THE pcb design MAGAZINE

December 2017

an I-Connect007 publication

Dr. Johannes Adam Sounds the Alarm for Thermal Design p.16

Thermal Management Update with Doug Brooks p.22

Streamlining Thermal Design of PCBs p.28

Much More!

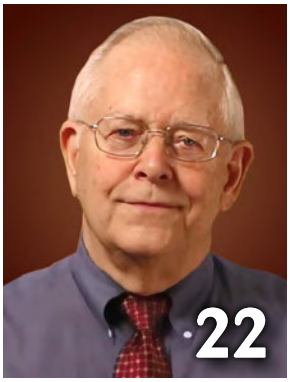
Thermal Design: Sounding the Alarm

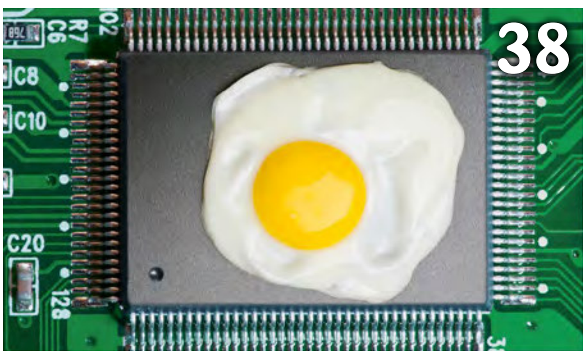
pg. 16

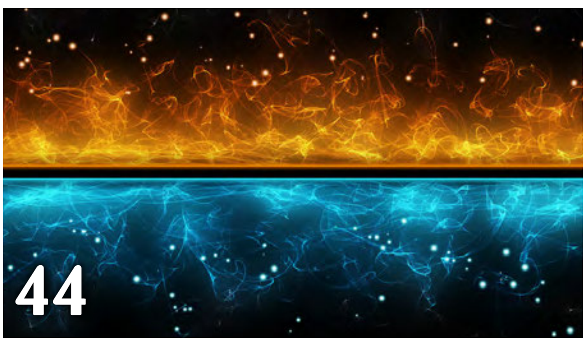

my
Premium Content
Download free books from our micro eBook library!
THE PRINTED CIRCUIT ASSEMBLER'S GUIDE TO...™
Conformal Coatings for Harsh Environments
Phil Kinner
Electrolube
I-007e Books
THE PRINTED CIRCUIT DESIGNER'S
Signal Integrity by
Secrets o
Design
Flex and
Written by industry experts, these free micro eBooks each contain valuable information on relevant topics in an easy-to-read format.
Log in or register to access all of our premium content at: myiconnect007.com
my
I-Connect007
GOOD FOR THE INDUSTRY

December 2017

Featured Content







Thermal Management

December is here, and much of the East Coast is under a blanket of snow. So, to take the chill off, we're going to turn up the heat with December's thermal management issue.

16 **Dr. Johannes Adam Sounds the Alarm for Thermal Design**
Interview

22 **Thermal Management Update with Doug Brooks**
Interview

28 **Streamlining Thermal Design of PCBs**
by Dr. John Parry

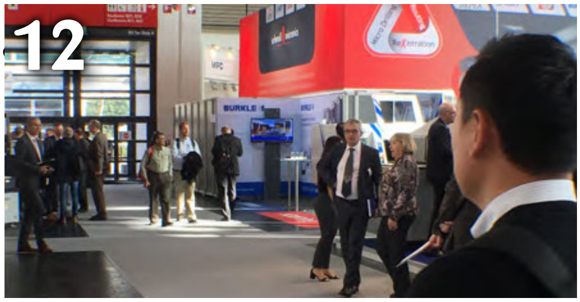
38 **That's Hot: Ventec's Goodwin on Thermal Management**
Interview with Mark Goodwin

44 **Thermal Management: A PCB Manufacturer's Perspective on Insulated Metal PCBs**
by Anaya Vardya and Dave Lackey

THE pcb design MAGAZINE

December 2017

More Content



SHORTS

20 **Solar Cells With a Quantum Shift**

42 **Simulating High-Temperature Superconductors**

62 **Revolutionizing Electronics Using Kirigami**

70 **New Robots Can See Into Their Future**

SPECIAL SECTION

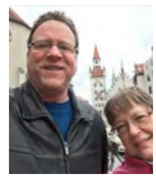
12 **productronica 2017 Photo Review**

ARTICLE

66 **IEEE's Romanian SIITME Show a Success**
by Joe Fjelstad

COLUMNS

8 **Reporting from Germany**
by Andy Shaughnessy



58 **Signal Flight Time Variance in Multilayer PCBs**
by Barry Olney

HIGHLIGHTS

56 **PCB007**

64 **MilAero007**

72 **Top Ten PCBDesign007**

DEPARTMENTS

76 **Career Opportunities**

86 **Events Calendar**

87 **Advertisers Index & Masthead**

Reporting from Germany

by Andy Shaughnessy
I-CONNECT007

I'm sitting here in Atlanta in a house with no power, watching the snow blow sideways while I work on whatever I can do in Word. It's starting to get cold inside, but after a week in Munich for productronica, I'm used to cold weather!

It was freezing every day in Munich, which is fitting, because parts of the city are frozen in time. Much of Marienplatz looks like it must have in the 1500s, despite the Allies' carpet-bombing efforts. We visited a church that probably dated to the Renaissance, and there were photos on the wall showing a priest giving Mass in 1945 with sunlight shining through the missing roof.

Munich is a resilient city. After WWII, the residents rebuilt it, brick by brick, with the "Trümmerfrau," or "rubble women," leading the way, because of the absence of adult males. They took care of business.

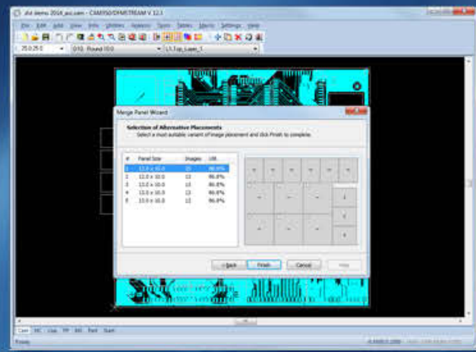
Munich isn't afraid to take a chance on a new idea. Their city government switched from Windows to a customized version of Linux 10 years ago, and just recently voted to switch back to Windows. (In case you're curious, other users of desktop Linux include the DoD, CERN, China, and North Korea.) And Munich is clean. I didn't see any trash on the street. There aren't a lot of public trash cans, because someone could put a bomb in a trash can.

Even in the cold and sometimes rainy weather, everyone I met in Munich was as nice as they could be. Most people in Munich speak English, and total strangers would walk up and ask if I needed help finding the right train. The trains were pretty easy to use, once I'd ridden them a few times. The main train station, the Hauptbahnhof, runs like its own city, with dozens of little restaurants, bars, and grocery stores. Editor Patty Goldman and I spent a day sightseeing in Marienplatz and trying to figure out the train system before the show started.

Solutions to Ensure PCB Manufacturing Success!

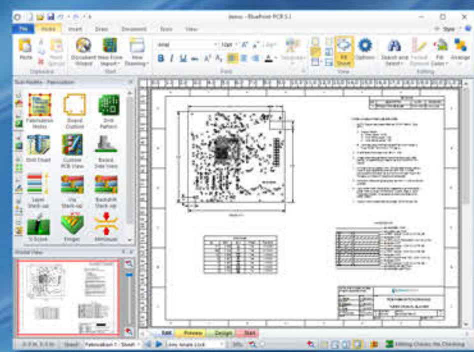
CAM350®

Verify and Optimize PCB Designs for Successful Manufacturing.



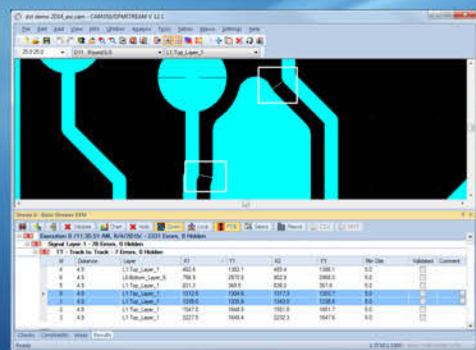
BluePrint·PCB

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



DFMStream™

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



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

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

DownStream Technologies


Andy Shaughnessy and Patty Goldman explore Munich during productronica 2017.

I'd never attended productronica, and it was something else. It must have been four times bigger than IPC APEX EXPO. Not quite as big as NAMM or a consumer show like CES, but it's one gigantic show, and spread out in separate halls. We did a lot of walking; the health app on my phone figured out that I walked 30,000 steps in one day. The Messe München venue has a room full of lockers, so you don't have to carry a winter coat around with you at a trade show.

I have never seen so much food at a trade show. Some companies had whole kitchens in their booths, cranking out lasagna, waffles, giant pretzels, and beer. Lots and lots of beer. I witnessed quite a few 10 a.m. booth meetings with every participant drinking a pilsner. That's just how it is in Germany. They really like beer.

The great thing about attending productronica was meeting the people from a company's European office, the ones who don't always come to shows in the U.S. Automotive was the biggest driver (pardon the pun) for most of the technologists I interviewed at productronica. Wireless was a close second. Every attendee and exhibitor I spoke with was bullish on our industry. Many companies were hiring, or planning to hire soon. One laminate rep said, "It's a great time to be in our industry."

Now, December is here, and much of the East Coast is under a blanket of snow. So, to take the chill off, we're going to turn up the heat with December's thermal management issue. First, we have an interview with Dr. Johannes Adam of Adam Research, a thermal expert who took the train in to meet with me during productronica. He discusses his TRM software, the books he's writing, and the need for electrical engineers to have a better understanding of thermal issues.

Next, Douglas G. Brooks discusses his collaboration with the same Dr. Adam on the book *PCB Trace and Via Temperatures: The Complete Analysis,* as well as some surprising findings, and why temperature charts based on a trace in isolation are inaccurate. Then, John Parry of Mentor explains how mechanical engineers and engineers, working together early in the process, can streamline thermal design time.

Ventec COO Mark Goodwin weighs in with a look at some new materials that feature a thermal capacity, and the effect of the automotive market on laminate development. And Anaya Vardya of American Standard Circuits discusses some of the choices facing PCB designers using insulated metal PCBs. We also have a column by Barry Olney on signal flight time variance in multilayer PCBs, and a review of the SIITME design show in Romania, courtesy of Joe Fjelstad.

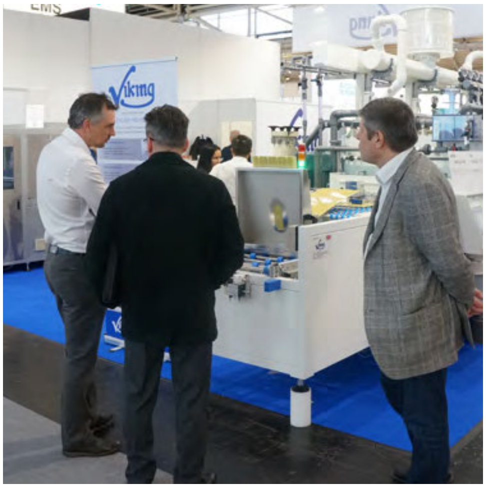
It's almost the new year, and DesignCon and IPC APEX EXPO are not far behind. We'll be covering both shows, so if you can't make, we have you covered.

Have a great holiday and Happy New Year!

PCBDESIGN


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

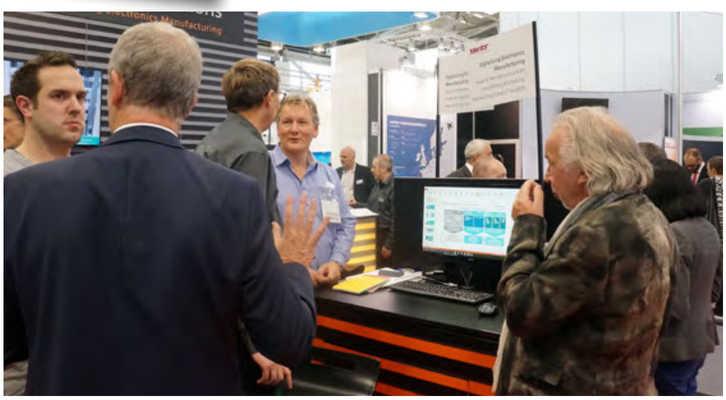
productronica 2017



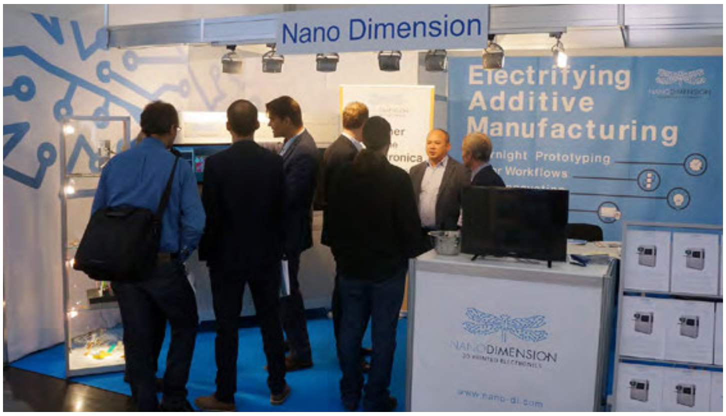
This was my first visit to productronica and it was all they said it would be. Hall after hall were filled with technology, though only one was focused on PCB manufacturing. All of the familiar equipment and chemistry manufacturers were there, along with a number of ones new to me.

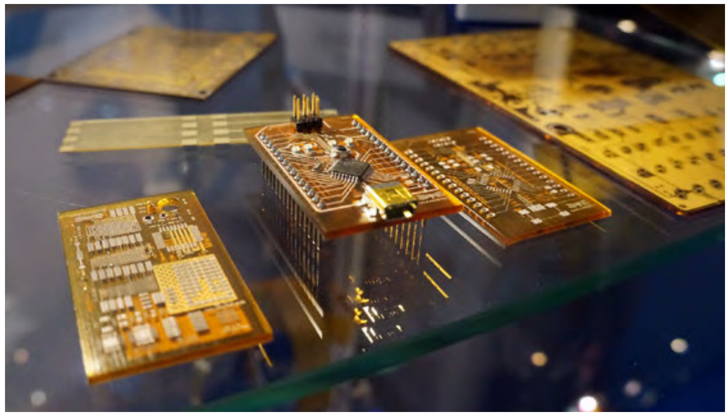
Hopefully these photos help to capture the flavor—literally, as food was the big giveaway at most booths—no pens, no gadgets, just wonderful food and pretzels! Click here to see our entire photo gallery.

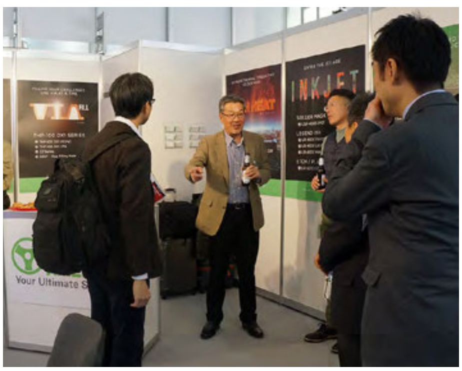
—Patty Goldman
I-Connect007

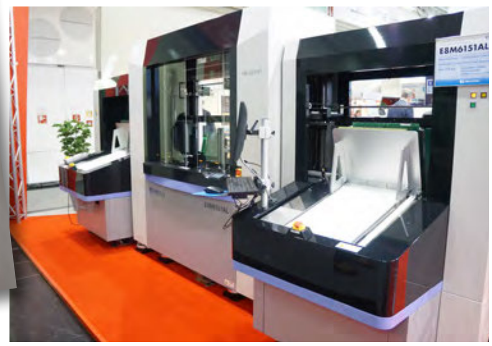


productronica 2017

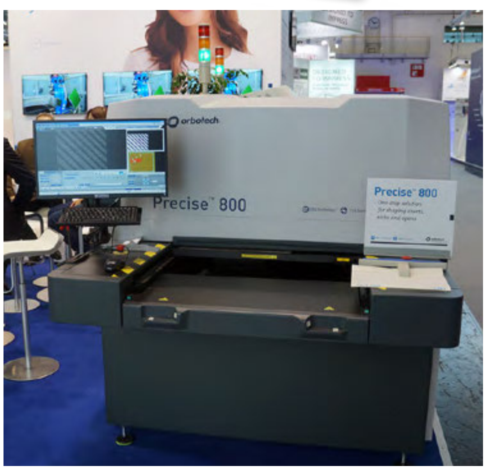


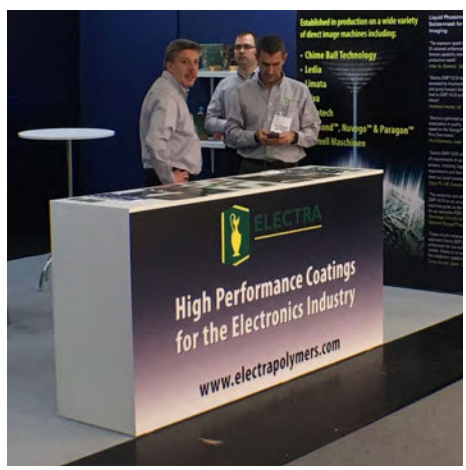




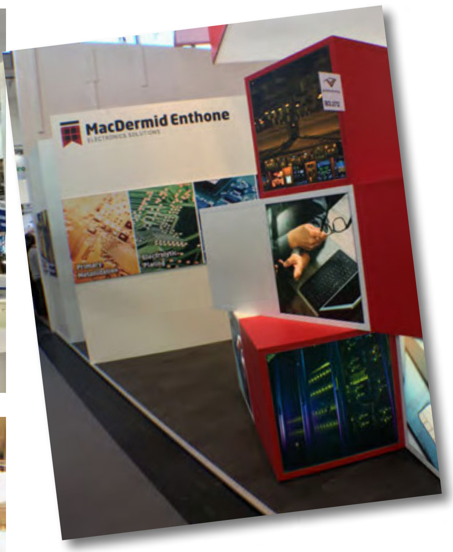


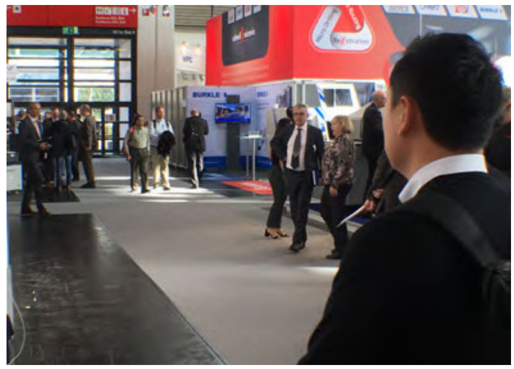
productronica 2017

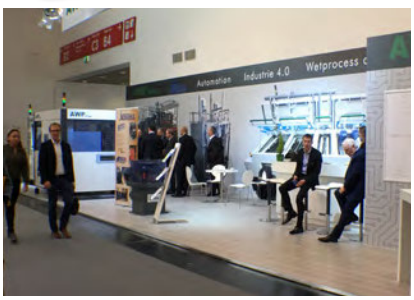


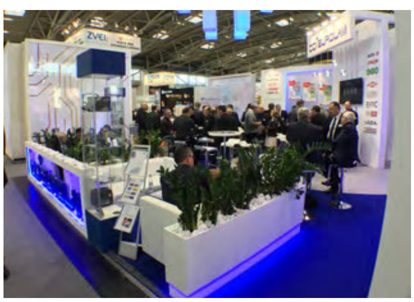


productronica 2017









Dr. Johannes Adam Sounds the Alarm for Thermal Design

by Andy Shaughnessy
I-CONNECT007

Thermal management expert Johannes Adam, PhD, was kind enough to take the train from his home in Leimen, Germany to meet with me during productronica in Munich. He is the creator of TRM (Thermal Risk Management) software and contributor, with Douglas G. Brooks, of *PCB Trace and Via Currents and Temperatures: The Complete Analysis, 2nd Edition.* He's also working on his own book on thermal management. Johannes sat down for an interview, and I asked him to share his views on the current state of thermal management for PCBs, and what the industry can do to put the spotlight on what's hot in PCBs.

Andy Shaughnessy: Johannes, why don't you give us a brief background of your career and how you got into thermal management?

Dr. Johannes Adam: I started in 1989 when I entered my first position in the industry. That position dealt with electronics cooling and the simulation of air flow in telecom racks of that time. The company that employed me was looking for a physicist who was able to do programming and to do user support. That was perfect for me, although I'm a professional astrophysicist.

Shaughnessy: There are not many astrophysicists in the PCB world.

Adam: A few. I never did serious observations with a telescope; instead I did programming solving differential equations in astrophysical scenarios. But, of course, there are intersections between both. We talk about the physics of heat flow and heat transfer in PCBs, and all this can be formulated in differential equations, which of course theoretical physicists are educated to solve.

Shaughnessy: What came next for you?

Adam: After seven years I joined another company, Flomerics. They were doing this business more professionally: solving airflow and heat transfer inside enclosures, cooling by heat sinks, cooling by fans, etc., all done numerically on the computer. I was doing some basic analysis of PCBs. In 2009, I decided to build up my own business and to concentrate on the heat flow and temperature in PCBs. For more than a decade I presented a seminar on thermal issues in PCBs for electrical engineers and PCB designers.

From those, and from customers, I learned what data they have and what they would need in order to calculate PCB temperatures before the manufacturing or the lab process. In my eyes, designing printed boards is an art and manufacturing is amazing.

All of this experience, previous and new, then went into my new TRM software. The basic idea is that you have an existing design, whether finished or just in the experimental state. You import the Gerber files from the layout software, add components, the dissipated power, and optionally assign Ampere values to traces, and then simulate what temperature can be expected under certain ambient conditions.

Johannes Adam

Shaughnessy: So, you think there is kind of a disconnect? Perhaps the EEs don't realize that they don't have all the data?

Adam: Yes, and some don't even know the proper data. I guess that university education is rare as far as this subject is concerned. Typically, thermal resistance is all about thermal management. But thermal resistance depends on anything and everything. It depends on the boundary conditions, on the layer count, on the vicinity of a component or the trace you are interested in.

Shaughnessy: What do you think PCB designers and EEs should take away from this? What would be a couple of things that they should do or should think about, going forward?

Adam: They should understand that any component or trace has neighbors, heated or cold, and they are mounted in or on a very complex geometry of copper on a dielectric. This whole mixture makes it almost impossible to have a single value for a temperature, or a value you can print into some data sheet. And there is the issue of boundary conditions. Electrical engineers get some crude specification, for example, that it should operate in 85°C ambient. But there's a principle difference: whether the ambient 85°C is the ambient air, or whether it's a solid piece of metal which is 85°C, whether it is air around or some other fluid. Each case will give completely different results.

I'm always claiming that no application note can reflect your individual board and application. If we look at design rules, it's even worse. Typically, they consider a very low layer count where the temperature created by a small amount of heat is very high. But, if you deal with multilayers, much more heat will work fine, won't it? In fact, you are mostly on the safe side applying design rules, but you are at risk of overdesigning and losing money. Applying laws of physics to the specific board gives a much more precise answer.

The horizon should be wider, because the electrical engineer is not the only one who contributes to the design of a PCB or a device. There is the product management, purchase department, marketing and design studio who, for example, want to decide on time and cost. The EE has to be firm and has to have the proper arguments to back up his ideas. Regarding temperature one of the best methods is to show pictures from simulations comparing different options as early as possible. He can do that already in early stages of the design cycle, of course, with some assumptions on power dissipation, currents, and so on. Finally, he will take a thermal image of the real PCB, but by then has got to be manufactured, to have components and has to be operating, all of which costs at least time and shouldn't require much iterations.

Shaughnessy: It seems as if thermal has only become a "hot" topic, pardon the pun, in the past

10 years or so. It didn't even get much coverage at conferences. And suddenly the big EDA companies all offer some kind of thermal analysis capabilities.

Adam: It is a matter of convenience. My TRM software can do very high-resolution multilayer calculations going down to sub-millimeter resolution in XY space. That's better than many thermal imaging cameras and sometimes they look very realistic. And with simulation you can look inside the PCB and see what happens there. I believe that, with respect to big commercial EDA and FEM tools, my tool is easier to use for both designers and developers. You could also treat a PCB model from scratch without any external data. By the grace of its design it is also an educational tool. By the way, it also calculates the voltage drop, optionally taking into account the local trace temperature.

Shaughnessy: Let's say you have a board that you know is going to have some thermal challenges. Other than putting in heat sinks and fans, is it possible to just design it for better thermal operation?

Adam: This can become difficult, because heat is also looking for its path from a hot spot to some cold spot. If there is complete coverage of the board with heating components, it is more or less hot throughout the entire volume of the PCB. The only way to make it cooler is to make it bigger and to separate the heat sources. Nobody likes to do that. Best is to add heat sinks or apply fan cooling, but there will be little improvement by making layers thicker, using a high-cost better conducting board material or drilling thermal via.

On the other hand, if heating is by few components or traces, then you can try to move the components or to make the traces thicker or wider and go down with temperature then. The software can be used to optimize the design and cost. I also know cases where traces were overdesigned.

Shaughnessy: So, you think that it should be taken into account just like all the other data that the EEs look at before getting the parts for the board?

Adam: What I claim is that the electrical engineering team should be able to do the thermal simulation, because they have the design under control. They know about the schematics and what the board should do at the end. It's the better way of thermal design rather to hand over this part to a mechanical engineer. Of course, they should talk to each other about how to add a proper heat sink, fans, air grills, and so on. But the electrical engineer should be able to do in what he's interested in regarding board thermal issues himself.

I can tell you a funny story. As a consultant, I had to calculate the effect of a high current pulse in a 14-layer board, and it was more than 100 amps. I asked the client, "What the hell? A hundred amps? I can tell you it will melt." "Not necessarily, it's just for a second." The detailed calculation then showed that within that second, the inner trace heated up to maybe 150°C, but you saw only little temperature signature on the outer layers. This was because a second was too short in time for the heat to penetrate the PCB. An infrared camera could never have detected what happened inside the PCB.

> "An infrared camera could never have detected what happened inside the PCB."

Shaughnessy: Well, maybe this will help educate some people, and get EEs talking more about thermal.

Adam: It's also an educational mission, yes! As I told you earlier, temperature is not a topic in the curriculum of an electrical engineer. They just learn, "Don't get hotter than 100°C and this is the thermal resistance, more or less." All the rest is more or less unknown to them. The electrical engineer will have to educate himself, doing the simulation, experimenting with trace thickness, prepreg thickness, board thickness, trace structure, and component placement. There is rarely a linear relation between input values and result.

Shaughnessy: What's the name of your company?

Adam: The company is called ADAM Research. You can reach me directly on the support line. I also am affiliated with another company, EasyLogix, or Schindler & Schill in Germany, where we cooperate to bring thermal issues and other physics topics into their DFM tool called PCB-Investigator.

Shaughnessy: I'm sure EEs will want to learn more about your thermal software.

Adam: On my website adam-research.de, they can find a list of features, case studies and more information. Just drop a note and either myself or a distributor will get in personal contact.

Shaughnessy: Well, best of luck with your upcoming book, and your quest to educate our industry about thermal issues.

Adam: Thank you for the opportunity, Andy.

PCBDESIGN

Solar Cells With a Quantum Shift

When sunlight strikes a typical solar panel, it creates pairs of electrons and positively charged holes. Normally, an electric field is used to separate these charges and produce electrical power, but this approach requires charges that have high mobilities and lifetimes, which makes it hard to develop new photovoltaic materials.

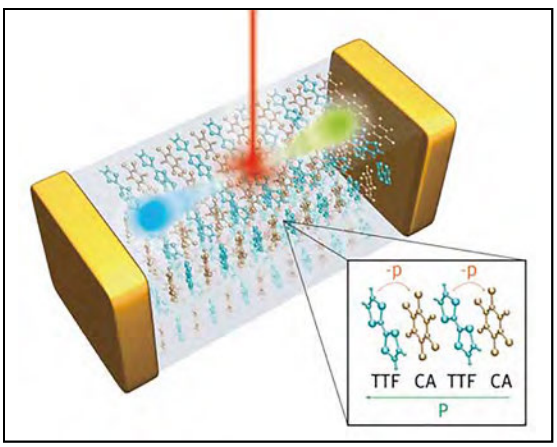
An alternative approach for extracting current from solar cells involves exploiting the symmetry of the repeating structural units that make up crystals.

For such semiconductors, light-induced transitions of charges to excited states become unbalanced, which creates a 'shift current' along a specific crystal direction. This shift current propagates rapidly and with less energy loss than a current generated by applying an electric field. But shift currents usually generate insufficient photovoltaic power for practical uses.

Now, Masao Nakamura from the RIKEN Center for Emergent Matter Science and colleagues have overcome this shortcoming by using ferroelectric organic molecules that spontaneously separate their positive and negative charges. Because ferroelectric materials naturally disrupt inversion symmetry, they have potentially large shift currents—particularly when charge separation occurs due to quantum-mechanical differences in the covalent bonds holding a crystal together.

The team investigated an organic ferroelectric with strong quantum polarization to explore its shift-current capabilities. Composed of alternately stacked tetrathiafulvalene (TTF) and p-chloranil (CA) aromatic rings, this complex undergoes instantaneous charge separation when cooled to around -200 degrees Celsius and is particularly sensitive to sunlight.

"Most ferroelectric materials need light with energy in the ultraviolet region to excite carriers over a large band gap," says Nakamura. "With TTF–CA, the band gap is narrow and responds to visible and infrared light, which is really important for applications like solar cells."



When the researchers measured the photovoltaic properties of the organic complex, they were taken aback by the amount of shift current generated—nearly ten times higher than comparable oxide ferroelectrics.

Thermal Management Update with Doug Brooks

by Andy Shaughnessy
I-CONNECT007

I had the opportunity to talk with our regular contributor Doug Brooks recently. He has been doing some research on temperature effects on PCB traces over the last few years, and I wanted to check the status of his latest thermal efforts. He discussed his work with Dr. Johannes Adam, why temperature charts based on a trace in isolation are inaccurate, and how the industry remained so wrong about PCB temperatures.

Andy Shaughnessy: You have done some work on thermal management lately. How did that project start?

Doug Brooks: I wrote an article in the mid-'90s on trace current/temperature effects, and I used two data sources: the then-current IPC data and some data I found in a 1968 Design News (DN) article. The DN temperatures were about 30–40% higher than the IPC temperatures and I wondered why. I began to suspect that it was because of the differences in the way the temperatures were measured or calculated. In looking for a way to confirm that hypothesis, I ran across an article about three years ago written by Dr. Johannes Adam, and I contacted him.

It turns out that Johannes had written a computer simulation program called TRM (Thermal Risk Management) [1] that was well suited for me to use to look at the data I had used in the article. He offered me a license for the software and we used TRM to simulate the IPC trace data in IPC-2152 [2] and the earlier data from DN. The simulations were very successful.

Shaughnessy: What did you find out?

Brooks: It turns out the DN data were unreliable!

Shaughnessy: And all of this took place over several years?

Brooks: No. That was just the beginning. It was so easy to simulate the IPC trace data that we began to simulate more realistic scenarios. The IPC data apply to a 6-inch trace in isolation. We began to look at what happens when we change things: change the length, change the pad sizes, add additional adjacent traces, add planes below the trace or on the other side of the board, more common layout conditions like those.


Doug Brooks

Then we wondered if we could simulate the temperature of a via, something that is difficult to do in practice and to our knowledge had not been done before. When that was successful, we looked at fusing temperatures. This is something I had written about earlier and had developed some basic rules for based on Onderdonk's Equation [3].

That opened some additional insights regarding whether traces heat uniformly or not (they don't). And that opened even more avenues for study. Soon, we had enough new information for a book [4].

Shaughnessy: After all that research, what thermal design issue really stands out in your mind?

Brooks: We discovered several very interesting insights, but by far the most dramatic was how wrong we had all been regarding via temperatures. Our industry-wide rule of thumb has been that a via's cross-sectional area should be the same as the trace cross-sectional area. If it is not, then multiple vias should be used [5]. It turns out that if the cross-sectional areas are equal, then the via is cooler than the trace. And if not, the via can take a lot more current than we had imagined. In most cases, if the trace has been sized correctly, only a single, small via is needed, almost regardless of current level.

Shaughnessy: How can you get by with a single, small via, and regardless of current? That seems counterintuitive.

Brooks: The reason is that the via length is very small compared to the trace. The trace acts as a heat sink for the via and conducts heat away from the via. We can easily push two to three times the expected current (and more) through a via, and the heat-sinking properties of the trace will keep the via temperature under control. These results are explained in detail in Chapter 7 of the book.

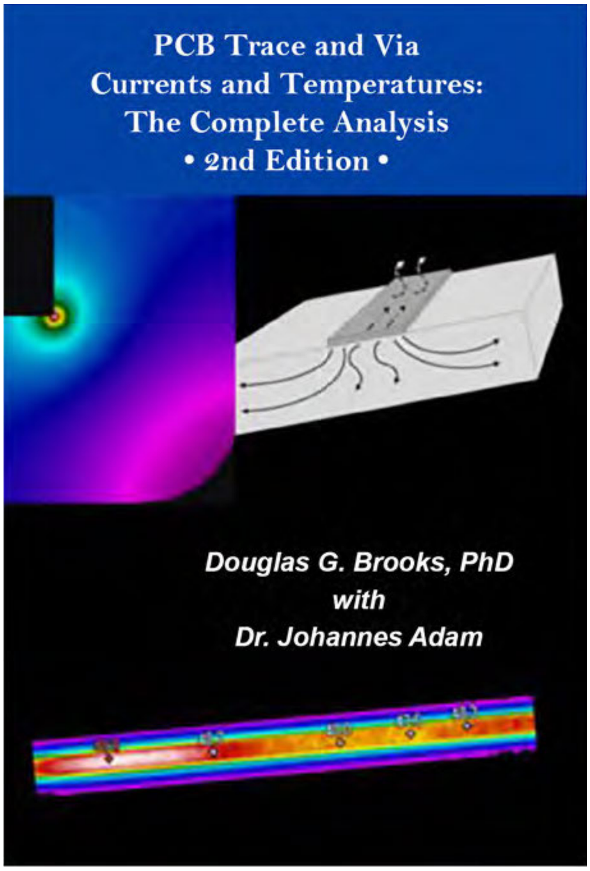


Figure 1: Doug Brooks recently collaborated with Johannes Adam on this book, and their research yielded some interesting results.

Shaughnessy: That result was based on simulations. Is that when the experimental work started?

Brooks: Yes. We knew that no one would accept those results without experimental verification. I went to Prototron Circuits in Redmond, Washington, and asked if they would provide some test boards for us. They were very generous in providing via test boards, and then several other boards for subsequent testing. The via experimental results confirmed the simulations, as described in Chapter 9 of the book. I could not have done everything that I did in the book without Prototron's contribution.

I was extremely fortunate that everyone asked for support was willing to help. Eight persons or companies are mentioned in the "Acknowledgement" section in the book, each of which provided invaluable services or advice. For example, C-Therm Technologies (Fredericton, New Brunswick) took board material samples and measured the thermal conductivity coefficients for us. The Jesse Garant Metrology Center (Windsor, Ontario) took X-rays of the via board for us. I am very grateful for it and humbled by it.

Shaughnessy: After all that, is there any general conclusion regarding trace currents and temperatures that you'd like to share?

Brooks: I'd like to highlight two. First, the IPC-2152 data are worst-case. By that, I mean that a single trace in isolation is a worst case. The data are correct, but we almost never have a trace in isolation. Anything we do in a real-world design sense lowers the temperature.


Design the Future You've Always Dreamed of
Don't let your 8-year-old self down. Create the future you always dreamed of, flying cars and all, with industry-defining OrCAD® PCB design technology. Your dreams can shape the future, OrCAD has the tools to help you make it possible.
Get The Free Trial ›
OrCAD™
CADENCE PCB SOLUTIONS
EMA Design Automation®

Second, all this stuff is too complicated to analyze with graphs or equations. Our industry has been here before. In the early '90s, many designers began to worry about and deal with controlled impedance traces. Back then, we used equations published in documents from IPC, Motorola, and National Semiconductor. Today, we now know that those equations are not adequate, and we need field-effect solutions for calculating trace impedance. The same thing is happening for trace current/temperature relationships. If we want to optimize our designs, we need to use thermal simulation tools.

> "If we want to optimize our designs, we need to use thermal simulation tools."

Shaughnessy: What's your next project regarding thermal management?

Brooks: Right after I gave a presentation on this topic in Tel Aviv last May, I was approached by Mentor's Nitin Bhagwath, who was really thinking outside the box. As we talked, we realized that there are two design paradigms board designers need to understand: The first is how to move the signal (current) from point A to point B. As rise times get faster, we have to start dealing with various signal integrity issues. As an industry, we already have a good understanding of how to do this. The second is how to move the power from one place to another. This is quite different from moving a signal. As the current increases, the I^2R drop increases, increasing the trace temperature (heating the trace). We need to manage this heat buildup and dissipate it some way. Currently, our primary answer is to increase the trace size, usually using the data in IPC 2152. But Nitin pointed out that there are other, perhaps many other, tricks we can use to manage the heat dissipation without impacting signal integrity. Many of these tricks employ the addition of non-current carrying areas of copper along the trace to increase the surface area, where we can, and ways to reduce the area in places where there is a high density of interconnects. Nitin will be presenting a paper on this with numerous examples at DesignCon in January. I have co-authored that paper with him, along with five other people [6]. I am excited about this new insight.

Shaughnessy: What's next for Doug Brooks? Didn't you say you were going to retire?

Brooks: I have been threatening retirement for about four years now. The time has finally come. My wife and I have just downsized into a condo near Seattle, and I am spending my time watching sports on a huge TV and playing with our seven grandchildren.

Shaughnessy: And writing articles for us! Thanks for talking with me, Doug.

Brooks: Thank you. PCBDESIGN

References

1. TRM (Thermal Risk Management) was originally conceived and designed to analyze temperatures across a circuit board, taking into consideration the complete trace layout with optional Joule heating as well as various components and their own contributions to heat generation. You can learn more about it by clicking here.

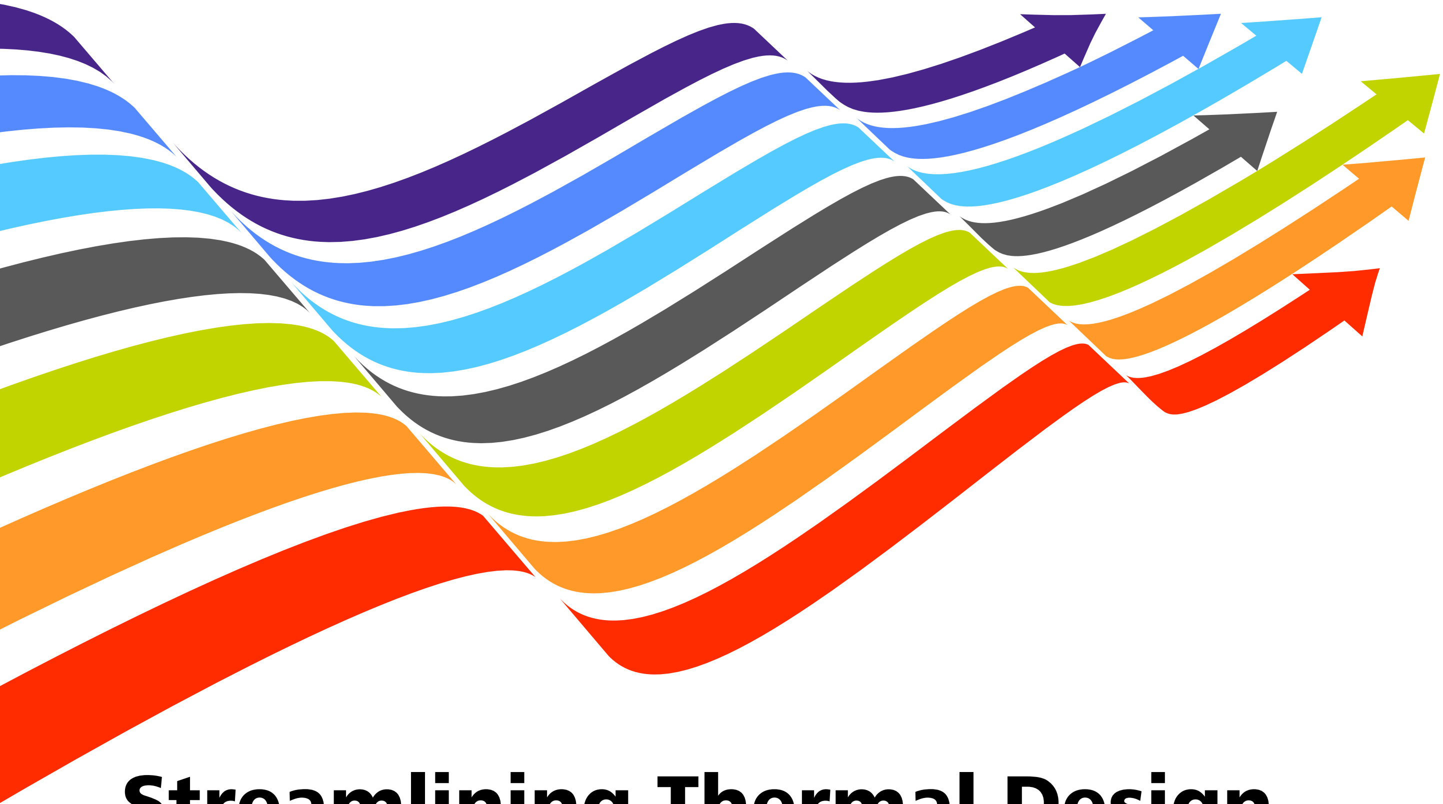
2. IPC-2152, "Standard for Determining Current Carrying Capacity in Printed Board Design," August 2009

3. This first article, "Fusing Current: When Traces Melt Without a Trace," was published in Printed Circuit Design, (Miller Freeman), in December 1998. It has been withdrawn from the UltraCAD website because of the new information we have discovered in this research.

4. Douglas G. Brooks and Johannes Adam, *PCB Trace and Via Temperatures: The Complete Analysis, 2nd Edition, 2017,* available at Amazon.com.

5. IPC-2152, op. cit., page 26.

6. The other co-authors are Robin Bornoff, Praveen Anmoula, and Pat Carrier (Mentor Graphics), Joseph Aday (Raytheon), and Robert Carter (Oak-Mitsui).



Streamlining Thermal Design of PCBs

by Dr. John Parry
MENTOR

When designing a PCB, thermal issues are often locked in at the point of selecting and laying out the chip package for the board. After that, only remedial actions are possible if the components are running too hot. Assumptions made about the uniformity of the airflow in these early design stages can mean a disaster for the commercial viability of a PCB if those assumptions are incorrect. A different approach is needed to improve reliability and to optimize board performance.

This approach starts with considering the overall flow environment, which is especially critical for good operation of air-cooled electronics. Start early, keep it simple, and focus on collaboration between mechanical and electrical design. If you are the mechanical engineer responsible for the thermal integrity of the product, you can provide as much useful feedback as possible to the electronic engineers about effects their choices will have on the thermal issues in their PCB design.

This collaboration entails advising with package selection and the positioning of components to best use system airflow for cooling. Layout and package selection usually are determined by electronic performance and cost; however, temperature and cooling can inevitably affect operation and cost so the consequences of design choices on thermal performance should be made as clear as possible, as early as possible.

Before Placement and Layout

First step is to optimize the enclosure-level airflow. Begin with a simple representation of the enclosure[1] to provide information about the airflow profile over the board. Smear the total board power over the total board surface to get a temperature map indicating any hot regions caused by badly distributed airflow. You can treat the board as a block with an isotropic thermal conductivity between 5 and 10. The results at this stage will not be affected by whichever value you choose in that range.

Do not use the board temperature to estimate component temperatures at this stage because components inject heat locally into the

board, which means that the heat flux density into the board below a component is higher than the board average. If the temperature of any section of the board is close to the maximum component case temperature, when you later refine the model to represent the component heat sources discretely, the component temperature limit will then likely be too high.

Make a Best-Guess Estimate for Component Power

Make a best-guess estimate of the individual power budgets for the main heat dissipating components that will be used in the design, and the approximate size of those packages. Then you can describe these packages as footprint heat sources in your simulation, which will smear the remainder of heat uniformly over the board surface.

The system architect will already have some idea of what key components will be required, where they will need to be positioned, and their size, etc. For example, some components may be used that were selected for another product or retained from the previous generation product.

Run Thermal Simulations as Early as Possible

If possible, include some form of 3D component model in the simulation before component selection is finalized so you can share the thermal results that can then be considered as part of the package-selection criteria. Some chips are available in more than one package style, and not all package styles perform equally well from a thermal point of view. Adding a heatsink later may be unnecessary if the design team has access to this information early.

Case temperature or junction temperature is the key measure to indicate whether the design is acceptable from a thermal perspective. At this stage, however, we are still only working with a rough estimate of individual component temperatures. The simplest 3D component model that can be used is a conducting block. (The 3D thermal simulation software FloTHERM includes material properties that provide case temperature prediction for different package styles.) For plastic components, a thermal conductivity of 5 to 10 is recommended [2], and 15 for any ceramic components. A thermal conductivity of 5 will give you a worst-case figure for case temperature.

If you have the 3D model, the effect of the component on the local airflow and any downstream components can be considered in the simulation. For example, large components can shield smaller, lower-profile components from cooling air. The wake formed behind a component is where the same air gets recirculated, so any components in that region are likely to be hot. Try to align any rectangular components so that their long side is parallel to the primary flow direction. This practice reduces the overall pressure drop because the flow encounters less of an obstruction and produces a smaller wake, which minimizes the effect on downstream components.

Share Thermal Results Among the PCB Design Team

Now, you can start to feed information about the PCB's performance to the design team. Although the simulation is relatively coarse at this stage, the airflow distribution over the board and the resulting board temperature map can illustrate what you have for available cooling air and what that may mean for component temperature.

When you share these initial results, emphasize that these nominal component case temperature values are subject to change because they are based on:

- an assumed layout
- rough power estimates
- uncertainty about package selection
- unknown layer stackup and copper distribution within the PCB, and
- preliminary heatsink size and design (if already known to be necessary)

This early model is useful for investigating the effect of component placement on the temperature of a component and its neighbors. Adjustments can be made easily and the model rerun in a matter of minutes. The results will give some indication as to which components, if any, might need some form of heatsink, which

can be investigated next. You can also use this information to prioritize which component thermal models need to be refined once more information is known from package selection.

Decide on Heatsinks Early

For any components that may be too hot, you can now investigate how effectively a heatsink lowers a component's temperature. If the airflow is mainly normal to one side of the package, a plate (or extruded) fin heatsink will likely work best. If not, then consider a pin-fin heatsink.

The thermal-simulation software includes pre-made parts that you can use to parametrically define heatsink geometry. Start by making the base of the heatsink the same size as the package and investigate different numbers of fins, fin height, and fin thickness. The aim is to see if the heatsink can be mounted on top of the package or if a larger heatsink might be needed, which will require board space for the mechanical attachment (Figure 1). The PCB design team needs this information as early as possible. Select an existing heatsink that provides adequate cooling performance or design a custom heatsink before the board can be routed because the mechanical attachment for the heatsink may affect component placement.

When including a heatsink, include the thermal resistance of the thermal interface material (TIM) between the package and the heatsink in your analyses. At this stage of modeling, a standard thermal pad with a thickness of around 0.2 mm and a thermal conductivity of 1.0 is a good conservative choice.

Refine the Thermal Models for Greater Accuracy

At this point, you can consider the most relevant thermal metrics to compare the thermal performance of candidate components. For components without a heatsink, the most relevant thermal metric is the junction-to-board resistance [3]. For components that are expected to have a heatsink, the junction-to-case resistance is the most relevant because the resistance usually is defined for the face that is in contact with the heatsink [4]. For TO-type packages, this face is soldered to the PCB. If both metrics are available, you can create a JEDEC Standard two-resistor model (Figure 2), and rerun the thermal model to get a first estimate of junction temperature [5].

The next level up for predictive accuracy is a Delphi model [6]. Delphi models are better for heatsink selection than two-resistor models because the top surface is subdivided into inner and outer regions that have different temperatures. Thus, you can use them to initially investigate the effect of heatsink-base thickness. However, for thermally critical packages that require a heatsink, use a detailed model.

You can do an Internet search for the components' datasheets to obtain more specifications. Or, if a thermal-simulation model is not already available, you can request one from your

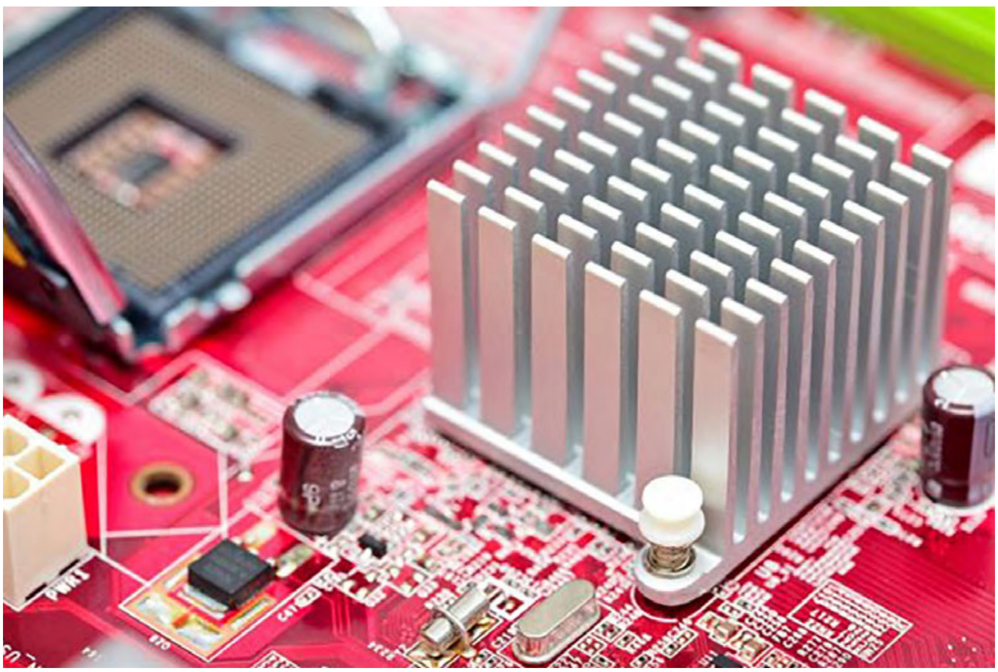


Figure 1: Heatsink with retaining pins that extends beyond the package body.


MULTI
LAYER
PCB

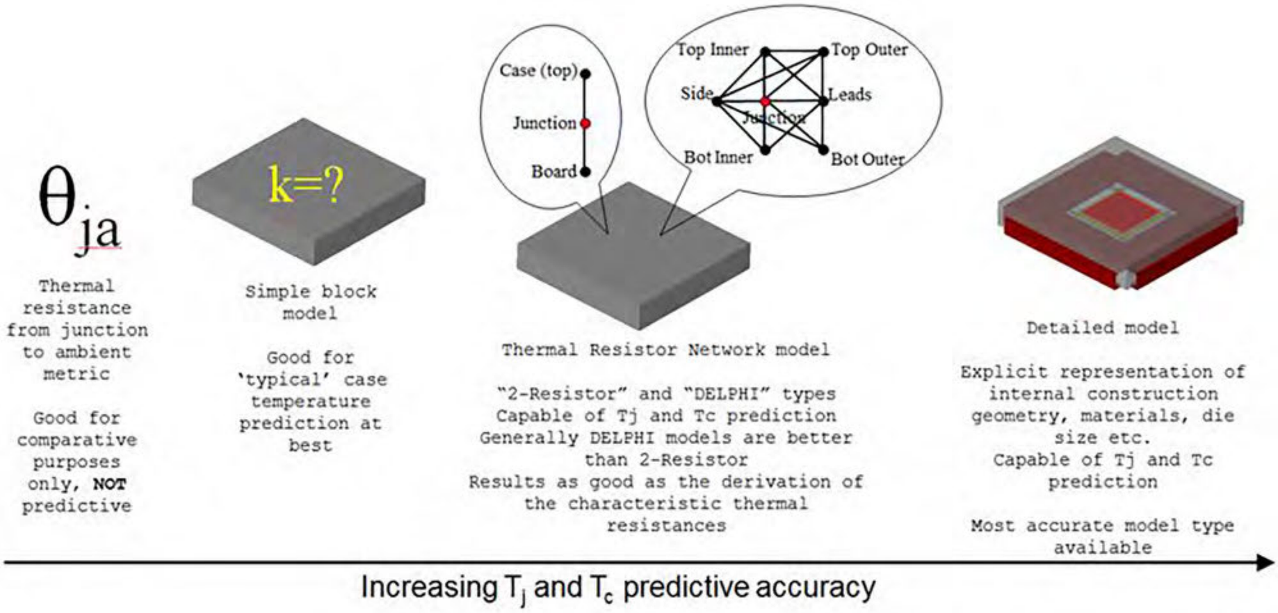


Figure 2: Hierarchy of package thermal information for design.

supplier. Sometimes they are provided under a non-disclosure agreement (NDA).

Add the Board Effects

Now that you have component footprints and heat-source estimates and modeled the components as 3D conducting blocks, the sensitivity of the results to the board thermal conductivity is also something that can be investigated. If you can, do this in parallel with refinements you make to the component models.

No single value exists for PCB thermal conductivity. PCBs are made of copper and dielectric material. The copper is about 1,000 times more thermally conductive so the dielectric material thermally insulates the layers from one another and insulates individual traces. Before the board has been tracked, you can use a simple isotropic conductivity value varied between 5 to 15 thermal conductivity to see how much of an effect the PCB's thermal data has on the simulation results. As your team gets into more detailed design, this thermal representation of the board will need to be refined.

Once the placement has been broadly defined, the next step for the PCB design team is schematic capture and electrical simulation (timing). The most useful thermally relevant information that can be obtained after the schematic capture, but before the board is routed, is the layer stackup of the board. Estimate how many signal, and power or ground layers, the board may have. Local electrical traces on the surface of the PCB spread heat away from the package interconnect (leads or solder balls), whereas buried power and ground planes increase the in-plane thermal conductivity at the macroscopic scale.

From a thermal perspective, the contribution of these copper-containing layers on the performance of the PCB is influenced by their thickness. The most common thicknesses are 0.5 or 1.0 ounces of copper, spread evenly over a one square foot area [7] (1 oz. is equal to 1.37 mils (thousands of an inch) or 0.0347 mm).

Once you have an estimate of the number of each type of layer in the PCB, you can upgrade the model of the PCB to include each of these layers individually. Before routing, an estimate needs to be made for the thickness and percentage copper coverage of each non-dielectric layer. One-ounce copper should be used for power and ground planes and 0.5 oz. copper for trace layers, with an assumed copper coverage of 80 and 20 percent, respectively.

The dielectric contributes little to the area-averaged conductivity, both in-plane and through-plane. The minimum thickness of the

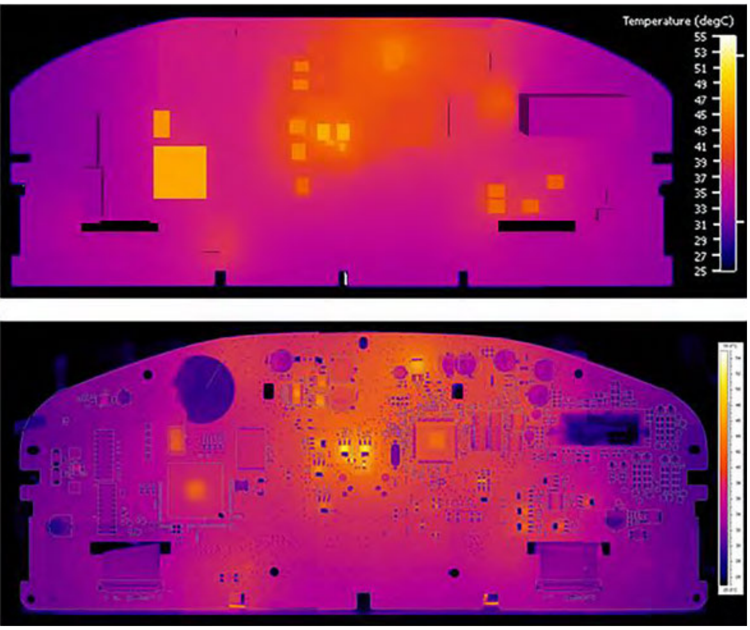


Figure 3: Thermal simulation modeling result (top), compared with a PCB measured using an IR camera (bottom).

dielectric layers depends on the thickness of copper either side to allow for the difference in the coefficient for thermal expansion [8], which gives the total thickness of the board.

For small, high-power, low-pin-count packages, the length scale of traces on the board are a similar order of magnitude to the package. Model these features in a similar level of geometric detail to the package before this information is available from the electronic design automation (EDA) system. For example, you can represent the copper pad that a TO package is soldered to, and traces local to the package, when the package is modeled in detail. The same applies to modeling any thermal vias below the pad used to conduct heat down to a buried ground plane. Once you get down to this level of detail, the thermal simulations are highly accurate (Figure 3).

Import Data from the Design System

Importing component placement data from the EDA system ensures that placement within the thermal tool is correct, and you should reimport it whenever the layout is changed. Detailed PCB modeling involves importing the stackup, the routing of trace layers from the EDA system, the distribution of vias, and the copper shapes on power and ground planes.

When thermal design from a mechanical perspective is done in parallel with the thermal design from an electrical perspective, the thermal design aspects of the board can close faster, more reliably, and with a better outcome than if it is undertaken in only one flow. Key to this is a shared understanding of what work can and should be done in each flow and how they can interact with, and affect, each other. **PCBDESIGN**

References

1. "12 Key Considerations in Enclosure Thermal Design… A High-Level 'How To' Guide," Mentor white paper.

2. Tony Kordyban (1994) "Estimating the Influence of PCB and Component Thermal Conductivity on Component Temperatures in Natural Convection," Third International FloTHERM User Conference, September 1994, Guildford UK.

3. Integrated Circuit Thermal Test Method Environmental Conditions—Junction-to-Board. JEDEC JESD51-8, October 1999.

4. Transient Dual Interface Test Method for the Measurement of the Thermal Resistance Junction to Case of Semiconductor Devices with Heat Flow through a Single Path. JEDEC Standard JESD51-14, November 2010.

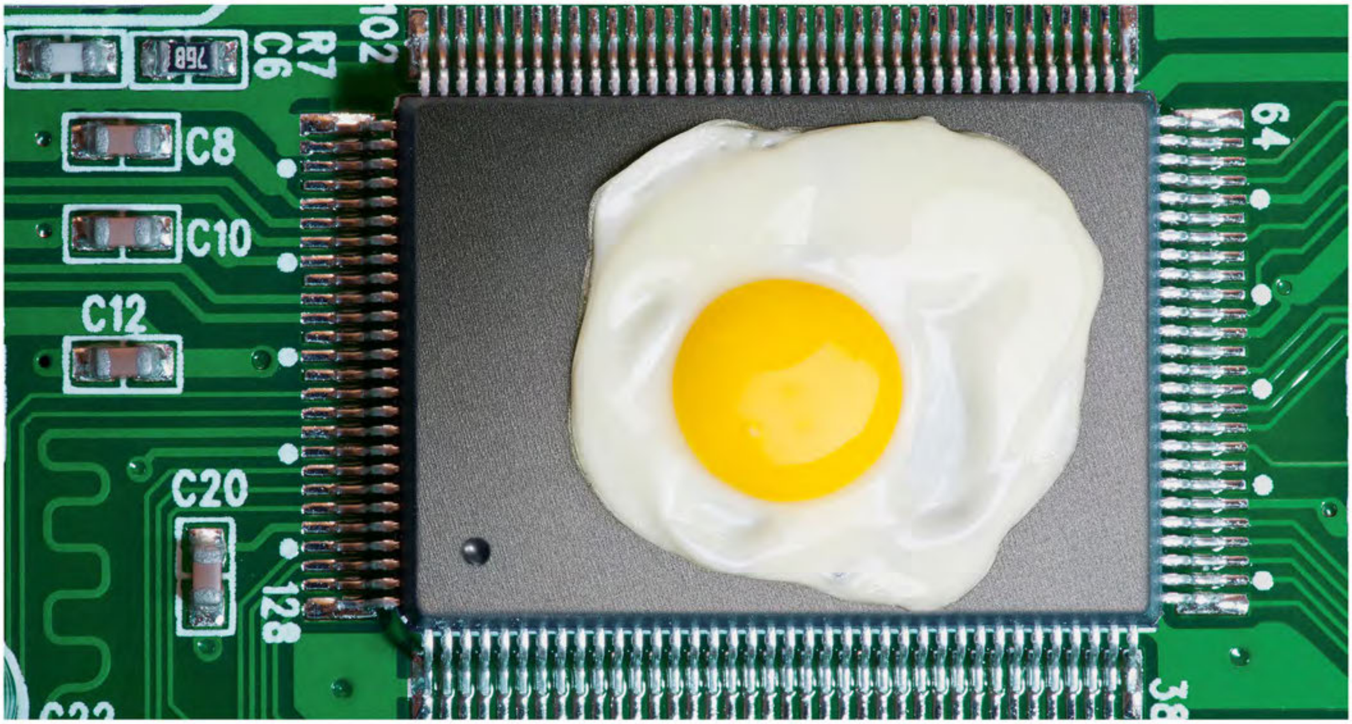
5. Two-Resistor Compact Thermal Model Guideline. JEDEC Guideline JESD15-3, July 2008.

6. DELPHI Compact Thermal Model Guideline. JEDEC Guideline JESD15-4, October 2008.

7. Definition of PCB

7. Copper thickness FAQ

John Parry is the electronics industry manager, mechanical analysis, at Mentor. He holds a PhD in chemical engineering.



THAT'S HOT:
Ventec's Goodwin on Thermal Management

by Patty Goldman
I-CONNECT007

IPC's fall committee meetings were held in conjunction with SMTA International, as has been the case for several years now. I sat in on some subcommittee meetings, including one on laminates, where I met up with Ventec COO Mark Goodwin for a discussion on thermal management from a laminate supplier's perspective.

Patty Goldman: Good to see you, Mark. How about we talk about thermal management?

Mark Goodwin: Yes, let's. From our perspective, there are really two material sets that we consider for thermal management. There's the IMS (insulated metal substrate) materials, the metal-back materials, which started off as an LED lighting story. But it's much, much more than an LED lighting story now, particularly with the development in e-vehicles and hybrid vehicles. There's a lot of e-powertrain, so heavy current moving power around in cars.

The second strand is a new direction for us and the technology is a thermally conductive thin core material, thermally conductive prepreg for building multilayers with thermal conductivity or using those layers and those prepregs in standard multilayer constructions, but as a hybrid, so only putting the thermal management where you need to put it.

Goldman: So, is it sort of self-cooling, then?

Goodwin: The materials have a thermal capacity. They can take heat away from embedded components. For example, people using coin technology (embedding a metal inside the PCB under the high-power components so that heat can be dissipated) may not need to use that, or maybe they can use it in conjunction with thermally conductive prepregs, thermally conductive cores. As a company, we're publishing an eBook through I-Connect007 on thermal management and the materials for thermal management. It's predominantly an IMS story, but there is some discussion in there about the new materials, thin cores and prepregs, and our thinking about thermal management. So that's thermal resistance and thermal impedance compared with thermal conductivity. There's

a lot of confusion, particularly at the design stage, I believe.

Goldman: Thermal management seems to encompass a rather broad area. You must separate the parts.

Goodwin: When we talk about thermal management, we're talking about getting the heat out of components, be that an LED chip, or be that an embedded 'whatever' in a multilayer circuit board.

Goldman: Does this facilitate embedding?

Goodwin: Yes, the new materials do.

Goldman: This is interesting. I hadn't really thought about embedded components before and I realize now it's a big deal if they're heat-producing; that's an issue until you've got a thermally conductive material.

Mark Goodwin, Ventec.

Goodwin: Everything is running at increasingly higher frequencies as well, and high frequencies generate more heat and you've got to do something with it. We've got copper in there, which is good at getting rid of heat or transferring heat, but then you've got to get the heat from the component through the substrate to a heat sink or something to transfer the heat out of the build-up. So, it's a development. If you remember, the first IMS materials produced were made using a glass-reinforced prepreg. They were just heavily ceramically loaded prepregs. Well, we've just taken that technology and made cores from those and improved the flow characteristics. When we first did this, there was just so much filler in these things—tons of pressure and very high heat rise was the only way to get these things to flow. We've improved that now. We can encapsulate up to three-ounce copper with these materials now. We've got some live projects on this, but again, a lot of it is driven by electronic vehicles, brake energy regeneration, and there are other applications as well.

Goldman: Would you say automotive is the main driver right now?

Goodwin: Oh, we're spending a lot of time at automotive shows and automotive electronic shows at the moment. That's the driver for us. I think it'll be bigger than LED and the reason is LED boards are generally very small. A powertrain board for an e-vehicle could easily be a three-square-foot panel or more. That's one circuit board instead of several hundred on that panel as with LED. And there are lots of cars, of course. Every announcement you see these days is either a government policy decision on e-vehicles or hybrid vehicles, or one of the big automotive companies saying that they're going to produce an electric or a hybrid variant of every model they produce, or that by 2021 they're going to be all e-vehicles.

Goldman: Yes, I read that.

Goodwin: This is a market with double-digit compound average growth rates every year, so it's got to be interesting.

Goldman: How does the push towards autonomous driving play into this?

Goodwin: I don't know that it works with thermal management, but what I do think is that it's where the core material and prepreg comes in because, now, the thermal issue is dealt with a thermal management PCB. But eventually, people are going to want to start putting logic onto those boards as well and do it all on one board, and then we're going to have to interface them with something. Over here, there's a heat dissipation issue. Over there, it's standard material, or even an SI material, so this is now one of our reasons for developing the thin cores and prepregs. They're compatible with all our other materials. If you want to start putting these things together you can start doing that.

One of the other things that's going to happen in the automotive business is in the use of aluminum and copper: They use copper in some applications, but copper is heavy and expensive compared to aluminum. Aluminum is going to become a structural part of the design as well. We're seeing a lot of thick aluminum on these projects. Then you might want to mount your logic board on top of that directly without pillars and screws. It's a single process, so all this sort of thing is coming. We see designs already.

Goldman: The automotive industry is always very conscious of weight and always looking to remove as much as possible. They'll fight to reduce ounces.

Goodwin: Yes. One mil of copper and one mil of aluminum is a very different story. And when they're putting all these batteries in, they've got to get the weight out somewhere else.

Goldman: All the batteries with the lead in them! One day, people will wake up to that. So, automotive seems to be driving the industry. It used to be military that seemed to be driving the industry.

Goodwin: Right, we see less drivers from military. We see drivers from commercial aerospace more than military aerospace now. In fact, in some ways, military is stuck 20 years in the past. They're extremely conservative with materials, extremely conservative with change.

Goldman: And I think automotive was like that, but they're really being pushed.

Goodwin: They've been pushed in two directions: cost and technology. And to be honest, unless you're talking about real high-end radar or something like that, military applications are generally not high technology in material terms, in the same way as some of the new stuff we're doing in cars. But cars encompass everything, don't they? They've already built autonomous driving, communication, information, etc. and collision avoidance with microwave materials, all this kind of stuff. It's all in there, so it's a very, very interesting business.

Goldman: All the industries meld together in your automobile, don't they? Europe is the hotbed for all of that.

Goodwin: Certainly, in terms of OEMs. But manufacturing globally, Europe and Japan I think drove the LED front-lighting in cars. I mean, it's only been what, 18 months or two years since the United States approved high-powered LED lighting for front-lighting on cars? There's catch-up there, and the U.S. can be a huge market as well, but where do these products get made? Probably, frankly speaking, in Asia, because the volume is so high. There's a lot of machining. That's the thing with metal-back boards. It's not particularly high technology in terms of circuit board manufacturing, but a lot of mechanical processing, drilling, routing, punching, with aluminum, which is not an easy job.

Goldman: Of course, it's always the prototypes here and the volume there.

Goodwin: Yeah. Prototypes, pre-production, and ramp-up series, but the volume will be in Asia. We do a lot of OEM work all over the world and we see the boards. Where we do the sales activity is often different to where we invoice or deliver the goods, because increasingly they're unrelated. We have to do the sales activity wherever we do it and the boards are made wherever. There is no U.S. market, European market, China market. There's just a global market now for these things.

Goldman: What do you see coming down the pike? What do you see next?

Goodwin: Well, we have still a lot of things to do with thermal management, so this is just starting in my opinion, but we're still working away on our signal integrity materials as well. But thermal management is going to be a big, big driver for us. And if you look at the profile of it, the real change and the real growth is going to be in the next 10–15 years. It's not something that's got a two-year life cycle.

Goldman: I read your recent interview with Barry about where Ventec was headed, which was interesting because you guys seem to be really moving.

Goodwin: We are moving! We haven't got ambitions to be the biggest in the world, but we've got ambitions to be in all the right places at a level that makes sense, which isn't always the biggest. Because you can't react quickly enough to market change. See, that's the thing I see in automotive now. Automotive never used to change anything, and even the automotive market now is starting to push change.

Goldman: That's what I meant by being pushed. Things are changing so rapidly and they can't have their three-year design cycles because they'll be left in the dust.

Goodwin: Exactly.

Goldman: It will be interesting to see how that's going to play out. Anything else we should mention?

Goodwin: We touched on collision avoidance. Some of the big players in the market have got some patents expiring in the next few months, so I think we'll start to see some "me too" products produced by companies such as Ventec who have the right cost position, have the right attitude towards customer service, and a strong supply-chain that ensures required lead-times. We all know where people have a position in the market that allows them to get away with long lead-times and high prices. Well, some of the guys producing in Asia have got their eyes on some of that business. It'll be a long haul because a lot of that's specified at an OEM level, but all the laminators in Asia now have OEM teams. We do, so it'll start to happen.

Goldman: Things don't stand still. They don't stay the same. They change very rapidly. In fact, more and more rapidly…

Goodwin: Absolutely.

Goldman: We'll have to talk every now and then about the automotive industry. I find that very intriguing. Because you're right, they had long design cycles and were very conservative in what they did.

Goodwin: When I was with Isola 15 years ago, I was supplying the materials to the automotive guys and you couldn't talk about changing anything, and now you don't always get the change, but you can have the discussion now and it's not just shut down immediately. It's very different.

Goldman: Mark, thanks for speaking with me today.

Goodwin: My pleasure. **PCBDESIGN**

Simulating High-Temperature Superconductors

An international group of physicists, among them UvA-Institute of Physics researcher Philippe Corboz, has now made important progress in simulating unconventional materials which are superconducting at higher temperatures.

It was proposed that a very simple model of interacting electrons moving on a two-dimensional lattice—called the Hubbard model—could capture the relevant physics of high temperature superconductivity.

By combining the latest numerical methods in large-scale simulations, the researchers have found a definite answer, namely a so-called "stripe" state in which the electron density is not uniform, but modulated in space. Stripes had also been found in previous studies, but not with the required level of precision to distinguish them from solutions with uniform density.

by Anaya Vardya and Dave Lackey
AMERICAN STANDARD CIRCUITS

A designer may choose from among many options to help dissipate the heat generated by various PCB components. This article primarily focuses on options that utilize metal to attach directly to the PCB during the manufacturing process to help with the heat dissipation process. Note that some people refer to these types of PCBs as thermal clad or metal clad PCBs (MCPCBs) while we call them IMPCBs.

When metal is attached to the PCB, the bonding material can either be thermally conductive but electrically isolative (insulated metal PCBs or metal core PCBs) or, in the case of RF/microwave circuits, the bonding material may be both electrically and thermally conductive. RF designers usually have the bonding material thermally and electrically conductive because they are using this not only as a heatsink but also as part of the ground layer. The design considerations are quite different for these different applications.

This article will focus on the IMPCB design considerations and things you should be discussing with your PCB supplier to ensure that you get a good quality PCB. It is not possible to go in great detail so we always recommend collaborating with your PCB supplier about your specific design and how to end up with the most cost-effective solutions.

Some of the applications of IMPCBs are:

- **Power Conversion:** An IMPCB offers a variety of thermal performances, is compatible with mechanical fasteners, and is highly reliable
- **LEDs:** Using IMPCBs assures the lowest possible operating temperatures for maximum brightness, color and life
- **Motor Drives:** Dielectric choices for IMPCBs provide the electrical isolation needed to meet operating parameters and safety agency test requirements
- **Solid State Relays:** An IMPCB offers a very thermally efficient and mechanically robust substrate

PCB Thermal Management

Optimizing PCB Thermal Performance with Altium Designer

- Common passive cooling techniques and their benefits
- The advantages and disadvantages of thermal relief pads vs. pastes
- Requirements for implementing thermal relief pads into PCB designs
- How to properly create thermal relief pads using CAD system rules

Check out our latest blogs about PCB Heat Dissipation Techniques and Thermal Pads and Paste for PCB Thermal Management for helpful tips!

For more blogs and whitepapers, please visit:

resources.altium.com

Altium® | ALTIUM DESIGNER

- **Automotive:** The automotive industry uses IMPCBs as they need long term reliability under high operating temperatures coupled with their requirement of effective space utilization

Single-Sided IMPCBs

In its simplest form, an IMPCB is a piece of copper foil that is bonded to a thermally conductive dielectric and a metal substrate (Figure 1). Typically, a PCB supplier can buy the copper foil laminated to the base metal from a number of different laminate manufacturers. Some of the key design factors to consider for single-sided IMPCBs are listed here.

Copper thickness

Typical thicknesses range from 1 oz. to 6 oz. The most commonly used ones are 1 and 2 oz. The thicker the copper, the more expensive the PCB.

Thermally conductive prepreg

This is one of the most important elements of the construction and what typically differentiates the various suppliers. This is the substance that both electrically isolates the copper circuitry from the main metal and helps with rapid transfer of heat between the two. It ensures that heat generated by the components is dispersed to the base metal (heatsink) as quickly as possible.

The prepreg is typically an organic resin with ceramic fillers to increase thermal conductivity. The filler type, size, shape, and percentage are some of the factors that determine thermal conductivity performance. The usual ceramic fillers are Al_2O_3, AlN, BN, etc. The performance of the various prepregs is measured by the thermal conductivity (watts per meter Kelvin or W/mK) and thermal impedance (Kelvin, meter squared per watt or Km^2/W). The higher the thermal conductivity the better the heat transfer; the lower the thermal impedance the better the heat transfer.

It is important to understand that the better the heat transfer associated with the prepreg, the greater the cost. It is therefore critical not to over-design. To put this in perspective, the thermal conductivity of FR-4 is approximately 0.4 W/mK, whereas the thermally conductive prepregs that are available on the market today range from 1 W/mK to 7W/mK. Apart from thermal conductivity, the thickness of the dielectric can be critical. Typically, the thickness of the dielectric ranges between two and six mils.

Base metal

Aluminum is the most commonly used base metal. The two most common types are 5052H32 and 6061T6. The former is typically less expensive and a lot more available than 6061T6. The thickness of the aluminum typically ranges between 40 and 120 mils, with 40 mils and 60 mils the most common thicknesses available. There are also cases where copper is used as a base metal. This is a significantly costlier solution. A brief comparison of the various base metals is illustrated in Table 1.

Panel utilization

The IMPCB laminate materials are significantly more expensive than FR-4 materials. It is therefore extremely important to understand how your board/array designs utilize the production panel. You should work with your PCB supplier to help you with this. The most popular size for a working panel on these materials tends to be 18" x 24".

Solder mask

Single-sided IMPCB designs are used for LED lights. Many of these applications require white

Figure 1: Single-sided IMPCB.

Metal Base Material	Thermal Conductivity (W/mK)	Thermal Expansion (ppm / K)	Comments
Aluminum 5052 H32	138	25	Al-Mg-Cr alloy: Best for bending, mechanical forming, most popular choice, low cost
Aluminum 6061 T6	167	25	Al-Mg-Si-Cu alloy: Best for CNC machining and V-cut scoring, medium cost
Copper C110	386	17	Pure Cu: Low CTE, high thermal conductivity, high cost

Table 1: Properties of various base metals.

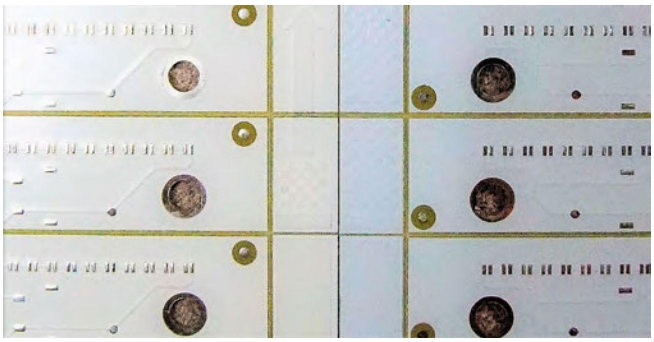


Figure 2: Different colors on two different types of solder mask.

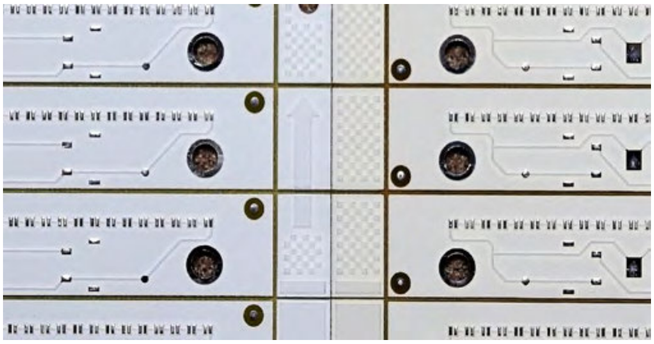


Figure 3: Solder mask "browning" with multiple reflow cycles.

solder mask, which is important to address. All white solder masks are not made equal. A lot of LED customers are looking for consistency in the color of their white solder mask. The marketplace today has a number of different solder masks that are marketed as LED solder masks. The issue is that they visually look different when you put them side by side.

Some solder masks have a bluish hue to them and others have a yellowish hue (Figure 2). Also, the colors look different with one coat versus two coats of solder mask, so this is another decision that will need to be made. In addition, there can be an interaction between the surface finish, the solder mask and subsequent heat processing steps in the assembly process. Some solder masks tend to change colors more than others with additional heat. Boards with lead-free HASL tend to become yellower the more heat they are subjected to. It is generally best to permit only one pass through the lead-free HASL process (therefore, no re-work at this station).

Figure 3 illustrates the same solder mask after lead-free HASL versus a board that has been through two assembly reflow cycles. Boards with ENIG after the solder mask process may turn slightly pink with subsequent reflow. This is caused by the formation of a complex between gold residues in the final rinse of the ENIG process with the titanium pigment in the solder mask which can shade the solder mask pink during the high temperature assembly process. It is thus important for the PCB

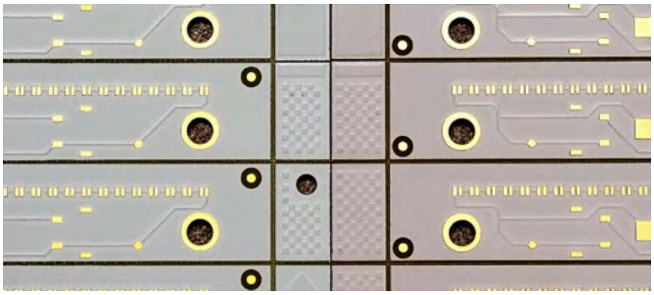


Figure 4: "Pinking" solder mask on ENIG board with multiple reflow cycles.

supplier to manage the rinses on the ENIG bath very carefully (Figure 4).

Machining/Fabrication

Scoring is the most common process used for square or rectangular shapes. The advantage of scoring is that it assists in maximizing material utilization since zero spacing is needed between parts to score them. In contrast, routing is the most expensive process since it is slower and requires spacing between parts and will likely reduce the material utilization. Make sure that your PCB fabricator has a scoring system that is specifically designed for scoring aluminum. The scoring machine should be equipped with a lubrication system. It is recommended to use diamond-coated scoring blades and router bits when dealing with aluminum base metal.

Double-Sided/Multilayer IMPCB

The PCB supplier manufactures a double-sided or multilayer IMPCB and then bonds it utilizing a thermally conductive prepreg to metal (Figure 5). The bonding process is done in the same multi-layer press that is used to manufacture a multi-layer PCB.

Many design factors and considerations that were discussed in the single-sided IMPCB section apply here, plus there are some additional considerations to think about as listed here:

Copper weights on all the layers

The thicker the copper, the more expensive it is. Also remember that the outer two layers will receive additional copper since the vias will need to be plated. Lines and spaces should follow the design guidelines of the PCB shop based on the copper weights of each of the layers.

Double-sided/multilayer construction

It is important to decide whether you can use FR-4 for your multilayer construction or if you require thermally conductive prepregs and cores. If you need thermally conductive cores and prepregs, there are a number of options available—but core thicknesses are limited so it is best to work with a PCB supplier or a laminate supplier on constructions that make sense. The prepregs tend to be low flow; it is important

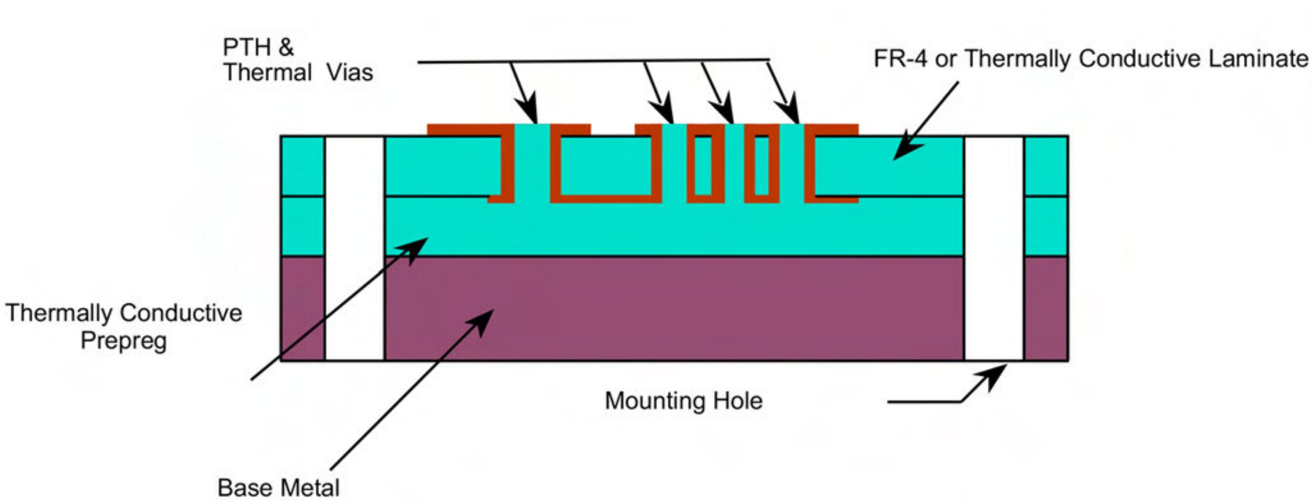
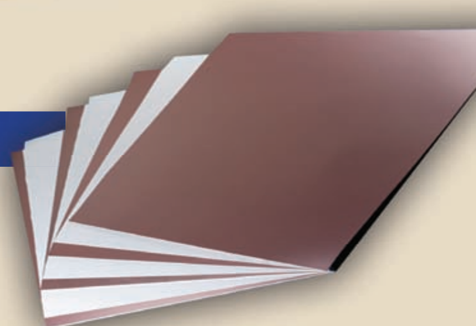


Figure 5: Schematic of a double-sided IMPCB.

Don't Let Your Temperatures Rise

Let ROGERS' Thermal Management Solutions *Keep You Cool*



COOLSPAN® TECA Film • 92ML™ StaCool™ Laminates • ML Series™ Laminates & Prepregs

Heat can be damaging, especially when it is not managed. That's why Rogers Corporation invested so much time and energy into creating an array of material-based thermal management solutions to keep heat rise to a minimum in printed circuits. From automotive circuits to LED modules to power supplies, ML Series laminates and prepregs effectively conduct heat away from the source, while COOLSPAN thermally & electrically conductive adhesive (TECA) materials enhance the thermal management of new and existing designs. And for that extra cooling edge, 92ML StaCool laminates feature a thermally conductive metal bottom plate to enhance the heat dissipation.

Product	Thickness	Thermal Conductivity (Z-Axis), W/mK	Thermal Impedance, C-cm2/W	Tg, C	CTE (Z-Axis), ppm/C		Dk, 1MHz	Df, 1MHz	Breakdown Voltage, kVAC	Flammability
					<Tg	>Tg				
92ML	8mils	2.0	0.52	160	22	175	5.2	0.013	>50	HF V-0

Don't let your temperatures rise. Use Rogers' thermal management solutions.

ROGERS CORPORATION

Advanced Connectivity Solutions

www.rogerscorp.com/acs

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

to work with a PCB supplier that fully understands the lamination press cycles to be used with these materials.

Thermally conductive prepreg

Choose the prepreg to bond the PCB (double-sided or multilayer) to the metal based on thermal conductivity required and thickness of the copper circuitry. From a PCB manufacturing perspective, a number of different factors need to be accounted for in the process of bonding the PCB to base metal:

- Ensure there is no delamination between the PCB and the metal. There are design factors that can impact this and process conditions in the lamination process
- Have a method to control the flow of prepreg through the plated through-holes to the top side; have a method to remove any flow that ended up on the top surface of the PCB
- There are number of mismatched CTEs in this package. It is important to try to balance the copper in the construction as much as possible from a PCB perspective and have a press cycle that helps to minimize warpage

Base metal

Aluminum is the most common metal in use, however there are many applications that will also use copper as the base metal. In general, when aluminum is the preferred metal, the 6061T6 alloy is the best choice for this type of construction.

Metal Core Boards

Conceptually, a metal core board is exactly what it sounds like: the metal is in the middle of the PCB, sandwiched between layers on both sides. Metal core PCBs usually have blind via layers located on both sides of the metal core substrate. There are also plated through-holes (PTH) going through the entire package. From a PCB perspective, it is important to isolate the metal from the through-hole, otherwise the board would short out completely. To accomplish this, one has to start by drilling the metal core approximately 40–50 mils larger than the plated through-holes, slots or cutouts. These then need to be filled with a non-conductive epoxy filler and then pressed. After pressing the metal core will need to have the filler compound removed from the surface and then prepared for lamination with the innerlayer cores. After lamination, the PTH is drilled and processed in the normal manufacturing process.

While thermal management is a factor in these PCBs, another reason that metal core boards are used is to help with vibration reduction so that components don't fall off the PCBs in high-vibration applications.

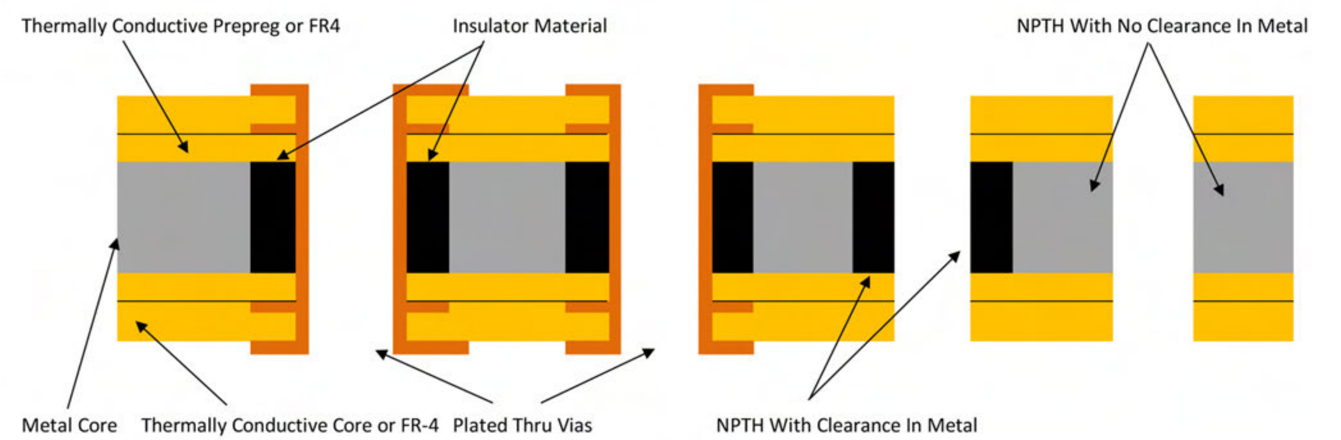


Figure 6: Schematic of a multilayer metal core board.

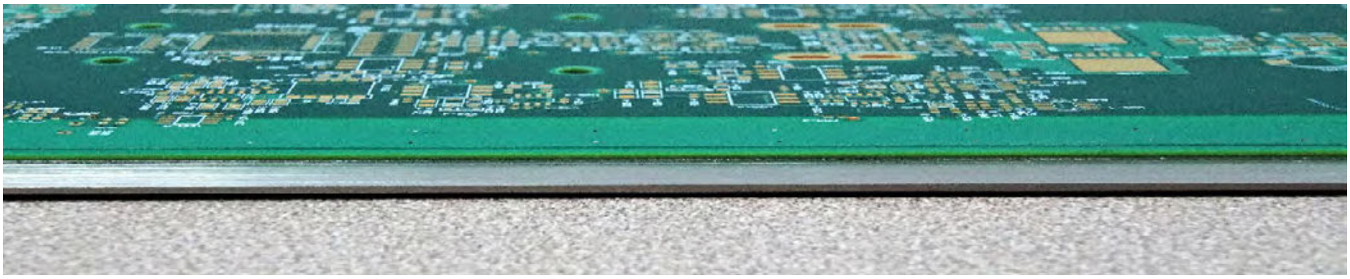


Figure 7: Milling on a metal core PCB.

Design factors to be considered:

Core materials/prepregs

The core materials and prepregs could be any material PCB raw material. We have seen metal core boards manufactured with polyimide, FR-4 or highly thermally conductive materials. It really depends on the application and what is desired from an electrical and/or thermal management perspective.

Metal core materials

Typically, we see either copper C110 or aluminum 6061T6 used. In the case of metal core boards, we see almost as many designs with copper as we do with aluminum.

Drilling the metal core

The metal core boards are drilled oversized with the entire drill pattern associated with the PCB, both the plated through-holes that go from the top layer to the bottom layer and the non-plated holes going from the top layer to bottom layer. Occasionally, mounting or grounding holes have no clearance in the metal core.

Insulator/filler

The insulator/filler material acts to insulate the PTH from the metal core so the entire PCB does not short out. The filler is initially in powder form and then applied to the surface and holes and put in a multilayer lamination press. This is a critical process; there can be no voids in the filler or when the PTHs are drilled, as chemistry can leach back to the metal core and cause a short. The core is then sanded to remove the excess filler on the surface. The filler material is a ceramic, epoxy combination.

Stack-up

The stack-up should be symmetrical in terms of number of layers on top of the metal core and number of layers below the metal core. Also, copper weight symmetry is preferred between all the layers just as one would want on any multilayer PCB. Lack of symmetry can lead to excessive warpage issues. In general, the typical IPC warpage specifications do not apply to these types of PCBs.

Milling

Most of the metal core boards have some kind of milling associated with the PCB that results in exposing the metal core layer (Figure 7).

Surface finish on exposed metal core

We recommend putting a surface finish on the exposed metal. Typically for aluminum we recommend chromate conversion and for copper we recommend a minimum of 50 micro-inches of electroplated nickel.

Selecting an IMPCB Laminate Supplier

There are many suppliers, with a majority based in China. In selecting a supplier, several factors should be considered:

- It is important to realize that when one looks at data sheets there are many different ways that laminate suppliers test these materials for thermal conductivity and there are no IPC standards for this. You really need to understand the test methods utilized since all materials advertised as 2W/mK may not result in similar performance

- Some suppliers 100% hipot test their materials, some will test them if you request it at an extra cost, and many don't have the ability to test their material
- Many suppliers will supply single-sided IMPCB materials. If you are thinking of utilizing multilayer constructions, only a handful of suppliers will supply you cores with high thermally conductive prepreg and thermally conductive prepreg that can be used to bond the multilayer PCBs to the metal
- Supply chain ease is important. Lead times vary between the different suppliers. Do you want to source in the U.S. and China or just China? If you are interested in doing both regions, choose a supplier that has a support structure in both regions of the world
- Another important factor could be R&D that is being done by the laminate supplier. As an example, a couple of laminate suppliers have developed special laminate materials where the aluminum can be bent and formed without compromising the copper circuitry or the dielectric layer

Selecting a PCB Supplier

It is important to partner with an appropriate PCB supplier for your IMPCB needs. Some criteria to consider are:

- Level of experience with manufacturing IMPCB materials
- A supplier that is educated and has manufactured a variety of different types of IMPCBs and materials from different suppliers
- A supplier that has a good relationship with the material suppliers in the space
- A supplier that is willing to work in partnership with you keeping an open mind when it comes to your ultimate needs
- If UL is important to your application, ensure that the supplier has the requisite UL paperwork
- Finally, consider a supplier that has the process controls and disciplines in place

Conclusion

We have focused this article around the designs we see most commonly. We have dealt in a number of situations where people's requirements for a variety of reasons deviate dramatically from these norms. Many other options are possible, so the key is to work closely with your PCB fabricator.

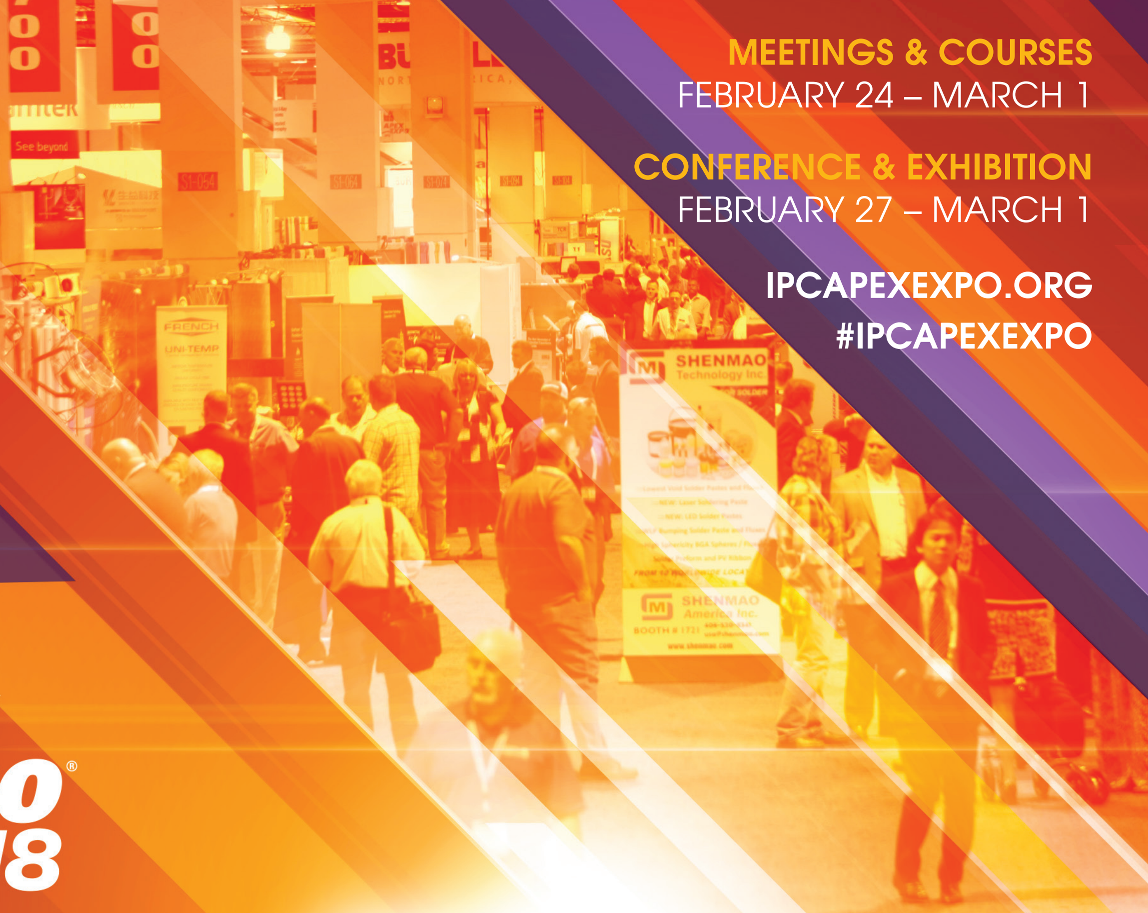
As we conclude this discussion of thermal management materials and substrates, it is important to remember that everything in technology is a trade-off. When choosing between the different thermal management substrates it is vitally important to define the end-product in terms of the actual conditions it will experience during its lifetime and deciding which options will be best suited for that product. Finally, selecting the right thermal solution is a like a three-legged stool, with the legs being: the end product, the actual design, and the manufacturing process—none of which can be ignored to create reliable thermally sound PCBs.

Acknowledgements

The authors would like to thank Jim Zeman, director of quality, and John Bushie, director of technology, for their work on the white solder mask evaluation. Our discussions with some of our suppliers—Electra, Taiyo, Aismalibar and Vente—helped to shape our thoughts. Finally, we would like to acknowledge all the IMS customers and designers we have worked with over the years who have challenged us in many ways and helped us grow in this sector. **PCBDESIGN**

Anaya Vardya is president and CEO of American Standard Circuits.

Dave Lackey is vice president of business development at American Standard Circuits.



PCB007 Highlights

New I-Connect007 Team Members Tour American Standard Circuits

Recently, Anaya Vardya, CEO of American Standard Circuits, invited two of I-Connect007's newest team members, IT coordinator Jonathan Zinski and Editor Kiersten Rohde, to tour his facility in West Chicago, Illinois. Happy Holden, resident PCB expert, also joined the newbies on their field trip to ASC. In the following articles, Jonathan and Kiersten describe their experience touring ASC.

It's Only Common Sense: A Frustrated PCB Customer Vents

I recently received an e-mail from a good friend at a top mil/aero company. Here's just one snippet: "Why do I have to call and ask, 'Where is my stuff?' If it were just one board shop, I would say good riddance and move on. But it is almost universal among our supplier base that late deliveries only get discovered when we ask…"

Rex Rozario's Next Big Thing, Part 1

Barry Matties joins Rex Rozario at Lympstone Manor, one of Rex's investments outside of the electronics industry. The property, once an old country house in the Exeter countryside, has been transformed into a splendid hotel and restaurant.

RTW SMTAI: NCAB's Perspectives on HDI and Miniaturization

John Piccirilli of NCAB Group discusses the trends in HDI and miniaturization he's seeing around the world. He also talks about other interesting technologies such as metal-backed PCBs and stretchable materials.

All Flex Hires Experienced CFO and Sales Manager

All Flex has hired two key individuals to bolster the management and expansion of its high-performing business areas.

Walt Custer's Annual Update from productronica 2017

Industry consultant Walt Custer of Custer Consulting sat down with me at productronica 2017 to once again inform our readers with his yearly update and perspective on what's happening in the industry. And guess what? It's good news!

The Right Approach: Steve's Particular Set of Skills (to become a World-Class Quality Manager)

Being a quality professional today is nothing like it was 20 or 25 years ago; on a personal level, I can attest to this fact. It is no longer adequate to appoint a quality manager simply based on a person's command of acceptance criteria and industry specifications; in the 21st century, a truly hybrid executive is needed.

Planning a PCB: Signal Integrity and Controlled Impedance Considerations

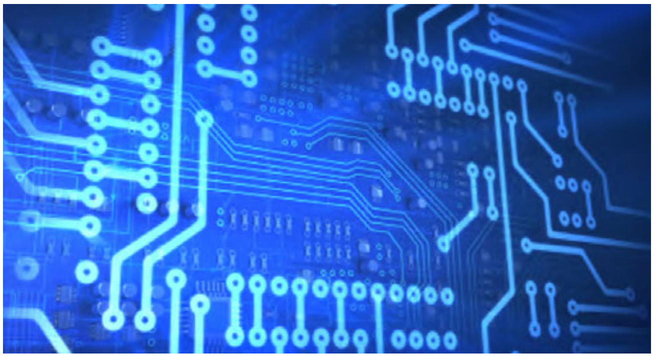
Knowledge and experience are the two key elements when planning a PCB. Today's PCB designers must have far more knowledge and understanding of the PCB production process than in the past. This is especially important when they plan and how they plan the stackup, via span, routing and power distribution.

PCB/PCBA M&A Top 10 FAQs

We frequently talk with owners about the possible sale of their businesses. Here are the top 10 questions asked by PCB/PCBA shop owners about the process.

Ladle on Manufacturing: Fabricating for Signal Integrity

Signal integrity! In a world which is increasingly high-speed and digital, the chemical-dependent and mainly analogue-controlled world of PCB manufacturing is not always a comfortable partner.



Signal Flight Time Variance in Multilayer PCBs

by Barry Olney
IN-CIRCUIT DESIGN PTY LTD / AUSTRALIA

Energy in the form of electromagnetic waves is transferred through the dielectric materials, of the multilayer PCB substrate, through vibrations of electric and magnetic fields. A transmission line does not carry the digital signal itself but rather, guides electromagnetic energy from one point to another. Signals travel at the same speed, given the same medium. However, the microstrip (outer layer) traces are embedded in a mélange of dielectric material, solder mask (if required) and air. This lowers the effective dielectric constant and increases the propagation speed compared to that of stripline (inner layer) traces. This month, I will look at the disparity in signal propagation in multilayer PCBs.

When we watch waves rolling onto the shore, we tend to think that the water is moving towards us, but that's not actually the case. The individual particles that make up the waves move up and down perpendicularly to the direction of the wave. But they do not move significantly themselves until they break, and hit the shore, which disperses the energy. The particles take part in the wave by bumping into one another and transferring energy. The waves travel as a transverse wave which is characterized by particle motion that is perpendicular to the wave energy.

Surfers hang out well offshore, sitting on their boards behind the break, patiently watching the horizon for the next set and the magic wave. As the wave (swell) passes, they bob up and down vertically. The waves come in sets because the amplitude of the waves is modulated by another longer wave. The first wave in a group is small, the next one is bigger and so on until the largest wave appears in the middle of the group. Then they get smaller again. Way back in my surfing days, we used to say that the third wave was always the biggest, but others say it is the seventh; it is all relative to the number of waves in the set.

Figure 1: Propagation of a wave around a stadium. Source: Science Learning Hub.

If you have ever experienced a "wave" at a football stadium (Figure 1), you will be amazed at the actual speed at which the wave travels. It takes less than 60 seconds for the wave to complete a circuit of a typical stadium but nobody moves (apart from standing up/down). Here the medium, in which the wave propagates, is people. If the perimeter of the outer edge of the stadium is 1km, then the wave propagates at ~60km/hr—and nobody has to wear a safety belt.

Many stadiums have a members-only section, to which are admitted only an elite group of persons whose apathy appears

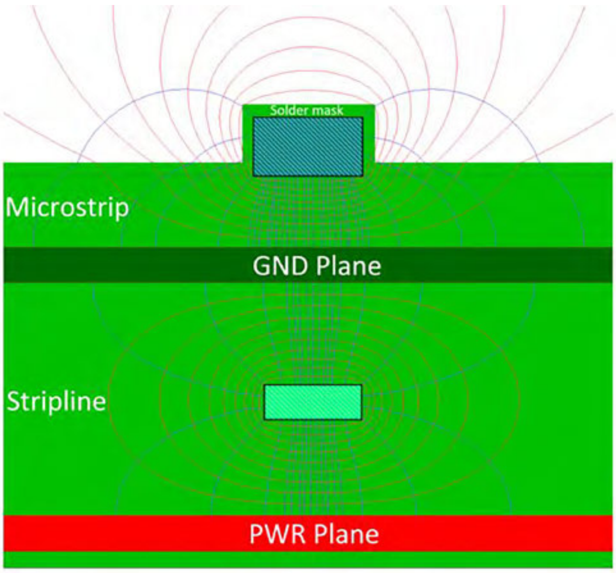


Figure 2: Microstrip and stripline electromagnetic fields, simulated in HyperLynx.

to be higher than average. They typically do not rise to participate in the wave. Nevertheless, the wave seems to jump across the impenetrable barrier continuing the circuit, just as electromagnetic energy is coupled between traces and components of a PCB without physically touching.

Similarly, the speed of a computer does not depend intrinsically on the speed of electrons, but rather on the speed of energy transfer between electronic components. The actual velocity of electrons through a conductor is very slow (~10 mm per second), however the "knock on" effect is very fast as it follows the electromagnetic field. The energy propagates as an electromagnetic wave. And, the speed of this wave varies depending on the layer, in the multilayer substrate, and the surrounding dielectric materials.

A stripline is any trace sandwiched between reference planes on both sides, as in Figure 2. The electric fields (blue) of a stripline are totally contained between the two solid planes, so the speed of propagation for signals guided by the trace is entirely determined by the dielectric constant of the surrounding materials.

On the other hand, a microstrip is any trace fabricated on the outer layers of a PCB. A microstrip has dielectric material and a plane on one side and air on the other. An embedded microstrip is similar but is covered in a conformal coating such as solder mask or another dielectric material. In this case, the effective dielectric constant should be calculated by a field solver and represents a combination of the surrounding materials. There are also other variants of microstrip and stripline, such as build-up microstrip and dual asymmetric stripline.

The electric fields surrounding the microstrip exist partially within the dielectric material(s) and partially within the surrounding air. Since air has a dielectric constant of one, which is always lower than that of FR-4 (typically 4.3), mixing a little air into the equation will speed up the signal propagation. Even if the trace widths are adjusted on each layer, so as the impedance is identical, the propagation speed of microstrip is always faster than stripline, typ-

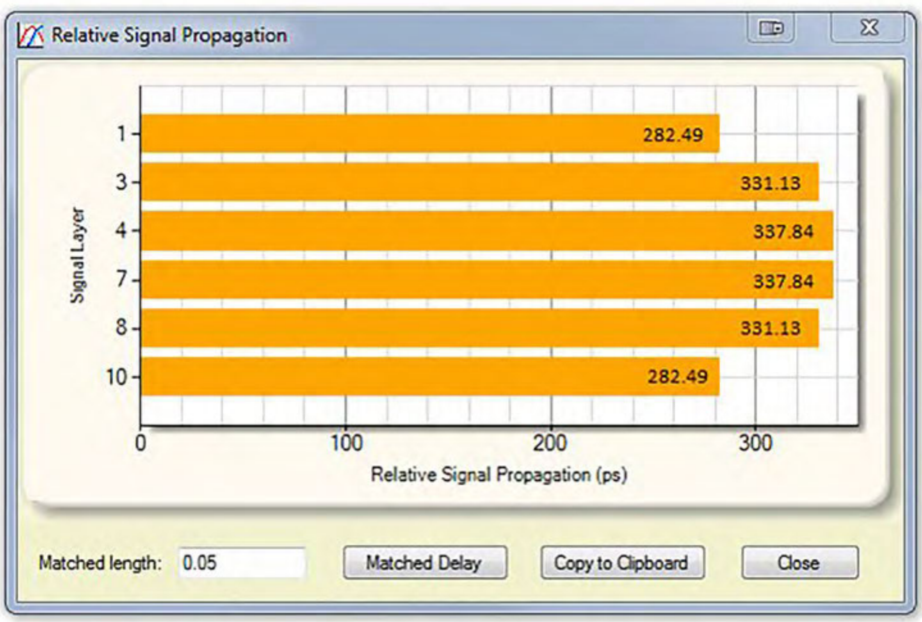


Figure 3: Relative signal propagation of microstrip and stripline, simulated in iCD Design Integrity.

ically by 13-17%. The speed of propagation of digital signals is independent of trace geometry and impedance.

If you are aware of this issue, then the trace delays (as shown in Figure 3) can be matched to compensate for the varying flight time, so that at the nominal temperature, all signals running on either microstrip or stripline will arrive at the receiver simultaneously. However, this does not solve the problem of temperature skew between microstrip and stripline layers. Figure 4 shows the impact of temperature on Rogers' high-speed RO4003 laminate. The black curve is measured while the red line is the calculated -40 to 90°C temperature range. Rogers' materials are specifically developed for high-speed designs. However, over this temperature range, the dielectric constant (Dk) of other materials can vary as much a 10% depending on the stability of the material. And since the propagation velocity is proportional to the square root of the dielectric constant, it will vary about ~5% over the same temperature range. Dielectric loss (Df) also tends to increase as temperature increases.

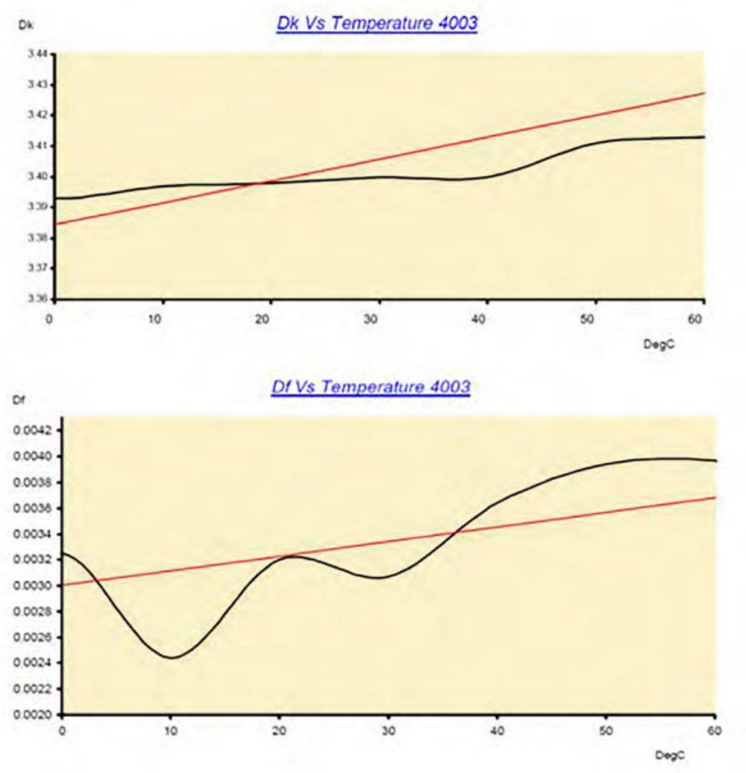


Figure 4: Dielectric constant vs. temperature (top), and dielectric loss vs. temperature (bottom). Source: Rogers Corporation

In contrast, microstrips react differently as the surrounding medium is a mixture of dielectric (prepreg and solder mask) and air. When the temperature changes, the portion of the field travelling in the FR-4 is affected, but the portion of the field in air is not. This is because the dielectric constant of air is steady and does not vary with temperature. As a result, the temperature coefficient for microstrip traces is less severe than for stripline. This implies that even if the delay of microstrip and stripline traces are perfectly matched, they will still vary independently with temperature. This skew could be as much as +/- 20ps, which eats away a large portion of the board level timing budget of high-speed signals.

The obvious way around this issue is to either route entirely on microstrip or entirely on stripline. Since microstrip traces tend to radiate emissions, and there is limited space on the outer layers due to component footprints, one would be well advised to only route critical signals on inner stripline layers. Microstrip and stripline traces should not be mixed, except for the unavoidable short fanouts from the IC lands to interconnecting vias.

Points to Remember:

- A transmission line does not carry the digital signal itself, but rather guides electromagnetic energy from one point to another.
- Signals travel at the same speed, given the same medium.
- Digital signals travel as a transverse wave which is characterized by particle motion that is perpendicular to the wave energy.
- The speed of a computer does not depend intrinsically on the speed of electrons, but rather on the speed of energy transfer between electronic components.
- The speed of this wave varies depending on the layer, in the multilayer substrate, and the surrounding dielectric materials.

- A stripline is any trace sandwiched between reference planes on both sides.
- A microstrip has dielectric material and a plane on one side and air on the other. An embedded microstrip is similar but is covered in a conformal coating such as solder mask or another dielectric material.
- Even if the trace widths are adjusted on each layer, so as the impedance is identical, the propagation speed of microstrip is always faster than stripline, typically by 13-17%.
- The speed of propagation of digital signals is independent of trace geometry and impedance.
- Trace delays can be matched so that either microstrip or stripline will arrive at the receiver simultaneously.
- The dielectric constant of materials can vary as much a 10%, depending on the stability of the material, creating temperature dependent skew.
- The temperature coefficient for microstrip traces is less severe than for stripline because it partially travels in air which does not vary with temperature.
- Even if the delay of microstrip and stripline traces are perfectly matched, they will still vary independently with temperature.
- For stripline, materials with similar dielectric constants should be chosen. This will ensure that the signal flight times are identical.
- One would be well advised to only route critical signals on inner stripline layers.

References

1. Barry Olney's Beyond Design columns: New Functionality Improves Designer's Productivity, Faster than a Speeding Bullet.
2. Waves as Energy Transfer, Science Learning Hub.
3. IEEE 802.3ap Backplane Ethernet, Joel Goergen, Manny Hernandez.
4. High-Speed Signal Propagation, Howard Johnson.

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

Revolutionizing Electronics Using Kirigami

A research team in the Department of Electrical and Electronic Information Engineering and the Electronics-Inspired Interdisciplinary Research Institute (EIIRIS) at Toyohashi University of Technology has developed an ultrastretchable bioprobe using Kirigami designs.

The Kirigami-based bioprobe enables one to follow the shape of spherical and large deformable biological samples such as heart and brain tissues. The results of their research will be published in Advanced Healthcare Materials (Ultrastretchable Kirigami Bioprobes).

High stretchability and deformability are promising properties to increase the applications of flexible film electronics including sensors, actuators, and energy harvesters. However, conventional elastomer-based stretchable devices require a large strain-force to stretch it, that arises from an intrinsic material property. This makes it impossible to follow the deformation of soft biological tissues, thereby preventing natural deformation and growth.

"To realize the ultrastretchable bioprobe with low strain-force characteristic, we used a Kirigami design as the device pattern. The stretching mechanism is based on an out-of-plane bending of the thin film rather than stretching of the material; therefore, the strain-stress characteristic is extremely low compared to that of elastomer-based stretchable devices," explains the first author of the article, Ph.D. candidate Yusuke Morikawa.

MilAero007 Highlights

All About Flex: More on UAVs and Flexible Circuits
The use of drones or unmanned aircraft vehicles (UAVs) is growing at a nearly exponential rate. This includes drones used by the government, private companies and hobbyists. If you have $100, you can find a long list of drones for purchase.

Nine Dot Connects: Good Design Instruction is a True Value-Add
Nine Dot Connects has certainly blazed an interesting trail. The company started out as an Altium reseller, but in less than a decade, Nine Dot Connects has also become a design service bureau and a provider of PCB design instruction, training, and consulting services.

Coast to Coast Names Ed Porter President and CEO
Coast to Coast Circuits has named Ed Porter president and CEO. Porter has served in a variety of managerial and executive leadership roles over the last 20 years within leading organizations in the printed circuit board industry, including TTM Technologies and Sanmina SCI.

Catching up with...Brigitflex
Today, Brigitflex is building unique custom-made boards for companies all over the world, from large defense and aerospace companies to small incubator companies inventing new products. They have become well-known as the shop to go to when nobody else can solve your problems.

American Standard Circuits Appoints Tony Monaco to Sales Team
American Standard Circuits has recently added industry veteran Tony Monaco to the company's sales team.

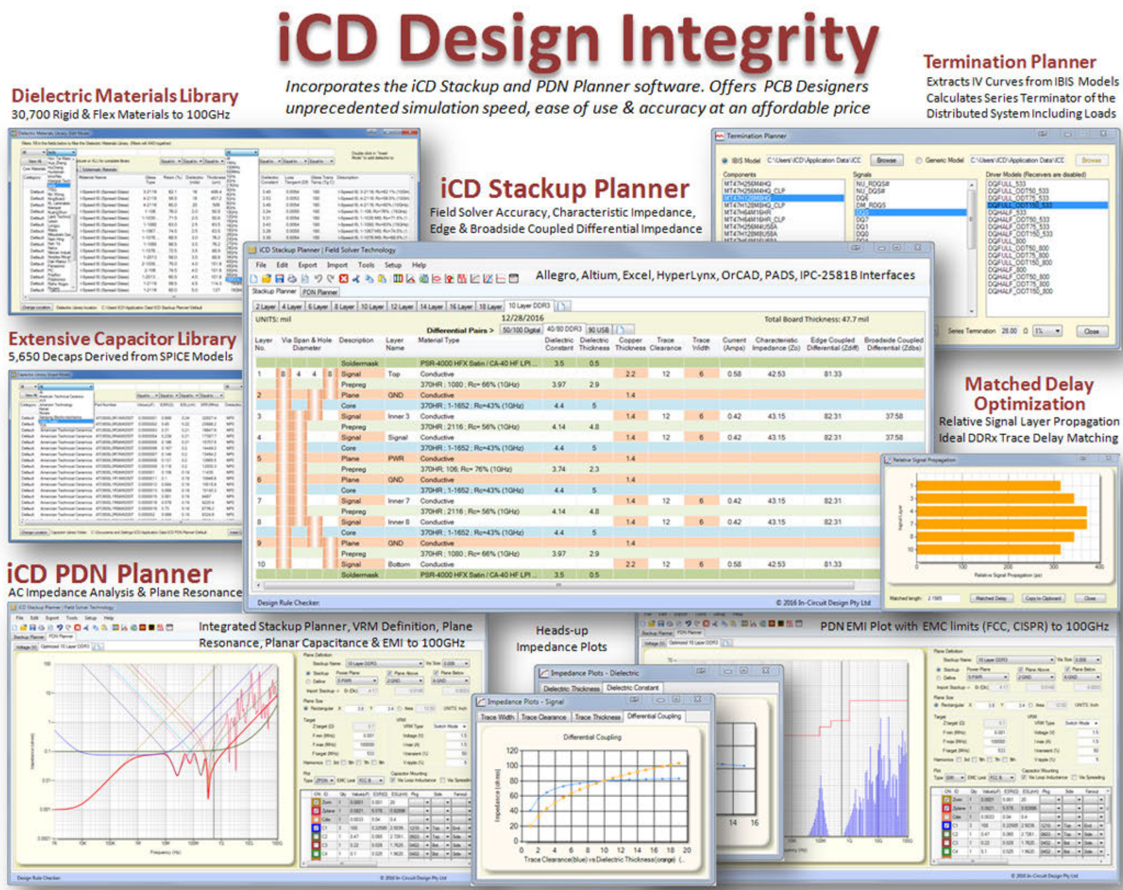
NPL and SMART Group to Hold Design, Process & Reliability Seminar
The National Physical Laboratory and SMART Group will hold a seminar November 9, 2017, that will showcase the latest research and results from NPL projects looking at solder joint and contamination failure, coating thickness measurement, solder joint reliability, and high temperature reliability for alternative solders and substrates materials.

Leveraging Industry Shows to Create a Presence
At the recent SMTA International show in Rosemont, Illinois, I had an opportunity to meet with Mark Osborn, president and owner of Colonial Circuits, based in Fredericksburg, Virginia. Colonial Circuits is a supplier of PCBs, mainly for the defense industry.

'Lighten Up' - Deep Space Communications via Faraway Photons
A spacecraft destined to explore a unique asteroid will also test new communication hardware that uses lasers instead of radio waves.

New RoboBee Flies, Dives, Swims and Explodes out the of Water
New floating devices allow this multipurpose air-water microrobot to stabilize on the water's surface before an internal combustion system ignites to propel it back into the air. This latest-generation RoboBee, which is 1,000 times lighter than any previous aerial-to-aquatic robot, could be used for numerous applications.

Marine Communication Systems Market Witnessing Surge in Advanced Technology
Technavio market research analysts forecast the global marine communication systems market to grow at a CAGR of more than 8% during the forecast period, according to their latest report.



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

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

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

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

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

In-Circuit Design Pty Ltd

www.icd.com.au

ARTICLE

Figure 1: The oral session opened with a presentation by Jose Sartori.

IEEE's Romanian SIITME Show a Success

by Joe Fjelstad
VERDANT ELECTRONICS

The 23rd annual IEEE International Symposium for Design and Technology in Electronic Packaging (SIITME) was held in Constanta, Romania, October 26-29, 2017. It attracted more than 190 participants from Romania, and 13 other counties. While there might be some who question the suitability of Romania as a venue for such a conference, the country has a long technological history and has produced some top innovators and pioneers in aviation including Aurel Vlaicu, Traian Vuia, Henri Coanda (who prototyped an early jet aircraft in 1910), and great engineers like Anghel Saligny, and George Constantinescu, a pioneering scientist credited with developing the theory of sonics.

The convention was created to bring together members of the electronic packaging community from near and far to discuss technology developments and needs. Ongoing efforts by the organizers have been intensified over the last decade in their effort to bring together academia and industry to share knowledge and experiences. The SIITME conference creates a shared environment where students, teachers, and senior researchers present their latest works and interact with industry representatives who highlight the state-of-the-art in the industry.

As Professor Dan Pitica of the Technical University of Cluj Napoca, Romania, observed,

Figure 2: The Industrial Workshop was moderated by Cosmin Moisa and Stefan Techau. Yours truly is shown discussing my disruptive solderless Occam technology.

Figure 3: Occam was one topic that generated a good deal of discussion.

Figure 4: Gaudentiu Varzaru presenting CETTI's first trials in Occam technology.

"SIITME is more than a just a conference. The Association for Promoting Electronics Technology, APTE, succeeded in bringing together eight top international professionals to give keynote talks including Klaus-Jurgen Wolter (Technische Universitat Dresden, Germany), Gheorghe Brezeanu (Politehnica University of Bucharest, Romania), Stefan Techau (ASM Assembly Systems GmbH & Co KG), Frederic Kratz (National Institut of Applied Sciences INSA, Centre Val de Loire, France), Joseph Fjelstad (Verdant Electronics, USA), Jose Sartori (OSRAM Opto Semiconductors), Etienne Sicard (INSA Toulouse, France), Radu Sporea (Advanced Technology Institute, University of Surrey, Guildford, UK), and many specialists from Continental Automotive, Tecnometal (Italy), ES Srl Electronic Solution (Italy), SEM Communication & GEST Labs SRL (Italy), Comtest, Miele Tehnica, NTT Data."

To focus on the future, the first day of the event was dedicated to the Industrial Workshop "Advanced Interconnection and Disruptive Technologies, Debate for the Future Sustainable Electronics Packaging." This event was moderated by industry representatives Cosmin Moisa, Continental Automotive Romania; Stefan Techau; Paul Svasta, University Politehnica of Bucharest; and myself. Topics included the reliability challenge for the electronic products, front-end assembly capability, managing PCB supplier challenges, and CETTI's early steps in exploring Occam technology were also discussed.

The final two days were divided between keynote speeches, industrial sessions, plenary oral and poster sessions. The works judged

Figure 5: The exhibition took place in parallel with the conference.

Figure 6: Discussions took place during the poster sessions.

Figure 7: Award winners were selected by organizers Professor Paul Svasta (far right) and Associate Professor Mihaela Hnatiuc (second from right). Alexandra Fodor (Technical University of Cluj Napoca), winner of Best Poster Award, stands between Iulian Nastac (far left) and Andrei Drumea (middle, Politehnica University of Bucharest), who also received recognition for their excellent poster efforts.

best of show were awarded with honorary certificates. The objective of creating a forum for close industry-university links is to harmonize the training of the human resource with an eye to meeting the near and future requirements of the industry. Industry participants accounted for nearly 40% of attendees, a number of whom presented papers of their scientific or industrial research work, including Continental Automotive Romania, BEIA Consult and Syswin Solutions. The latter two companies are members of the Electronic Innovative Cluster, ELINCLUS, who, together with other companies, presented at the exhibition section. They are prime examples of the continuous effort of the Cluster's management entity, APTE, to help to promote the interests of their members, specifically, facilitating contacts, promoting research results, products and services on an international level. The local organizing committee, headed by Associate Professor Mihaela Hnatiuc from Constanta Maritime University, has made a significant effort to help in the success of the conference, including by facilitating a visit through the institution's laboratories.

During the closing ceremony, the general chair, Professor Paul Svasta, emphasized that the SIITME 2017 event was the largest event yet and attracted the most diverse group to date. In closing, Professor Daniela Tarniceriu, Dean of the Faculty of Electronics, Telecommunication and Information Technology from Gheorghe Asachi Technical University of Iasi, and Marian Petrescu, Continental Automotive Romania, SIITME Conference and Exhibition committee member, announced that the next event, SIITME 2018, will be held in Iasi, one of the most beautiful cities in Romania, and one of the four cities where Continental Automotive chose to set up operations. Surely, not accidentally. **PCBDESIGN**

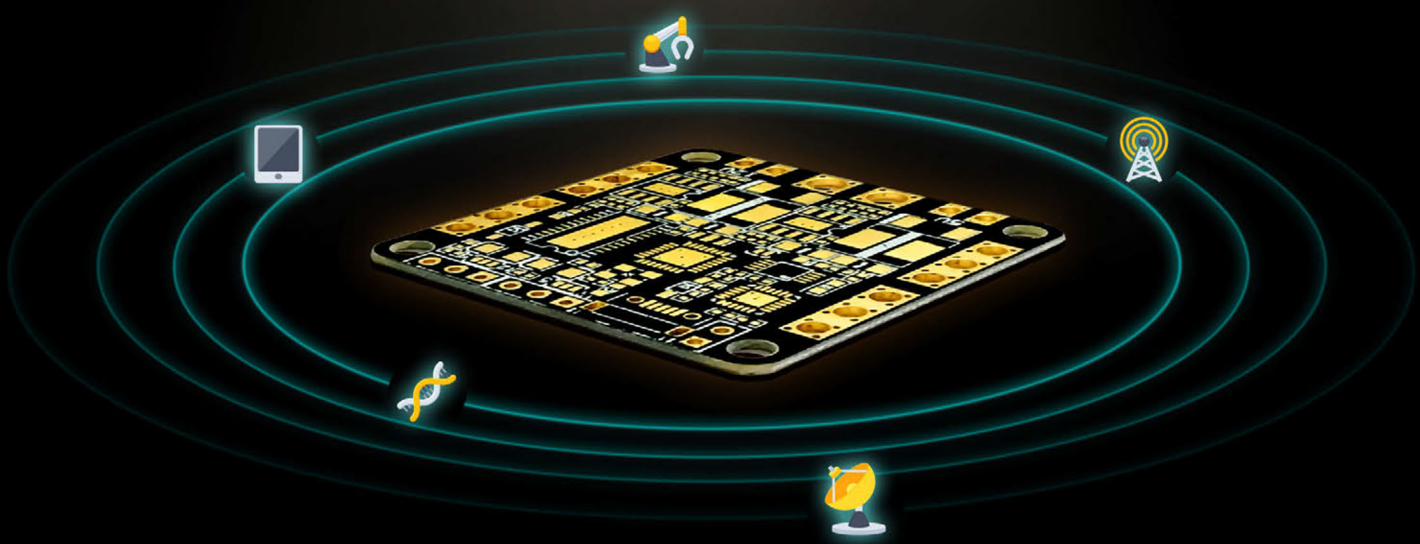
Joe Fjelstad is CEO of Verdant Electronics. To read past columns or to contact Fjelstad, click here.

New Robots Can See Into Their Future

UC Berkeley researchers have developed a robotic learning technology that enables robots to imagine the future of their actions so they can figure out how to manipulate objects they have never encountered before. In the future, this technology could help self-driving cars anticipate events on the road and produce more intelligent robotic assistants.

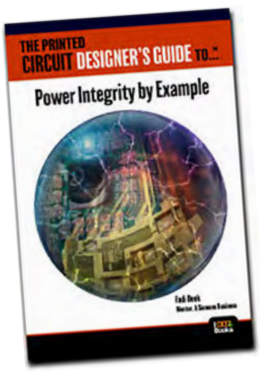
Using this technology, called visual foresight, the robots can predict what their cameras will see if they perform a particular sequence of movements.

The scientists are continuing to research control through video prediction, focusing on further improving video prediction and prediction-based control, as well as developing more sophisticated methods by which robots can collected more focused video data.



Recent Highlights from PCBDesign007

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



The Printed Circuit Designer's Guide to... Power Integrity by Example is authored by Fadi Deek of Mentor, A Siemens Business. This title is a follow-up to Deek's previous book on signal integrity, which is also available for download in our library.

2 Nine Dot Connects: Good Design Instruction is a True Value-Add

I recently had the chance to interview Paul Taubman, technical services director for Nine Dot Connects. We discussed the company's expansion from VAR to service bureau and content provider, and the changing landscape of PCB design instruction.

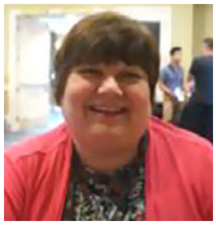
3 Cannonballs, eBooks, and Signal Integrity

I-Connect007 editors Andy Shaughnessy and Pete Starkey recently met with Polar Instruments Managing Director Martyn Gaudion at productronica. They discussed the success of "The Printed Circuit Designer's Guide to...Secrets of High-Speed PCBs," the ongoing challenge of facilitating communication between designers and fabricators, and the influence of chemical bond-enhancement processes on insertion loss.

4 Happy Holden and Charles Pfeil Discuss the Past and Future of PCB Design, Part 2

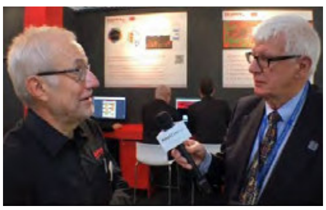
In Part 2 of this feature interview, Barry Matties continues his conversation with industry icons Happy Holden and Charles Pfeil at the recent AltiumLive 2017 event in San Diego. In this half of the interview, we discuss the potential for artificial intelligence in EDA software tools, the graying of the designer pool, and what can be done to draw more young people to PCB design.

5 Nancy Jaster Brings Manufacturing, Design Background to Designers Council



Nancy Jaster was recently named the head of the IPC Designers Council. At the recent AltiumLive 2017 event, Barry Matties spoke with Nancy about her unique background in both design and manufacturing, and how she hopes to use that experience and mindset to revitalize the Designers Council going forward, particularly with the International and student chapters. We also discussed her plans to bring more young people, particularly women, into the industry.

6 Video from productronica 2017: Karel Tavernier on Ucamco's New Communic8tor



European Editor Pete Starkey and Ucamco Managing Director Karel Tavernier discuss Ucamco's cloud-based Communic8tor platform which facilitates two-way communication between the CAM engineer and the PCB designer, or any other party involved in the manufacturing process. This gives real-time access to image data and annotations, enabling queries to be resolved, changes to be approved, and a full communications history to be maintained.

7 Beyond Design: When Do Traces Become Transmission Lines?

At low frequencies, traces and components on a PCB behave simply as lossless lumped elements—as taught in Circuit Theory 101. But as the frequency increases, the copper trace and adjacent dielectric(s) become a transmission line, the skin effect forces current into the outer regions of the conductor and frequency-dependent losses impact on the quality of the signal. The PCB trace now behaves as a distributed system with parasitic inductance and capacitance characterized by delay and scattered reflections.

8 Bay Area Circuits Updates InstantDFM Tool

Bay Area Circuits is on a quest to help PCB designers and design engineers. For the past few years, the company has been holding facility tours and open house events to help designers understand more about the fabrication process. At PCB West, I spoke with President Stephen Garcia and COO Brian Paper about their recent design tool update, and some of the other services they offer for PCB designers.

9 Cadence Appoints Anirudh Devgan as President

Dr. Devgan will report to Lip-Bu Tan, Cadence chief executive officer. Together, they will further the company's System Design Enablement strategy by accelerating the momentum in the core EDA business and delivering to the expanding needs of its growing customer base.

10 Signal Integrity: The Experts Weigh In

When we began planning our issue on signal integrity, we arranged a conference call with a variety of industry experts. Mike Steinberger of SiSoft, Mark Thompson of Prototron Circuits, and Yogen and Sunny Patel of Candor Industries joined editors Andy Shaughnessy, Patty Goldman, Happy Holden and Publisher Barry Matties on the call for a spirited discussion about the challenges related to signal integrity and some of the tricks of the trade for helping ensure SI.

PCB Assembly Supervisor—full time
Accurate Circuit Engineering—Santa Ana, CA

Position Summary: Responsible for all assembly processes to ensure continued growth as directed by management.

Essential Job Functions:
- Create, implement, and supervise in-house manufacturing facility
- Recruit, hire, train, and supervise assembly floor personnel
- Extensive hands on experience with all aspects of PCB assembly
- Understanding of IPC-A-610 standards
- Research and acquire additional assembly resources
- Gather data on product shortages, lead times, price changes, etc.
- Coordinate the assembly activities with sales to ensure 100% on-time delivery
- Create, implement, and supervise daily quality processes to ensure 100% accuracy
- Document, monitor and review progress of the business unit
- Respond to internal and external customers in a timely manner
- Coordinate walk-through, site audits, etc.

Qualifications:
- Minimum 3 years as operations supervisor of electronics assembly house
- 5+ years' experience in the electronics industry
- Previous experience as a quality or operations supervisor preferred
- Ability to solve practical problems using pre-established guidelines
- Strong facility in Microsoft Office applications
- Excellent verbal and written communication skills
- Ability to work with people of diverse backgrounds
- Highly organized/excellent time management skills
- Ability to perform at the highest level in a fast-paced environment
- Valid California driver's license.

apply now

PCB Process Planner

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

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

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

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

apply now

orbotech

Electronics Expert Engineer

Orbotech is looking for an Electronics Expert Engineer to handle various hardware activities, including communication, data path processing, device interfaces and motion, as well as system supporting functions in a multidisciplinary environment.

What Will Your Job Look Like?
- Providing cutting edge hardware solutions for challenging product line needs
- Developing board design and Logic in VHDL
- Defining and managing interfaces (software, algorithm, mechanics and electricity)
- Successfully integrating hardware with other product disciplines
- Supporting the product needs during and following release

What Do You Need to Succeed?
- BSc in electronics engineering
- At least 5 years of R&D experience in complex board design, mainly FPGA (communication interfaces, DDR controller, algorithm implementation)
- Experience in an Altera/Xilinx development environment
- Experience in ECAD design tools (DxDsigner, ModelSim) is an advantage
- Knowledge in laser interfaces, RF and analog is an advantage

Who We Are
Virtually every electronic device in the world is produced using Orbotech systems. For over 30 years, Orbotech has been a market leader in developing cutting edge inspection, test, repair, and production solutions for the manufacture of the world's most sophisticated consumer and industrial electronics.

apply now

orbotech

Electronics Team Leader

Orbotech is seeking an Electronics Team Leader to join our electronics team, which develops multi-disciplinary systems, including vision/laser, image processing, and control and automation missions.

What Will Your Job Look Like?
- Lead a team of electronics engineers in a multi-disciplinary environment
- Lead electronic activities from requirement phase to development, integration and transfer, to production
- Be the focal point for other disciplines and projects managers
- Maintain and improve existing electronics platforms

What Do You Need to Succeed?
- BSc/MSc in electronic engineering/ computer science from a well-recognized university
- 5+ years' experience in digital board design, high-speed links, computing embedded systems, and HW/SW integration
- 2-3 years' experience in leading a team of engineers
- Solid skills in complex FPGA design with multi-modules
- Solid skills in high-speed board design, DDR3/4, PCIE, USB, IO, and optic links
- Ability to design and execute end-to-end solutions

Who We Are
Virtually every electronic device in the world is produced using Orbotech systems. For over 30 years, Orbotech has been a market leader in developing cutting-edge inspection, test, repair, and production solutions for the manufacture of the world's most sophisticated consumer and industrial electronics.

apply now

INDIUM

Technical Content Specialist

Indium Corporation is seeking a technical content specialist to guide the development of data-rich, high-level content for the company's semiconductor and advanced assembly materials (SAAM) sales and technical literature. The technical content specialist will work with multiple departments to ensure that all externally-facing technical and sales collateral and internal training materials are consistent in format and of superior quality.

The technical content specialist will:
- Assist in the development of key content and ensure consistency of message and format across platforms
- Develop a technically-detailed understanding of Indium Corporation materials and offerings to the SAAM industry
- Curate a library of technical conference papers and associated materials, including content related to Indium Corporation materials and their performance
- Assist in the development of, and ensure consistency for SAAM promotional materials, such as product datasheets (PDS), images, brochures, whitepapers and presentations (technical and sales)
- Attend at least one technical conference and its paper session per year

Requirements:
- Technical undergraduate degree (BS in Chemistry/Physics/Metallurgy/Materials Science or Engineering discipline)
- 5 years of work experience in semiconductor assembly or advanced electronics assembly
- Excellent written and spoken English language skills; fluency in Chinese desirable
- Proven ability to work independently with verbal or written instructions

apply now

ventec
INTERNATIONAL GROUP
騰輝電子

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

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

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

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

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

www.venteclaminates.com

apply now

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

Prototron Circuits

Experienced PCB Sales Professional

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

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

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

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

apply now

Process Engineer (Redmond, Washington)

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

apply now

ARLON

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

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

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

more details

Events

IPC Calendar of Events, click here

SMTA Calendar of Events, click here

PCB Design Calendar of Events, click here

47th NEPCON JAPAN
January 17–19, 2018
Tokyo Big Sight, Japan

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

EIPC 2018 Winter Conference
February 1–2, 2018
Lyon, France

MD&M West
February 6–8, 2018
Anaheim, California, USA

2018FLEX
February 12–15, 2018
Monterery, California, USA

IPC APEX EXPO 2018 Conference and Exhibition
February 27–March 1, 2018
San Diego, California, USA

China International PCB and Assembly Show (CPCA)
March 20–22, 2018
Shanghai, China

KPCA Show 2018
April 24–26, 2018
Kintex, South Korea

Thailand PCB Expo 2018
May 10–12, 2018
Bangkok, Thailand

Medical Electronics Symposium 2018
May 16–18, 2018
Dallas, Texas, USA

2018 EIPC's 50 Years Anniversary Conference
May 31–June 1, 2018
Bonn, Germany

JPCA Show 2018
June 6–8, 2017
Tokyo, Japan

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

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

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

EDITORIAL:

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

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

MAGAZINE PRODUCTION CREW:

PRODUCTION MANAGER: **SHELLY STEIN**

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

INNOVATIVE TECHNOLOGY: **BRYSON MATTIES**

COVER: **SHELLY STEIN**

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

THE pcb design MAGAZINE

The PCB Design Magazine® is published by BR Publishing, Inc., PO Box 1908, Rohnert Park, CA 94927

December 2017, Volume 6, Number 12 • The PCB Design Magazine© is published monthly by BR Publishing, Inc.

ADVERTISER INDEX

Accurate Circuit Engineering......... 7
Altium.. 45
American Standard Circuits......... 29
Candor Industries........................ 11
Coast to Coast Circuits, Inc......... 63
DesignCon................................. 53
Downstream Technologies...... 9, 31
Eagle Electronics........................ 23
Electrolube................................. 21
EMA Design Automation............. 25
General Circuits.......................... 71
I-Connect007.............................. 85
I-Connect007 eBooks................ 2, 3
In-Circuit Design Pty Ltd.............. 65
IPC.. 55
Isola.. 5
Mentor, a Siemens Business......... 51
Miraco.. 39
NCAB............................ 33, 35, 57
The PCB List......................... 27, 69
Prototron Circuits........................ 67
Rogers Corporation..................... 49
Sunstone Circuits......................... 17
US Circuit.................................... 43
Ventec International Group.... 37, 59