic one: Now that many people are working from home, they are using the hours they spent commuting to now help out with industry standards. And finally, we have noticed that folks, for lack of a better term, are bored without their usual outlets and hobbies so they are giving standards development their full attention.

Now that many people are working from home, they are using the hours they spent commuting to now help out with industry standards.

I would like to take a moment and thank all our industry volunteers. Truly, we at IPC are grateful for your efforts in the past few months. We would not be able to produce any of our published work without you. But looking elsewhere, I do not believe that this kind of engagement should stop at standards development.

Last month, I wrote about the importance of competition for improving the technical skills of printed board design professionals and how competition can be an avenue for technical networking among industry veterans and new professionals alike. (Shameless plug: I am still recruiting design professionals to donate their time to plan and execute the IPC Design Competition.) This was intentionally forward-looking, and it envisioned a future when social distancing rules had relaxed a bit. Hopefully, that future is near.

But right now, the situation is still fluid. Some areas of the world have done well to flatten their curves, yet others are continuing their efforts to turn the tides of infection. While scientists and policymakers are working hard to resolve the myriad economic and sociological issues that come with COVID-19, one fact is true for most areas of the globe: Having large-group gatherings—especially of more than 10 people—might be a long way off.

I hope you will not mind if I backtrack from the future to the present and discuss how design professionals can become and stay involved with one another from the comfort of their own lockdown, and as the world slowly learns what the "new normal" looks like. I firmly believe in the power of the human spirit; I have been comforted to watch as humans from around the world work through their isolation by connecting more than ever before. In a world of closed doors, we have turned to Windows (or MacOS, or any number of Linux flavors).

This returns us to my introduction. While the world stands still, IPC's technical standards committees are teeming with activity. However, there are plenty of other ways that printed board professionals can donate their time toward building a stronger community and forging new avenues for their careers.

Out of an abundance of caution and respect for the gravity of the rapidly developing situation, we decided to temporarily delay the creation of new chapters after the debut of IPC Design at IPC APEX EXPO 2020. However, based on the level of engagement we have seen in our technical standards committees, we now feel comfortable in pursuing these openings. We have started to organize design engineers in Italy, Israel, Salt Lake City, Asia, New England, Texas. Every day, more design engineers are coming together to take part in IPC Design.

Above all, we want to empower folks to continue to practice distancing hygiene while still enabling them to exercise their creative, social, and technical minds. To that end, we are beginning to develop a new speaker series that combines board design engineering with more general product design in the industry. As an example, a talk might consist of a mechanical engineer discussing the design of a regenerative braking unit in an electric car while a board design engineer discusses how the unit's control board was designed given the mechanical and electrical constraints of the system.

Another new effort will be the creation of a technical paper library that includes board design engineering and related technologies for all IPC Design affiliates. To build this library, we may choose to embrace our pillar of competition by awarding the best submissions with recognition or prizes at industry events. In any case, we will be accepting submissions from all affiliates.

Of course, the future of IPC Design is contingent on the future state of the electronics industry and the world at large, but we have hope. If the activity in IPC's standards committees and the initial response to IPC Design are bellwethers for the future of the printed board design engineering industry, then these first programs will only be the tip of the iceberg insofar as what we can accomplish. DESIGN007

Patrick Crawford is the manager of design programs and related industry programs at IPC. To read past columns or contact him, click here or email PatrickCrawford@ipc.org.

UCLA Team to Develop Breathalyzer-Like Diagnostic Test for COVID-19

A research team led by Pirouz Kavehpour, professor of mechanical and aerospace engineering in the UCLA Samueli School of Engineering, is developing an inexpensive and fast Breathalyzer-like diagnostic tool to test for the novel coronavirus that causes COVID-19.

"Unfortunately, the COVID-19 pandemic has exposed a critical weakness in health care security infrastructure, which is a substantial deficiency in our capabilities to conduct rapid, simple, point-of-care diagnostic and environmental sample collection and testing," said Kavehpour, who is the principal investigator on the research. Kavehpour's team has received a one-year, $150,000 research grant from the National Science Foundation.

"The goal in this research is to develop cheap, massively deployable, rapid diagnostic and sentinel systems for detecting respiratory illness and airborne viral threats," said Kavehpour, .

The design could also be altered to detect other infectious diseases and airborne viral threats by continuously monitoring the air of indoor environments, such as hospitals, schools and airports, for the presence of the dangerous levels of virus.

The concept is based on an environmental water condensation technology developed by Kavehpour and his research group. They have applied for a patent for the design.

Although similar in use to Breathalyzer tests designed to check blood alcohol levels, which use infrared light to measure blood alcohol levels, the method behind the COVID-19 diagnostic test is different. For the coronavirus test, a person would exhale into the device for about a minute. Water vapor from their breath would condense on a special plate. Live virus and virus RNA could then be screened by using fluorescent genetic tags that light up if the virus is present. It could take about 10 minutes to show results.

If the design is successful and meets all federal criteria, test kits could be in production as early as fall 2020, said Kavehpour, who is also a professor of bioengineering.

(Source: Matthew Chin, UCLA Newsroom)

Pirouz Kavehpour

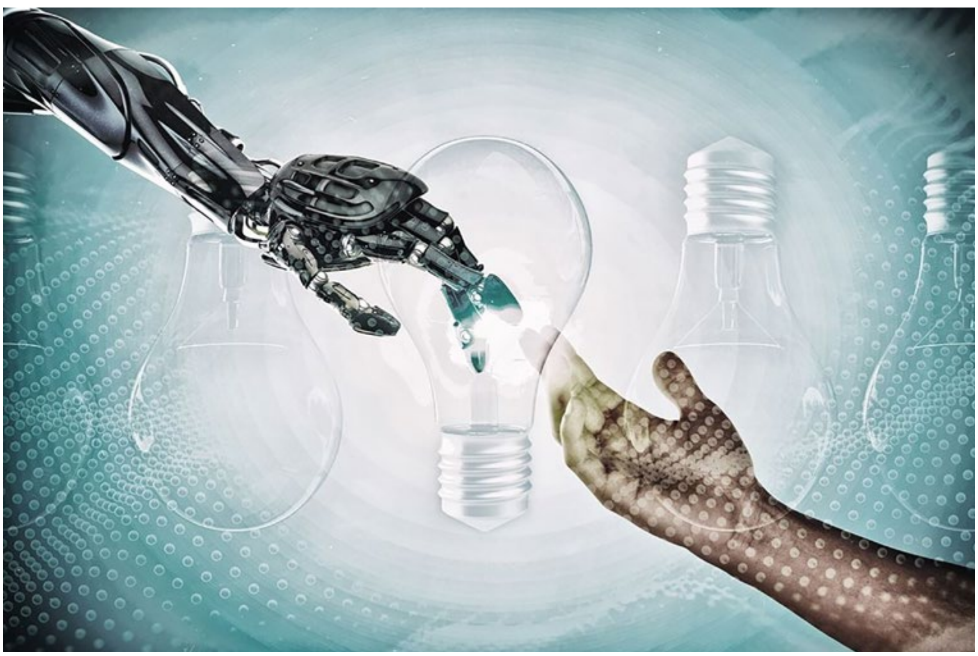
The Power of Forward Thinking

Connect the Dots

by Bob Tise and Matt Stevenson, SUNSTONE CIRCUITS

Innovation comes in many forms and from more places these days. Last year, the staff of *Popular Science* locked themselves in a room and came up with its annual list of 100 greatest innovations. These technological advances will shape how we live and work in the coming years. Included were products like Omrom's HeartGuide, HTC's VIVE Pro Eye, and Pro Display XDR by Apple, as well as the national rollout of 5G cellular from all four mobile carriers.

Breakthroughs result when visionary individuals and organizations identify needs that are not being served, platforms that can be improved, or entirely new products capable of transforming daily life. What do these electronic devices and an industry-wide technology shift have in common? They are all innovative, and they all contain PCBs.



Turning Visionary Ideas Into Innovative Products

Whether it's a new electronic device or advancement of broader technology infrastructure, moving from idea to innovation requires processes—the journey from concept to design to prototype—and manufacturing. Bringing innovation to the market requires many skills across multiple disciplines, all working together.

From mechanical and electrical engineering to sales and marketing, a team needs to have all these resources available to them to be successful. PCBs are at the heart of these products and product innovation; designing, prototyping, and manufacturing them is vital to success.

To the uninitiated, PCB design may seem like a rigid, by-the-book process, but It's not. Because it takes creativity to turn a concept into design, PCB design is very much an art form. Granted, boards aren't quite the same as blank canvasses, and there's nothing artsy-sounding about schematics or a bill of materials or documentation. PCB design is a world filled with limits, tolerances, and parameters—in other words, rules. But, like in "The Matrix," some of those rules can be bent and others broken.

When you focus on design as an opportunity to be creative, rather than a task standing between you and production, you put yourself in a position to better fulfill the promise of innovation.

Let's start with components. The placement and use of components aren't just pragmatic; there is also an element of creativity in their use. There is no single right way to place them, but there is something unspoken about the elegance and aesthetic beauty of well-arranged components. It's important to remember, and this is where "The Matrix's" rules cannot be broken, how you orient and organize components will heavily influence the manufacturability of design and, ultimately, the functionality of your finished board (Figure 1).

Pro Design Tips:

- Orient similar components to facilitate efficient and error-free soldering
- Do not place components on the solder side of a board that would sit behind through-hole components; your assembly shop will tell you how much room they need
- Organize through-hole and surface-mount components to minimize assembly steps
- Avoid components that will give you heating issues when the board is in use or take steps to control or remove that heat

Next, consider routing. Almost every piece of PCB design software has an auto-router feature that can be useful in a lot of situations, but when you're designing something brand new, you can benefit from taking over the controls yourself.

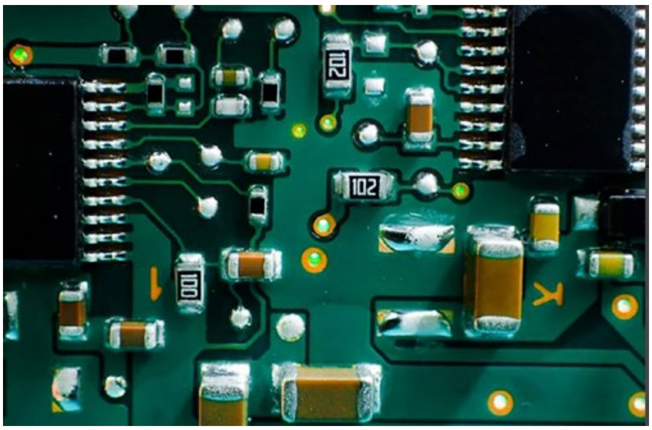


Figure 1: Components.

Leave the auto-router in charge, and the results can be asymmetrical, even look and feel messy. These configurations are not easily manufactured and can be lacking when it comes to performance. It's like a pilot who doesn't know the destination of the flight. Use this tool as a complement to your routing efforts—not a replacement for you.

If you have time, hand route the entire design; it will look more professional, work more efficiently, and be easier to update if you must rework a section of the board. If you cannot avoid having to auto-route some of the board, hand route the critical traces yourself, and then auto route the remaining ones. In this capacity, the auto-router can help with schedule compression while also making the bulk of your non-critical connections. There will be some messy routing and too many vias—plus editing will be harder down the road—but you'll have something to work with much sooner.

Innovation increasingly goes hand in hand with PCBs getting smaller. A multilayered PCB can offer requisite functionality if it occupies a small space in your device. If you have limited or no experience with multilayer board design, the prospect of an attempt can be daunting. Multilayer boards don't have to be a hassle. Think of them as multiple single-layer boards stacked together.

Pro Design Tips:

- Become familiar with multilayer design tips and guidelines provided with your CAD tool
- Familiarize yourself with components, tolerances, and requirements unique to multilayer PCBs
- Make sure your design tool library is set up for multilayer boards
- Know your manufacturer's requirements for multilayer designs
- Stay away from blind and buried vias unless you really need them
- Don't hesitate to ask for help

To be considered an innovator—a profitable one, at least—it's important that you're first to market with your product. This presents a seemingly compelling reason to head straight for full production, but designing your PCB is just step one for ensuring its manufacturability and eventual functionality. Prototyping is the critical next step in the process, one that can make you more confident about your production runs.

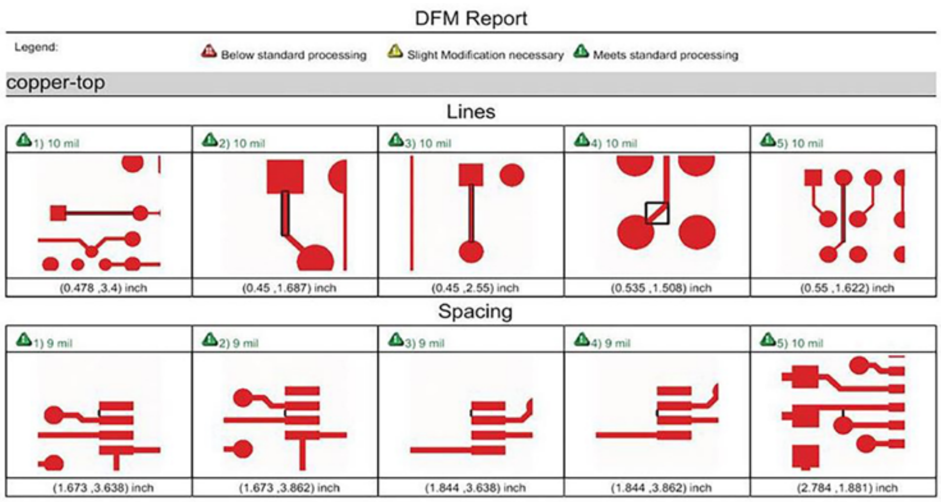


Figure 2: DFM.

Some feel prototyping costs too much, takes too much time, or must be outsourced abroad. Even if you are in a hurry, prototyping makes sense. It also doesn't have to cost a lot or happen 5,000 miles away. We believe prototyping is vital to a smooth, successful production run. It cost-effectively validates manufacturability of design, helps avoid incredibly expensive mistakes during full production, creates more opportunities to innovate, and tests your boards for quality and durability.

Prototype fabrication services can vary, and if you are working on something cutting-edge, there are several critical elements to keep in mind. Many manufacturers offer a feasibility assessment of your design to ensure PCB manufacturability and viability. This can help you avoid costly and unnecessary iteration during the prototyping phase. Layout review may uncover expanded manufacturing needs, so don't be afraid to ask questions and make sure your prototyping partner uses processes and equipment that match your desired speed of production and quality of the product (Figure 2).

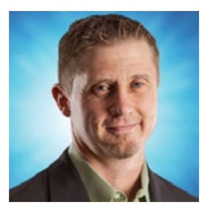
Once an acceptable prototype has been manufactured, assembled, and proven, your board moves on to full production and then assembly. At assembly, functional components for power regulation, I/O interfaces, and processing are connected with the wiring of your PCB. The resulting printed circuit assembly (PCA) will be key to end-product functionality.

In terms of time, money, and reputation, the cost of a do-over after assembly is more than at the design or prototyping phases—especially at higher volumes. A PCA with performance problems at this point could result in delayed order fulfillment, project cost overruns, or a competitor beating your product to market—a missed opportunity for innovation. That is why choosing your assembler wisely is so important. A reliable assembly partner will help you avoid costly failures associated with slipshod production, as well as defective and improperly installed components.

Moving your PCB design through prototyping, manufacturing, and assembly is the journey within the journey for product innovation. Every part of it is a continuous learning process that offers an opportunity for improvement. That's what makes innovation both challenging and exciting. **DESIGN007**

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


Bob Tise


Matt Stevenson

MilAero007 Highlights

What It Takes to Be a Milaero Supplier, Part 4 ▶

The decision to pursue military and aerospace (milaero) certification impacts every facet of the organization, and not every shop is prepared to make this transformation. This is the final article in a four-part series, breaking down what it takes. In Part 4, Anaya Vardya explores what it takes to be a milaero supplier in the area of manufacturing.

Photo Slideshow From Historic U.S. Launch Into Space ▶

Hugs and cheers followed NASA astronauts Bob Behnken and Doug Hurley as they made their way from the Dragon capsule into the International Space Station just before 3:30 p.m. EDT Sunday, marking the first successful leg of America's return to a full-fledged space program. We've put together a slideshow of some of our favorite images provided by NASA, from the preparations to liftoff to the final docking and entry into the International Space Station.

From the Hill: MIL-PRF-31032 Offers a Rewarding Twist ▶

If you are fabricating PWBs to military specifications, the master drawing will state: "Fabricate to MIL-PRF-55110, MIL-PRF-50884, or MIL-PRF-31032." This sounds very complicated on the surface, but there is a rewarding "twist" if the fabricator is certified to MIL-PRF-31032. Mike Hill explains.

Boeing Rolls Out First Loyal Wingman Unmanned Aircraft ▶

A Boeing-led Australian industry team has presented the first unmanned Loyal Wingman aircraft to the Royal Australian Air Force, a historic milestone for the company and the Commonwealth.

Defense Speak Interpreted: What's an RCV, and What Do Electronics Have to Do With It? ▶

In "Defense Speak," RCV does not stand for ranked-choice voting, a remote control vehicle, a riot control vehicle, or a refuse collection vehicle, although the second one is close; it stands for a remote combat vehicle. Denny Fritz explores this concept and its defense applications.

Airbus Wins ESA Contract to Construct Third European Service Module for NASA's Orion Spacecraft ▶

The European Space Agency (ESA) has signed a contract with Airbus for the construction of the third European Service Module (ESM) for Orion, the American crewed spacecraft. The contract is worth around €250 million.

SpaceX Manned Mission Promises More Success for Milaero and Electronics Manufacturing ▶

On May 30, SpaceX became the first non-governmental organization to send human cargo into orbit and to a successful docking rendezvous with the International Space Station (ISS). On June 1, Nolan Johnson spoke with milaero consultant and I-Connect007 columnist Mike Hill about the significance of this mission.

Raytheon Technologies Supporting the Supply Chain ▶

To support its small business suppliers, Raytheon Technologies formed a Small Business Supplier Stimulus Team focused on coordinating support to its supply base. This team brings together cross-functional members from each of the company's businesses, creating a hub of knowledge, expertise, and experience.

FLEX007

A SPECIAL DESIGN007 MAGAZINE SECTION

Lead-Free Solder: Panacea or Pandemic?

Flexible Thinking

by Joe Fjelstad, VERDANT ELECTRONICS

Solder has been used as the primary means of interconnecting electronic components for more than seven decades. For the first five-plus years of those decades, life was simple, and the solder of choice was fundamentally tin-lead solder in the form of one of two basic alloys: Sn60Pb40 and Sn63Pb37—the latter being the eutectic variant of the timeless and tireless servant of electronics assembly.

Today, however, the number of solder alloy options available to make electronic connections run into the many tens—if not hundreds—of different alloy choices. Read your journals, and you will see that the subject of solder dominates the content in terms of new or recurring problems and their detection, solutions in the form of processes and equipment, and concerns about the reliability of solder joints, which remain the primary root cause of failure of electronics.

For the benefit of all those who are new to the electronics interconnection industry (and there is a constant flow of new people entering the industry), I believe it is worth recalling


We go where others will not.™

how we got to this point for your knowledge and edification. This is my opinion, and it is a cautionary tale.

In 2006, the EU legislated a ban on lead in electronic solders. This was covered under the moniker of Restriction of Hazardous Substances (RoHS), which restricts the use of a number of naturally occurring elements that can be toxic under certain circumstances, along with some man-made organic compounds commonly used as flame retardants, which have been shown to have possible links to cancer.

RoHS was conceived as a prospective means of protecting both the environment and the health of Europe's citizenry (and presumably those of others around the world). However, most high-profile targets of the effort and promotion were executed out under the banner of "lead-free." This was arguably done because lead has long been identified as the potentially poisonous element that harms the health and development of children, mostly from ingestion of paint chips tinted with lead pigments, which are sweet to the taste. (The Romans used to store wine in lead vessels to increase its sweetness as lead is quite soluble in vinegar, and the common name for lead acetate formed in the process was "lead sugar.")

While lead in paint and gasoline were definitely vectors for ingestion and poisoning, there was little to no evidence to show any risk of lead in electronics solder making its way to children (or adults, for that matter). The exception was found in the injury of those unfortunate individuals in Asia and South Asia who were tasked to recycle electronics without being given any education, training, tools, or protective gear to carry out those tasks. It was both calculated, callous, and inhumane, but it was also highly profitable to those in charge.

It was an opening for lead-free, and since that time, lead-free has, in general, been a moneymaking machine that continues to crank out profits because it is a mandate that will never have a single solution. There are too many variables to control, and too many chasing the "perfect" solution. I have some dear and highly respected friends and colleagues in this industry who have—year after year—tried to solve the problem, but like those who play the fun but insidious game "whack-a-mole," it is unlikely they will ever win because one never knows when the next best thing will be found that will make everything perfect at last. It has not happened yet. Instead, the horizon has just gotten more cluttered with prospective solutions.

With that said, and again for the benefit of the newcomers to the electronics interconnection industry, the recounting of some of the important reasons why lead-free should never have been adopted are hereafter provided so that those newcomers might have these reasons to ponder and reflect upon as they themselves step up with mallet in hand to give the game of "whack-a-mole" their best shot. I suspect when one considers these reasons, one might as well ask themselves, "Is lead-free solder really a panacea, or is it a pandemic?

Reason 1

At the time of the ban, lead in electronic solders accounted for approximately 0.5% of all lead used globally. To target such a small quantity with such sweeping and overwhelming legislative force was the equivalent of using a shotgun to exterminate a mosquito.

Reason 2

Lead in electronic solder has never been proven to cause a single individual any harm from normal use and standard and reasonable sanitary practices. (If any reader has data to the contrary, please do not hesitate to share it with me so I can be corrected.)

Reason 3

Legacy lead from electronic solders that has ended up in landfills has not been shown to leach into groundwater. Even with years of such legacy lead from many other sources, such as batteries and painted structures, studies done at the time showed that all but a few landfills around the nation had significant quantities of lead leaching, and these were in landfills which took in significant waste from nearby high lead use industries [1].

Reason 4

Most lead-free solders contain substantial percentages of silver (~3–4%), and silver is toxic to microbial life and fish larva. This has often been ignored perhaps because it was felt that it would be recaptured and recycled.

Reason 5

The global conversion cost to lead-free ran to perhaps $100 billion or more with no real benefit [2]. The conversion cost continues to this day.

Reason 6

Lead-free solders typically require significantly higher temperatures to solder, meaning that more energy is required and will be required on an ongoing basis. The higher temperatures required also negatively impact and influence the PCB material choice, as well as PCB and semiconductor devices and other component reliability.

Reason 7

The definitive reliability of lead-free solders remains elusive to this day. Alloy after alloy is trotted out to run the assembly gauntlet with varying degrees of success and it remains an uncertainty how well they will perform over time, as there are numerous effects (cracking, voiding, tin whiskers [3], etc.)—all of which have causes that remain, in many cases, and are neither well understood nor predictable.

Reason 8

Several early studies from both universities [4] and the U.S. EPA [5] (and which were largely ignored) indicated at the time that, on-the-whole, lead-free electronics appeared to be much less environmentally friendly than those using tin-lead solder. Again, the primary case against lead was its toxicity, which for lead—as metal or in an alloy—was and remains unlikely to cause harm unless directly ingested. (Most common lead salts, hydroxides, chlorides, and sulfates are insoluble in water and many acids.)

Reason 9

Traditional tin-lead solders continue to be used by military and aerospace manufacturers that are exempt because they must have the highest levels of reliability, resulting in a dual path that has created an ongoing logistical headache; this is becoming ever more challenging as sources of tin-lead finished components continue to dry up.

Reason 10

Interconnection innovation remains hobbled as engineers are diverted from solving real technological problems to solving a "lead problem" that never existed in the opinion of most scientists.

Reason 11

Tin resources have always been significantly lower than those of lead; as a result, new tin mines were opened often in areas where rival factions fight to the death for control of the mines. It is also highly unfortunate that some of the richest tin mines are found in rain forests, which must be clear-cut to get to the tin deposits, contributing to global warming [6].

Reason 12

The provisions of the Waste Electrical and Electronic Equipment (WEEE) regulations mandate the recycling of electronic products, making all the concerns of lead in electronics completely moot.

Conclusion

As the timeless adage goes, "The road to hell is paved with good intentions." If you can feel the heat as you are trying to make your products better and more reliable, you may well get the impression you have already arrived there. Then again, as Winston Churchill sagely said, "When you are going through hell, keep going."

Keep up your spirits, and stay well. **FLEX007**

Editor's Note: Read the *Solderless Assembly for Electronics: The SAFE Approach* e-book written by Joe Fjelstad to consider an alternative

approach to manufacturing electronics without solder. Visit I-007eBooks.com to download this and other free, educational titles.

References

1. G.C. Munie, L. Turbini, D. Berneir, J. Gamalski, & D.W. Bergman, "Examining the Environmental Impact of Lead-Free Soldering Alternatives," Proceedings of IEEE International Symposium on Electronics and the Environment, San Francisco, California, May 2000.
2. H. Miller, "Lead-free Electronic Solder, Why?" InfraFOCUS, 2002.
3. nepp.nasa.gov/WHISKER
4. L. Turbini, "The Real Environmental Cost of Lead-Free Soldering," *Solid State Times,* Volume 4, Issue 3, July 2002.
5. "Solders in Electronics: A Life-Cycle Assessment," U.S. Environmental Protection Agency August 2005.
6. K. Hodal, "Death Metal: Tin Mining in Indonesia," The Guardian, November 23, 2012.

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.

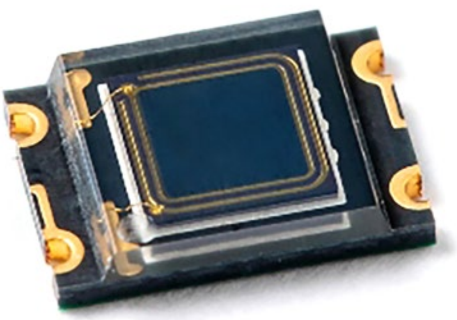
Kyoto Semiconductor Develops Broad Range Sensitivity Photodiode in Industry's Smallest Class

In the 40 years since the company's founding, Kyoto Semiconductor Co., Ltd., Tsuneo Takahashi, president and CEO, and headquartered in Fushimi Ward, Kyoto, has led the industry with optical solutions built from world-standard technologies and with Japanese quality and attention to production detail.

They have announced the development of a two-wavelength Photodiode, the KP-2 Two-tone photodiode KPMC29, in the industry's smallest class and capable of being surface-mounted, with silicon and indium-gallium-arsenide photodiodes, with photosensitivity for a wide range of wavelengths, from 400 to 1,700 nm, arrayed along the same light axis.

Development Background

Spectroscopic analysis technology, that allows for the identification of objects and their characteristics without needing direct contact with those objects, by reflecting light of those objects and measuring the degrees of transmission and reflection, is increasingly needed in a wide range of fields, including medicine, production, and security. Spectroscopic analysis relies on two factors for successful use in identifying a wide range of objects: a wider range of wavelengths, or colors, for the source wavelengths, and a wider range of sensitivity for those wavelengths with the photodiode end.



Development was therefore undertaken to meet these product needs.

Product Characteristics

1. A wider range of wavelength sensitivity (400~1,700nm). To expand the wavelength range, we layered both silicon photodiodes, which are sensitive to shorter wavelengths, and indium-gallium-arsenide photodiodes, which are sensitive to longer wavelengths, along the same light axis.

2. A compact surface-mountable package in the industry's smallest class. To successfully make this KP-2 two-tone PD smaller, the indium-gallium-arsenide photodiodes were laid down in pits arranged on the base for the silicon photodiodes, which ordinarily allows light to pass through. This allowed for the package height to be as low as possible. (Patents have been filed.) Compared to earlier products from Kyoto Semiconductor, the volume ratio has been reduced to 1/8.

(Source: Business Wire)

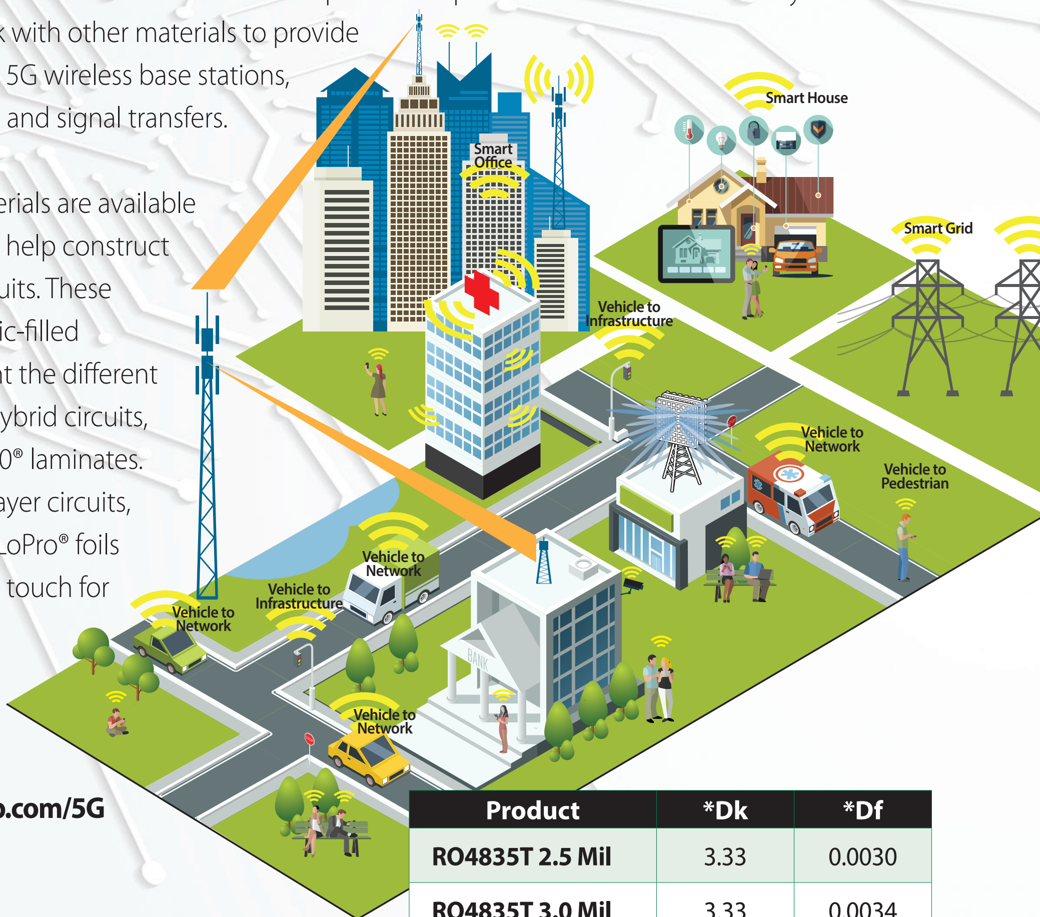
5G: Higher Frequencies!

Do you have the **right** circuit materials?

Frequencies at 28 GHz and higher are being used in Fifth Generation (5G) wireless communications networks. 5G infrastructure depends on low-loss circuit materials engineered for high frequencies, materials such as RO4835T™ laminates and RO4450T™ bonding materials from Rogers Corporation!

Rogers RO4835T spread-glass-reinforced, ceramic-filled laminates are low-loss materials in 2.5, 3.0, and 4.0 mil thicknesses. They are well suited for millimeter-wave frequencies as part of the inner cores of 5G hybrid multilayer PCBs. They can work with other materials to provide the many functions needed by 5G wireless base stations, including power, signal control and signal transfers.

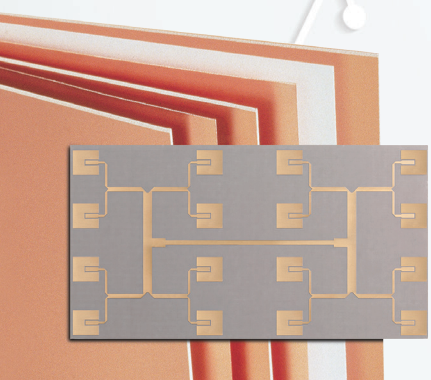
Rogers RO4450T bonding materials are available in 3, 4, and 5 mil thicknesses to help construct those 5G hybrid multilayer circuits. These spread-glass-reinforced, ceramic-filled bonding materials complement the different materials that will form these hybrid circuits, including RO4835T and RO4000® laminates. And for many 5G hybrid multilayer circuits, Rogers CU4000™ and CU4000 LoPro® foils will provide a suitable finishing touch for many hybrid multilayer circuit foil lamination designs.



5G is here! Do you have the right circuit materials?
Learn more at **www.rogerscorp.com/5G**

Product	*Dk	*Df
RO4835T 2.5 Mil	3.33	0.0030
RO4835T 3.0 Mil	3.33	0.0034
RO4835T 4.0 Mil	3.32	0.0036
RO4450T 3.0 Mil	3.23	0.0039
RO4450T 4.0 Mil	3.35	0.0040
RO4450T 5.0 Mil	3.28	0.0038

** IPC TM-650 2.5.5.5 Clamped Stripline at 10 GHz - 23˚C*



ROGERS CORPORATION

Advanced Connectivity Solutions

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

Flex007 Highlights

Lenthor Engineering Adds Dale Smith in the Role of Chief Technology Officer ▶

Lenthor Engineering Inc.—a California-based designer, manufacturer, and assembler of rigid-flex and flex PCBs—is pleased to announce the addition of Dale Smith in the role of chief technology officer.

Price Circuits LLC Installs ESI 5335 Laser Drill ▶

Price Circuits LLC, a division of Circuitronics LLC, is pleased to announce another addition to our recent expansion. The ESI 5335 laser has been installed and fully operational. With our focus on supporting complex PCBs technologies in rigid, flex, and rigid-flex technologies, our technical capability has expanded in 2020.

Ynvisible and Evonik Collaborate in Printed Electronics ▶

Ynvisible Interactive Inc. and Evonik Creavis GmbH, the strategic innovation unit of Evonik Industries AG—a global specialty chemicals group headquartered in Essen, Germany—announced a collaboration in the field of printable electronics and introduced a first joint technology demonstrator.

NextFlex Leads Recipients of $5M in Naval Research Funding ▶

NextFlex, America's Flexible Hybrid Electronics (FHE) Manufacturing Institute, announced the expansion of its manufacturing workforce education activities with $5 million in funding from the Office of Naval Research (ONR).

New Smart Wearable Promotes Employee Safety as Business Reopens ▶

IoT connectivity and security startup Nodle, nonprofit foundation Coalition Network, and leading global technology solutions company Avnet announce the creation of a new smart wearable contact tracing device—the Nodle M1.

ESA Approval for Ventec VT-901 Material in ACB Belgium Rigid and Flex-Rigid Production ▶

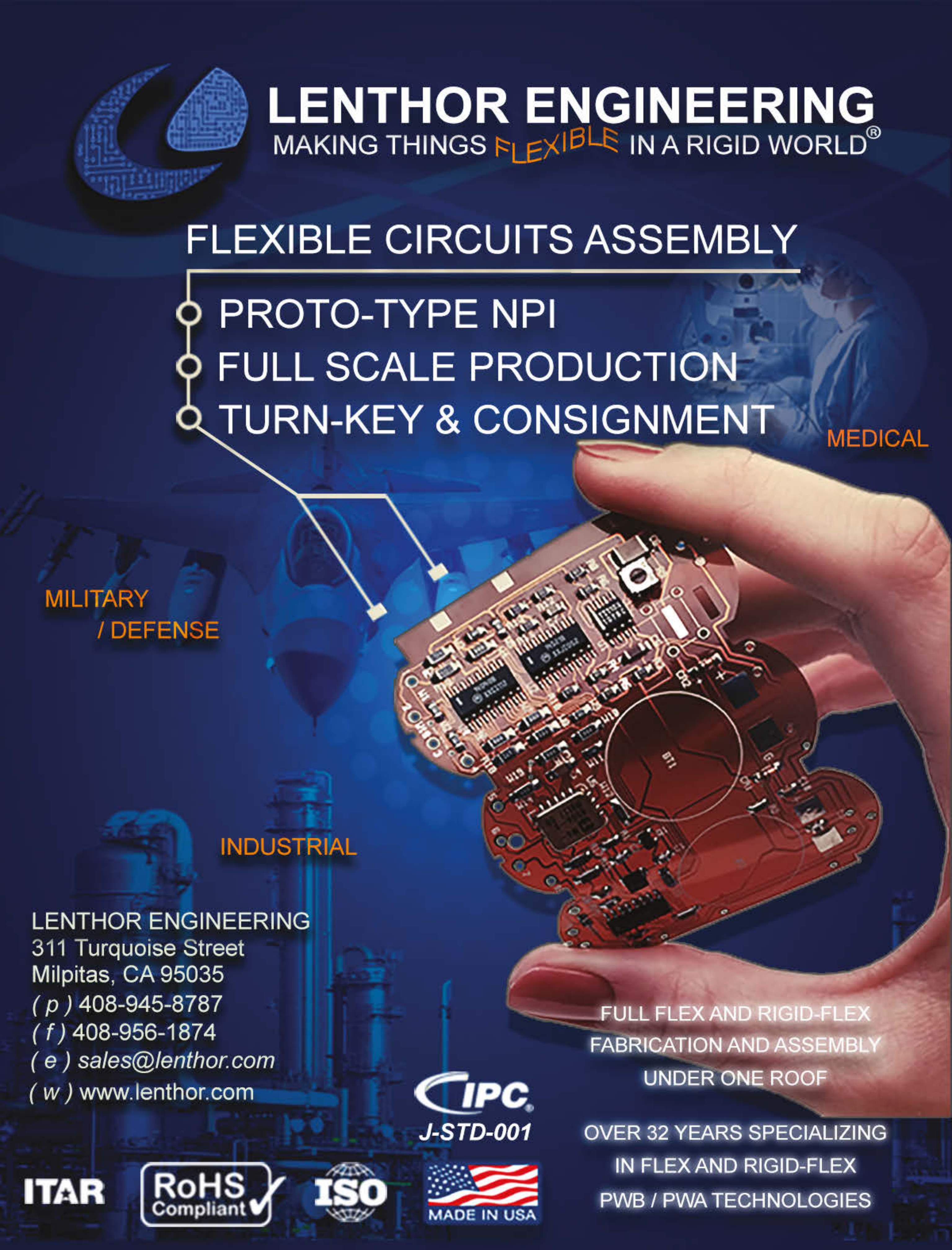
Ventec International Group Co. Ltd. is pleased to announce that its VT-901 polyimide material is now fully qualified by ESA in ACB Belgium's manufacturing process for rigid and rigid-flex polyimide PCBs and HDI PCBs.

PCB Connect Expands With New Office, New Employee, and Opens New Company in Denmark ▶

Due to PCB Connect's successful growth in Denmark, the company decided to invest further in the Danish Market. They are very glad to announce a new and bigger office location, a new employee, and that they will have a Danish company PCB Connect A/S registered.

Powercast and Liquid X Announce Printed Electronics Venture to Enable Durable, Washable e-Textiles That Seal in Wireless Charging Electronics ▶

Powercast Corporation, the leader in RF-based long-range over-the-air wireless power technology, and Liquid X, an advanced manufacturer of functional metallic inks with prototype-to-production design and manufacturing capabilities, announced a printed electronics venture to enable garment manufacturers to easily integrate wireless power functionality into durable, flexible, high-performance, and washable e-textiles.


LENTHOR ENGINEERING
MAKING THINGS FLEXIBLE IN A RIGID WORLD®
FLEXIBLE CIRCUITS ASSEMBLY
PROTO-TYPE NPI
FULL SCALE PRODUCTION
TURN-KEY & CONSIGNMENT
MEDICAL
MILITARY / DEFENSE
INDUSTRIAL
LENTHOR ENGINEERING
311 Turquoise Street
Milpitas, CA 95035
(p) 408-945-8787
(f) 408-956-1874
(e) sales@lenthor.com
(w) www.lenthor.com
FULL FLEX AND RIGID-FLEX
FABRICATION AND ASSEMBLY
UNDER ONE ROOF
IPC
J-STD-001
OVER 32 YEARS SPECIALIZING
IN FLEX AND RIGID-FLEX
PWB / PWA TECHNOLOGIES
ITAR
RoHS Compliant
ISO
MADE IN USA

Next-Generation Flex Circuits: Elastic

EPTE Newsletter

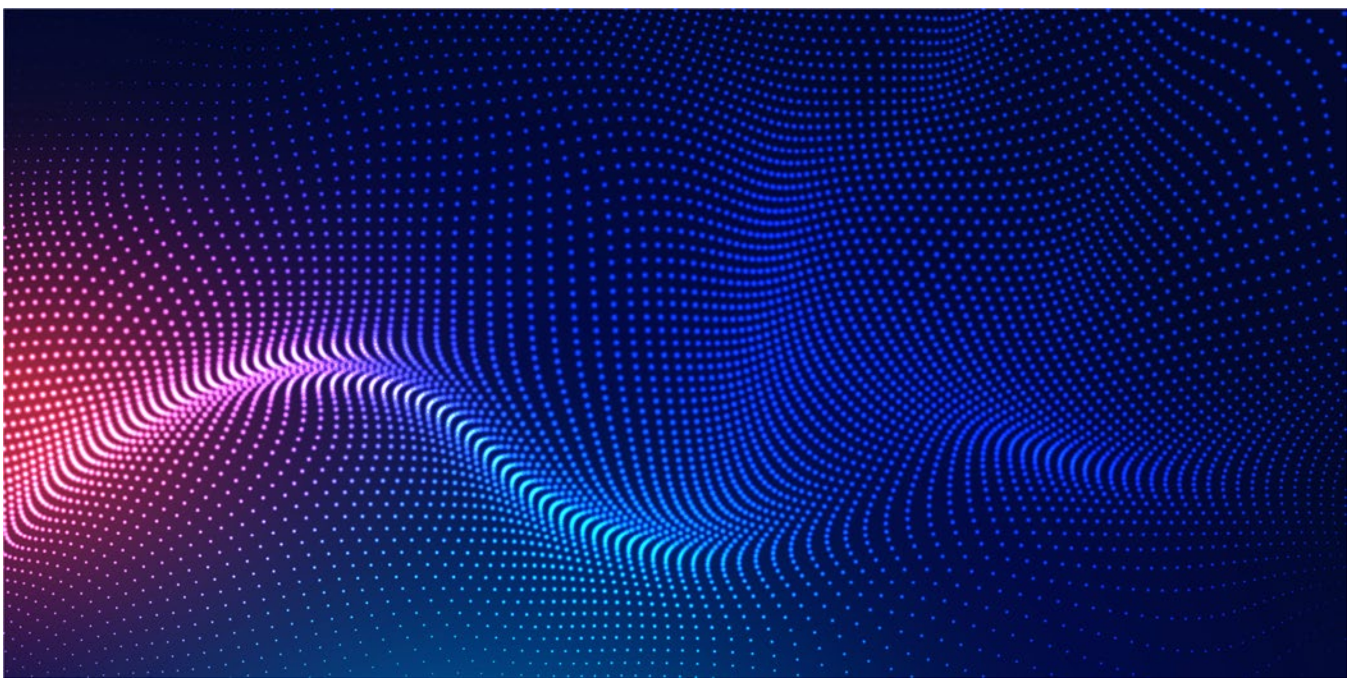
by Dominique K. Numakura, DKN Research LLC

My experience in the flex circuit industry spans almost 40 years. Industry production volume grew exponentially during my career as circuit manufacturing shifted from North America and Europe to Eastern Asia. Japan was the manufacturing leader for the consumer market, but that has spread amongst new companies in Taiwan, Korea, and China.

The number of countries that manufacture circuits has expanded, but the basic construction and manufacturing process remains the same. While circuit density is much higher, and automated processes lower manufacturing costs, circuit designers still use single-sided, double-sided, and rigid-flex constructions in their designs. Several new design capabilities are now available using flex circuits. They are revolutionary due to their unique performances and cannot be created using traditional flex circuits. Let's look at some of these next-generation flexible circuits.

Wearable electronics and health care devices brought a need for elastic, flexible circuits. Traditional flexible circuits are built on thin, flexible, plastic films, such as polyimide films and polyethylene terephthalate (PET) films. The standard construction of these flexible circuits allows them to flex over 10 million times and is the reason they are employed as the major wiring material for mechanical and electronic devices found in hard disc drives and inkjet printers.

New applications from wearable and medical devices demanded elasticity for circuits that attach to the human body. Traditional flexible circuits cannot satisfy the new requirements of




A BRIGHT FUTURE FOR LEDS
PSR-4000 CR01MW
Bright White
Excellent Crack Resistance
Matte Finish
Low Image Growth
Excellent Chemical Resistance
PSR-4000 CR01MW after ENIG
TAIYO AMERICA
UNMATCHED INNOVATION
Phone [775] 885-9959 • www.taiyo-america.com • info@taiyo-america.com

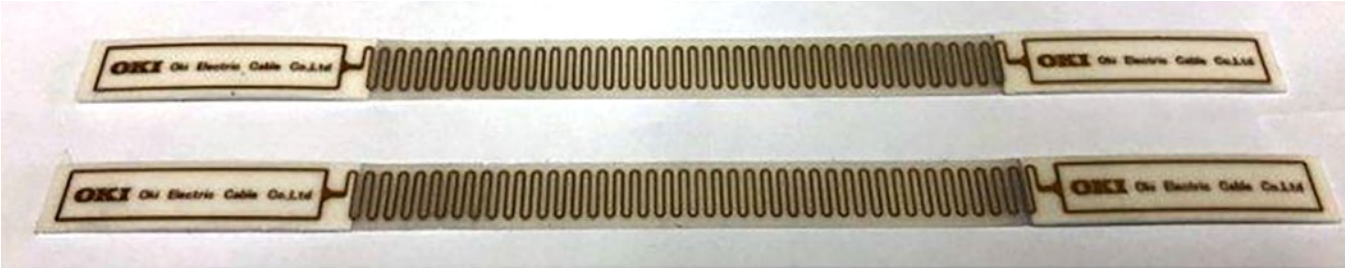


Figure 1: Elastic, flexible circuits with meander pattern provided by Oki Cable.

these products. Generating fine traces from the circuits' base films relied on dimensional stability, but new applications are completely different.

Several circuit manufacturers and material suppliers are collaborating to develop new elastic, flexible circuits. Urethane rubber could be an appropriate base material once its relatively low-heat resistance is improved. Silicone rubber is the perfect material to use as the base layer and cover coating, due to its heat resistance and transparency. The increase in cost is the only problem.

Several circuit manufacturers and material suppliers are collaborating to develop new elastic, flexible circuits.

Elastic conductor material has to be addressed. One solution is to have laminate suppliers develop copper-clad laminates with elastic sheets. Circuit manufacturers could etch the copper foil to generate the circuits using a typical manufacturing process for flexible circuits. Copper foils do not have the equivalent elasticity as rubber materials, so conductors have to use a meander pattern (Figure 1). This pattern allows the circuits to stretch by 50%.

Creating flexible conductors using a screen-printing process could be another choice for elastic conductors. Printed conductors can have a much higher elasticity compared with copper foils depending on the ink used for the binder material, but the circuit's conductivity can decrease by one hundredth.

Another challenge with elastic conductors is they are not very stable. The resistance of the conductor is proportional to the elongation of the circuit. For example, when the conductor is stretched by 100%, the resistance doubles. This necessitates the use of meander patterns for printed conductors. Unfortunately, when using meander patterns instead of straight patterns, the amount of space needed for circuits increases. The circuits are larger, and a rise in costs is inevitable due to an increase in the usage of materials. It may need supplemental ideas and experience to build high-density circuits on the elastic substrates.

Creating elastic circuits does not employ conventional thinking. Because the resistance from the elastic conductor is proportional to its elongation, we can estimate how much the conductor is stretched measuring its resistance. This is the basic mechanism of the strain gauge.

The demands for elastic, flexible circuits will grow significantly in the next few decades as the market for wearable, and medical electronics will continue to grow. This segment forces us to think outside of the box for material usage, design ideas, and manufacturing processes. Anything is possible. FLEX007

Dominique K. Numakura is the managing director of DKN Research LLC. Contact haverhill@dknreseach.com for further information and news.

1 Dana on Data: The Importance of PCB Technology Roadmaps ▶


Dana Korf

Dana Korf explains how it is critical that PCB fabricator technology roadmaps and capacity planning align with their customers' product development and volume requirements to ensure that optimum cost, reliability, and performance goals are achieved.

2 This Month in *Design007 Magazine:* What Did You Expect From Me, Anyway? ▶


Todd Westerhoff

As engineers, we work in the middle of a (usually long) process chain. It seems easy enough. So, why is it that so many of the requirements we're supposed to meet and so much of the data we receive is downright bad?

3 A Design Economics Horror Story ▶


Rick Hartley

During a recent conversation with Rick Hartley, he shared one of his favorite PCB design horror stories. This is a cautionary tale about what can happen when design teams place too much faith in app notes and do not follow cost-aware design techniques. Enjoy!

4 Ventec Book Excerpt: Thermal Management With Insulated Metal Substrates ▶

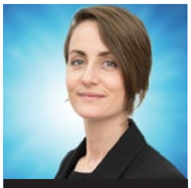
The Printed Circuit Designer's Guide to... Thermal Management With Insulated Metal Substrates, written by Ventec International Group's Didier Mauve and Ian Mayoh. In this free eBook, the authors provide PCB designers with the essential information required to understand the thermal, electrical, and mechanical characteristics of insulated metal substrate laminates.

5 Connect the Dots: Increased Focus on Health and Wellness Transforms the PCB Industry ▶


Stevenson and Tise

Our increased focus on health and wellness drives technology advancement for personal devices and those used in the delivery of healthcare. Bob Tise and Matt Stevenson explain how this trend also drives both PCB production innovation and a long-overdue update of the employer/employee relationship.

6 Sensible Design: Thermal Management—Five Tips for Application Success ▶


Jade Bridges

With so much to consider when choosing a thermal management material, it's important to do your calculations, consider the equipment's operational, and environmental conditions and experiments. Underestimating these could compromise the reliability of an electronic assembly and shorten its life expectancy.

7 A Design Economics Primer ▶


Chris Young

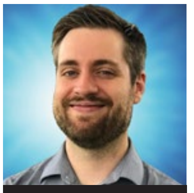
Chris Young begins cost-aware design before the design cycle has even begun. Andy Shaughnessy and Nolan Johnson recently interviewed Chris, an engineer with The Goebel Company and founder of Young Engineering Services, and asked him to explain his approach to design economics.

8 The Pulse: Communicating Materials From Design to PCB Fabrication ▶


Martyb Gaudion

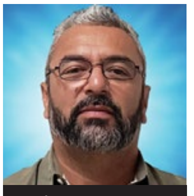
Designer and fabricator communication—especially for high-speed PCBs—should be a bidirectional "thing." It is so easy for a designer to say, "Just build this," and hand over a challenging design to a fabricator who could have performed better with some preliminary conversation or dialog before placing the order. Martyn Gaudion explores communicating materials from PCB design to fabrication.

9 Design Circuit: Competition as a Tool for Growth ▶


Patrick Crawford

In his debut column, Patrick Crawford—manager of design programs and related industry programs at IPC—highlights the new IPC Design program and announces plans for an international printed board design competition, which will also include a student division.

10 Fresh PCB Concepts: Why Design and Produce PCBs Beyond Industry Specification? ▶


Ruben Contreras

While questions may be an annoyance, especially when you finally have a board designed and are ready to have it built for a product, Ruben Contreras explains the importance of asking these questions and requiring specifications.

Career Opportunities

Career Opportunities

BÜRKLE NORTH AMERICA, INC.

Service Engineer

Schmoll Laser Drilling and Direct Imaging

Burkle North America seeks a full-time service engineer in the Northeastern U.S. This position will provide expert-level service on multiple laser drilling and direct imaging product lines. Install, commission, and maintain Schmoll products at multiple customer sites across the Northeast. The candidate will perform modifications and retrofits as needed. Maintain complete and detailed knowledge of Schmoll products and applications and handle a wide variety of problems, issues, and inquiries to provide the highest level of customer satisfaction. Assist customers with the potential optimization of their machine functions and work with clients on application improvements.

Qualifications

Required: Bachelor's degree from a technical college/university in an associated field. Three years directly related experience, or equivalent combination of education and experience. Must possess a valid driver's license and have a clean driving record.

Preferred: Experience in control systems and electronic troubleshooting, as well as in general electrical and mechanical service tasks. Experience and knowledge in the PCB manufacturing process, with a focus on laser drilling and/or direct imaging.

Send resume to hr@burkleamerica.com.

apply now

WHELEN®
TRUSTED TO PERFORM™

Process Engineering Director

Whelen Engineering Co., Inc. seeks full-time process engineering director in Concord, NH, to develop, plan and execute GreenSource Fabrication, LLC Div.'s process technology business strategy; manage process engineering activities, staff and compliance; improve process design, cost, quality and resource utilization; interact w/ customers and incorporate feedback; develop financial capital and labor projections; travel internationally for conferences, supplier and customer visits (15-25% worktime); write white papers, IP applications and give talks re. Division's products/processes.

Min. req.: U.S. Bachelor's or foreign equivalency in environmental science or engineering; min. 10 yrs. work exp. in: PCB fabrication process engineering; comprehensive and current experience in PCB fabrication/substrate markets w/ SAP tech; developing chemical and mechanical processes, chemistries and equipment for PCB manufacturing demonstrated by international experience implementing complex processes; ability to direct and troubleshoot PCB manufacturing problems; min. 5 years exp. leading, managing and training process engineering teams, developing and executing process technology business strategies and plans in worldwide PCB markets, including Japan, Taiwan, China, Europe; min. 3 years exp. giving talks, writing and presenting white papers; ability to travel internationally (15-25% worktime).

Send CVs to: Corinne Tuthill,
ctuthill@greensourcefab.com
or GreenSource Fabrication, LLC,
99 Ceda Road, Charlestown, NH 03603.

apply now

Career Opportunities

FREEDOM CAD SERVICES

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

eptac
TRAIN. WORK SMARTER. SUCCEED.

Become a Certified IPC Master Instructor

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

Qualifications and skills

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

Benefits

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

apply now

Career Opportunities

Career Opportunities

Career Opportunities

U.S. CIRCUIT

Sales Representatives (Specific Territories)

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

- Florida
- Denver
- Washington
- Los Angeles

Experience:

- Candidates must have previous PCB sales experience.

Compensation:

- 7% commission

Contact Mike Fariba for more information.

mfariba@uscircuit.com

apply now

ZENTECH

Zentech Manufacturing: Hiring Multiple Positions

Are you looking to excel in your career and grow professionally in a thriving business? Zentech, established in Baltimore, Maryland, in 1998, has proven to be one of the premier electronics contract manufacturers in the U.S.

Zentech is rapidly growing and seeking to add Manufacturing Engineers, Program Managers, and Sr. Test Technicians. Offering an excellent benefit package including health/dental insurance and an employer-matched 401k program, Zentech holds the ultimate set of certifications relating to the manufacture of mission-critical printed circuit card assemblies, including: ISO:9001, AS9100, DD2345, and ISO 13485.

Zentech is an IPC Trusted Source QML and ITAR registered. U.S. citizens only need apply.

Please email resume below.

apply now

Career Opportunities

BLACKFOX
Premier Training & Certification

IPC Master Instructor

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

For more information, click below.

apply now

Professionals Seeking Employment

D.B. Management Group L.L.C. is currently working with many professionals who are seeking new positions. If any of these qualified professionals sounds like someone you would like to learn more about, contact **Dan Beaulieu** at **207-649-0879** or danbbeaulieu@aol.com. If you are a qualified professional looking for a new opportunity, contact Dan as well. Fees are 10% of candidates' first year's annual compensation. There is no fee for candidates.

Click here to learn more ▶

President, Company Leader, Business Builder
This professional has done it all. Built new businesses and turned around hurting businesses and made them successful. A proven record of success. This candidate is a game-changer for any company. He is seeking a full-time leadership position in a PCB or PCBA company.

General Manager PCB and PCBA
Senior manager with experience in operations and sales. He has overseen a number of successful operations in Canada. Very strong candidate and has experience in all aspects of PCB operations. He is looking for a new full-time position in Canada.

Regional Sales Manager/Business Development
Strong relationship management skills. Sales experience focused on defense-aerospace, medical, high-tech PCB sales. Specializes in technical sales. Also has experience in quality, engineering, and manufacturing of PCBs. He is looking for a fulltime position in the Southeastern U.S.

Field Application Engineer (FAE)
Has worked as a respected FAE in the U.S. for global companies. Specializes in working alongside sales teams. Large experience base within the interconnect industry. He is looking for a full-time position.

Business Development Manager
Understands all aspects of interconnect technical sales from PCB design and fabrication to assembly and all technologies from HDI microvias to flex and rigid-flex. Has also sold high-tech laminates and equipment. Proven record of sales success. He is looking for a full-time position.

CEO/President
Specializes in running multi-million dollar companies offering engineering, design, and manufacturing services. Proven leader. Supply chain manager. Expert at developing and implementing company strategy. Looking to lead a company into the future. He is looking for a full-time position.

PCB General Manager
Forty years of experience serving in all capacities, from GM to engineering manager to quality manager. Worked with both domestic and global companies. Available for turn-around or special engineering projects. He is looking for long-term project work.

Process Engineering Specialist
Strong history of new product introduction (NPI) manufacturing engineering experience: PCB/PCBA. Held numerous senior engineering management positions. Leads the industry in DFM/DFA and DFX (test) disciplines. He is looking for either a full-time position or project work.

VP Sales Global Printed Circuits
Worked with a very large, global company for a number of years. Built and managed international sales teams. Created sales strategies and communicated them to the team. One of the best sales leaders in our industry. He is looking for a full-time position.

Plant Manager
This professional has years of experience running PCBA companies. Led his companies with creative and innovative leaderships skills. Is a collaborative, hands-on leader. He is looking for a full-time position.

National Sales Manager
Seasoned professional has spent the past 20 years building and growing American sales teams for both global and domestic companies. Specializes in building and managing rep networks. He is looking for a full-time position.

Global Engineering Manager/Quality Manager
Has experience working with large, global PCB companies managing both engineering and quality staff. Very experienced in chemical controls. She is interested in working on a project-by-project basis.

CAM Operators and Front-end Engineers
These candidates want to work remotely from their home offices and are willing to do full-time or part-time projects.

WHAT DO ALL OF THESE LEADING COMPANIES HAVE IN COMMON?

atg LUTHER & MAELZER

ATOTECH

CHEMCUT
BOUNDLESS INNOVATION | UNBEATABLE PRECISION

CIMS
Creativity in Motion

CYBEROPTICS

DownStream Technologies

ELECTROLUBE
THE SOLUTIONS PEOPLE

esi
Designed for Brilliance. Engineered for Production.

KIC Making Ovens Smarter™

KYZEN

MacDermid Alpha
ELECTRONICS SOLUTIONS

Mentor
A Siemens Business

MIRTEC

NANODIMENSION
Electrifying Additive Manufacturing

orbotech
The Language of Electronics

pluritec

PULSONIX

rehm THERMAL SYSTEMS

schmoll asia pacific

ventec
INTERNATIONAL GROUP
騰輝電子

VJ ELECTRONIX
Process Control Solutions
CONFIDENCE DELIVERED

YXLON

They all benefit by advertising with us–PCB007 China Magazine.

Should your name be on the leaders' list?

GET STARTED NOW!

I-Connect007
GOOD FOR THE INDUSTRY

pcb007china.com

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

ASSOCIATE EDITOR: **MICHELLE TE**
michelle@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 © ALPHASPIRIT**

DESIGN007 MAGAZINE

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

June 2020, Volume 9, Number 6
DESIGN007 MAGAZINE is published monthly, by BR Publishing, Inc.

ADVERTISER INDEX