THE pcb design MAGAZINE

an I-Connect007 publication

February 2017

I·007e Books

Introducing a New Micro eBook Series

Brought to you by: I-Connect007 GOOD FOR THE INDUSTRY

" The timing of this book release couldn't be better. Marrakchi takes the reader on a guided tour through the entire DFM process! "

-Kelly Dack

Free Download

I-Connect007 is excited to announce the release of the first title in its ***Printed Circuit Designer's Guide to...***™ series, *The Printed Designer's Guide to…Design for Manufacturing (DFM).*

Written by David Marrakchi, a Senior Technical Engineer at Altium, this micro eBook provides designers of all experience levels all the DFM knowledge they need to produce a manufacturable and functional board.

This is a must-read for any board designer who wants to get a good board back, every time.

The Printed Designer's Guide to…Design for Manufacturing (DFM) provides an in-depth look at DFM: what DFM entails, why it's so critical today, and how to implement DFM best practices.

We welcome you to download this book free by visiting the book's website.

Look for these other exciting titles in our new micro eBook series to be released soon:

- ***The Printed Circuit Buyer's Guide to... Solderless Assemblies*** *by Joe Fjelstad*
- ***The Printed Circuit Buyer's Guide to... Flex Design*** by American Standard Circuits
- ***The EMS Buyer's Guide to… Procuring High Complexity Electronics*** by Zentech
- ***The Printed Circuit Buyer's Guide to... Simulation and Analysis*** by Mentor Graphics

We hope you enjoy *The Printed Designer's Guide to…Design for Manufacturing (DFM)!*

"Where Customers and Technology INTERACT"

Are you bored and feel trapped in a nowhere job? Come join our team!

now hiring

We are actively seeking to enhance our current operation and are looking to fill the following positions:

- Director of Sales & Marketing
- Director of Quality
- Inside Sales/Customer Service
- Front End Engineering - Planner
- Production Control Manager
- Senior Process Engineer
- Shipping/Recieving
- Outside Sales/Manufacturer's Representative
- Maintenance Technician
- Production - all areas
- Lab Technician

Click Here for more information.

E-mail resume to: Kelli Cain

WWW.MULTILAYER.COM

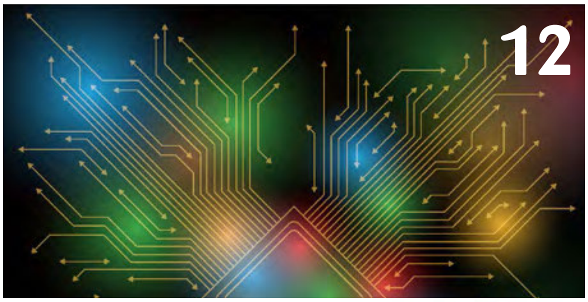
Mil-PRF-55110 AS 9100

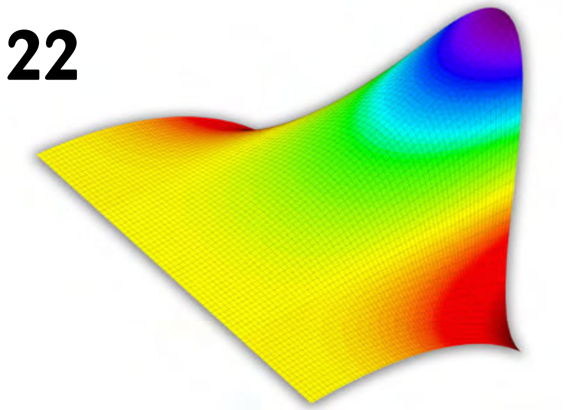
Multilayer Technology
3835 Conflans Rd
Irving, TX 75061-3914

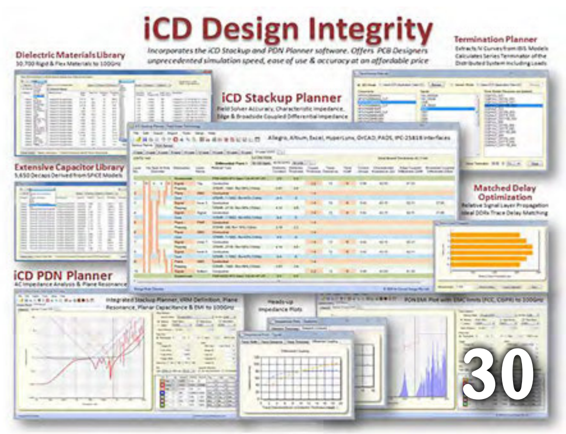
(972) 790-0062

THE pcb design MAGAZINE

Feature Content







New Technology

Technological innovations have been coming at us so fast that it's often difficult to keep track of them. We asked our contributors to discuss new technology that they were excited about. In this issue, Douglas G. Brooks, PhD, focuses on a variety of experiments conducted with the Thermal Risk Management (TRM) tool, such as finding the temperature inside the barrel of a via. Columnist Barry Olney of In-Circuit Design explains how his new iCD Design Integrity platform can make PCB designers much more productive. Dingru Xiao of Cadence Design Systems discusses new tool options that integrate the functions needed by power integrity engineers and PCB designers in one platform. And columnist Tim Haag of Intercept Technology wonders if PCB designers will ever be able to design boards with a hologram, just like the one in the first "Iron Man" movie.

185HR

High-Tg, Low-CTE Materials with High Thermal Reliability

FEATURES

- High Thermal Performance
  - Tg: 180°C (DSC) (Base Laminate)
  - Td: 340°C (TGA @ 5% wt loss)
  - Low CTE for reliability
- T260: 60 minutes
- T288: >15 minutes
- Lead-free Compatible and RoHS Compliant
- UV Blocking and AOI Fluorescence
  - High throughput and accuracy during PCB fabrication and assembly
- Superior Processing
  - Closest to conventional FR-4 processing
- Industry Approvals
  - IPC-4101D-WAM1 /98 /99/ 101 /126
  - UL – File Number E41625
  - Qualified to UL's MCIL Program
- Core Material Standard Availability
  - Thickness: 0.002″ (0.05 mm) to 0.060″/0.062″ (1.5 mm)
  - Available in full size sheet or panel form
- Prepreg Standard Availability
  - Roll or panel form
  - Tooling of prepreg panels available
- Copper Foil Type Availability
  - Standard HTE Grade 3
  - RTF (Reverse Treat Foil)
- Copper Weights
  - ½, 1 and 2 oz (18, 35 and 70 μm) available
  - Heavier copper available upon request
  - Thinner copper foil available upon request
- Glass Fabric Availability
  - Standard E-glass
  - Square weave glass fabric available

KEY APPLICATIONS

- Heavy copper, 6 & 12 oz automotive applications
- Military
- Medical
- Industrial

ORDERING INFORMATION:

Contact your local sales representative or visit **www.isola-group.com** for further information.

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

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

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

isola

THE pcb design MAGAZINE

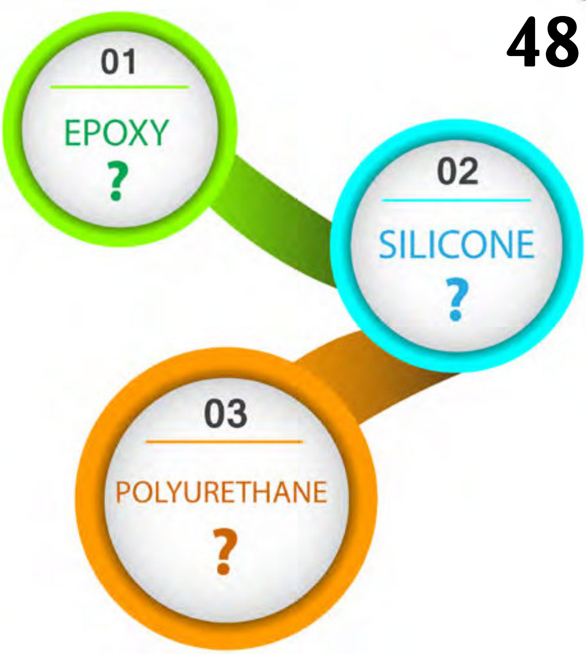
February 2017

Vol. 6, No. 2

thepcbdesignmagazine.com

60

48



More Content

Are You a Luddite?

by Andy Shaughnessy
I-CONNECT007

Luddite: *One of a group of early 19th century English workmen destroying labor-saving machinery as a protest; broadly: One who is especially opposed to change. <The Luddite argued that automation destroys jobs>*
—Merriam Webster's online dictionary

I recently discovered that I'd become a Luddite. Sure, it was only in one very specific case, but as the IT guy for our home, I like to think I'm on top of technology.

Lately, my girlfriend Rita and I have been listening to her podcasts in the car, primarily "My Favorite Murder," "Serial," and "These Are Their Stories," which is devoted to tongue-in-cheek discussions of the entire "Law and Order" franchise. It turns out podcasts are all she listens to, with the occasional exception of public radio. She has hundreds of podcasts saved on her phone. She even listens to podcasts that feature the stars of podcasts talking about podcasts.

So, I tried to set up podcasts on my phone. No dice. I followed the directions…I think. Finally, Rita took my phone, mashed a few buttons, and handed it back to me. "Here, you're ready to go. Any questions?" At least she didn't tousle my hair and call me "kiddo."

Of course, as the editor of a technical magazine, I try not to get too far behind the technology curve. But if we're not paying attention to

every single tech news item, we might miss out on innovative new products and systems that can make our lives healthier, more productive, or just more fun.

The following are just a few of the innovations we've seen in the last few years:

- Google and Tesla made self-driving cars a reality. Google did so with a splash, but Tesla went under the radar with a software upgrade. The Autopilot update included a steering feature which allowed drivers to sit in the passenger seat, hands-free, and scare the hell out of all of the other drivers on the road. One Tesla driver posted a video of himself riding on the roof of his Tesla as it navigated through traffic.
- Robots are not only moving toward artificial intelligence—they're learning to teach other robots. And they can teach other robots faster and better than humans can. Some swimming robots are now transparent, made up primarily of water and covered in a rubbery gel.
- Scientists have mounted tiny backpacks on dragonflies, hoping to eventually control their flight patterns and create a tiny air force that could launch tiny missiles at our enemies. (OK, I made up the last part. But it's clearly not out of the question.)
- Doctors used genetically engineered immune cells to save dying cancer patients. Is this the beginning of the end of cancer?
- Researchers used genome editing to create fungus-resistant wheat. Can they eventually engineer plants with traits such as drought and disease tolerance?
- Now we can talk to our TV, and ask it what our schedule looks like for tomorrow night. My new laptop has voice-operated everything.

All of this would have been considered pie-in-the-sky nonsense a decade ago. But this is nothing like what the next decade, or the next century, will bring. As futurist Ray Kurzweil once explained, "An analysis of the history of technology shows that technological change is exponential, contrary to the common-sense 'intuitive linear' view. So we won't experience 100 years of progress in the 21st century—it will be more like 20,000 years of progress (at today's rate)."

This month, for our New Technology issue, I asked our contributors to discuss new technology that they were excited about. Douglas Brooks, PhD, discusses a variety of experiments conducted with the Thermal Risk Management (TRM) tool developed by his partner Dr. Johannes Adam, including exploring the temperature in the barrel of a via, thermal profiles around right angles, and current density at each point on a trace. Columnist Barry Olney of In-Circuit Design explains how his company's iCD Design Integrity platform can make PCB designers much more productive. Dingru Xiao of Cadence Design Systems discusses some new tool options that combine the functions needed by power integrity engineers and PCB designers in one integrated environment. And Columnist Tim Haag of Intercept Technology wonders if, and when, PCB designers will ever be able to design boards by interacting with a hologram, just like the one in the first "Iron Man" movie.

We're happy to welcome back Joe Fjelstad, whose column espouses the benefits of the standard grid pitch for PCB design. Columnist Alistair Little of Electrolube explains how to select the perfect resin for your PCB. And Dick Crowe of Burkle USA offers his review of NAMM 2017, the National Association of Music Merchants show that draws tens of thousands of musicians and music retailers to Anaheim, California each year.

We're all getting ready for IPC APEX EXPO and the Design Forum in San Diego. If you're at APEX, stop by our booth. If you can't make it, we'll have plenty of video coverage of the show, from start to finish.

I hope to see you in San Diego. And don't be a Luddite! **PCBDESIGN**

Andy Shaughnessy is managing editor of *The PCB Design Magazine.* He has been covering PCB design for 17 years. He can be reached by clicking here.

FEATURE

EXCITING NEW TECHNOLOGY: Thermal Risk Management

by Douglas G. Brooks, PhD

Two years ago I entered into a collaboration with Dr. Johannes Adam, from Leimen Germany. Johannes has written a software simulation tool called Thermal Risk Management (TRM). We used it to look at the thermal characteristics of PCB traces under a variety of conditions, and it is hard for me to contain my excitement and enthusiasm for what it does and what we learned about traces using it. Our collaboration resulted in the publication of numerous articles and a book [1]. In this article, I'll talk about some of the capabilities of TRM that really caught my attention. (Note: TRM has much more power than I will have space to discuss in this article. See the technical note at the end of the article for more on this.)

TRM allows us to model a PCB trace and predict its thermal properties. So the first thing we did was model a variety of traces and compare the results with the data in IPC-2152 [2]. The data fit very well, giving us confidence that the process was valid. But if the software could fit a single trace in isolation, then it could also predict the temperature of a trace under a variety of other conditions that board designers more typically face, such as changes in length, the proximity of adjacent traces, and the presence of planes. For example, Figure 1 illustrates the thermal patterns around a trace with and without a plane directly under the trace. The presence of a plane clearly lowers the trace temperature and widens the area of the board where the thermal profile spreads.

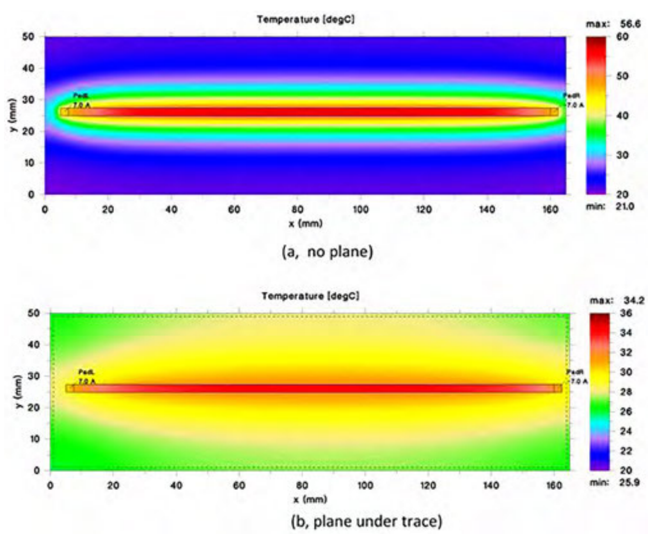


Figure 1: Thermal profile of a trace with and without an underlying plane.

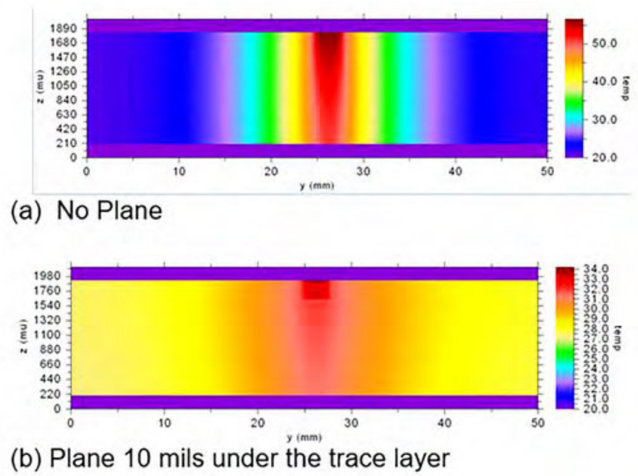


Figure 2: Thermal profiles underneath the traces and through the boards in Figure 1.

But TRM also allows us to view things in many other dimensions. For example, Figure 2 shows the thermal profile of a cross-sectional view of the board underneath the trace. The thermal profile underneath the trace through the board without the plane extends all the way to the far side, but the thermal profile of the board with the plane changes dramatically at the plane. I know of no other way to look at this profile other than with a computer simulation.

But the result that really got us excited, and led to an entire series of studies that culminated in the book, came about when we used TRM to look at the temperature of a via. There is no other practical way to explore the temperature within the barrel of a via. What we found was, for a normal trace, the via is cooler than the trace! (Figure 3). It turns out that it is not the current through the via that determines the temperature of the via; it is the temperature of the adjacent trace [3]. Therefore, if the trace is sized correctly, only a single via is required (in most practical situations) and its size can be much smaller than previously assumed.

The software allowed us to look at the temperature profile around a right-angle corner. There is some suggestion that the inside of the corner is hotter than the outside of the corner. Our simulations in Figure 4a showed this to be true. But the reason is a little surprising. The reason is not because the current density is concentrated at the inside corner. When we look at Figure 4b, we see that the thermal profile extending out from the trace into the dielectric around the inside of the corner is different than

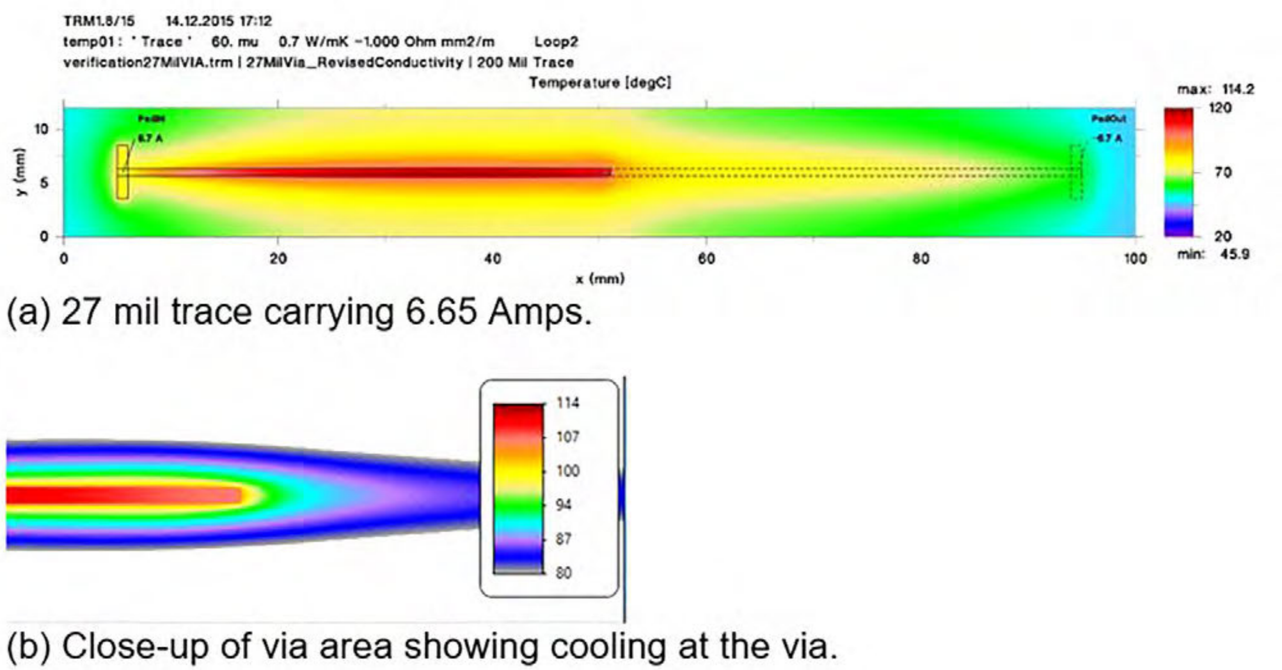


Figure 3: Thermal profiles around a typical via.

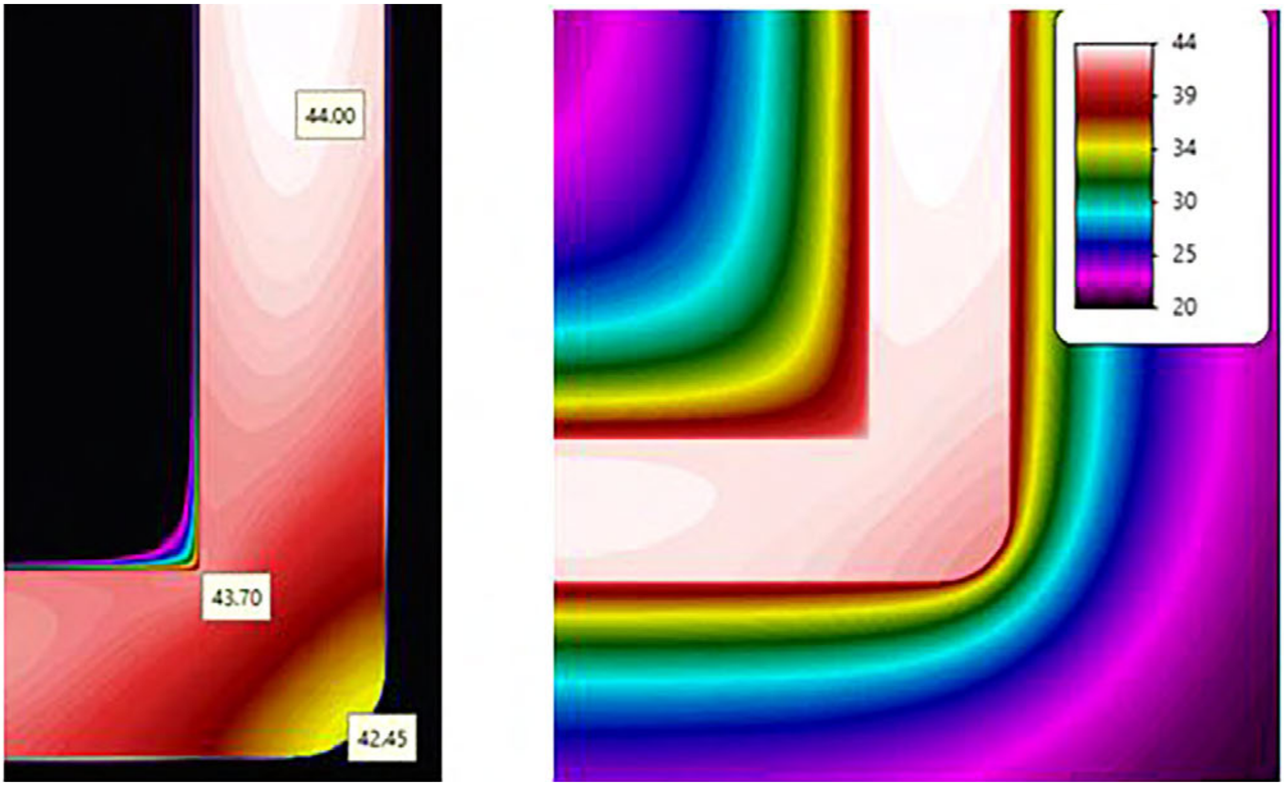


Figure 4: Thermal profiles of a right-angle corner; (a) is for a narrow temperature range, and (b) is for the full temperature range.

it is around the outside of the corner. The outside of the corner is cooler because there is more board area for the trace to "cool into," not because the current density is lower there.

TRM has the capability to look at the current density at every point along a trace. Figures 5 illustrates the current densities for the right-angle corner, above, while Figure 6 shows the current densities around a trace with 4 vias. When we analyzed the current densities in the vicinity of the vias, especially the "near vias," there were some surprising discoveries. Again, I know of no other way to look at the current density at a point along a trace other than with a tool like TRM.

TRM has some capability of looking at temperatures related to current pulse streams of arbitrary frequency and duty cycle. In this sense it is analyzing the temperatures related to AC currents, although the capability does not extend all the way to analog waveforms. But a lot of insight can be gained from looking at traces carrying a pulse width modulated (PWM) AC

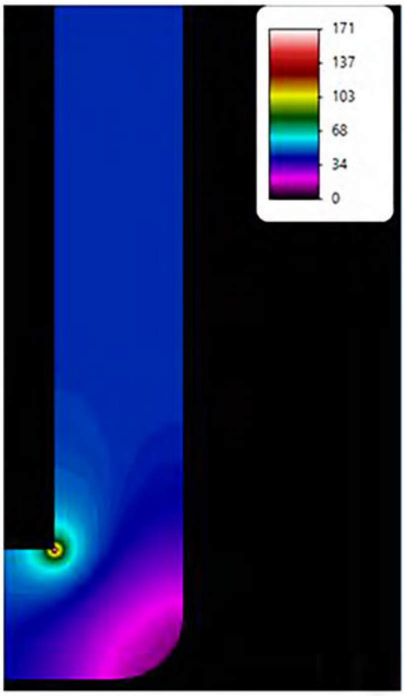


Figure 5: Current density around the right-angle corner shown in Figure 4.

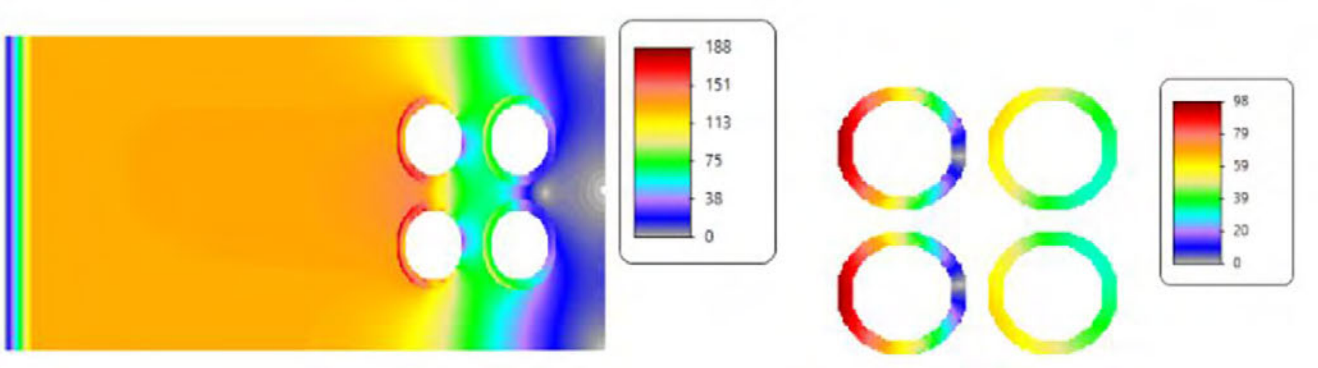


Figure 6: Current density around vias.

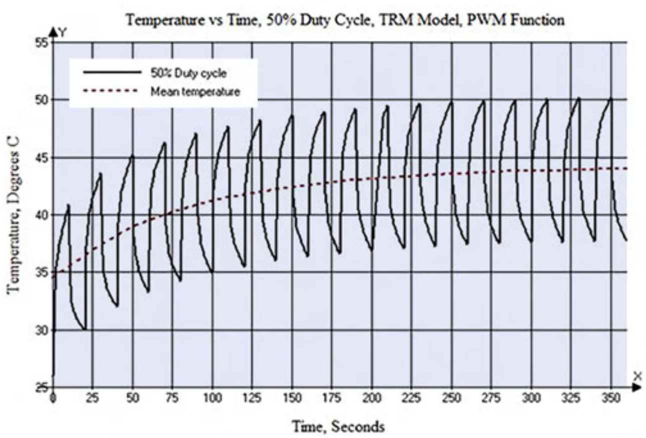


Figure 7: Temperature profile of a trace carrying a pulsed waveform with 50% duty cycle.

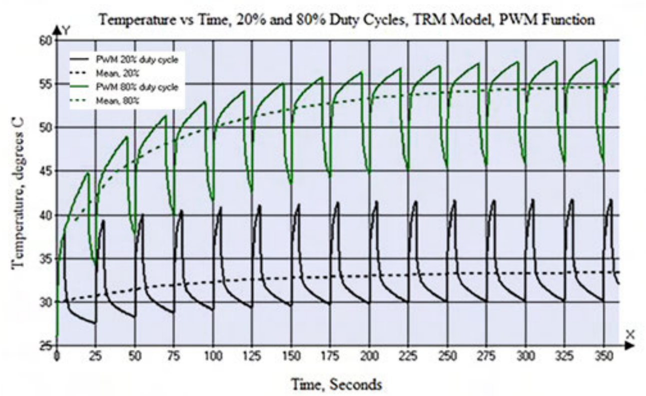


Figure 8: Temperature profiles of a trace carrying a pulsed waveform with 20% and 80% duty cycles.

waveform. Figure 7 illustrates the temperature profile of a trace carrying a pulsed current waveform with a 50% duty cycle. Figure 8 illustrates the same trace carrying pulse streams with 20% or 80% duty cycles. The software can show us the thermal pattern through time, but also the ultimate stable temperature that will be reached for any pulse condition.

My final illustration of TRM's power relates to what we could discover about the fusing process (i.e., what happens when a trace is subjected to a sudden overload). (Those who know me know that this is one of my favorite topics!) The early work on this topic was done by W. H. Preece in the 1880s, and I. M. Onderdonk, probably in the 1920s. Onderdonk is the author of a famous equation that bears his name that is frequently referenced in fusing investigations (Equation 1).

[Eq. 1] $$33\left(\frac{I}{A}\right)^2 S = \log_{10}\left(\frac{t}{234+Ta}+1\right)$$ Eq. 1

Where: I = the current in amps
A = the cross-sectional area in circular mils
S = the time in seconds the current is applied
t = the rise in temperature from the ambient or initial state
Ta = the reference temperature in degrees C

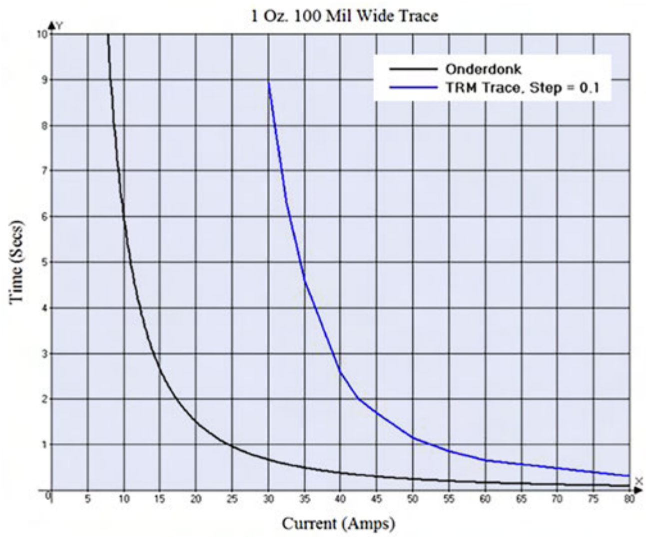


Figure 9: Fusing equations for a 1.0 oz. 200 mil wide trace.

Onderdonk's equation applies to a single copper wire in space and assumes there is no cooling of the wire. Therefore, it only applies for a very short period of time, perhaps up to five or 10 seconds, depending on conditions. We used TRM to simulate the conditions Onderdonk assumed for a bare wire in space and also for conditions on a normal PCB. The results are shown in Figure 9.

The simulation shows that at very high currents, both the bare wire and the trace fuse very quickly. But at lower currents, it takes much longer for a trace on a board to melt than it does for a bare wire. The separation of the curves in Figure 8 is a direct result of the cooling mechanisms on the board.

TRM allowed Johannes and me to explore paths that had never been available to us before. And it turned out, several of these new paths led to additional paths, and they to yet more paths. I am now convinced that designers who really need to know the thermal properties around their traces, or who really need to optimize the thermal properties of their designs, will need to use a tool like TRM.

Graphs and equations are no longer adequate. And we have been here before. A great many designers needed to start worrying about impedance-controlled traces in the early 1990s. At that time, impedance equations were available to us in a variety of publications. But today we all know that the equations are no longer accurate enough and we need "field effect" (i.e., computer model) solutions. And today that is starting to happen on the area of PCB trace thermal design.

Technical note: In this article, I have shown how the TRM (Thermal Risk Management) software tool can be used to analyze thermal issues of and around individual traces, and I've found that the tool is very well suited to doing this. But this may leave the reader with the mistaken believe that this is the limit of what TRM can do. In fact, TRM is *much* more powerful than this!

Dr. Adam originally conceived and designed TRM to analyze temperatures across an entire 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. Although the program could be adapted to the measurement of an individual trace, as we have done here, it was not originally conceived with that use in mind. TRM has the ability to look at an *entire board* whose data can be entered into the software using, for example, the entire set of Gerber and drill files.

This article includes images of thermal diagrams the software produces relative to a single trace under analysis. But Figure 10 illustrates

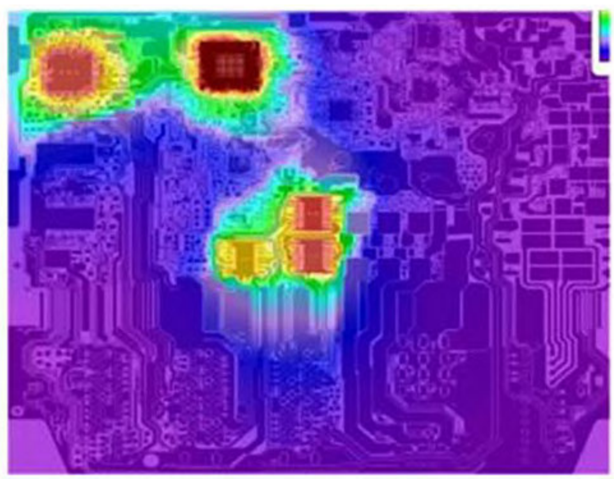
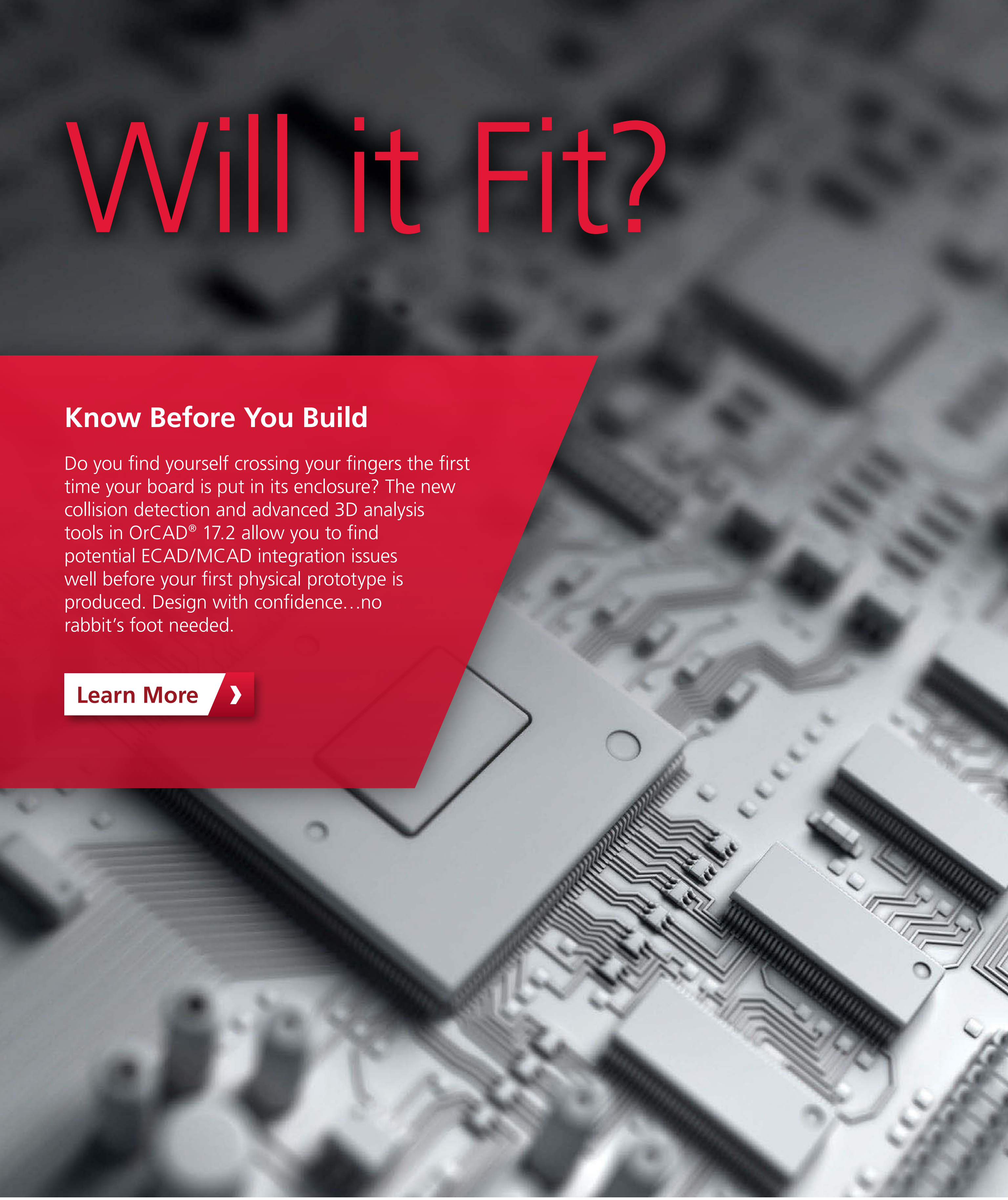


Figure 10: With TRM, a thermal profile can be generated of an entire board, including components, under load.


Will it Fit?
Know Before You Build
Do you find yourself crossing your fingers the first time your board is put in its enclosure? The new collision detection and advanced 3D analysis tools in OrCAD® 17.2 allow you to find potential ECAD/MCAD integration issues well before your first physical prototype is produced. Design with confidence…no rabbit's foot needed.
Learn More ›

how a similar thermal profile can be generated of an *entire* board, including components, under load. Imagine how powerful this could be for a systems or package designer worrying about the thermal performance of an actual product in a difficult environment, such as the engine compartment of a car, or the confines of a very small enclosure! As I said, this is exciting technology. **PCBDESIGN**

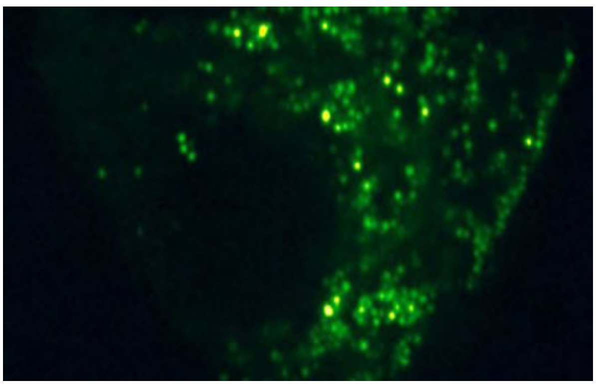
You can learn more about TRM and its capabilities at www.adam-research.de/en/.

References

1. Brooks and Adam, "PCB Trace and Via Temperatures; The Complete Analysis, 2nd Edition," 2017, available at Amazon.com.
2. IPC-2152, "Standard for Determining Current Carrying Capacity in Printed Board Design," August, 2009, www.ipc.org.
3. The examples in this paper are taken from Reference 1, Chapters 7, 8, 10, 13, and 14.

For the last 20 years, **Douglas G. Brooks** has owned a small engineering service firm and written numerous technical articles on printed circuit board design and signal integrity issues, and published two books on these topics. He has given seminars several times a year across the U.S., as well as Russia, China, Taiwan, Japan, and Canada. His primary focus is on making complex technical issues easily understood by those without advanced degrees. His latest book, *"PCB Trace and Via Currents and Temperatures: The Complete Analysis,"* was released in 2016.

New Method Improves Accuracy of Imaging Systems



New research provides scientists looking at single molecules or into deep space a more accurate way to analyze imaging data captured by microscopes, telescopes and other devices.

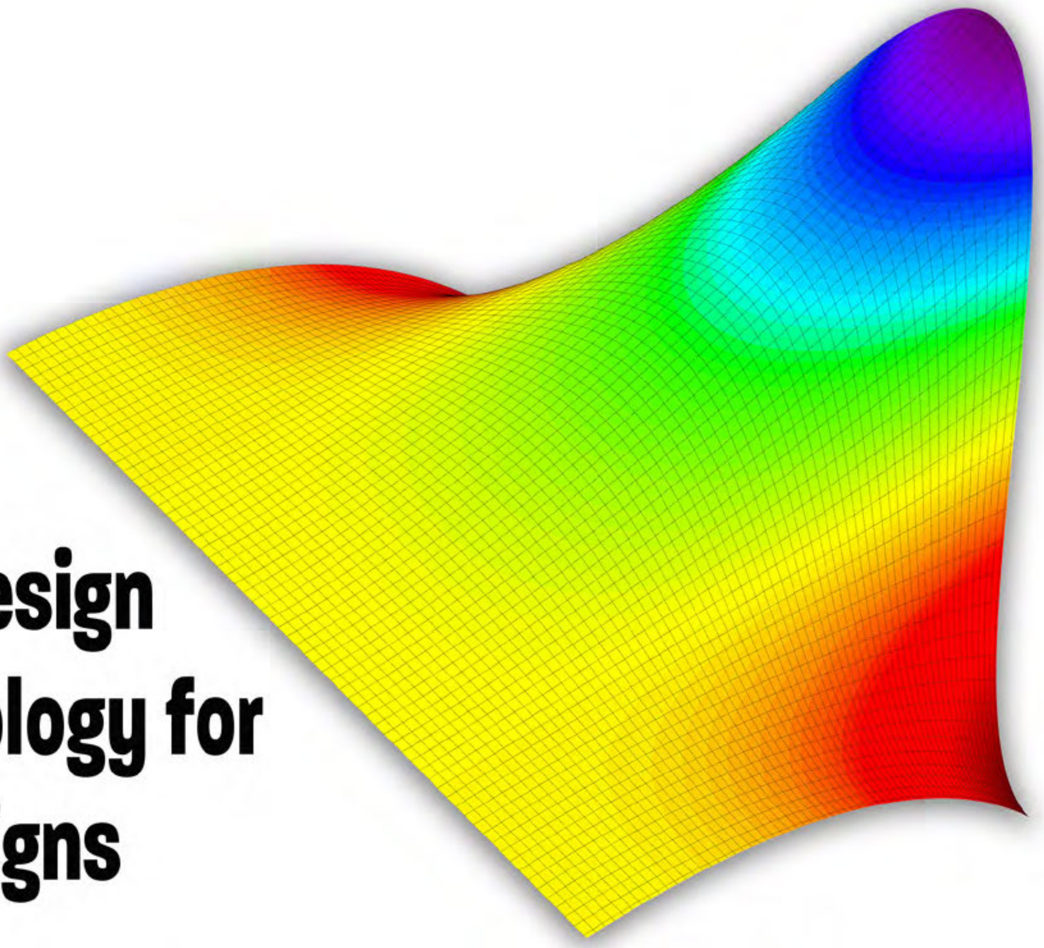
The improved method for determining the position of objects captured by imaging systems is the result of new research by scientists at the University of Chicago. Many imaging systems and image-based detectors are constituted of pixels, such as with a mega-pixel cell phone. So-called particle tracking allows researchers to determine the position of an object down to a single pixel and even explore sub-pixel localization to better than one-tenth of a pixel accuracy.

But such sub-pixel resolution depends on algorithms to estimate the position of objects and their trajectories. Using such algorithms often results in errors of precision and accuracy due to factors such as nearby or overlapping objects in the image and background noise. SPIFF can correct the errors with little added computational costs, according to Scherer.

Applicable to many disciplines

"Analyzing an image to obtain a rough estimate of an object's position isn't too difficult, but making optimal use of all the information in an image to obtain the best possible tracking information can be really challenging," said David Grier, professor of physics at New York University, who was not involved in the research.

The research described in the paper applied SPIFF to experimental data on solids (i.e., colloidal spheres) suspended in a liquid, but the researchers have now applied their method to many other datasets, including nanoscale features of cells (e.g. vesicles), metallic nanoparticles and even single molecules, Scherer said, adding that the SPIFF method is applicable to all tracking algorithms.



A New Power Design Methodology for PCB Designs

by Dingru Xiao
CADENCE DESIGN SYSTEMS

Abstract

Advanced PCB design is an iterative process of analysis-fix-analysis. Historically, this process is very time-consuming, requiring analysis experts and PCB designers to work together to find and fix layout problems. This article describes a new PCB design methodology that allows a PCB designer to perform the power design without having to run expert-level analysis tools. This methodology provides the setup automation for advanced analysis without the need to understand every minute parameter, and can be completed in a few steps.

User-friendly analysis reports guide the PCB designer to the exact location where design changes must be made to meet specifications. IPC standard-based constraint of automatic calculations helps PCB designers understand how good is good enough for the layout changes. All violations of IPC standards can be marked directly in the layout, so PCB designers can easily find the problems in the layout and fix them before handing the design over to the power integrity (PI) experts. This allows PI experts focus on performance optimization, cost reduction, and other issues, so a high-quality design can be produced in a shorter amount of time.

PCB Power Design Challenges

For modern electronic systems, power design has become more important with the requirements of low power, minimization, high density and high-speed data rate for high-end applications. Usually, the planes, copper pours, routed power traces and vias on a PCB serve as power distribution, signal return paths, heat dissipation and so on.

The following questions must be considered:

- How does the PI engineer communicate with the hardware engineer and PCB designer about problems with the design and guidance on how to fix it?
- How does the PCB designer communicate with the PI engineer about solutions to those problems?
- How can the design engineer and PCB designer determine whether a solution is good in the early stage?

Generally, PI engineers communicate with PCB designers by email, phone calls, or meeting face to face to discuss the issues and the solutions to fix problems with the layout.

Unlike when analyzing signal integrity, PI engineers are not usually involved in the early stages of the design because of the lack of pre-layout analysis tools for power analysis in the industry. The first cut of PCB power design usually is based on experience and industry conventions, so many power problems only surface late in the process, leaving PI engineers to focus mainly on the post-layout verifications for power systems. Also, PCB designers generally do not want to use professional analysis tools because of their complicated settings and different EDA tools/platforms.

This lag time greatly affects the efficiency of design and the time-to-market of the products.

PI-Aware PCB Design Methodology

To solve these challenges, a new design methodology is needed for PCB power design that combines the design and analysis together. The workflow of this methodology is shown in Figure 1.

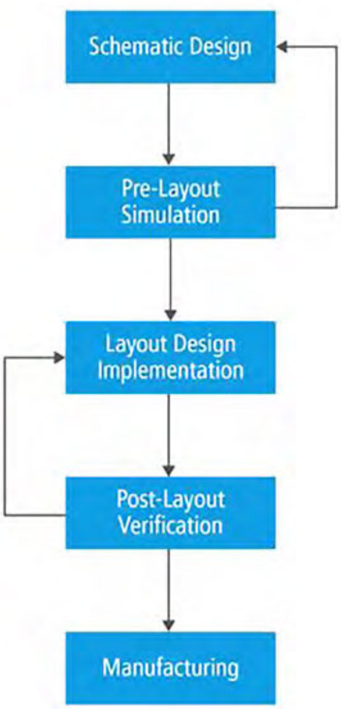


Figure 1: Advanced PCB design/analysis flow.

With this methodology, PI engineers are involved in the design process at very beginning of the design process, in the schematic design stage. They can prepare the component models, settings, and constraints for component instances, and for the entire design when the schematic is complete. These settings and constraints are one-time efforts, and they can be passed to layout automatically for future post-layout analysis.

PCB designers can complete the setup for simulation quickly by clicking few buttons; they do not need to understand all the ins and outs of complicated analysis concepts and parameters. All violations are recorded in an HTML report and can be noted in the layout. With cross-probing between the report and the layout, PCB designers can locate the problems quickly and accurately, and fix them directly. To verify the effects of the layout changes, a single click reruns the simulation with a refreshed report. The same engine is used as the power expert, so the simulation results are consistent.

All engineers—including hardware engineers, PCB designers and SI/PI experts—can use the same platform for design and analysis, and all settings in the pre-layout simulation can be reused by the post-layout analysis.

New Technologies

Some new technologies introduced in this new methodology support the analysis-fix-analysis process include Analysis Model Manager (AMM), Cadence Allegro PowerTree technology, IPC-based constraint generation, integrated design/analysis environment and so on. These technologies enable designers to set up automated analysis, making their process much simpler.

In the early stages of design, PI engineers can create a PowerTree flow with the schematic data. If available, AMM data can be applied to the PowerTree flow, as well. Based on the PowerTree flow, the PI engineer quickly generates a workspace that includes all component settings, constraints, and relevant power/ground nets for enabling and selection.

In the layout stage, PCB designers can use the workspace for an iterative analysis-fix-analysis process without special support from PI ex-

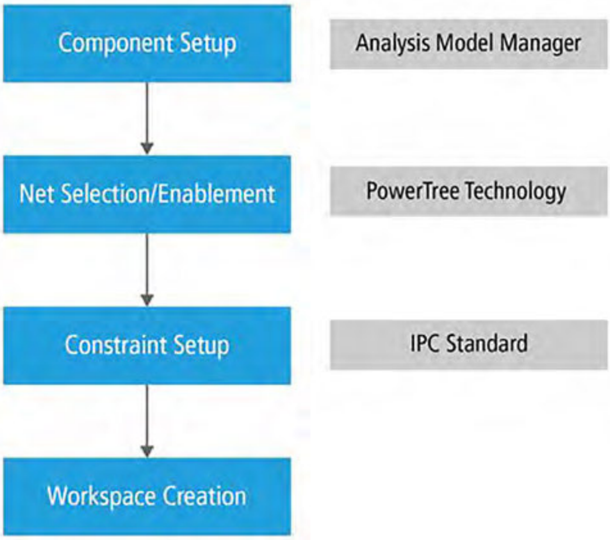


Figure 2: IR drop analysis setup automation flow.

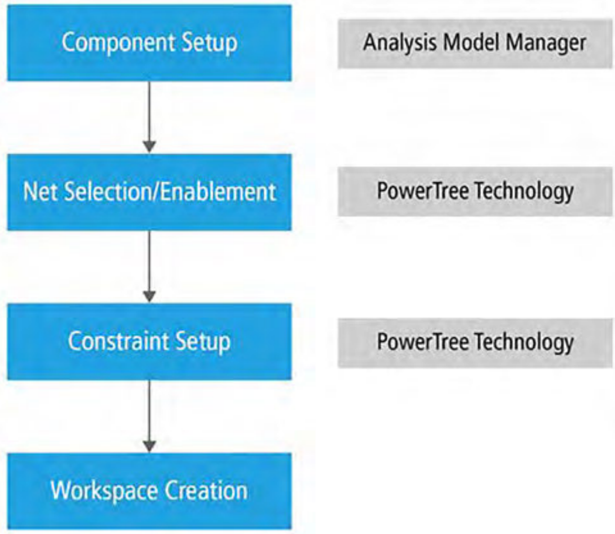


Figure 3: Decap selection and optimization setup automation flow.

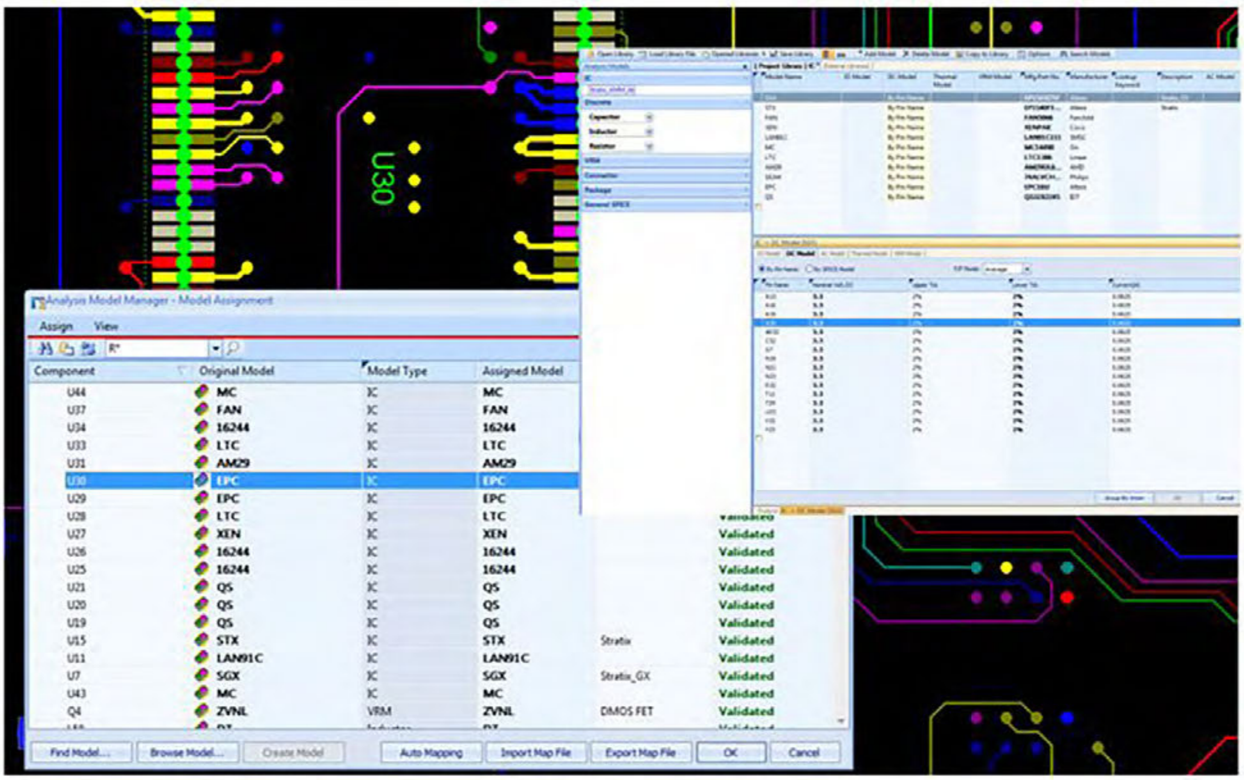


Figure 4: AMM model editing/assignment.

perts. For example, the IR drop analysis setup automation flow is shown in Figure 2.

For decap selection and optimization, the setup automation flow is shown in Figure 3.

Because the simulation setup is automated, either the layout designer or PI engineer can validate that target impedance constraints are met as an in-design task. Both pre- and post-

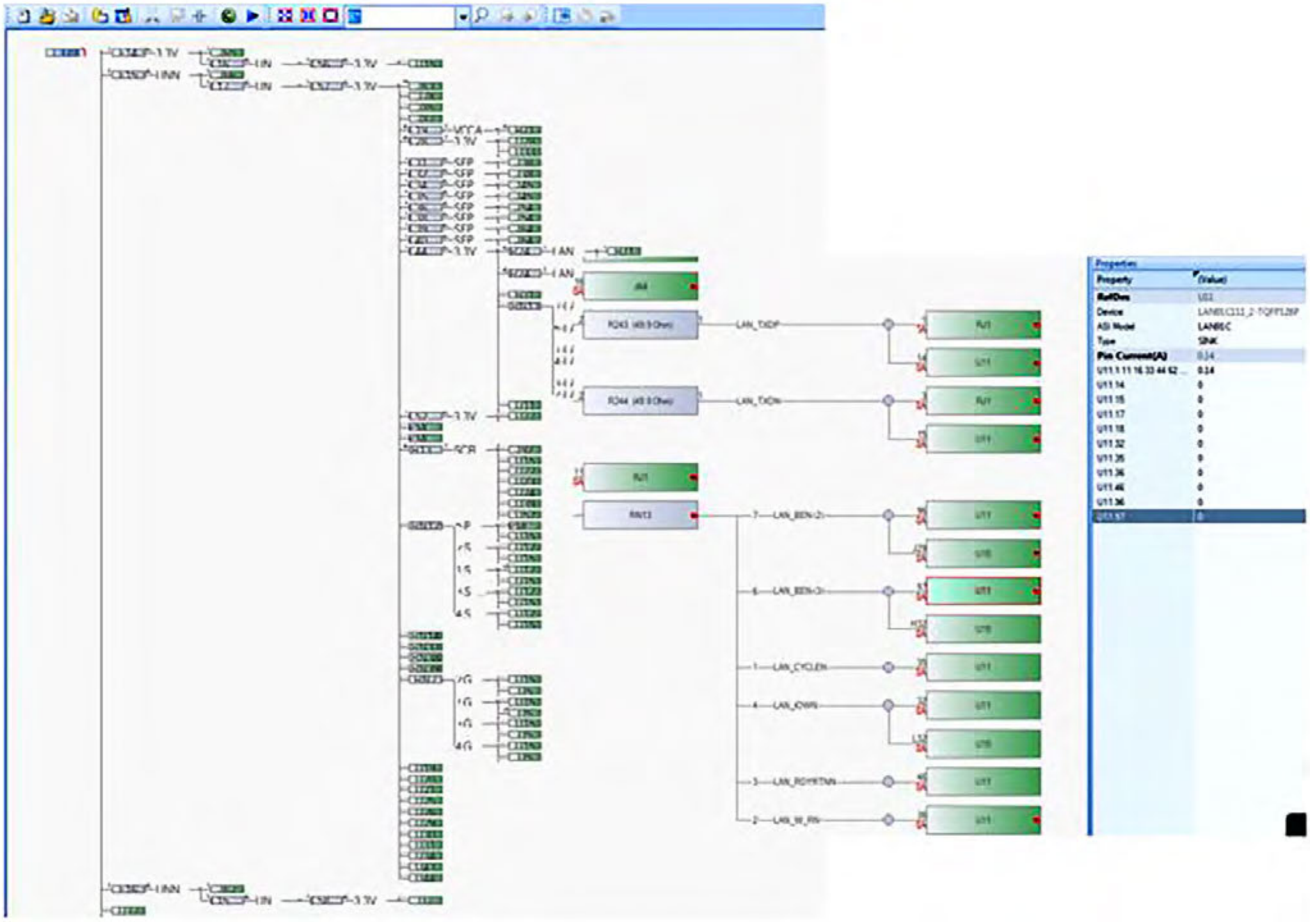


Figure 5: A PowerTree diagram.

layout can share the same analysis setup, and the optimized decoupling scheme can be updated automatically using PCB design tools.

AMM

AMM is a utility used to manage all analysis models including SPICE, IBIS, VRM, and sink and thermal parameters based on components. Using the unified AMM data, all components—including ICs and discrete components with the same part number or model name—need only one-time model assignment in the same design, without duplicating each one. The model data can be reused by different designs, thus avoiding the manual setup for each design.

AMM data can be prepared by manual entry in advance, or extracted from existing designs. Figure 4 shows the AMM model editing and model assignment dialog.

PowerTree

The PowerTree technology is a visualization tool to view power topologies from schematic or layout data. All component properties—including the resistance value for a resistor, the voltage property for a power/ground net, and so forth—can be extracted during the schematic or layout stages and used to set up the pre-layout simulation using the PowerTree tool. Based on the simulation results, users can determine whether all of the selected components meet the functional specifications before the layout is implemented.

AMM is integrated into the PowerTree tool as well, and users can reuse the model data in both pre- and post-layout analysis. Using PowerTree topologies for the post-layout analysis, only the relevant power nets/ground nets are selected and enabled, isolating the pertinent

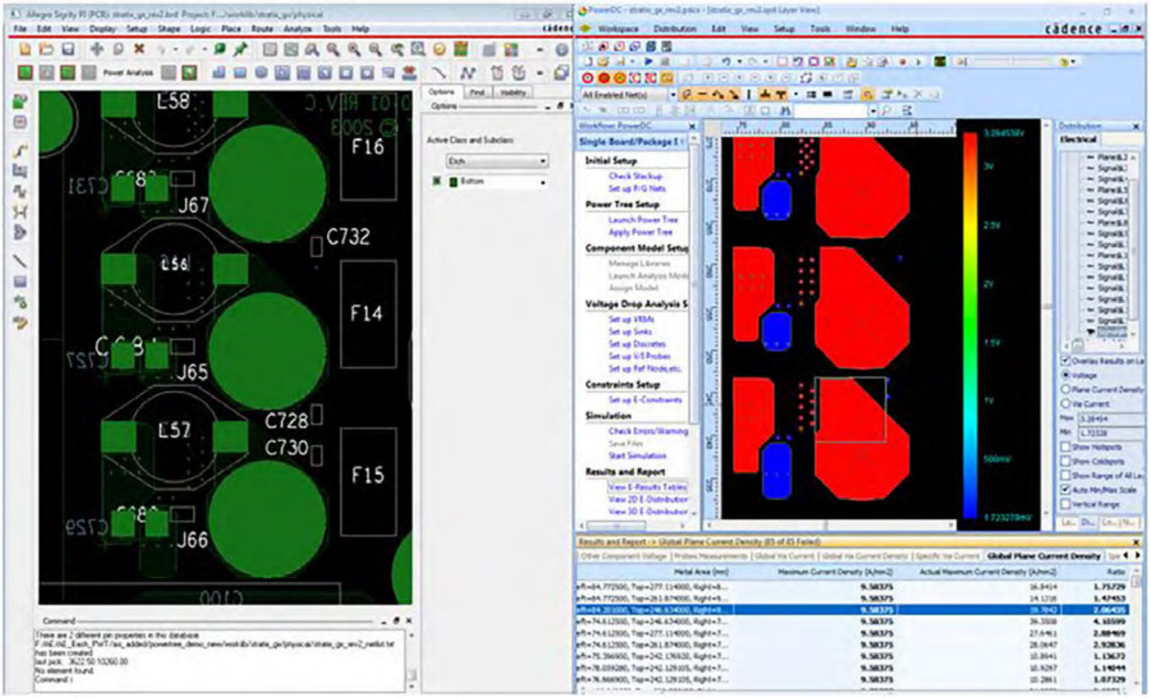


Figure 6. Cross-probing between PCB design and PCB analysis.

data, thus saving time and avoiding errors. The tool can also track the schematic changes for the power system, and check the effects of the changes made to the design. It supports both DC and AC analysis setup automation.

With PowerTree technology, the design engineer can access early visualization stages and can define the simulation criteria. This ensures that the layout engineer makes the correct layout changes and can pursue decap placement, so the production design meets the target requirements for each component.

Integrated Design/Analysis Environment

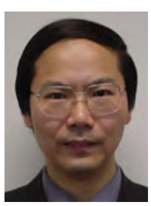
Using this integrated design and analysis model, users can debug the layout with side-by-side windows, showing analysis results for the specified layout. Using the cross-probing capability (Figure 6) and incremental layout update capabilities, layout editing is intuitive and goal-oriented.

If PI experts have already prepared the workspace, then PCB designers can do the batch mode analysis to get a HTML report and then use it to track all violations in layout environment. This helps PCB designs to quickly handle the obvious layout issues before the design is handed over to the PI engineers.

Summary

With the automated analysis in place, design and analysis data unified in one system, and using an integrated design and analysis environment, PCB designers can better understand PI issues from the early stages of the design, find the PI problems, and fix the layout during the layout design stage to avoid severe issues with power integrity. This allows hardware engineers, PCB designers, and PI experts work together smoothly and efficiently.

Streamlining the process results in the design delivered on time, on budget, and of a superior quality. **PCBDESIGN**



Dingru Xiao has more than 27 years of electrical and electronics industrial experience covering software, hardware and system development and testing, digital signal processing, etc. He joined Cadence Design Systems in 2002.

New Functionality Improves Designer's Productivity

by Barry Olney
IN-CIRCUIT DESIGN PTY LTD / AUSTRALIA

Productivity features not only save time-to-market, but they also curb frustration. One of the most common issues that managers and PCB designers face today is lack of time. Unlike money, time is a non-renewable resource. Once time has passed, it is gone forever. With that said, getting more work done, in the same period of time, by using the right tools results in higher profits. Add it all up and it equates to cost savings and improved productivity. Integrated software is the key to efficient PCB design.

One of the main details lacking in today's PCB design software is the flow of impedance control from design capture through to board fabrication. If the impedance of all the required technologies, used in the design, is determined up-front at the time of capture, the engineer's intent should be preserved and flow through to downstream tools. However, that scenario rarely happens. Many PCB designers simply select the trace width and spacing, that they typically use assuming all FR-4 is the same, and start routing, thinking that the fab shop can fix it later. This is due to their inability to define the requirements as they rely on their fab shop's feedback, on impedance control, after completion of the Gerbers. This is analogous to shutting the gate once the horse has bolted.

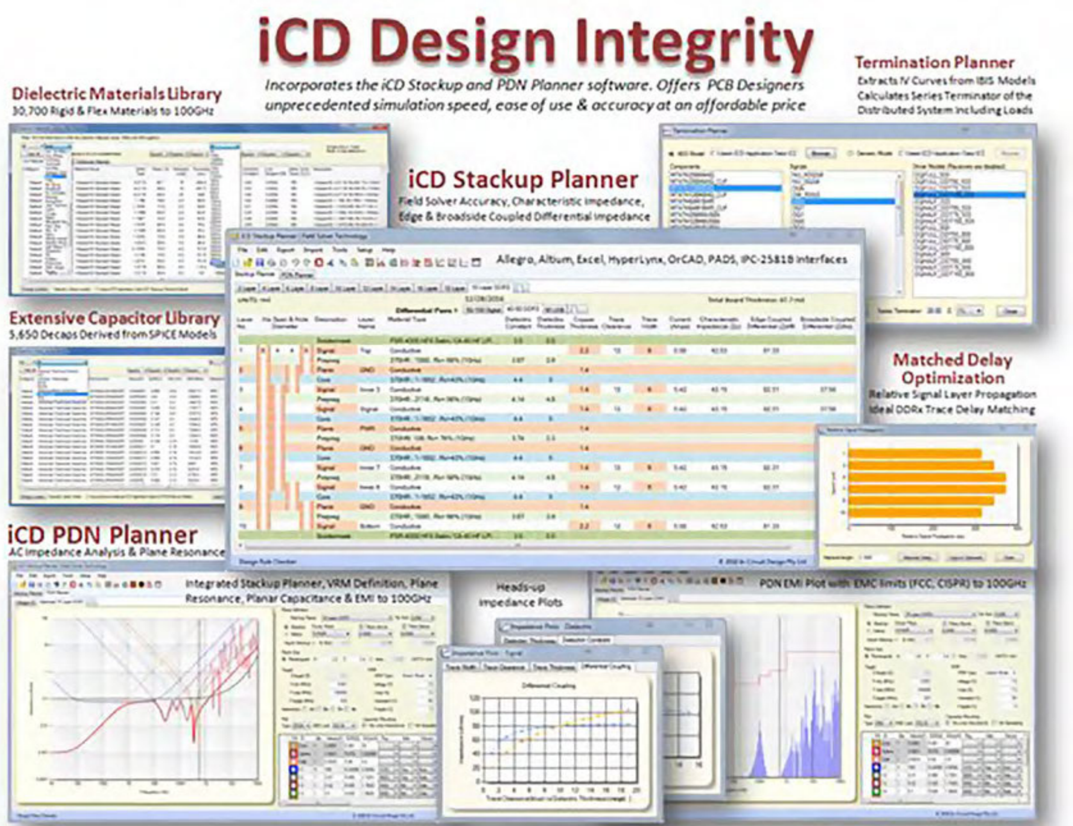


Figure 1: iCD Design Integrity productivity features.

I originally came up with the concept of an online impedance calculator way back in 1994 when I was working on the PCB layout and design for a new generation of SPARC 20 servers. We basically reformatted a Sun SPARC 20 pizza box motherboard to fit into a 5.25-inch drive slot. This was of course a tight exercise, but my idea was this: If I could physically fit all the required chips and connectors on the board, then it could be routed. This design required a 12-layer motherboard, and I used the IPC-317 "Design Guidelines for Electronic Packaging Utilizing High Speed Techniques" closed-loop equations to determine the signal layer impedance. The maximum CPU frequency was 200MHz, so there was plenty of margin.

To cut a long story short, after months of development, the project was completed and launched into the market. In 1995, the development team received an IEEE Engineering Excellence Award for the design. The server was subsequently licensed to a US company, jointly owned by Fujitsu and Sun Microsystems. Under license, the manufactured machine, a US-built SparcPlug Station, won the Best Performance category in the 1997 US AIM "Hot Iron" Awards.

During the development phase, as with all complex equations, I kept getting different results each time I manually calculated the impedance of a signal layer. Realizing that other designers have a similar problem, and a need to control the impedance of high-speed designs, we decided to write the code for an online impedance calculator based on the closed loop equations to simplify the process. This was first launched in 1995 and has developed, from its humble beginnings, into the iCD Stackup Planner, which today has very accurate field solver technology. Since then, we have had over 15,000 registered users globally. Of course, the product today has matured and is quite different from the original online tool, but our philosophy remains the same: a focus on simulation speed, ease of use and accuracy at an affordable price.

We have created a centralized, shared, impedance planning environment that connects materials, PDN analysis, stackup planning, signal integrity, PCB design and fabrication, consolidating the impedance control from schematic to fabrication. The impedance is planned pre-layout and flows through the design process to fabrication–achieving right first time design.

"We have created a centralized, shared, impedance planning environment that connects materials, PDN analysis, stackup planning, signal integrity, PCB design and fabrication, consolidating the impedance control from schematic to fabrication."

Bi-directional Interfaces were developed for the most commonly used EDA tools: Allegro, Altium Designer, HyperLynx, OrCAD, PADS, and more recently the new IPC-2581B format. The IPC-2581B format interface has now been thoroughly tested by the IPC-2581 Consortium. This new feature gives the iCD Stackup Planner the ability to import/export Cadence Allegro and OrCAD stackups and to import Altium Designer, PADS Pro, Xpedition and Zuken CR-8000 stackups.

This allows the designer to either extract the stackup from their layout tool into the Stackup Planner or create a new stackup from scratch. Materials can then be inserted from the 30,700 available rigid and flex materials to greatly increase the accuracy of impedance, allowing for multiple differential pair technologies on the one substrate. The completed stackup is then exported back to the layout tool and design rules are automatically created to match the routing requirements. An Excel fabrication drawing can also be automatically created to inform the fabricator of the stackup and materials required single ended, edge and broadside coupled differential pairs and microvia spans.

The iCD Design Integrity suite of tools incorporates both the Stackup and PDN Planner, plus a myriad of new functionality specifically

developed for high-speed PCB design as shown in Figure 1. Over the past year, we have added the following productivity features:

- Relative signal propagation with Matched Delay Optimization feature
- Termination Planner—IV curves extracted from IBIS models
- Heads-up impedance plots created by multiple field solver passes
- PDN EMI Plot with FCC, CISPR and VCCI EMC limits
- IPC-2581B format, bi-directional interface
- Dielectric materials library of 30,700 rigid and flexible materials up to 100GHz.
- Capacitor library of 5,650, with Samsung caps added

Today's high-speed interfaces simply cannot be modeled using a match-to-length methodology—an approach that conventional PCB design tools support. This is because of the tight timing required. A matched length of 2.3 inches for a DDR3/4 data lane, for instance, can produce up to 70ps delta, between signal layers, leaving the timing way outside the required setup and hold times.

Consider, for example, a DDR3 interface with eight byte lanes. Each byte lane has data signals, strobe signals, and mask signals. And each grouping of signals has its own set of signals with eight signals in a byte. The timing within each byte lane must be within ~30ps for the highest speed memory.

Signals propagate at the speed of light in free space. However, this speed varies dramatically depending on the surrounding dielectric materials. Each layer of a multilayer PCB can have a very different propagation speed. This is particularly important for the latest high-speed DDR3/4 memory devices. The new Matched Delay Optimization feature of the iCD Stackup

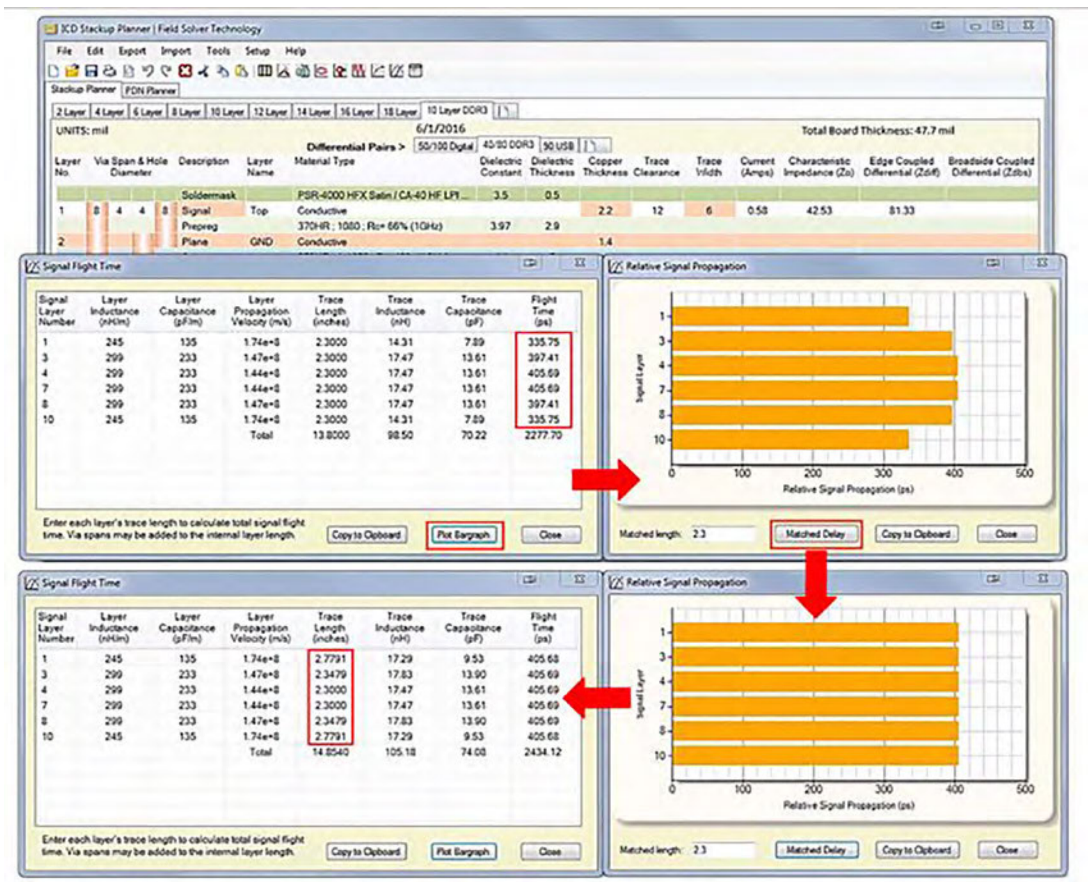


Figure 2: Matched delay optimization of each signal layer's relative flight time.

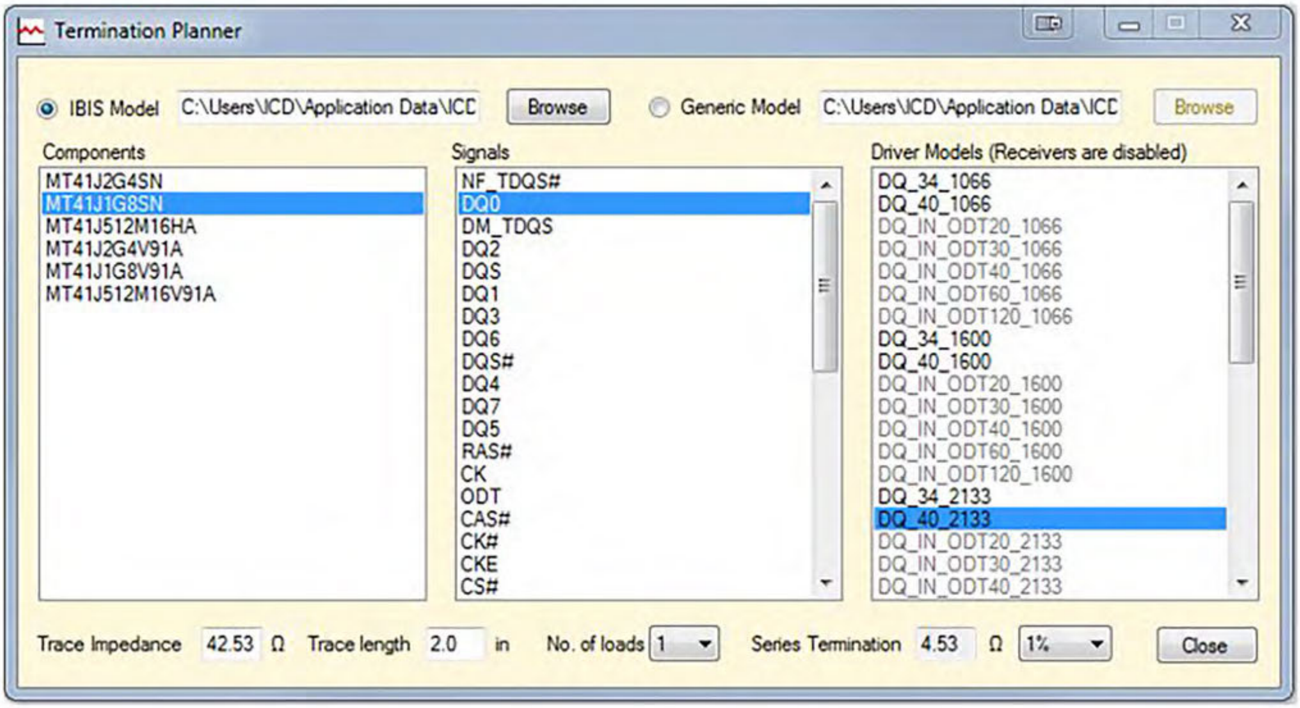


Figure 3: Matching a DDR3 driver IC to the transmission line.

Planner allows you to not only match the length of busses, but to take this one step further by automatically calculating the appropriate length required to match the delay exactly. The integrated field solver simulates the flight time, of each signal layer, to quickly give you the results you need to effectively route memory.

The relative signal propagation, of each signal layer, is displayed as a bar graph, once the matched length has been set (Figure 2). Selecting Matched Delay automatically optimizes the length, of each signal layer, to match the maximum delay. The users can then route the data lane to the exact delay, in their preferred design tool.

Also, it is one thing to perfectly match the delay of the transmission lines. But unfortunately, when using mainstream PCB layout software, one really has no idea what the driver impedance is, let alone the capability to match the driver to the impedance of the transmission line. The iCD Termination Planner addresses this issue.

Firstly, the attributes required to determine the source impedance of the driver, are extracted from an IBIS model IV curves. Then the required series termination resistance is calculated, based on a distributed system, to match the transmission line for the selected layer in the iCD Stackup Planner. If the IBIS model is not available (or produces an error) the user may use Generic Models to calculate an approximate series termination. Generic models include: typical DDRx, Display Port, ECL, HDMI, LVCMOS and LVTTL gates, Mini-LVDS, NAND Flash, PCI, SDRAM, HSTL and SSTL models.

The number of loads on the transmission line also has an effect on the required value of series termination; as the IC input inductance and capacitance tend to roll-off the signal rise time. This can be adjusted from 1–6 loads and automatically compensated for in the calculation.

Benefitting from 22 years of customer feedback and product development, the iCD Design Integrity software now provides numerous productivity tools for the high-speed PCB designer. The software was designed and built by PCB designers, specifically for PCB designers. We know what you need and what you want as we also do many real-world PCB designs and simulations ourselves. The tools are easy to use

and integrate with all the popular EDA tools to enhance your process efficiency. Beyond 2016, iCD will continue to develop new features to enable productivity gains as new methodologies and technologies arise. **PCBDESIGN**

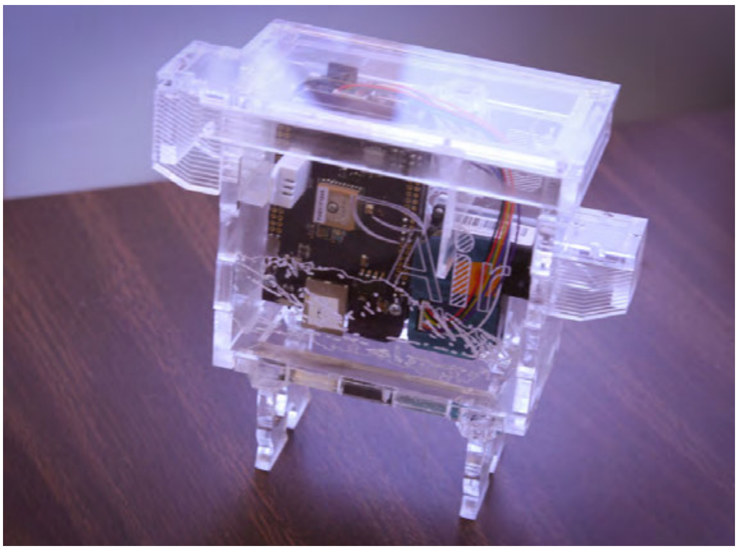
References

1. Barry Olney's Beyond Design column: Rock Steady Design.

2. A full list of manufacturers and materials available in the iCD Dielectric Materials Library is available here.

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

The Building Blocks for Bad Air



This season, assistant professor Kerry Kelly and associate professor (lecturer) Tony Butterfield have launched the AirU program for Salt Lake County high schools in which they visit science classrooms to talk about air quality and help students build a functional air pollution detector kit out of toy blocks and an inexpensive computer board. Beginning this month, the professors also will leave their own low-cost portable air-quality sensor with each classroom they visit so students can test and maintain them as well as collect data for researchers about pollution throughout the Salt Lake Valley.

"We started out with candy containers like Tic Tacs and other things and decided that wouldn't work," he says. "Then we realized [toy blocks] were the way to go because they [students] could build them into any shape they want."

So far, the team has built 15 kits. The professors use a small fog machine to demonstrate how the sensor works.

In addition to the toy-block sensors, Kelly, Butterfield and students from the University of Utah's electrical and computer engineering department also designed and created low-cost portable air pollution monitors that they will begin leaving in each classroom they visit. These monitors — about the size of a small box of tissues — are similarly powered by a low-cost computer board but are higher-grade than the toy-block monitors and have sensors that can specifically detect and measure particulate matter, temperature, humidity, carbon monoxide and nitrogen dioxide. They also are equipped with GPS and will be connected to the Internet wirelessly. The professors will ask the students to maintain the sensors and help test their reliability.

Ultimately, Butterfield and Kelly want to distribute as many as 50 of their research-grade sensors in schools around Salt Lake County to track pollution throughout the valley. They also want to eventually sell the kits for personal home use once they get the cost down.

Design Tools of Tomorrow: A Real "Marvel"

by Tim Haag
INTERCEPT TECHNOLOGY

You've probably noticed that I sometime discuss movies in this column to make a larger point about PCB design. Yes, I confess that I enjoy a bucket of popcorn and a couple of hours wasted in the local cinema just as much as the next guy. One of the movies that I have enjoyed the most over the past several years is the first "Iron Man" from 2008. Lately, the whole Marvel cinematic universe has gotten wild, with gods, monsters and space aliens.

This isn't a bad thing but they have gotten away from the purity of engineering that made the first Iron Man movie so good. Let's face it; the lead character, Tony Stark, is the engineer's engineer. In that first movie, he designed the Iron Man suits right in front of us on the big screen, and he used some awesome-looking design tools in his lab. That got me to wondering: What would our world of design be like if we had some of that magic?

I think everyone would agree that advances in system resources, improvements in data specifications, and more automation would be welcome enhancements. But I'm trying to get past the obvious incremental advances that we normally see in the next feature release of our favorite CAD tool du jour. Instead, I'm looking for those improvements that we can only imagine or dream about today. So, if we're going to dream, then let's dream big. Just how would you spec out the design system of tomorrow?

It would seem to me that the most obvious improvements would be in the user interface. A common complaint that I hear from designers is being constrained by having to use a mouse. One user even suggested that instead of a mouse that they would like to see the application follow their eye movements instead. Would that be helpful to you, being able to use your eyes to control the place and route of your design in-

stead of mousing around? I'm not so sure. But if eye-controlled movements were implemented, then there would have to be other enhancements as well to complement it.

Voice recognized commands would be a very helpful enhancement to have. In that first "Iron Man" movie, Tony Stark gave commands with his voice, and his design system could understand and respond to that input. Imagine if you were designing your board and the routing followed your eye movements, and you could tell the system through voice commands when to drill a via, what size via to use, and what layer to end up on. Or if that is too much, imagine if you could at least use your voice to input strings of text for a drawing or a schematic. Since our smartphones can already do this, why shouldn't our layout tools be able to do it as well?

Of course, you would probably always want the ability to input information directly with your hands somehow, and in that movie you see a virtual keyboard in use quite often. Since the need for direct system-level communication with our CAD tools will likely never go away, a keyboard seems like a logical way to do this. But what if we had a virtual keyboard instead of the mechanical keyboard that most of us are accustomed to now? That could be very helpful; the keyboard could change its input buttons depending on the needs of the design (language, symbols, tool commands), and change the contour of the physical keyboard to provide the best ergonomic interface for the user.

Beyond that, just imagine if you could interact with your design as a hologram floating in front of you the way Tony Stark did in the movie. Wouldn't it be amazing if you could pick a section on your holographic design with your hands and expand it to the point where you could peer into it, spin it around, and manipulate it as you desired? Want to push a trace down to a different layer? Just give it a nudge in the right direction and the holographic display changes it to the next layer. Don't like the way a certain area fill looks? Then just grab it with your fingers and pull it out and throw it into the virtual garbage can. Want to split off a portion of circuitry to put into a reuse block for use in another design? Then grab it and pull it over to the side and tell your system to save that portion under a new file name.

Figure 1: Could the PCB designer of the future use a holographic CAD tool similar to the one used by Tony Stark in the first "Iron Man" movie?

The system interacted with Tony Stark through a voice interface named Jarvis. This interaction was often laden with Jarvis' sarcasm and, to tell the truth, I'm not too sure that I would want my CAD system questioning what I do and coming up with suggestions that I don't really want. But then again, maybe my pride needs an adjustment here because there could be some real advantages to this. Imagine the possibilities if we could verbally interact with our DRCs, and after informing us of a net clearance problem our design system could suggest a range of solutions based on our design specifi-

cations and requirements. The question would then be, "Are you the kind of person who would argue with your design system?" If so, then who would win?

Farfetched thinking, you say? Perhaps. But this wouldn't be the first time that the entertainment industry has driven advances in technology. Think about the popularity of cell phones, which rose dramatically many years ago because of the introduction of the flip phone. The consumer market was ready for that kind of technology because they had already seen it in use on "Star Trek" with their hand-held communicators. The movie "2001: A Space Odyssey" showed the astronauts using flat portable viewing devices that they could use for reading news and reports just as we do with our tablets today.

But don't downplay the time-tested method that we have now of driving technology enhancements by user requests. Your input will help transition today's dreams into tomorrow's realities. So, keep logging those enhancement requests and bug reports with your CAD vendor, stay involved with your user groups, and do all you can to make sure your voices are heard.

Early on in my design career I was laying out boards with tape and dollies on a drafting table. I then digitized those tape-up designs, and eventually I started using some of the early CAD systems that often needed more hands-on help than they were worth. How many of you remember working on a monochrome monitor? To display different layers, you had to differentiate them with different fill patterns because the monitor only had one color. I had no idea in those days that advanced design automation would one day be available to help with my work. Back then, if you wanted to make a change to a circuit, you would have to manually recreate it. Now what was tedious work can be done in a fraction of the time using CAD tools with automated dynamic editors and externally connected simulators.

If we had been able to see back then what kind of tools we would have to work with today, it would have been a mind-blowing experience. And now, seeing just how far we've come over the years with design tool technology, we naturally wonder, "What's next?"

Yesterday we knew with absolute certainty that laying out a board required tape and dollies on a drafting table, and today we know with absolute certainty that laying out a board requires the assistance of advance automation in our CAD tools. I wonder what we'll know with absolute certainty tomorrow.

Perhaps, at the end of the work day in the future, PCB designers will close up shop with a conversation like this:

"Will that be all, Mr. Stark?"

"Yes, Jarvis, goodnight." **PCBDESIGN**

Tim Haag is manager of customer support and training at Intercept Technology.

Advances in AI Will Help Machines Understand Human Thoughts

The technological complexities inherent in developing perceptions for machines, as well as the verification and validation of these tools, have been eased to a large extent by the advancements of AI technology.

"With Brain Computer Interface (BCI), AI can power future machines to understand human thoughts and emotions, even without physical or vocal communication," noted Frost & Sullivan TechVision Senior Research Analyst Debarun Guha Thakurta.

Leveraging convergent ideas with technologies, AI can open up new horizons in groundbreaking applications.

"AI is being widely used across sectors like healthcare, automotive, banking and finance, and aerospace and defense due to its ability to understand patterns in data and make highly accurate predictions and simulations," noted Guha Thakurta.

New ON IPC EDGE

IPC Essentials & IPC ESD Certification

IPC ESSENTIALS

IPC EDGE

- Simple to use online portal
- Convenient for you, open 24/7
- Keeps track of your training transcripts

IPC ESD Certification for Trainers (CET), and IPC ESD Certification for Operators (ECO)

ESD certification control for your company

- Fills a gap in the industry, developed by IPC and EOS/ESD Association Inc.
- Operators are trained by certified ESD trainers
- Certification complies with ANSI/ESD S20.20
- Verifiable and trackable certification record

IPC Essentials

IPC Essentials delivers

- Enhanced knowledge of IPC Standards and their applications
- Modules covered — standards development, impact on the electronics industry, standards training
- Increases knowledge and improves productivity for line operators, managers, executives and all those impacted by IPC standards
- Improvement to the certification programs
- A program requirement for CIT and MIT professionals

Visit www.ipc.org/edge for more details.

IPC EDGE Online Education and Training for the Electronics Industry

IPC EDGE

PCB007 Highlights

CES 2017: Disruptive Technologies

Those of you that have read my previous columns covering CES 2017 know that at recent CES shows I have seen many drones, autonomous cars, IoT devices, robots, and other items ranging from robots who stand in for your doctor to smart trash cans who tell you via Alexa, Google, or soon, Cortana, on your own computer, that since you have thrown away two empty milk cartons in the last few days it may be time to order more milk.

CES 2017: Press Day, Launch.It and Showstoppers

Before the show opens, CES provides two days and evenings not open to the general attendees, to enable companies, large and small, an opportunity to present their new offerings to the press. These opportunities range from small meetings with individual members of the trade press to huge events.

Flex Talk: A Glimpse into PCB Sales

Summarizing the feedback from both customers and manufacturers, the most successful PCB salespeople are organized, take a genuine interest in their customers' needs and business challenges, and have a better than average understanding of the PCB industry.

Weiner's World

Here are a few thoughts for the New Year and beyond from a long-term colleague and friend of the past half century, Harvey Miller, electronics market researcher, consultant and founder of Fabfile Online.

Emmy Ross Discusses the New I-Connect007 E-Book Series

One thing that is long overdue in our industry is a series of guidebooks focused on helping companies with all their needs, from qualifications like AS9100 and Mil-P-31032 to various technologies, heavy copper, rigid-flex and microvias. I-Connect007 is now providing our industry with an entire series of these guidebooks, starting with the first, "The Printed Circuits Buyers Guide to… AS9100 Certification."

On Location at HKPCA: IPC's Mitchell and Carmichael on Asia, the Global Industry Outlook, and Trump

While in Shenzhen, China, recently, with the PCB industry's biggest trade show as a backdrop, Publisher Barry Matties sat down once again to interview IPC President John Mitchell and IPC Asia President Phil Carmichael. This time they discussed trends they're seeing in the Asia-Pacific region and what a Trump presidency means to the industry.

Top 10 Most-Read PCB News Stories of 2016

As the year ends, look back at the industry's highs and lows. The editors at I-Connect007 have compiled a list of the top 10 most-read news stories from the pages of PCB007. Join us for a look back at the most popular news highlights of 2016.

Punching Out! Types of Company Buyers in the PCB and EMS Sectors

Mergers and acquisitions in the U.S. PCB sector have been in the news recently, with at least 12 deals completed over the past year, and several more in the works. In contrast, the EMS sector has been relatively quiet, but that may change now that the presidential election is over.

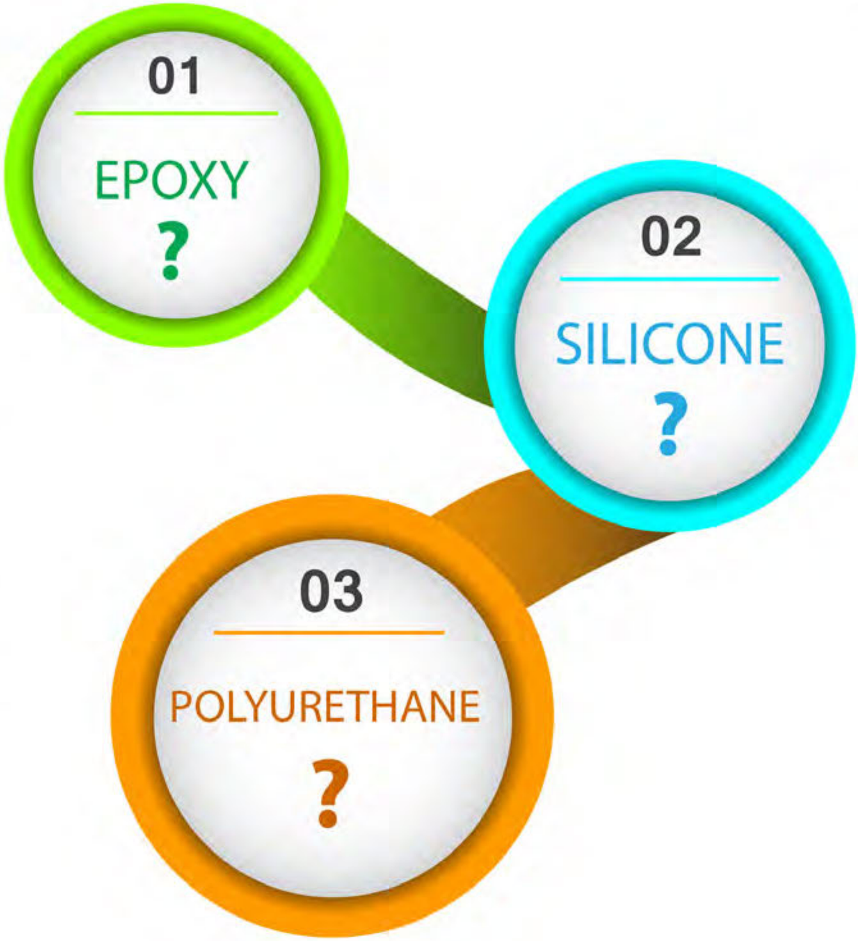
PCB Maker Unimicron Developing SLP Products

Unimicron Technology has set aside a capex budget of NT$4.946 billion ($153.19 million) for 2017 to finance the development of substrate-like PCB (SLP) products and ramp up the production capacity of HDI boards and automobile boards.

All About Flex: Are Manufacturing Companies Susceptible to Ransomware?

Every business (and every individual) needs to pay attention to cyber security. There are many sophisticated hackers throughout the world looking for ways to access or corrupt systems. While manufacturing companies have not been a common target, there are certainly risks that need to be considered.

SELECTING THE RIGHT RESIN FOR THE JOB



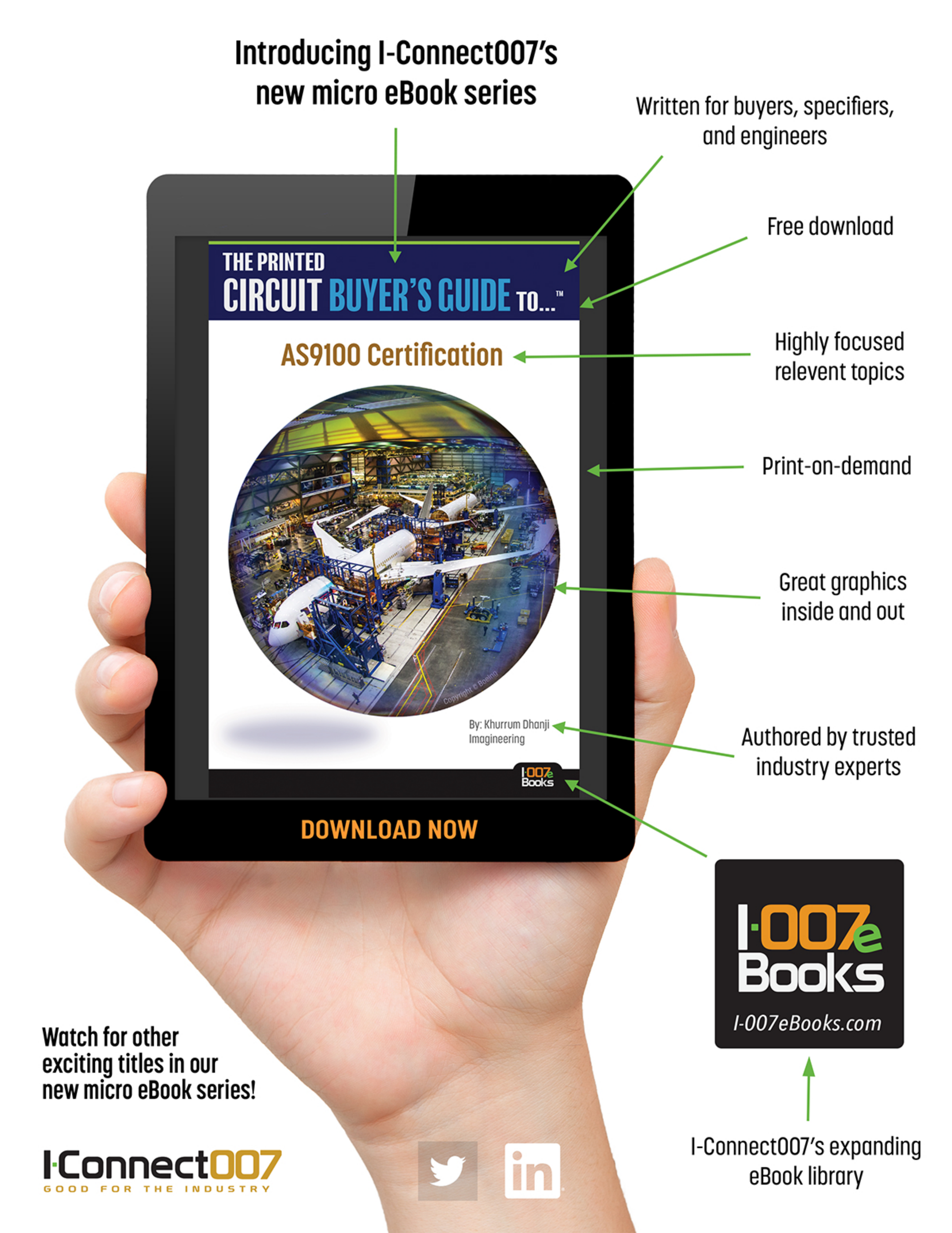
by Alistair Little
ELECTROLUBE

So far in these columns, we've looked at the various reasons for circuit potting and encapsulation using resins, outlined the various resin properties and how they relate to particular types of circuit protection, seen how to apply these resins in a production environment by deciding on which mixing and application techniques are appropriate to specific production needs, and finally, how best to achieve a satisfactory cure.

But, I hear you ask, which type of resin is best for my project and production methods? How are my choices affected by the physical constraints of the board or area to be covered? What resins are suitable for rework? These and other issues I plan to cover in this column.

Basically, you have a choice of three main types of resin: epoxy, polyurethane and silicone. Each has distinctive properties and your choice will be guided by how these properties can be exploited to meet your particular production circumstances and application needs. Let's first take a look at production volumes. Are you prototyping, delivering a short production run, or manufacturing on a much larger scale? Your answers to these questions will determine whether you use manual or machine-based application methods, and this will pose a further question: Which resin types are compatible with these application techniques?

For example, if you are potting prototypes or small-volume runs you will most likely use manual methods, while much larger-volume runs will best be served by automated, machine-based mixing and dispensing systems. Should you be looking to work up from the prototype



Introducing I-Connect007's new micro eBook series
Written for buyers, specifiers, and engineers
Free download
Highly focused relevent topics
Print-on-demand
Great graphics inside and out
Authored by trusted industry experts
THE PRINTED CIRCUIT BUYER'S GUIDE TO...™
AS9100 Certification
Copyright © Boeing
By: Khurrum Dhanji
Imagineering
I-007e Books
DOWNLOAD NOW
I-007e Books
I-007eBooks.com
I-Connect007's expanding eBook library
Watch for other exciting titles in our new micro eBook series!
I-Connect007
GOOD FOR THE INDUSTRY

stage to a full production run, then the differences between manual and machine mixing and dispensing will have to be taken into account when choosing your resin.

When mixing and dispensing manually, a longer useable life and gel time will generally be required compared with machine mixing, which is a faster process that doesn't necessarily need these properties. The process could involve the use of two different resins with different mixed properties, but having similar cured properties in order to take full advantage of the two different mixing conditions. Useable life is defined as the time that the mixed resin can flow and remain just workable; the gel time is the interval between mixing and the point at which the resin has just set and cannot flow. In the latter state, the resin is still relatively soft and can often be reshaped by applying slight pressure. However, note that the larger the volume of mixed resin, the shorter the useable life and gel time.

"If a rigid to semi-rigid encapsulation is acceptable, then the slower curing rate of a two-part epoxy resin would provide the desired longer useable life and gel time for manual mixing."

If a rigid to semi-rigid encapsulation is acceptable, then the slower curing rate of a two-part epoxy resin would provide the desired longer useable life and gel time for manual mixing. Epoxy resins are also suitable for machine dispensing but caution should be exercised if the resin contains abrasive fillers as these will accelerate the wear of dispensing equipment. To mitigate this problem, epoxy resins are available without fillers. Should a more flexible cured finish be required, then polyurethane or silicone based resins would be suitable alternatives to epoxies.

Two-part polyurethane resins tend to have a lower exotherm (less heat is generated) compared to epoxy resins, which may be advantageous when potting delicate components. While silicone resins tend to be less popular than either epoxy or polyurethane resins, they are highly flexible on cure and they tolerate continuous high operating temperatures—often in excess of 180°C.

When the resin is mixed, next comes the question: How do you want it to be applied? If the parts of the circuit to be coated are defined, discrete areas, then generally either a dam-and-fill system is employed to isolate the areas that need to be potted or a more thixotropic resin can be used. Some single-part epoxies and especially silicone resins have these advantageous thixotropic properties.

On the other hand, if an entire board needs to be covered, then resins with good flow characteristics should be used as the resin will be designed to flow from a dispensing point to cover the board. Most two-part epoxy resins—even those containing fillers—are not only easily mixed but can flow between components and devices with limited spacing.

But how much resin should you apply? As a general rule of thumb, the amount of resin to be applied must be sufficient to cover the top of the highest component of the board, and the thickness of the resin layer must provide the desired level of protection. While most customers will determine the minimum thickness of resin layers for their particular applications by trial and error, the relevant technical data sheet will provide good guidance, and consulting your supplier will often help resolve a problem.

A note here about colour: Optically clear resins may be desirable for applications such as LED lighting fixtures, as the cured resin will obviously need to retain its clarity for the life of the unit. Colourless resins are also useful for prototyping applications, as the encapsulated components are easily viewed during and after environmental and mechanical testing. On the other hand, coloured or opaque potting and encapsulation resins conceal what lies beneath the encapsulant surface, providing an effective foil against counterfeiters or those wishing to copy a circuit layout, helping you to protect your intellectual property.

On the subject of prototype boards and systems, it is often desirable to be able to inspect the board and identify and remove failed components for inspection and replacement. There is a range of resins that are more suitable for use when reworking or repair may be necessary, as well as a range of products designed to allow the removal of cured resin from the board. Both polyurethane and silicone resins are more easily removed for rework purposes, and special solvents are available to assist with this process.

As previously mentioned, technical data sheets can be a great help when you embark on a new production schedule with new components and resins, but if you foresee any problems with matching resin types to your production procedures that are not easily resolved by studying the literature, be sure to contact your supplier's technical support team for further advice.

Next time, I shall be taking an in-depth look at some of the most frequently asked questions we get asked as resin experts and exploring options in response to these enquiries. **PCBDESIGN**

Alistair Little is technical director for Electrolube's Resins Division.

Looking Inside Materials the Smart Way

Just as cars in Germany need to be inspected every two years to ensure they are safe, other safety-critical objects – turbines, generators or high-pressure containers, for example – have to be examined regularly as well. This is especially important when the materials and products used are pushed to their outermost performance limits in order to increase economic efficiency. To carry out these assessments, inspectors receive a printout map of the factory grounds to help them find their way. Once they locate the structure to be inspected – say, a high-pressure container – they inspect it with help from a sensor. The difficulty is that they have to inspect the entire surface. But which parts have they already evaluated with the sensor, and what still needs to be done?

Support is on the way: the Fraunhofer Institute for Nondestructive Testing IZFP has developed 3D SmartInspect for intelligent inspection and quality control. In day-to-day work, the process would look like this: inspectors wear augmented reality (AR) glasses, though the system works with a tablet PC or a smartphone too. They view the object to be examined – let's take the high-pressure container again – through the glasses.

Once all data has been acquired, inspectors can see the results immediately on their AR glasses. Areas with any kind of a defect – a cavity where it doesn't belong, or corrosion – appear red on the display. Inspectors can immediately indicate where the repair team needs to intervene, either by using chalk on the actual object or via digital means.

The Benefits of Employing a Standard Grid Pitch in Design

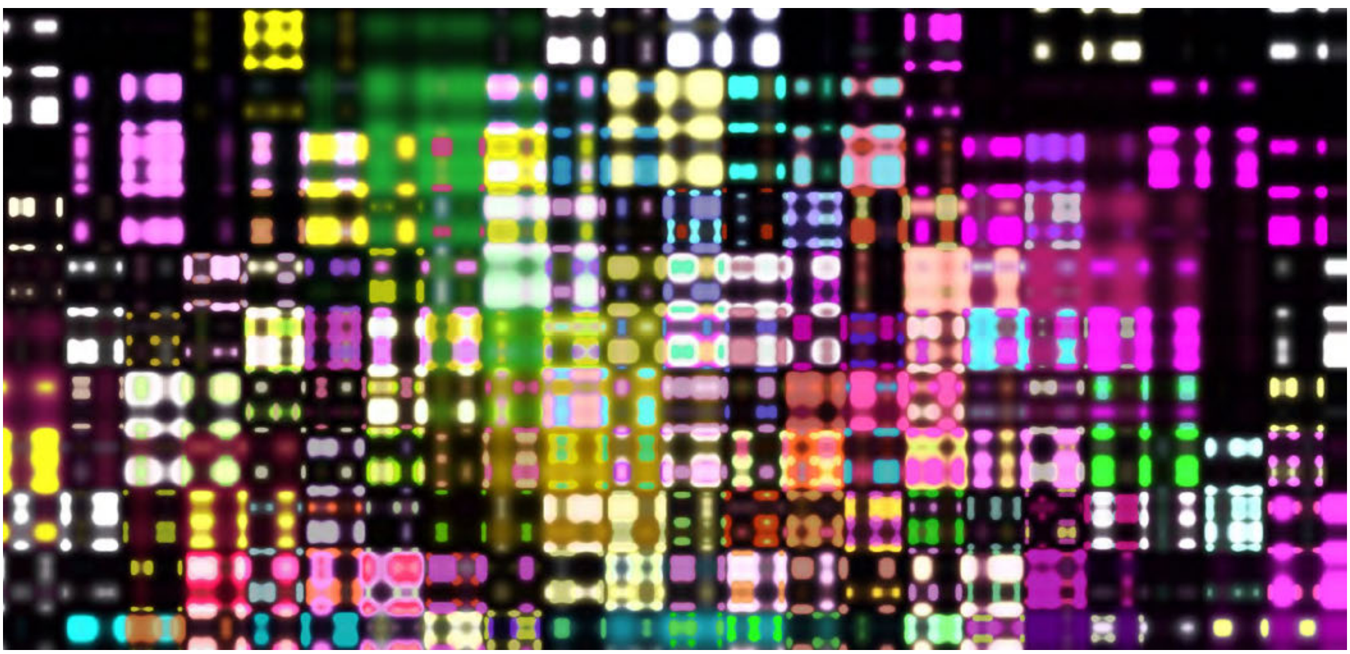
by Joe Fjelstad
VERDANT ELECTRONICS

Since the time of their introduction, area array packaging such as BGAs, area array CSPs, and wafer-level packages have been extolled for their ability to simultaneously improve product performance and assembly yield while reducing overall system cost. Because of these important advantages, the industry was enticed to adapt to area array packaging. The benefits of area array interconnection became abundantly clear when it was realized that the previous generation of packaging solutions based on peripherally leaded packaging concepts would not carry the electronics industry into the future.

As peripherally leaded packages grew in size to support increasing pin counts required by more highly integrated ICs, the packages began sapping device performance while consuming more PCB real estate. Area array packages were promptly recognized as a highly promising solution however there were lots of unanswered questions relative to manufacturing and inspection of these new devices. There was some knowledge but it was limited to OEMs who had embraced area array methods for internal designs.

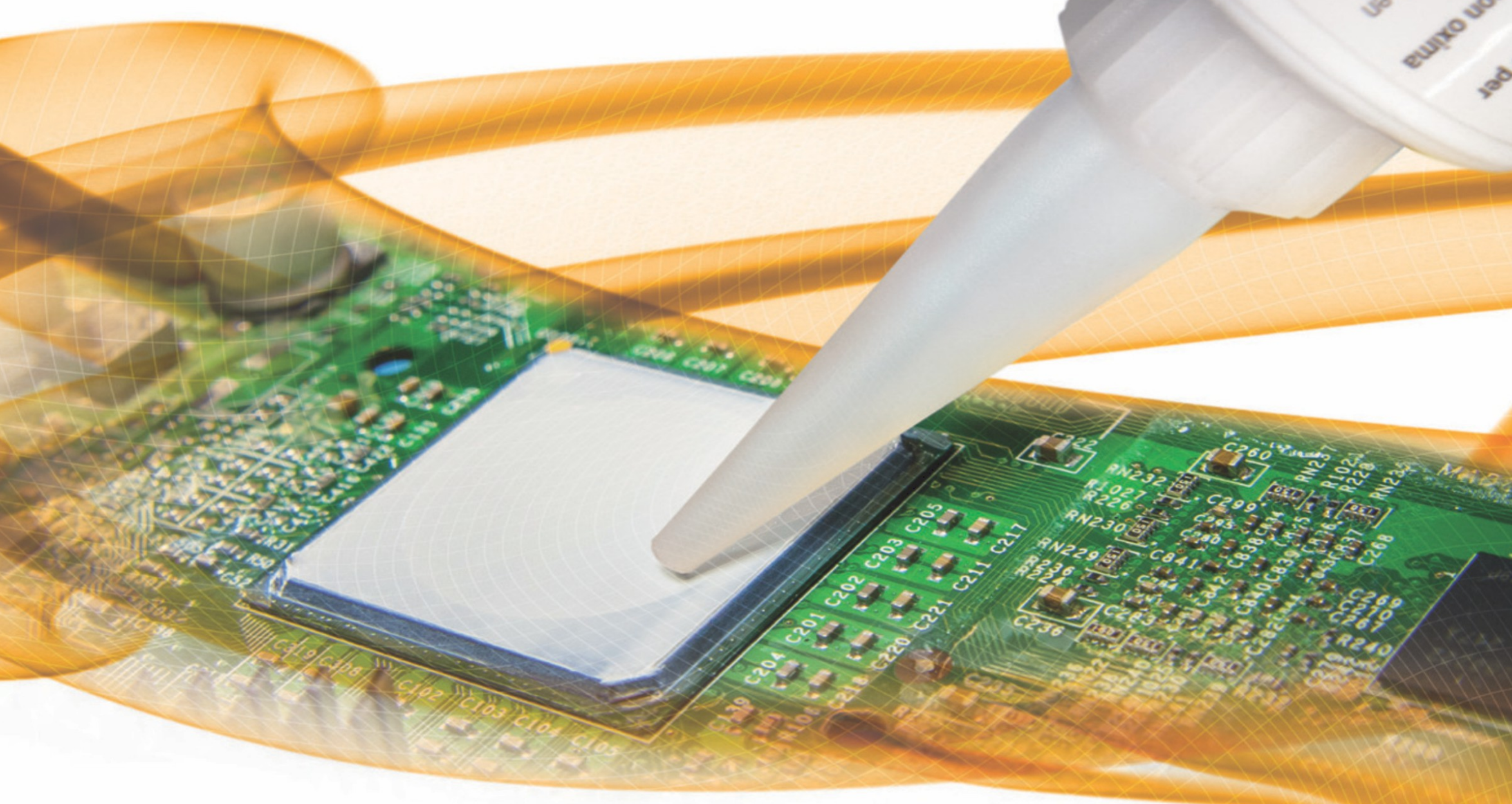
The industry at large needed to jump on the learning curve and overcome its fear of the unknown. One of the most vexing concerns at the time (an arguably still today) is that terminations beneath the area array package were un-seeable. Given the fact that then, as today, solder joints were a major cause of failure, there was much consternation over the quality of the joints. New equipment and methods, such as x-ray inspection, were applied and the industry made rapid progress in developing materials and processes that assuaged those fears and allowed the industry to continue to design and manufacture ever improved products at lower costs.

However, in the industry's rush to take immediate satisfaction of the benefits of area array packaging, it collectively failed to consider some highly significant overall benefits made possible



APEX EXPO IPC 2017

SAN DIEGO, CONVENTION CENTER, USA
FEBRUARY 14-16, 2017 - **Stands 525 & 527**



Thermal management solutions that perform when the heat is on

With a consumer requirement for ever-more diminutive devices and an expectation of improved efficiency and power, effective thermal management materials have become an increasingly essential part of product development.

From bonding and non-bonding thermal interface materials, to thermally conductive resins, our solutions offer the ultimate level of protection and heat dissipation.

With an expansive product range and a strong emphasis on research and collaboration, we provide a complete electro-chemical solution to the world's leading manufacturers across a variety of industries.

Isn't it time you discovered how Electrolube can serve you? Simply call, or visit our website.

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

Tel: 888-501-9203

www.electrolube.com

ELECTROLUBE
THE SOLUTIONS PEOPLE

Electronic & General Purpose Cleaning	Conformal Coatings	Encapsulation Resins	Thermal Management Solutions	Contact Lubricants	Maintenance & Service Aids

by area array packaging—the potential to design assemblies with every component having termination that could land on a standard grid.

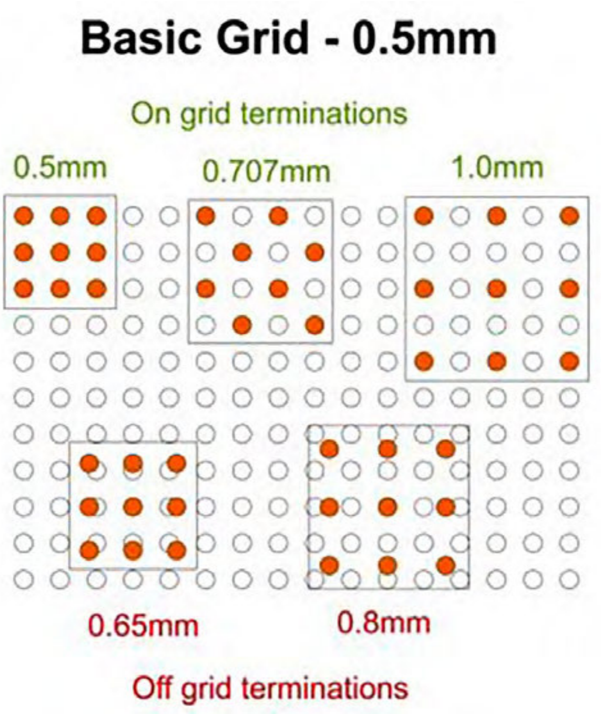


Figure 1: Basic common grid.

It is arguable that this is one of the most important and still largely ignored benefits offered by area array technology. It is a benefit that provided the opportunity to make electronics industry more coherent than it has ever been. The simple truth is that great benefit can be achieved by establishing a fundamental base grid pitch for all interconnections. This will allow for the creation of an underlying standard that will provide a much clearer view for the future. The logic is compelling that a single standard grid pitch for interconnection will allow for all electronic elements of an electronic assembly to be manufactured, tested and assembled in an almost snapped-together fashion, something akin to the children's construction toy Lego. Figure 1 illustrates how the termination on a common grid works.

Stepping back in time as alluded to earlier, coherent integration of electronic elements based on a standard grid is not really a new idea. IBM used the method internally with great success for many years with its through-hole interconnected devices. They even mass-produced PCBs with thousands of holes on grid suitable for personalization when and where required. The standard grid was also used by the rest of the industry and it was based on a 100-mil grid

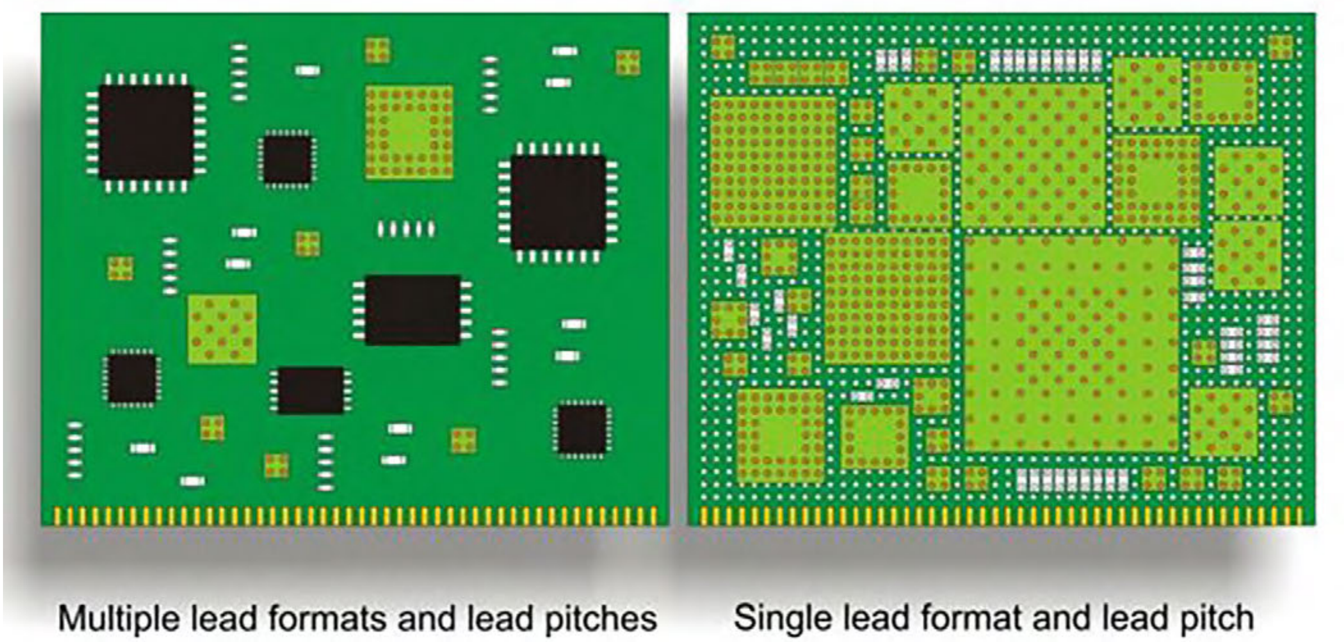


Figure 2: A variety of lead pitches shown at left, compared to the common grid on the right.

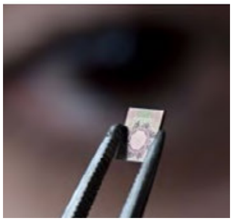
pitch. In fact, 100 mils was the standard lead pitch used for nearly all electronic components of that era, and especially dual in line (DIP) packages, and designing PCBs was an almost perfunctory task. However, as through-hole interconnection technology yielded primacy to surface mount technology a change took place. Lead pitches were based on yields in PCB board manufacture and assembly, causing the formula to be tossed out, and the 80% Rule was established, resulting in numerous lead pitches (e.g., 1.25, 1.0, 0.8, 0.65, 0.5, 0.4, etc.). This significantly challenged and challenges the design process compared to terminations on a common grid, as shown in Figure 2.

Now we collectively have opportunity to reconstitute the "magic formula" that worked so well in the early years and make it available to the entire global electronics community. Industry adoption of a standard grid does not preempt manufacturers wishing to maintain exclusivity from making "Rolls-Royce" electronic systems based on unique design rules and manufacturing concepts the customers might seek out for their products where cost is not a concern. Rather the concept put forth here is for all those who really just want and need "basic transportation."

Interestingly, should the concept of standards-based electronic assembly design be broadly adopted as logic (at least this individual's logic) dictates it should be, we might well see a day when the lowest-cost and the highest-performing electronics are one and the same. That will be the day when we have collectively stripped out all of the unnecessary elements from electronic design, thus reducing manufacturing cost to the lowest limit. Those who feel threatened by standards and those with rugged individualist attitudes may slow the adaptation of a standard grid concept, but they cannot diminish the intrinsic and compelling, if hidden, logic that a standard fundamental grid pitch has to offer. **PCBDESIGN**

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

Device Could Revolutionize Scanning, Spectroscopy and Wireless Communication



Your doctor waves a hand-held scanner over your body and gets detailed, high-resolution images of your internal organs and tissues. The physician then sends gigabytes of data instantly to a remote server and just as rapidly receives information to make a diagnosis.

Integrated circuit researchers at the University of California, Irvine have created a silicon microchip-based component that could make these and many other actions possible.

Known as a "radiator," the tiny gadget emits millimeter-wave signals in the G band (110 to 300 gigahertz). Most transmitters now generate linearly polarized signals, which can get "lost" when antennas and receivers are out of alignment. Emissions from one of the UCI radiators, if you could see them, would appear as tiny spinning tornados. Beams of this shape are particularly effective at penetrating solid objects and providing detailed pictures of what's inside.

But the new radiator can do a lot more than facilitate scanning and imaging. According to Heydari, it could be the key that unlocks millimeter-wave transmission as part of the fifth-generation wireless standard now in development. In addition, the tiny yet powerful chips can be embedded virtually anywhere. The Internet of Things will rely heavily on machines, buildings and other infrastructure being equipped with sensors and antennae. Driverless vehicles will only be possible if cars and trucks can detect each other.

"By using this millimeter-wave technology, cars all of a sudden become super-smart processing systems," Heydari said.

MilAero007 Highlights

[Supplier Spotlight: American Standard Circuit's AS9100 Journey]

I had the privilege of working with American Standard Circuits (ASC) over the past year with their successful pursuit of AS9100 quality management system certification. The company has chosen to share their approach, lessons learned and benefits gained by going through this formidable process.

[Aerospace Robotics Market to Hit $4.54B by 2022]

The market is projected to grow from $1.81 billion in 2016 to $4.54 billion by 2022, at a CAGR of 16.55% during the forecast period.

[Sparton and Ultra Electronics JV Inks $30.3M US Navy Sonobuoy Contract]

Sparton Corp. and Ultra Electronics Holdings plc have been awarded subcontracts valued at $30.3 million to their ERAPSCO joint venture, for the manufacture of sonobuoys for the United States Navy.

[Tim's Takeaways: 'Sparks' to the Rescue in RF Design]

Just like the early days of radio where Sparks the radio specialist was in demand to get the job done, we now need RF specialists to work together with electrical engineers to create the intricate designs required for RF circuits. You are now Sparks, the go-to specialist who will take care of RF design business.

[FTG Receives New Long Term Agreement Worth Over $12M]

Firan Technology Group Corporation has been awarded a new three-year long-term agreement (LTA) from one of the leading global OEMs supporting the aerospace market.

[NASA Aeronautics in 2016: This is the Story]

We are in the midst of an historic new era of X-planes research, including the continuing development of trailblazing green aviation technologies, and ever-expanding base testing of new air traffic management technologies.

[PCi Purchases Keyence IM-6225 Instant Dimensional Measurement System]

Rigid-flex circuit board manufacturer Printed Circuits Inc. has purchased an IM-6225 instant measurement system from Keyence.

[Aviation Industry to go Faster with Real-time Data Analytics]

The intensity of competition within the global commercial avionics systems market is expected to increase strongly over the coming years. Improvements in economy and the growing appeal and ease of air travel have generated a high buzz for all players in the global commercial avionics systems market and they are expected to ramp up development rates to match the demand.

[Declining Defense Spending by Developed Economies Emerged as Major Cause of Concern]

Only a handful of players dominated the global thermal imaging market in 2012, finds Transparency Market Research (TMR) in its latest report. Prominent companies such as DRS Technologies Inc., BAE Systems Inc., Raytheon Company, and FLIR Systems Inc., held dominance in the global market with a share of 75% in 2012.

[Global Nanosatellite and Microsatellite Market Report]

"The U.S. has continued offering lucrative opportunities for players operating in the global nanosatellites and microsatellites market. With an increasing number of companies looking to expand their footprint in the nation, the U.S. is likely to remain at the fore of the global market through forecast period," said a lead TMR analyst.

NAMM 2017: Bigger and Better

by Dick Crowe
BURKLE USA

The annual NAMM show (National Association of Music Merchants) in Anaheim, California is arguably the largest show of its kind. NAMM is to music what the CES Show is to consumer electronics, or the Auto Show is to cars. And electronics plays a major role in automotive and music these days.

NAMM, as reported last year, is a loud and diverse show. My guess is that there are well over 1,000 guitars on display along with all sorts of other musical instruments. I attend NAMM for two reasons; the first is my selfish interest in guitars and the second is to see how electronics also continues to impact this business.

It really is a giant show; not as big as CES, but far bigger than any show in the electronics manufacturing industry. When I first hit the show floor, my first thought was, "Wow, how do I bring that guitar home?" If you're a musician with a "day job" in the electronics industry, NAMM is the place to be. Music and electronics continue to interact. Amplifiers, effects pedals, electronic tuners, and voice and instrument modulation all are on display. This year I spent a little time looking over wireless 2.4 GHZ receivers for use on guitars and other instruments. Cables are a great nuisance and a good fidelity wireless system makes sense, but they can be rather expensive as well.

I reviewed one wireless system by Carol Electronics of Taiwan. It was advertised as be-

ing small and very portable and came with a receiver, condenser microphone and an instrument jack. I didn't get an exact price, but think that it is in the $1,000 range. It's a great way for a guitar player to eliminate all the cables at his feet.

Away from the cacophony of sound in the convention center halls was a very interesting presentation, focused mainly on guitar resellers, on the subject of endangered Dalbergia tonewoods like rosewood and cocobolo, among others. Many species of Dalbergia are important timber trees used on guitars and decorative wood products. Some Dalbergia woods are now controlled by CITES (the Convention on International Trade in Endangered Species of Wild Fauna and Flora, also known as the Washington Convention), a multilateral treaty to protect endangered plants and animals [1].

You might think a guitar made of wood and steel wouldn't fall under the purview of many regulatory agencies, but that's how it is these days. The revised regulation effective January 1, 2017 covers two issues: the first is the issue of the guitar manufacturer purchasing wood products that have been legally permitted, and the second is the out-of-country travel and/or sale of these endangered products. A great deal of detailed record-keeping is required.

Two main takeaways from this presentation is that the guitar manufacturer, in the case of the presentation that I attended, is more than willing to provide the user appropriate permit numbers to either sell the product across borders, or hand-carry the product across borders. Secondly, for residents of the United States this includes gigging or transporting your guitar into Canada, or Mexico, and it does work both ways. The EU countries all work in similar fashion. So, to be on the safe side, any guitar purchased after January 2, 2017 should have a permit number, and the guitar-carrying traveler should contact the guitar maker for the wood permit number prior to 2017.

It was quite a show. The Band's Robbie Robertson was presented with the NAMM "Music for Life" award for his many decades in music, first with Ronnie Hawkins and the Hawks, then as part of Bob Dylan's band during his infamous electric shows, and finally as founder and chief songwriter for The Band.

In other NAMM news, Aerosmith guitarist Joe Perry took home the Les Paul Award on behalf of the Les Paul Foundation, and engineer/producer Jack Douglas, the so-called "sixth member" of Aerosmith, was inducted into the NAMM TEC Hall of Fame. Douglas also engineered albums such as John Lennon's "Imagine" and The Who's "Who's Next."

I've been to NAMM for several years now, and it never disappoints. I look forward to coming back for NAMM 2018. **PCBDESIGN**

Dick Crowe is a special projects editor for I-Connect007. He is the former CEO of Bürkle USA and Bürkle North America. Dick is also a guitarist and harmonica player.

Recent Highlights from PCBDesign007

1 Help Wanted! Our 2017 Industry Hiring Survey

This month we conducted an industry survey on plans for hiring during the year. Included here is a summary of the results. We started by simply asking, "Do you plan to hire additional people this year?" More than half of the respondents answered yes while about a third said no—which we take as an optimistic sign that our industry plans to expand in 2017.

2 Mentor Graphics CEO Walden C. Rhines Named IEEE Fellow

Mentor Graphics Chairman and CEO Dr. Walden C. Rhines has been named a Fellow of the Institute of Electrical and Electronics Engineers (IEEE). Dr. Rhines is being recognized for leadership and technology innovation in integrated circuit design and automation.

3 True DFM: Taking Control of Your EDA Tool

We PCB designers are doing some truly great things with our layout tools. But we must remember that these tools are so powerful that they will sometimes allow us to design things that can't be manufactured! We must collaborate with our fabricator and assembly brethren and embrace the best DFM practices, or face the consequences downstream.

4 Ucamco Releases Integr8tor v2016.12

Ucamco has launched the v2016.12 maintenance release of Integr8tor. This release features important extensions and enhancements to existing functionality and addresses a number of software inconsistencies that were reported by our Integr8tor user base.

5 Altium's Sales Plan: Deal Directly with Designers

Altium has been shaking up the EDA world for quite some time. The Australian company once slashed the cost of Altium Designer by 75% to grow market share, and who could forget their famous (or infamous) "Bunny" ad campaign? In this interview with Lawrence Romine, Altium's global head of field marketing, he shares his views on sales and marketing in the EDA world, as well as Altium's philosophy on selling EDA tools.

6 IPC Designers Council San Diego Chapter Meeting: Flex Focus Fills the House

I recently attended the January meeting of the San Diego chapter of the IPC Designers Council. The meeting was hosted by the PCB design bureau San Diego PCB, and featured guest speaker John Stine, VP of Operations for Summit Interconnect, Anaheim.

7 Beyond Design: The Maturing EDA Industry

EDA companies generally innovate through acquisitions and mergers as it is easier to buy new technology than expend time and resources developing a product and risk losing market share. However, some tend to focus on partnering with other EDA vendors rather than acquisition. But this leaves them vulnerable to outside influence.

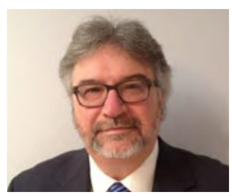
8 Cadence Posts Strong Q4 2016 Results; Revenue at $469 Million

Cadence Design Systems posted results for the fourth quarter and fiscal year 2016. Cadence reported fourth quarter 2016 revenue of $469 million, compared to revenue of $441 million reported for the same period in 2015. On a GAAP basis, Cadence recognized net income of $38 million in the fourth quarter of 2016, compared to net income of $80 million for the same period in 2015. Revenue for 2016 totaled $1.816 billion, compared to revenue of $1.702 billion for 2015. Net income for 2016 was $203 million, compared to net income of $252 million for 2015.

9 EMA Releases Ultra Librarian for OrCAD

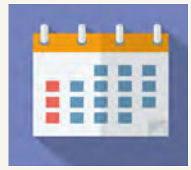
EMA Design Automation has just released Ultra Librarian for OrCAD providing symbols, footprints, and 3D models for an expanding library of currently over 8 million parts. "Symbols, footprints, and 3D models are the building blocks of any electronic board design," said Manny Marcano, president of EMA Design Automation. "We created Ultra Librarian for OrCAD so that our customers could simply download these parts rather than wasting time on the tedious and error prone task of creating them."

10 Flex and Rigid Sales and Marketing with Al Wasserzug



After decades in the PCB industry, Al Wasserzug of Cirexx International has seen marketing and sales trends come and go. I recently caught up with Al and interviewed him, covering the latest sales and marketing techniques, the value of traditional methods such as trade shows and conferences, and the particular characteristics of marketing flex circuits.

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



Events

For IPC Calendar of Events, click here.

For the SMTA Calendar of Events, click here.

For a complete listing, check out The PCB Design Magazine's event calendar.

IPC APEX EXPO 2017 Conference and Exhibition
February 14–15, 2017
San Diego, California, USA

China International PCB & Assembly Show (CPCA)
March 7–9, 2017
Shanghai, China

14th Electronic Circuits World Convention
April 25–27, 2017
Goyang City, South Korea

IPC Reliability Forum: Manufacturing High Performance Products
April 26–27, 2017
Chicago, Illinois, USA

KPCA Show 2017
April 25–27, 2017
Goyang City, South Korea

IMPACT Washington D.C. 2017
May 2–3, 2017
Washington, D.C. USA

Thailand PCB Expo 2017
May 11–13, 2017
Bangkok, Thailand

JPCA Show 2017
June 7–9, 2017
Tokyo, Japan

IPC Reliability Forum: Emerging Technologies
June 27–28, 2017
Düsseldorf, Germany

SMTA International 2017 Conference and Exhibition
September 17–21, 2017
Rosemont, Illinois, USA

electronicAsia
October 13–16, 2017
Hong Kong

IPC Flexible Circuits: HDI Forum
October 17–19, 2017
Minneapolis, Minnesota, USA

TPCA Show
October 25–27, 2017
Taipei, Taiwan

productronica 2017
November 14–17, 2017
Munich, Germany

HKPCA/IPC International Printed Circuit & South China Fair
December 6–8, 2017
Shenzhen, China

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

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

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

EDITORIAL:

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

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

MAGAZINE PRODUCTION CREW:

PRODUCTION MANAGER: **MIKE RADOGNA**
mike@iconnect007.com

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

INNOVATIVE TECHNOLOGY: **BRYSON MATTIES**

COVER: **SHELLY STEIN**

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

THE pcb design MAGAZINE

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

February 2017, Volume 6, Number 2 • The PCB Design Magazine© is published monthly by BR Publishing, Inc.

ADVERTISER INDEX