THE pcb design MAGAZINE

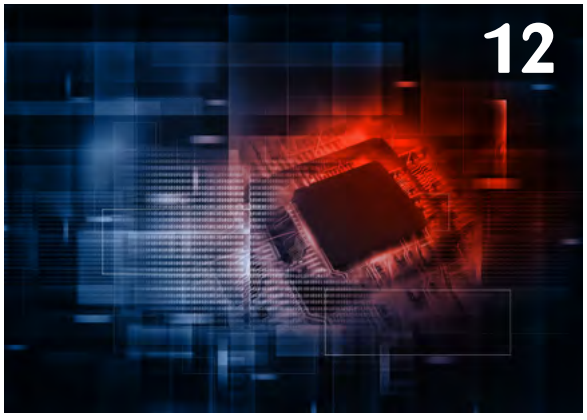
June 2016

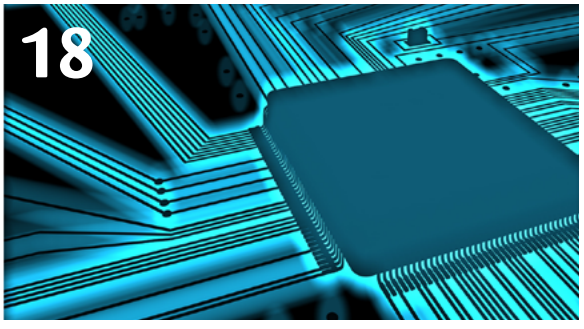
an I·Connect007 publication

DESIGNING WITH Tighter TOLERANCES

THE pcb design MAGAZINE

Feature Content





Designing with Tighter Tolerances

Ever since the first PCBs were designed, spaces, traces, and features have been shrinking at an almost exponential rate, along with available board real estate. For some designers, what was formerly a jigsaw puzzle has become a logistical nightmare. This month, our first feature story is an interview with Albert Gaines, owner of HiGain Design Services, who focuses on the ins and outs of designing with tight tolerances. David Ledger-Thomas of Honeywell Aerospace discusses some of his techniques for handling fine spaces and traces. And Mark Thompson of Prototron Circuits covers this topic from the viewpoint of the CAM operator, offering a variety of tips to help designers stay out of trouble when things get small.

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

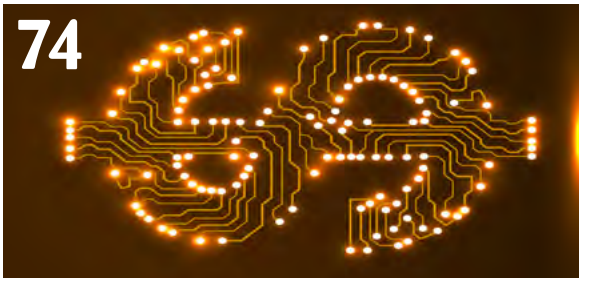
Isola GmbH
Isola Strasse 2
D-52348 Düren, Germany
Phone: 49-2421-8080
info-dur@isola-group.com

THE pcb design MAGAZINE

June 2016

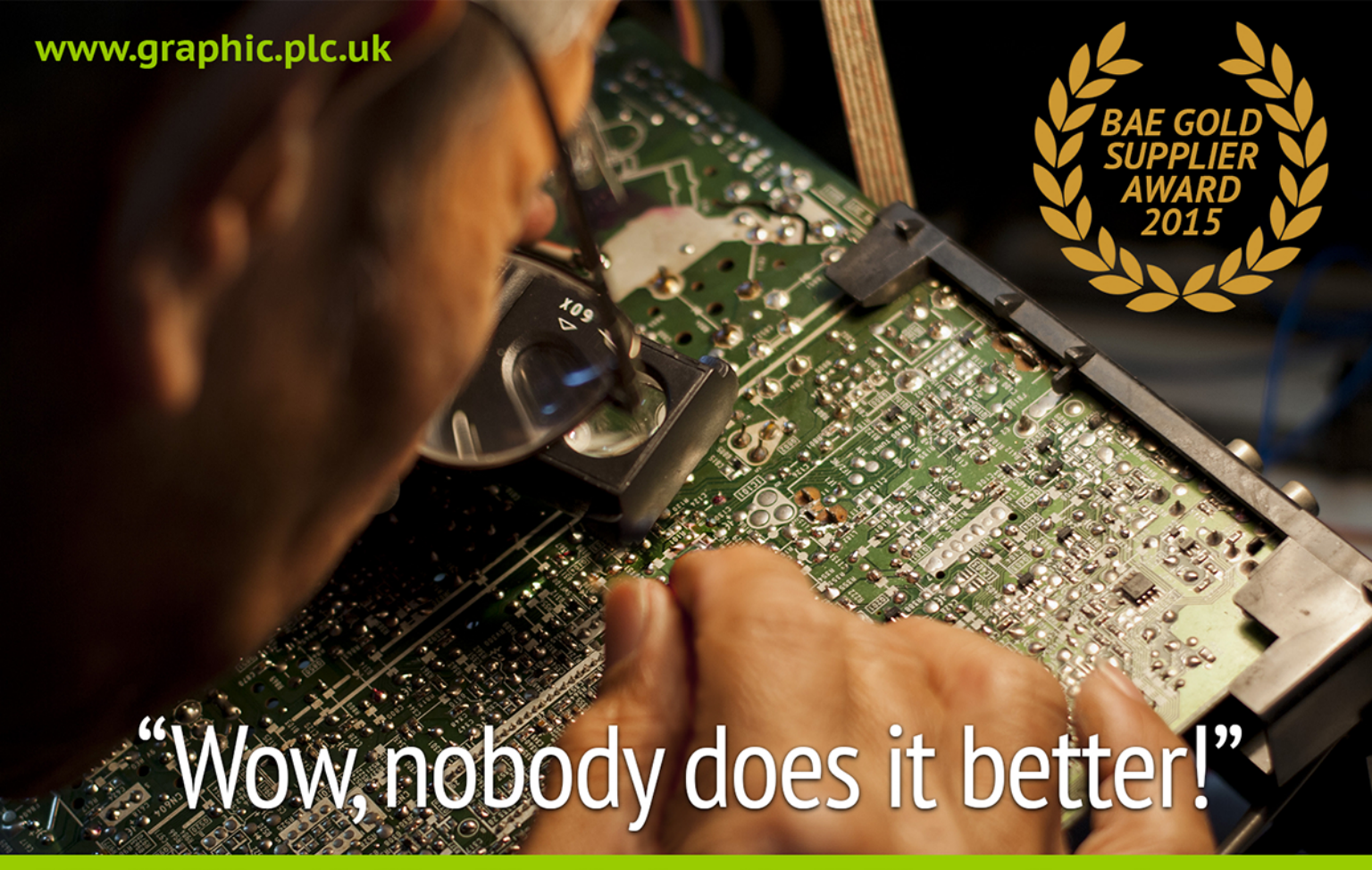
Vol. 5, No. 6

thepcbdesignmagazine.com



More Content

www.graphic.plc.uk



HDI • Rigid-flex • Flexibles • Micro Flex Assembly • Multilayers • Copper-invar-copper

- Over 48 years of experience in the PCB business
- The most PCB manufacturing approvals in the world
- Experts from prototype to full production runs
- Ultimate PCB interconnect solutions for Medical, Military (MIL-PRF-31032) and Aerospace (AS 9100)
- All work is done in-house at one of our facilities: USA, UK, Europe or China
- All orders are IST tested and delivered to your door

Application Engineering is included FREE with every job.

To learn the benefits of design-for-manufacture, click here.

'Where People Make the Difference'

Graphic USA
Scott Kohno & Carmen Gallegos
+1 (562) 404-0626

GRAPHIC GROUP

Graphic PLC Global Sales
UK: +44 (0) 1363 774874

Let's Get Small

by Andy Shaughnessy
I-CONNECT007

I sometimes wonder what people were thinking during great moments in history. For instance, did you ever wonder what was going through the minds of the technologists who created the earliest PCBs?

I imagine that when the first PCBs were developed, rather than just being satisfied that they'd created this great new piece of interconnect, the lead engineers were already thinking, "What if we could put more components on this thing? What if we shrank the traces? Could we use the ENIAC computer to design PCBs? That would be swell!"

Then it was on: The Tight Tolerances Arms Race. Available board real estate has been shrinking ever since those golden days of PCBs. Maybe it's just human nature; we tend to want more of everything, and there was plenty of empty space on those early boards.

Now, spaces, traces, pitch, vias—everything about the PCB is tiny. And for decades, most of you enjoyed the logistical challenge: How much more can you pack onto a circuit board? How small can you go before you reach a practical manufacturing limit?

Apparently, some of you have reached your own limit, so to speak. In our surveys, almost 20% of PCB designer respondents say that dealing with finer features and the accompanying lack of real estate, all while facing tighter dead-

Others Can Imitate, But Nothing Can Duplicate

BluePrint-PCB®

The Original and Still the Best.

AUTOMATING THE PCB DOCUMENTATION PROCESS FOR OVER 10 YEARS!

BluePrint-PCB® is the original PCB documentation solution and still the best most robust tool on the market today. BluePrint automates the PCB documentation process, creating comprehensive electronic drawings, successfully driving the procurement process and is used in final inspection to verify the product was built to specification.

- Lowers documentation cost and time
- Increases documentation detail & accuracy
- Shortens PCB design cycles
- Reduces documentation time by 60-80%
- Improves manufacturing & simplifies inspection

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

DownStream Technologies

lines, is taking the fun out of the job. They also believe that many OEM customers are not savvy, shall we say, about the basics of PCB design and manufacturing, which further frustrates the designer and exacerbates the problem.

Designers, you made it look easy, and now customers expect you to work your magic, and to do it quickly. It doesn't look like it's going to get any better. As I said, it's human nature. Your customers are going to keep squeezing every mil of available space out of your board real estate. Unless you plan to retire soon, you're going to have to keep working with increasingly finer spaces and traces. (And please don't retire. We don't have any youngsters in line to replace you.)

This month, we look into the drive toward increasingly tighter tolerances, with a variety of approaches that designers can take when things really get small. In an interview for our cover story, Albert Gaines of HiGain Design Services discusses some of his tricks for designing boards with fine spaces and traces, as well as methods for fanning out from ultra-fine-pitch BGAs. David Ledger-Thomas, a design engineer with Honeywell Aerospace, explains how techniques for designing boards with tight tolerances can vary from EDA tool to EDA tool, because of something as simple as a rounding error. Mark Thompson of Prototron Circuits discusses fine-line designs from his viewpoint in the CAM shop, and he lays out some DFM tips that can keep your own CAM guy from calling you late on a Friday with questions about your design.

We also have some great columns in this issue. Barry Olney of In-Circuit Design viewed Howard Johnson's newly released High-Speed Digital Design Collection, made up of seminars presented over 20 years at Oxford, and he breaks down many of Johnson's methods for taming the "black magic" of signal integrity. And John Coonrod of Rogers Corporation discusses some of the challenges related to applying soldermask over RF circuitry and components, and some of the trade-offs that should be considered as well. We also bring you an interesting interview with Mike Brown, CID, owner of the PCB design consulting firm Interconnect Design Solutions. He explains how collaboration is vital for success of each design project, and why designers should all strive to achieve DFP (design for profitability).

Be sure to check out the special section on IPC's recent IMPACT Washington, D.C. lobbying event, with numerous interviews conducted by Patty Goldman, managing editor of *The PCB Magazine*. Patty interviewed all of the industry's movers and shakers. You won't find coverage like this anywhere.

As we head into the summer, don't forget to take *The PCB Design Magazine* and Design007 with you to the beach or the mountains. We're ready when you are!

See you next month. **PCBDESIGN**

Andy Shaughnessy is managing editor of *The PCB Design Magazine.* He has been covering PCB design for 16 years. He can be reached by clicking here.

Flexible, reliable Supply Chain Solutions

High quality copper clad laminates & prepreg bonding materials

ventec
INTERNATIONAL GROUP
騰輝電子

Ventec International Group is a world leader in the production of high quality and high performance copper clad laminates and prepregs, with full R&D and product development capabilities.

Our global distribution means we can readily supply our products to all markets in all geographic areas.

Ventec – wherever technology takes you, we deliver

Ventec International Group

China: joan@ventec.com.cn
Europe: contact@ventec-europe.com
USA: ordersCA@ventec-usa.com

Follow @VentecLaminates

www.**venteclaminates**.com

Designing with Fine Lines and Features

by Andy Shaughnessy

Albert Gaines is the owner and senior PCB designer at HiGain Design Services in Norcross, Georgia. He's been a PCB designer since 1981; he designed a variety of boards at Scientific Atlanta, and then ViaSat, before deciding to open his own company in metro Atlanta. I asked Albert to talk about some of the finer lines and features that come through his shop, as well as some design techniques for boards with tight tolerances.

Andy Shaughnessy: ***What are the tightest tolerances you are currently designing?***

Albert Gaines: We have used 3.6 mil line and space on a few but most are 4 mil line and space for HDI.

Shaughnessy: ***What are the most challenging issues designers face regarding fine spaces, traces, and pitch?***

Gaines: As we have progressed into smaller-pitch components, the space in and around components has decreased even more. These smaller areas drive what the vias and routes can be, and sometimes we are restricted by our clients and vendors on the sizes we can use. This may be a costing issue on the bare boards, or the smaller lines and vias may be restricted by an internal spec that was conceived before ultra-fine-pitch came to be. We have actually had to get chief technical officers sign off on the line/space and via sizes required for layouts.

Placing bypass capacitors for these finer-pitched parts can also be an issue. The pitch may drive the use of smaller body caps than the engineers or clients want to use. We end up placing caps all around the part because the application notes said to use this quantity and this value.

Shaughnessy: ***Do your PCB design tools handle tighter tolerances well? What about PCB design tools in general?***

Managing a global PCB supply chain can seem like a puzzle.

Let us be the solution.

With over 35 years of experience and key partnerships around the world, we have what it takes to make your next project on time, on budget and hassle-free.

LEARN MORE ABOUT THE NUJAY SOLUTION ▶

NUJAY TECHNOLOGIES INC.

Your Global Connection For Interconnects

www.nujaytech.com

Gaines: Yes, we have tools that we can set up rules/constraints for those specific needs. In today's world, most higher-end tools allow you to set up rules/constraints for certain areas. This allows for those tight rules to be enforced in the area that drives that tight lines and spacing, while allowing more relaxed rules/constraints on the rest of the board. Setting up those rules could be made easier, but I appreciate what we have today to control layout, compared to what we had to use in the past.

Albert Gaines

I know you can find lower-end tools from a cost standpoint; however, these tools often do not have the features needed to perform the more complex mixed-technology boards required by our clients. I think the old saying "You get what you pay for" applies here. If you need the layout control, you may have to pay more for it.

Shaughnessy: ***With finer features, how do you determine whether it is cost-effective to use HDI?***

Gaines: Most of the components we have to work with are already selected by the time we get the design. I think more knowledge needs to be shared with the engineering and component teams that are selecting the core components. They need to better understand the impact those HDI components have on layout, board procurement and assembly. Many new-technology components were developed to make things smaller for end-products such as handheld devices. But most of our boards are not that small or dense.

These HDI components can increase layer counts and require the use of smaller lines/spaces and via sizes. Many times, one component drives the entire board technology from a fabrication standpoint. If more time could be spent on selection of a component that better fits the overall design and layout complexity of the board, the end-product would cost less due to reduced procurement and assembly issues.

I have asked before why an ultra-fine-pitch BGA was selected instead of a 1 mm pitch package. The answer was that the engineer thought it was sexy. This "sexy part" caused issues in the bare board procurement and the assembly process. Yes, process issues are fixable, but when you are trying to keep the cost down, the overall impact of these ultra-fine pitch parts should be considered.

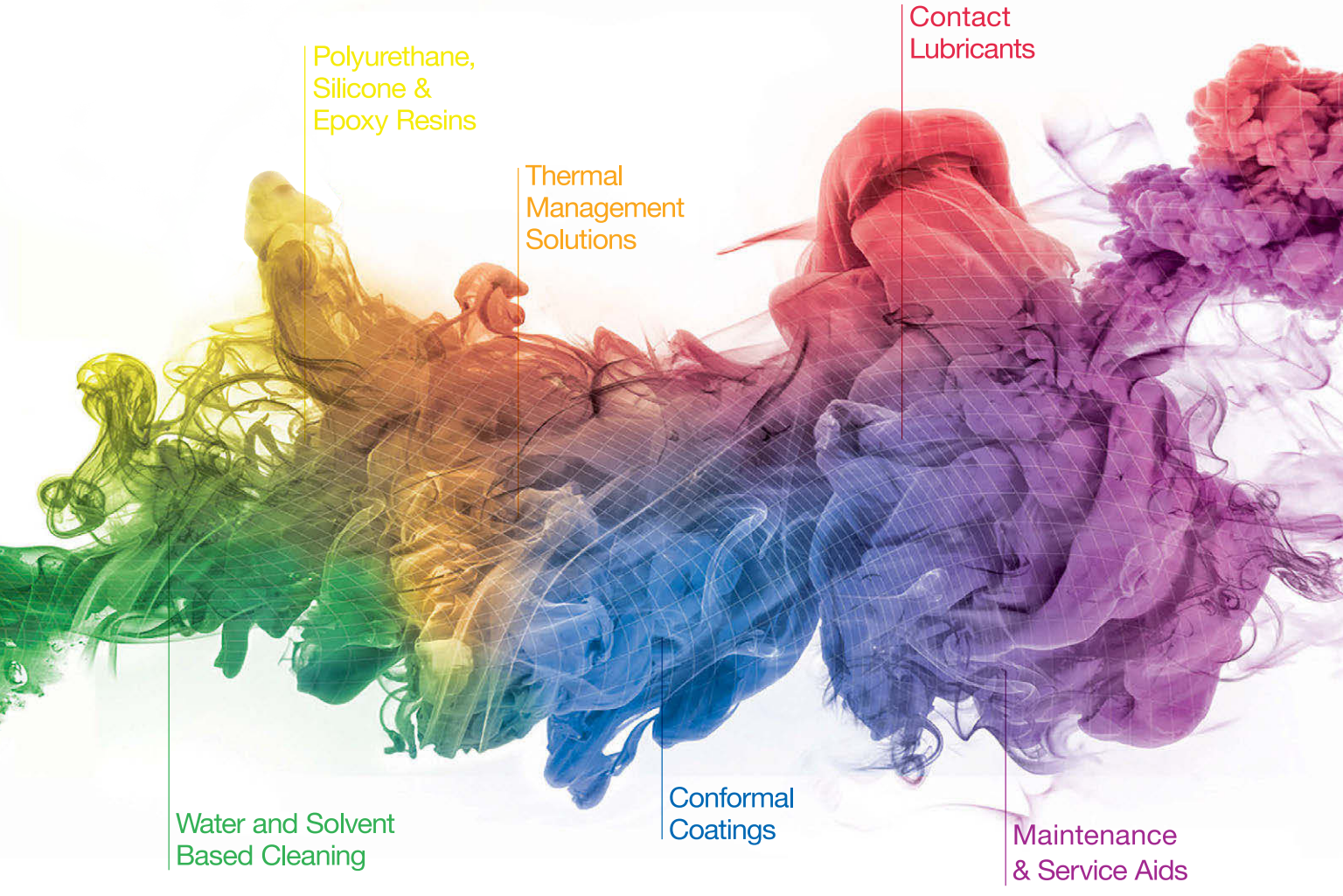
Shaughnessy: ***You work with ultra-fine-pitch BGAs. Do you have any special techniques for doing ultra-fine-pitch BGA fanouts?***

Gaines: We have used ultra-fine-pitch BGAs for projects in past, and I see this becoming more prevalent in the newer more complex designs. This may seem basic to many designers, but the idea is to think about what can be done to solve the clearance issues. You need to first look at the space that the part creates between pads. This space may be increased by making the ball pad as small as possible. Always verify your sizes with your assembly team.

Next, look at the max via pad size allowed based on minimum spacing. The via pad will dictate the hole size using the minimum annular ring allowed by your vendor. Topside soldermask tenting of the vias is also suggested but should be approved by your fabrication/assembly team. If vias are impossible, a fanout has to be created on the component side using even smaller lines and spaces. You often have to push the envelope of what your board supplier and assemblers want to process as a standard board.

Shaughnessy: ***What do you think is the most important thing to remember when facing tight tolerances?***

Gaines: Tight tolerances are caused by the empty spaces created by the part. We designers have always been all about using the space available. In today's designs, the space has gotten many times smaller. You have to change the way you think. The thought can't be that this part does not allow the sizes we have been using. You have to look at what sizes the space will allow. Do not think of what you can't do,



Where experience and innovation come together

In the formulation, manufacture and global supply of conformal coatings, thermal pastes, encapsulants, cleaners and lubricants, we have the solutions of the future. Our ethos of collaboration and research, combined with a truly global presence and manufacturing capability has led to the development of ISO standard, environmentally friendly products for the world's leading industrial and domestic manufacturers.

Our unique provision of the complete solution, combined with the scale and scope of our capabilities ensures a reliable supply chain, and exemplary service.

Find out how you could become part of the solution. Simply call or visit our website.

Scan the code to discover our full spectrum of superior 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

but what you can. The part space will drive what you can do.

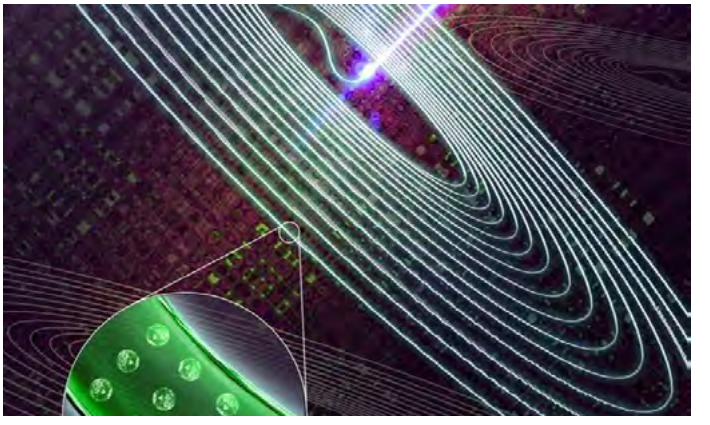
Things do not have to be small to have tight tolerances. We have clients who develop home appliances that may be powered anywhere in the world. We have to design for the different AC input voltages that are used globally. These AC voltage lines require minimum creepage distances maintained in the design. The AC circuits are mixed among the DC circuits due to the end-product configuration and controls. Often components have to be selected with lead spacing based on the minimum distances required. Once again it is the space that drives the layout.

Sometimes we do think newer ideas and newer technologies are sexy. But sexy may cost more than it is worth. Carefully select the HDI path. Often it is the best option for the added features per square inch derived. Overall cost, reliability of the layout, and assembly should be weighed and understood by all.

Shaughnessy: *Thank you, Albert.*

Gaines: Thank you, Andy. **PCBDESIGN**

Glass Now Has Smart Potential



Australian researchers at the University of Adelaide have developed a method for embedding light-emitting nanoparticles into glass without losing any of their unique properties—a major step towards 'smart glass' applications such as 3D display screens or remote radiation sensors.

This new "hybrid glass" successfully combines the properties of these special luminescent (or light-emitting) nanoparticles with the well-known aspects of glass, such as transparency and the ability to be processed into various shapes including very fine optical fibres.

"These novel luminescent nanoparticles, called upconversion nanoparticles, have become promising candidates for a whole variety of ultra-high tech applications such as biological sensing, biomedical imaging and 3D volumetric displays," says lead author Dr. Tim Zhao, from the University of Adelaide's School of Physical Sciences and Institute for Photonics and Advanced Sensing (IPAS).

"Integrating these nanoparticles into glass, which is usually inert, opens up exciting possibilities for new hybrid materials and devices that can take advantage of the properties of nanoparticles in ways we haven't been able to do before. For example, neuroscientists currently use dye injected into the brain and lasers to be able to guide a glass pipette to the site they are interested in. If fluorescent nanoparticles were embedded in the glass pipettes, the unique luminescence of the hybrid glass could act like a torch to guide the pipette directly to the individual neurons of interest."

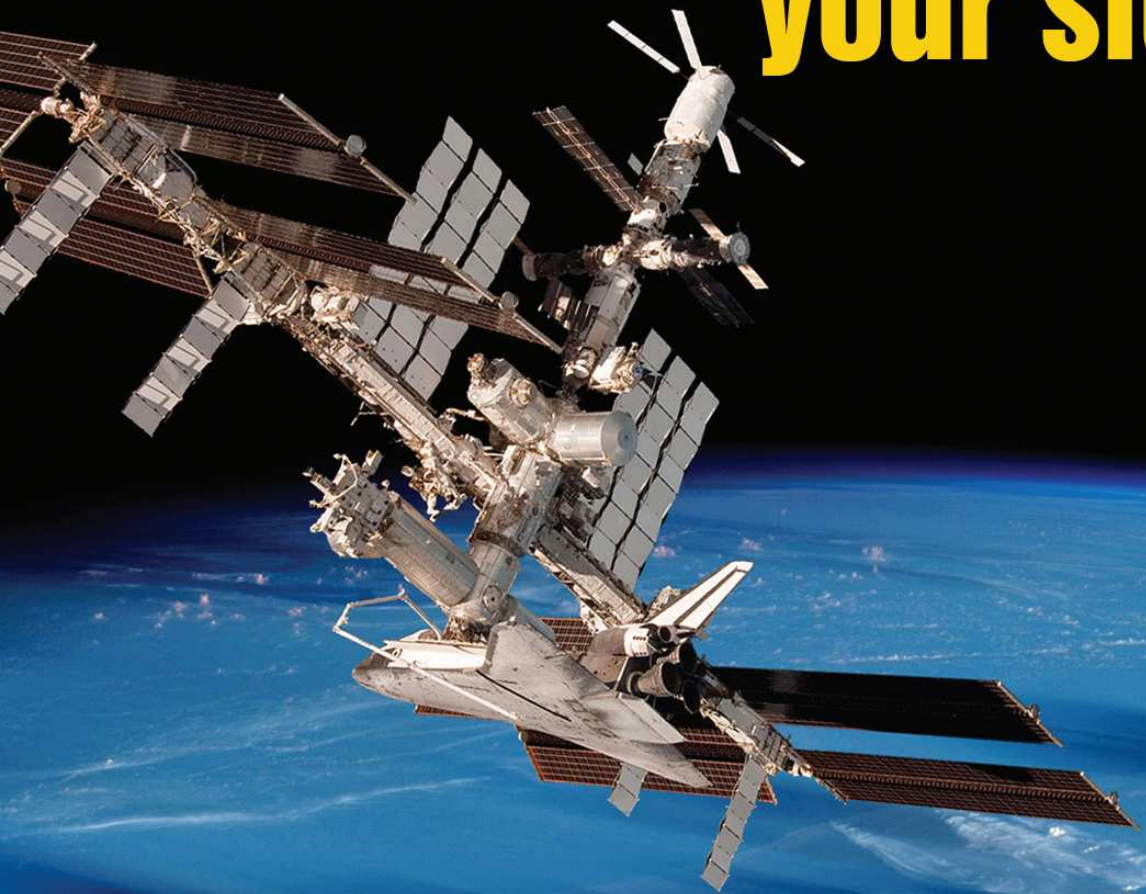
Although this method was developed with upconversion nanoparticles, the researchers believe their new 'direct-doping' approach can be generalised to other nanoparticles with interesting photonic, electronic and magnetic properties.

To date, the method used to integrate upconversion nanoparticles into glass has relied on the in-situ growth of the nanoparticles within the glass.

"We've seen remarkable progress in this area but the control over the nanoparticles and the glass compositions has been limited, restricting the development of many proposed applications," says project leader Professor Heike Ebendorff-Heideprem, Deputy Director of IPAS.

When failure is not an option, don't leave your high tech PCBs to chance.

Have an ACE up your sleeve!

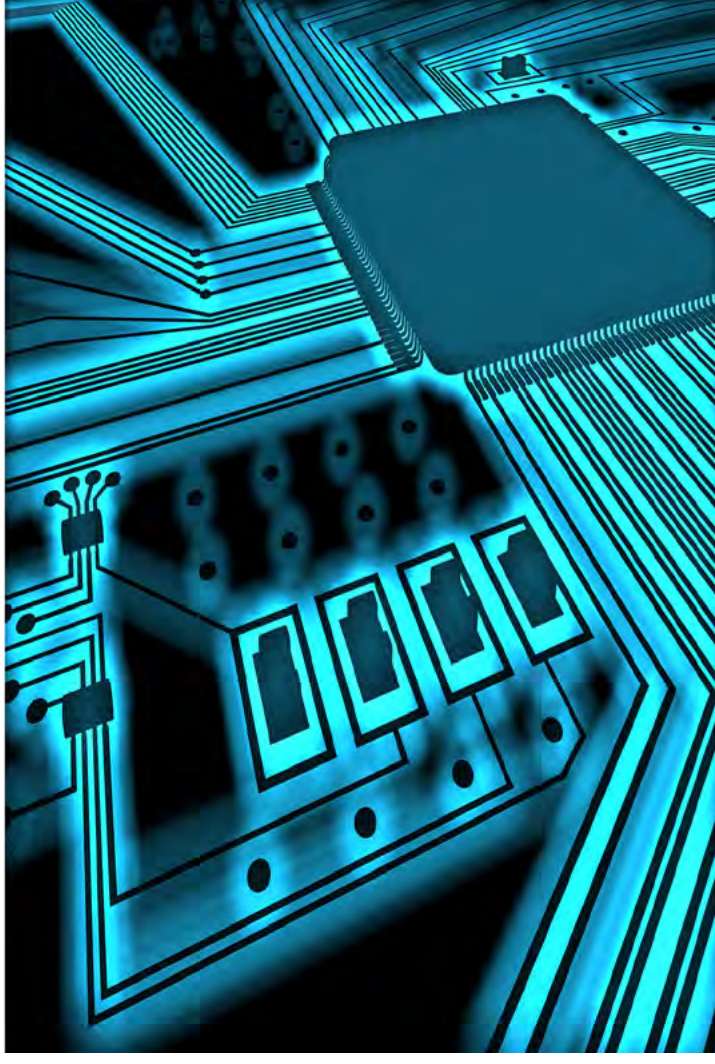


- Specializing in high technology quick turn prototypes, as fast as 24 hrs
- MIL-PRF-55110, MIL-PRF-31032A, AS9100, and ITAR certified
- Leading experts in RF, Hybrid PTFE/FR4 and Antenna/Microwave

ACE
ACCURATE
CIRCUIT
ENGINEERING

www.ace-pcb.com

sales@ace-pcb.com



From the CAM Shop: Tight Tolerance Design Tips

by Andy Shaughnessy

After you finish your design, it winds up in the hands of people like Mark Thompson, the man who runs the CAM department at Prototron Circuits in Redmond, Washington. He sees CAD data firsthand, and often has to address errors and inconsistencies in PCB designs. For this issue, we asked Mark to discuss today's tight tolerances, some of the problems they can cause PCB designers, and what designers can do when dealing with shrinking features.

***Andy Shaughnessy:** What are the tightest tolerances you are currently building?*

Mark Thompson: First off, that is a great question. We have long said in fabrication if you added up all the accumulative tolerances a fab shop has to deal with the part would be physically impossible to build. Having said that, unusual process tolerances CAN be achieved, such as plated holes with a +/-.002" tolerance for press-fit devices. One fab shop may say that the best they can do for plated holes and slots/cutouts would be +/-.003" but often we can do +/-.002". How is that possible, you ask? For one thing, we can tell the CAM system to select a tool that is +.004-.000". This selects a tool that works best for a +/-.002 final tolerance.

Another typical tolerance issue is with controlled impedance. Many fabricators ask for +/-15% tolerance for traces thinner than .0035". This is not uncommon considering that just 10% = less than half a mil of total accumulated deviation throughout the fabrication process. The good news is that most fabricators use a field solver for the impedances, which means they can adjust for process variables like plate, etch, mask thickness, etc. And ultimately this means that even in situations where a fabricator may ask for +/-15%, they may incur as little as 5% deviation if they have good process control. I guess I would end by saying if the customer has some unusual tolerances they need to achieve, I recommend speaking with your chosen fabricator to make sure they can be met.

Find the ideal balance between Tx and Rx equalization settings up to

100x faster

SiSoft's new OptimEye technology helps you optimize your design faster so you can ship your designs faster!

OptimEye™

- Replaces hundreds of blind sweep simulations with a single run
- Designed specifically for AMI-based analysis
- Generates custom per-channel equalization settings in a large system design

Evaluate OptimEye on your design today

CLICK HERE TO GET STARTED!

SiSoft™

We Are Signal Integrity

www.sisoft.com

***Shaughnessy:** What are the most challenging issues fabricators face regarding fine spaces, traces, and pitch?*

Thompson: Another great question. There are many. First and foremost is the chosen copper weight vs. trace and space. Many times this comes down to what a fabricator has to do for compensations for the process. In this case, we are talking about etch compensations.

Let's say you have a .1 mm trace and space design and you desire 3 oz. finish. The general rule of thumb is that for every half ounce of starting copper, we do a half-mil etch compensation. For three ounces we would need a .003" etch comp, and if the space is .00393" (.1 mm) we would be left with a .00093", space which is way outside of most folks' capabilities.

As far as pitch is concerned, there are a couple of things to consider. First, if you are looking at a .4 or .5 mm BGA pitch, you won't have the ability to run two sets of traces between the pads, so you would look to keep any differential pairs on the inside. Additionally, with the smaller pitch constraints it is difficult to maintain a minimum web of mask material to prevent wicking at surface finish. In order to maintain a .004" web of mask material, for instance, sometimes a fabricator will reduce the clearances down to as little as half or one mil per side to be able to maintain the web and not create other issues for the end-user at assembly.

***Shaughnessy:** What can designers do to alleviate some of that pain downstream when they're designing boards with such fine features?*

Thompson: So much of this just comes down to understanding the manufacturing process. Let's use the example of the mask clearances vs. web. For instance, if the designer knows that there will be SMTs less than .004" apart from each other, they may choose to create a mask-defined clearance where the clearance is smaller than the underlying SMT to be able to maintain that critical web of mask between mounts. In other cases, if the surface finish is a very thin deposition and does not tend to "wick" between mounts, they may choose to "gang relieve" the mask area so there are no physical webs between mounts.

Mark Thompson

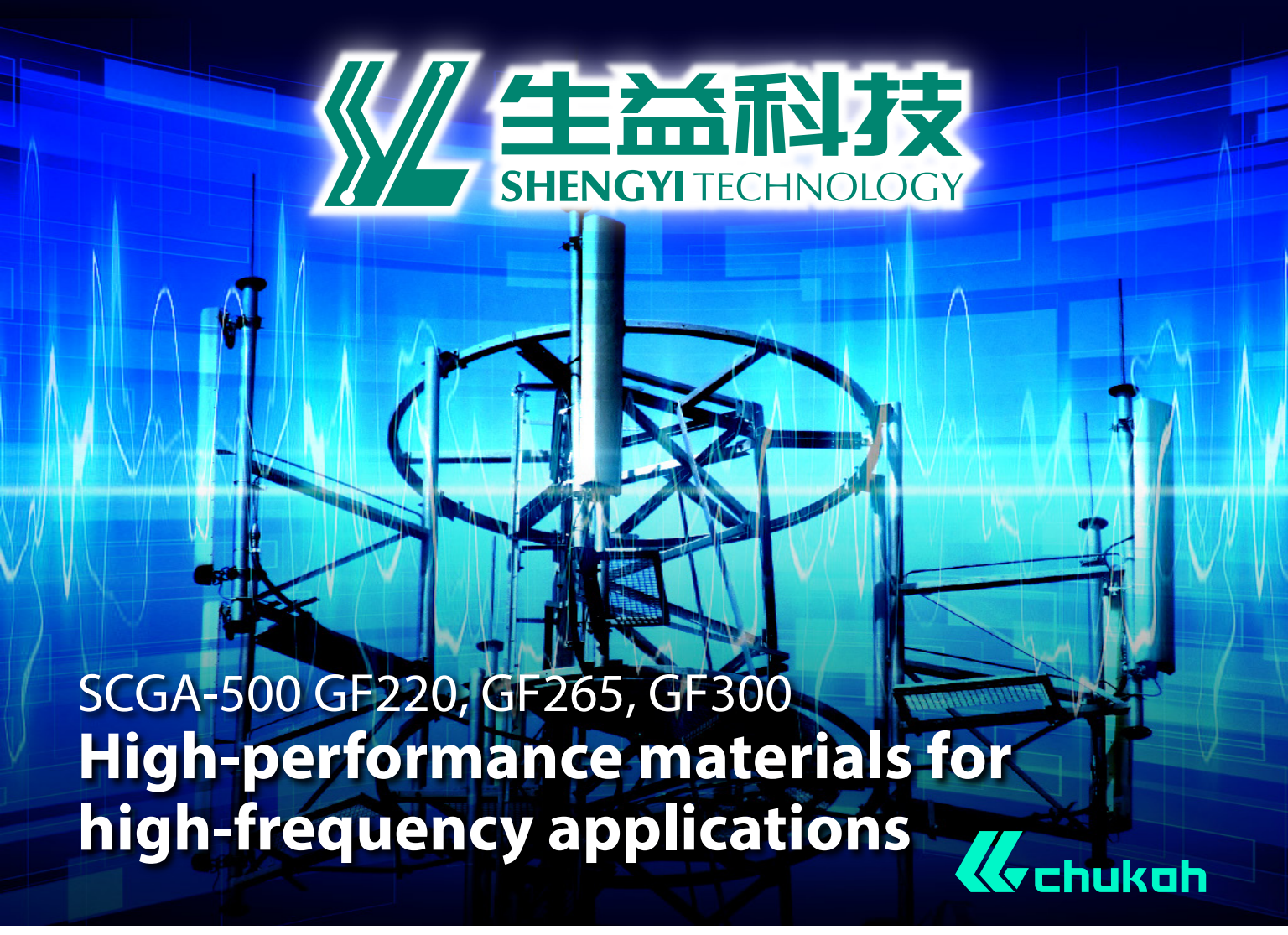
The same holds true with egressing from tight pitch BGA parts with traces. This is where you will see "neck down" areas where the traces get thinner to be able to get in and out of the BGAs.

***Shaughnessy:** What are some of the more common mistakes designers make with fine spaces, traces and pitch?*

Thompson: First off, space value on the design vs. copper weight is a big issue. So many times these days, we see a tight-pitch part that dictates the use of true .003" trace and space. This can only be done on either one quarter or three eighths ounce starting foils. Attempting to start a 3/3 design on half-ounce would mean .035"/.0025" after etch compensation, which is too small for most folks. Second, select a mask color that will allow for the tightest mask webs between surface mounts. Likewise, select a surface finish that does not tend to "wick" or short the mounts at assembly.

***Shaughnessy:** I hear a lot of horror stories about BGA fanout issues. Do you have any advice for designers dealing with ultra-fine-pitch BGAs?*

Thompson: Yes, as much as possible, keep breakout traces on the inside layers. Externally, try to



SCGA-500 GF220, GF265, GF300

High-performance materials for high-frequency applications

chukoh

Key product features:	GF220	GF265	GF300
Consistent permittivity in high frequency range	●	●	●
Low Df at high frequency	●	●	●
Excellent thermal resistance	●	●	●
Low moisture absorption	●	●	●
Stable properties in high frequency range	●	●	●

Learn more about SCGA-500 Series Materials

Photo Chemical Systems
www.photochemicalsystems.com

PARAMOUNT
www.paramount-usa.com

avoid having a termination point inboard on the BGA (attempt to keep the connections limited to the first two rows). And if you are truly pressed for real estate, consider a blind via approach.

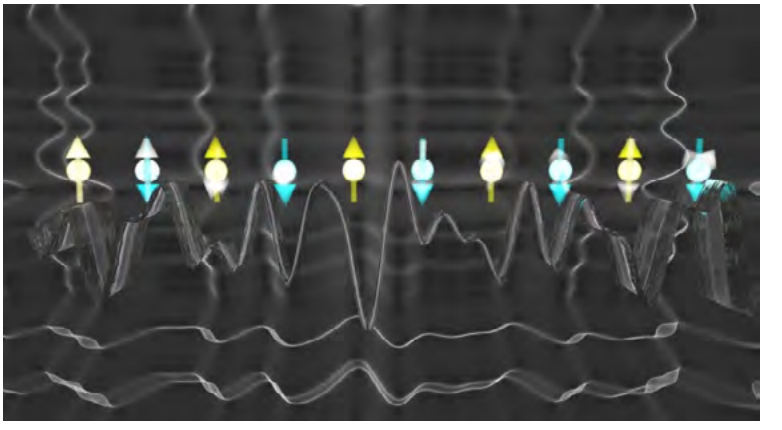
***Shaughnessy:** When do we "hit the wall" with tolerances? At what point is the piece just not manufacturable?*

Thompson: Another great question. I used to say we have hit the mechanical wall at .0059" drill sizes and .003" traces and spaces. I have to amend that now to whatever you can reasonably process at fab. If you can deal with .002" traces and spaces, this would allow for two tracks between BGA pads. This is good. Likewise, previously when we said .0059" was the mechanical "wall," we see some folks are going down to .003" and .002" holes! I think if geometry dictates smaller holes and features, fabricators must find a way to make that happen.

***Shaughnessy:** Thanks for the good info, Mark.*

Thompson: My pleasure, Andy. **PCBDESIGN**

Disorder Grants a Memory to Quantum Spins



The universe has been forgetting its own initial state since the Big Bang, a fact linked to the unrelenting forward march of time. Systems that forget where they started are said to have thermalized, since it is often—but not always—an exchange of heat and energy with some other system that causes the memory loss.

The opposite case is localization, where information about the initial arrangement sticks around. Such a situation is rare, like an ice cube that never melts, but one example is Anderson localization, in which particles or waves in a crystal are trapped near impurities.

Now, researchers working with JQI and QuICS Fellow Christopher Monroe have directly observed this localization in a system of 10 interacting ions, trapped and zapped by electric fields and lasers. Their findings were published June 6 in Nature Physics.

"The transition of quantum systems from thermalized to localized represents a boundary between states governed at long times by quantum mechanics and ones that follow classical physics," says Jake Smith, a graduate student at JQI and the first author of the paper. "It's important to know if a given quantum system will thermalize because if it does you can use techniques from classical physics to predict its long-time behavior."

By focusing a powerful laser to a diameter of just over a micron, the team also applied a random shift to the magnetic environment of each spin, creating the necessary disorder. Then, they tuned the strength of the interactions relative to the size of this disorder and traced the emergence of localization. They performed many experiments with random amounts of disorder, preparing each spin to point either up or down and then measuring all of the spins after a certain amount of time to see where they pointed.

"This work is a major advance in quantum simulation as our platform can be scaled to dozens of ions, where detailed modeling becomes impossible due to the complexity of many-body quantum states," Smith says.

Flex Your Design Skills

Innovative Flex Design Starts with OrCAD

The new flex and rigid-flex enabled OrCAD 17.2 has been shown to reduce rigid-flex design time by 25%—all while eliminating manufacturing errors common to flex circuit technology. Start your next rigid-flex project off the right way with OrCAD 17.2.

View Rigid-Flex Design Video ›

OrCAD™
CADENCE PCB SOLUTIONS

EMA Design Automation®

DESIGNING PCBs WITH TIGHTER TOLERANCES

by Andy Shaughnessy

David Ledger-Thomas is a PCB design engineer with Honeywell Aerospace. He's spent decades designing PCBs for a variety of applications, including defense, aerospace, computers, and high-performance audio. I asked David to share some of his thoughts on designing high-tech boards with increasingly finer spaces, traces and pitch.

Andy Shaughnessy: ***What are the tightest tolerances you are currently designing?***

David Ledger-Thomas: Out of the gate, the assumption for "tight tolerances" is referring to trace and space width. That would be 4.5 mil trace width and 4.5 mil trace spacing. But to add for conversation, tightest tolerance may also reference how much trouble is the PCB fabricator going to have with a particular design (i.e., layer count in a specified thickness, drill size aspect ratio, annular ring, cost, schedule, etc.) while still meeting customer requirements. The PCB design team must also take into account some design parameters such as mechanical/electrical performance, manufacturability, reliability and schedule. So all these could be a part of the "tight tolerance" vocabulary.

Shaughnessy: ***What are the most challenging issues designers face regarding fine spaces, traces, and pitch?***

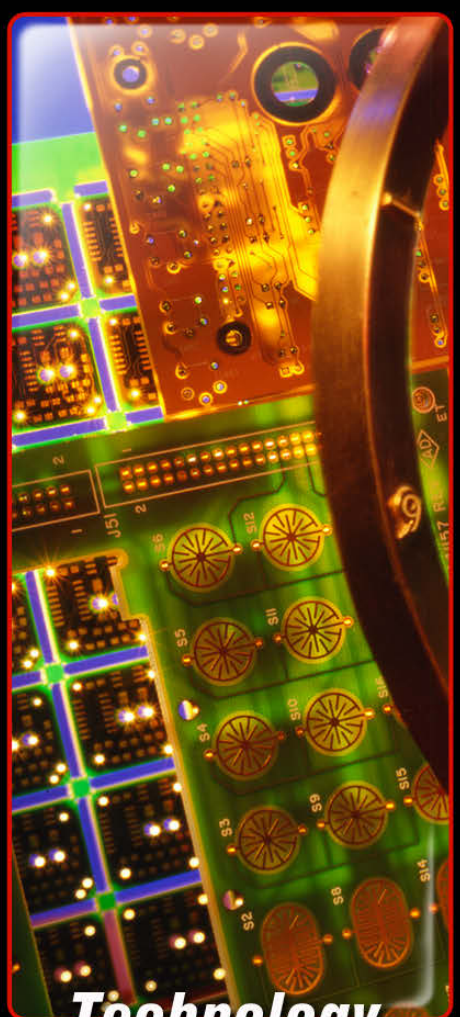
Thomas: Signal integrity would weigh heavy on the application of fine spaces, traces and pitch. Reliability would also be a factor, as in, will the board perform over temperature with a very low mean time between failures (MTBF). And of course can the board be fabricated to required specifications repeatedly and in a timely manner. If a fabricator had to manufacture 20 boards to get one good one, then that would be a poor performer for cost, time and reliability (even though the design was great in the PCB design tool).

Shaughnessy: ***Does your PCB design tool handle tighter tolerances well? What about PCB design tools in general?***

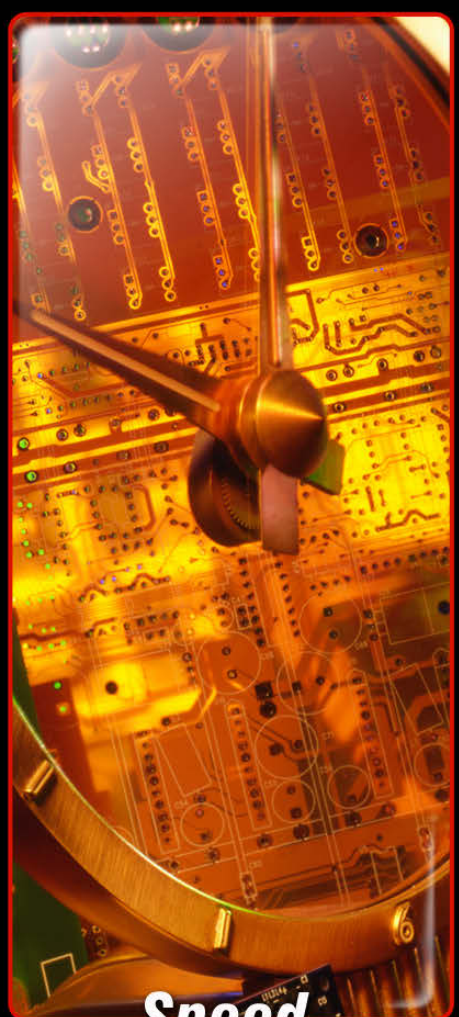
prototron.com

We Take the Time to do it Right

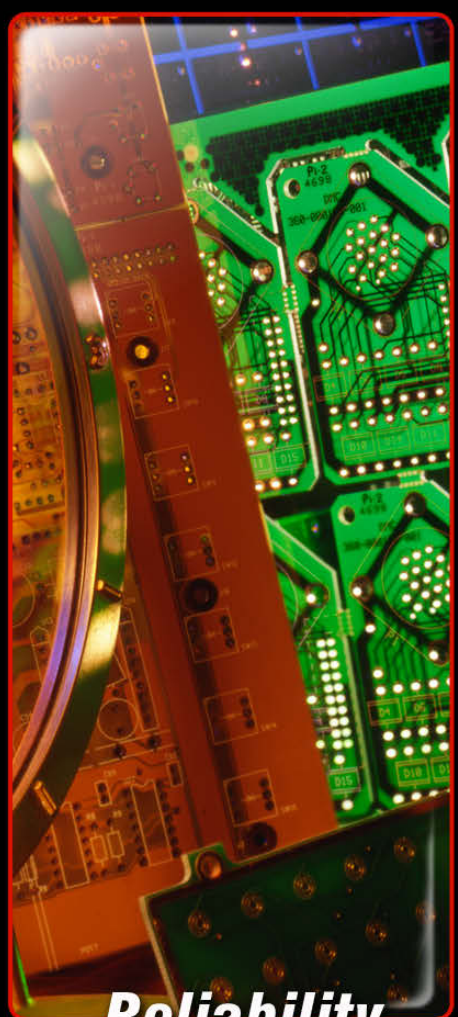
because you don't have the time to do it over.



Technology



Speed



Reliability

Prototron Circuits

High quality quick turn manufacturing, just another reason we are

America's Board Shop

Redmond Facility
15225 NE 95th Street, Redmond, WA 98052
Toll: 888.847.7686 Phone: 425.823.7000

Tucson Facility
3760 E. 43rd Place, Tucson, AZ 85713
Toll: 800.279.5572 Phone: 520.745.8515

David Ledger-Thomas

Thomas: That's a somewhat "loaded" question. To answer straight out, yes. Here's the caveat: Some PCB design tools do handle very tight tolerances well and some do not. So the "right tool for the job" rule applies, i.e., expecting a free/web-based PCB tool to design an HDI board would be rather troublesome. But remember that these are "design tools." Once the "right tool for the job" is selected, then, as with any other tool, it is the applied knowledge of the user that provides the level of results and quality for the design activity.

Now for the fine print. The PCB tool of choice may have issues with handling data of a tight tolerance board with generating outputs for fabrication/assembly. Meaning the as-designed PCB tool board data is not represented correctly in the fabrication/assembly outputs. Due to rounding issues, the Gerber format/ODB++/data output translation from the PCB tool may not have been correlated as expected. Or there may be other not-so-nice nuances of the PCB tool.

Another point would be the preferred/selected fabricator has not been informed/contacted (or received their feedback) as to the specific tight tolerance implementation. This means that the tight design constraint "as designed/specified" in the PCB tool could cause a resultant of fabrication issues.

Shaughnessy: *With finer features, how do you determine whether it is cost-effective to use HDI?*

Thomas: First, start by asking, "Is HDI the right technology application/approach? Or can other PCB technology be used to accomplish the same as HDI with lower cost, while still meeting requirements?" Second, what form would the implementation of HDI take in this particular application (i.e., using stacked vias, buildup layers, via-in-pad, laser/plasma/photo drilling vias, plating vias closed)? If these listed factors are negligible, then HDI would likely be a great design technology to utilize. It should also be noted that HDI may not be used for some cases and types of high-reliability designs. Customer specifications may also call out "No HDI."

Shaughnessy: *Do you work with ultra-fine-pitch BGAs? If so, do you have any special techniques for doing this type of fanouts?*

Thomas: Yes. Design techniques ensure the PCB footprint and fanout are correct. You have to take into account signal integrity, and getting the signal on and off the device with minimal impact. Refer to the device manufacturers' recommendations (electrical and mechanical), fabrication tolerances and assembly methodologies. For a bit more detail, always confirm that the fanout has been routed with PCB fabrication tolerances in mind. It might be a good idea to run the design by your fabricator to get their recommendations.

Shaughnessy: *What do you think is the most important thing to remember when facing tight tolerances?*

Thomas: I believe there are a multitude of items to be aware of when doing tight tolerance designs. First would be to meet signal integrity. Second, ensure the design is workable at the fabricator and assembly houses. Third, confirm customer requirements are met.

Shaughnessy: *Thanks for speaking with us, David.*

Thomas: Thank you, Andy. **PCBDESIGN**

Intercept's complete design flow gives you the power to keep moving, full speed ahead. Use Mozaix for flexible schematic capture and Pantheon for advanced, intuitive PCB/Hybrid/RF design to move quickly and seamlessly from concept to manufacturing. Our 30+ import translators make it easier than ever to switch tools. Let Pantheon and Mozaix drive your next innovation.

PANTHEON ADVANCED PCB, HYBRID, RF LAYOUT
MOZAIX NEXT GENERATION SCHEMATIC CAPTURE
REUSE BLOCK TECHNOLOGY
MULTI-BOARD & PANEL ARRAYS
LAYOUT-DRIVEN DESIGN & PROTOTYPING
SIMULATION & ANALYSIS INTERFACES
DESIGN AUTOMATION
DFM & ARTWORK VERIFICATION
HIGH SPEED DESIGN
ARTWORK PANELIZATION
AUTOMATIC DRAWING CREATION
30+ TRANSLATORS

pure engineering™
INTERCEPT

www.intercept.com

30+ Years of EDA Software Solutions

PCB007 Highlights

Happy's Essential Skills: Technical Writing
Technical writing is one of those topics that they don't really talk about in college—at least not where I went. Writing and English has never been a strong like of mine compared to science and math. So I did my required time in English and wrote my lab reports the best I knew how.

Weiner's World
3D printing, China's SMT equipment and robotics markets, IPC's mandate, counterfeiters, and Taiwan PCB makers' shift to automotive electronics—Gene Weiner talks about these things and more in this new article.

'Can Do' in CAM Outsourcing: A Case for Outsourcing CAM Engineering
In the West, outsourcing is sometimes considered taboo and many believe it is one of the causes for shifting our manufacturing base to the East—specifically China and other lower cost Asian countries. In this series of columns, I will make a case in support of CAM outsourcing—especially for North American and Western European PCB manufacturers.

KCA Electronics and MEI Acquired by HCI; Shane Whiteside Named President and CEO
HCI Equity Partners (HCI), a middle market private equity firm based in Washington D.C., announced today that it has acquired both KCA Electronics in Anaheim, CA and Marcel Electronics International (MEI) located in Orange, CA in separate stock transactions.

All About Flex: Imaging Methods for Etch Resist, Part 1
Imaging is a major process step in creating a copper circuit or flexible PCB. In single-sided circuit fabrication, the imaging process creates the resist pattern that protects the copper from the etchant. It is critical that this pattern precisely define the circuit traces, as issues with imaging will transfer to the subsequent processes.

EPTE Newsletter: Many Plants in Japan Shut Down After Earthquake
Large manufacturing companies including Sony, Panasonic, Honda and Toyota suspended operations citing damage at their plants. The supply chain disruptions could keep them idle for up to a week. Toyota Motors was the hardest hit amongst the large corporations.

Advanced Circuits Acquires Assets of Micom Circuits
Advanced Circuits, North America's 3rd largest printed circuit board fabricator, announced today that it has entered into a definitive agreement to acquire certain assets from Micom Circuits of New Brighton, MN.

North American PCB Business Growth Accelerates in March
Total North American PCB shipments in March 2016 came in at 10.3% above the same month last year, bringing the year-to-date growth rate up to 6.1% for the first quarter. Compared to the preceding month, March shipments were up 18.6%.

Teledyne Agrees to Divest Teledyne PCT to Firan Technology Group
"The divestiture of Teledyne PCT is consistent with Teledyne's ongoing evolution and focus on high-technology, proprietary engineered products," said Robert Mehrabian, chairman, president and CEO of Teledyne. "We will work closely with FTG to facilitate a smooth transaction and transition of the operations to FTG's U.S. operations, supporting the customers of Teledyne PCT."

A Conversation with Walt Custer: Market Report
In a recent conversation, Walt Custer shared current market data and industry trends, detailing those market segments and regions that are currently seeing growth and those that are in decline. Walt also offered his interpretation of the data, which he uses to forecast the upcoming month.

SPECIAL SECTION

IPC

IMPACT WASHINGTON, D.C 2016

IPC's IMPACT Washington, D.C. 2016: Who, What, Where, and Why

All photos in this section courtesy of IPC

by Patty Goldman
EDITOR

CEOs, CTOs, VPs, presidents, upper management, even engineers and worker bees: I call out to you to read this message. Your very life, at least your working life, may depend on it and I am truly not being dramatic.

I had the opportunity in mid-April to attend IPC's IMPACT 2016 conference in Washington, D.C., and it was quite a learning experience—and I didn't even get to most of the meetings. I'm not big on government, politics, our Congress, or probably 99% of the things that go on in our nation's capital (but I do love the museums). However, I learned that we have to work with what we have. So in this special IMPACT Washington, D.C. section, I have included nine interviews I conducted with people in our industry who can tell you in their own words what it was like to be involved and what they think of IMPACT—and whether it's worthwhile for you to attend.

A very serious and determined group of your peers—top management from IPC member companies representing PCB, EMS, equipment and materials suppliers—listened carefully to IPC's staff experts on the immediate, most pressing concerns of our industry. This year, three hot issues were chosen to follow up on with members of Congress and their staff. (It is best to limit the agenda to just a few items so as to not dilute the message nor distract the intended audience.)

The three major issues addressed at this IMPACT were:

- TSCA—The EPA's interpretation of the Toxic Substances Control Act makes it more difficult to recycle chemicals like copper etchant than to simply treat and dispose. IPC's argument: "We want to do the right thing and recycle as much as possible. Do you really want to discourage this?"
- Dept. of Labor—New proposed regulations would significantly raise the baseline salary of those who can be considered exempt from federal overtime pay regulations, effectively making more people eligible for overtime pay. Plus a formula is being proposed that would continue to raise this baseline on a yearly basis, pushing many salaried employees to become hourly, with attendant time card requirements.
- NNMI—the National Network for Manufacturing Innovation is a public-private partnership that draws on the resources of the federal government, local governments, universities, research institutes and industry to accelerate manufacturing innovation. IPC is urging full funding and long-term planning for the network.

Attendees were also asked to extend thanks for passing the now permanent R&D tax credit.

Since this time it appears that the DOL has issued the new regs affecting overtime pay, not the best of news for business. However, IPC is part of a coalition to continue to educate mem-

bers of Congress on the impact this will have on our industry.

On the other hand, as of May 23, IPC's language on by-products (TSCA item) has been included in the compromise being worked out between the House and Senate. This will indeed be a benefit to (mainly) PCB fabricators, keeping recycling practical and sensible. A vote is expected in House this week and the Senate possibly next week. This is a big win for our industry and is a direct result of efforts at IMPACT.

And now to the "Why." I think I'm like most of you—I abhor politics, politicians and all things that smack of them, which of course includes at least half the population of Washington, D.C. However, ya gotta do what ya gotta do, as they say. And as John Hasselmann says, "You are either at the table or on the menu," meaning that if we don't speak up and let Washington know what is important and vital to our industry, then we are at the mercy of whatever regulations suit their fancy—or are on the agenda of the myriad government agencies and/or special interest groups (think EPA, OSHA, Greenpeace, etc.).

It became obvious to me through conversations with the attendees that some of the congresspersons and their staff viewed corporations as the enemy, though others were more open-minded. It's so easy to look the other way (or vote the other way…) when a corporation or business is far away and seen as a big blob full of greedy people who don't want to share their wealth (magically produced, apparently). But when actually sitting down face to face, suddenly that abstract enemy entity becomes real, the company president becomes a real person and then he mentions the 10,000 or 1,000 or even 50–60 people that work for him (duh, voters!), and perspectives change.

And so it was and is. One thing I heard time and again was the importance, the criticality of a face-to-face meeting with one's representatives in Congress and/or a member of their staff. More than one attendee mentioned visits to their facilities by their representative and the very positive impression it made on some. A bonus for the CEO was that the tours sometimes became a town meeting for their employees, which is definitely a win-win.

All of this happens and happened at IMPACT Washington, D.C. 2016. Many of the participants had been to IMPACT several times before but some were newcomers. IPC's Washington staff carefully prepared the agenda, the talking points, so to speak, and thoroughly coached participants on how to approach various representatives. In one case, specific "hot buttons" were to be carefully avoided. This was serious, important business. I can't stress that enough—as important as that next piece of equipment or facility upgrade, in fact probably more important, considering the number of things in Washington working against staying in business.

So don't sit back and wait for someone else to go. Start thinking about and planning for IMPACT 2017, next April. There will be a new administration, new members in Congress, and more educating to be done. New bills will be proposed. Will they be pro-business? Will they help or hinder your business?

In the meantime, contact IPC's John Hasselmann and ask him to help set up a visit or tour with your representatives at your company. Bookmark and regularly check IPC's Government Relations page for updates on legislation and other info that could affect your company, along with the latest issue of the Global Advocacy Report. Do be proactive and take part. It's good for you, good for your business and good for our industry.

I hope you find this special section enlightening and inspiring. And thank you. **PCBDESIGN**

Veteran IMPACT Washington, D.C. Attendee Matt Turpin on the Event's Benefits

I made contact with Matt Turpin, CEO of Zentech, before the first evening's dinner. We sat down to discuss what he hoped to gain by attending the event.

Patty Goldman: *Matt, I'd like to know what your expectations are of this meeting.*

Matt Turpin: The IPC does this every year, and I've been here the last five years for IPC. It's a great opportunity for the IPC and members of the IPC to meet with their local officials on Capitol Hill as well as other people on Capitol Hill and kind of deliver the message of what's important to IPC and the IPC vendor, whether it's in terms of RoHS compliance or conflict minerals, etc. This year it's Defense Department labor regulations and things like that. It's a good way for IPC to get its point across and to influence what happens on Capitol Hill.

Goldman: *How has that worked in the past?*

Turpin: It's worked out well. Some of the issues in the past have been the R&D tax credit and it looks like that is permanent at this point. I think some of the things relative to changing the narrative with conflict minerals is going slow, but it keeps the issue alive and shows that it's not the slam dunk that Dodd-Frank thought it was going to be. There are other issues where they have had some success, like the NNMI (Na-

Zentech's CEO Matt Turpin and VP John Vaughan during a preliminary meeting.

tional Network for Manufacturing Innovation) that the White House was big on, getting that properly funded and through Congress. That was a big push and it's been a big success.

***Goldman:** Do you think this is directly attributed to IPC and its members being here in Washington?*

Turpin: Absolutely, yes. The IPC are tying up and spending member dollars doing this and they've got a local lobbying group that helps them with setting up. As part of that, they're making sure that they're getting back their bucks. Every year we talk about what are we going for, and what progress to aim for. The Government Affairs Committee orders routine board calls and committee calls to find out what we're working on, what outcomes to expect and what kind of progress.

"Every year we talk about what are we going for, and what progress to aim for. The Government Affairs Committee orders routine board calls and committee calls to find out what we're working on, what outcomes to expect and what kind of progress."

***Goldman:** You've seen real progress?*

Turpin: There has been real progress. Absolutely. There's always something new.

***Goldman:** I assume there is always something you have to worry about and work on.*

Turpin: Congress is always trying to come up with new ways…

***Goldman:** New ways to mess it up [Laughs]. What do you particularly want to get out of this session?*

Turpin: The sessions have a number of different purposes. One is that it's good to get IPC members together. They tend to bring in CEOs for this event. It's good to do the networking and to find out what other people are faced with separate from the regulatory issues. It's also good for the CEOs that come to this to understand what the regulatory climate is like and what those issues are. Because I know when I first started coming, I really didn't understand all the issues that IPC was going to bat to Congress for in terms of representing their constituents within the IPC. I enjoy that aspect of it.

For me personally this year, I'm taking a more active role in helping get across the message in terms of the new Department of Labor regulations that are being proposed—as related to exempt and non-exempt status and raising the baseline salary level of those who can be considered non-exempt.

***Goldman:** Anything else you would like to say about this?*

Turpin: I would say the only other thing is that anybody who is reading this article and is aware of the IPC, or some of the events the IPC does, whether it's APEX or whether it's IMPACT or another event, if they're a CEO, it's worthwhile to get involved and to help out. It helps them personally and it helps the industry as a whole.

***Goldman:** Some people would probably say it's expensive to come here, like paying for the hotel, travel and that kind of stuff. How do you feel about the money end of that?*

Turpin: Everything has a cost. You could certainly argue that not participating also has a cost. I personally think that it's worthwhile and that on the whole the cost is definitely worth the benefit to people individually and to members as a whole.

***Goldman:** Thank you, it's nice to talk with you.*

Turpin: Thank you. **PCBDESIGN**

IPC's Hasselmann on IMPACT Washington, D.C. 2016: Why it Matters

I spoke with IPC's VP of Government Relations, John Hasselmann, immediately following the welcome dinner at IMPACT Washington, D.C. 2016. Among the topics we discussed was the importance of industry executives coming to Washington to present a collective message to policymakers.

Patty Goldman: *John, how was the dinner discussion tonight?*

John Hasselmann: Thank you, Patty, it was good. This was an opportunity for those who have been here before, and those who haven't, to network and get acquainted. We also had two speakers, Republican senior strategist Charlie Black and Democratic senior strategist Scott Pastrick, to talk about the current state of politics, not just in Washington but nationally. When you go into a meeting with a member of Congress or a policymaker, it's important to remember there are other things happening in the headlines, and they may be distracted by that. We need to be the advocate and say, "We need to talk to you about this even though there's a lot of other stuff going on out there."

Goldman: *Right, your goal is to keep them focused on your issues, and of course they've got a million other issues that they are also focused on.*

Hasselmann: Exactly. But the collective group here is a very powerful voice. When you have these executives coming into town, representing the electronics industry, and they are here together on the same page, not necessarily as competitors, that makes an impact.

IPC's John Hasselmann(L) and John Mitchell (R) pose with Congressman Bill Johnson (R-OH) with the capitol building in the background.

RF/MICROWAVE CIRCUITS? AMERICAN STANDARD CIRCUITS MAKES IT EASY!

Our experts are just one of the ways ASC makes buying PCBs easy. Your expert for RF/Microwave PCBs is John Bushie.

Have a question about designing your RF PCB? Click HERE to ask John now!

Digital | RF/Microwave | Flex/Rigid-Flex | Metal Clad

American Standard Circuits

Creative Innovations In Flex, Digital & Microwave Circuits

www.asc-i.com

Goldman: *I think a lot of people don't understand that it can make a difference. They think, 'I'm just one little person, what can I possibly accomplish? The policymakers have a million other things on their minds. How do you make a difference?*

Hasselmann: Well, your voice as an individual is important, but when there is a collective voice that is unified, it's very important. I always point out that firefighters, nurses, policemen, teachers, etc., all have representation here.

Goldman: *Tremendous representation.*

Hasselmann: Exactly, and so should manufacturers and the electronics industry. We're just using the tools in our toolbox to represent the industry and be that collective voice. You mentioned that these members of Congress and their staff have all these issues to think about, and they're all over the map meeting with tons of people. IPC represents the whole supply chain. IPC members are in every continental state, and IPC represents almost a million workers. So when you tell that story to members of Congress, they'll listen. We're their constituents in almost every district. So if we come in and we're organized, if we have a succinct message and we have credible data, we can have a real impact.

"IPC members are in every continental state, and IPC represents almost a million workers. So when you tell that story to members of Congress, they'll listen."

Goldman: *What do you expect to accomplish over the next two days? What's the mission?*

Hasselmann: This year is a little different because we're in a presidential election year. Congress has very few legislative days this year, so we have a small window of opportunity to educate these policymakers, and to lay the groundwork for the coming year with a new President and a new Congress. A lot of our goals here are focused on meeting policymakers that may be even more influential next year. So we're meeting with surrogates from the presidential campaigns tomorrow; some are current members of Congress, others are former high-ranking officials who now represent these campaigns. But we're also solidifying and still educating a lot of these policymakers about IPC and who we are and what we stand for as an industry.

Goldman: *I can imagine what happens is you get some of them educated and then they move on. You've got to keep on educating.*

Hasselmann: That's why we're here, for the industry. The 48 hours that these executives come to town for IMPACT help us do our job for the rest of the year. That's not just in the U.S.—we're focused globally—but this particular event in D.C. is very important. It's exciting and I love doing it.

Goldman: *One of the things that I hope to accomplish is to get across to a lot of people that aren't here why there's good reason to be here.*

Hasselmann: Well, thank you for that. When I started here three years ago, one of my goals was to make sure that we had a stronger communications effort. What's that cliché? If a tree falls in the woods, and no one is around to hear it, does it make a sound? I think that is what happened in the past, like there was a small segment of the industry that knew the importance of what we were doing on the advocacy front, but it wasn't enough.

Now the industry is getting more involved—through the board of directors, through our government relations steering committee, and through various other groups. This is a good thing; we need to be at the table. You know, the other cliché is: If you're not at the table, you're on the menu!

I think because our industry is so connected to government, whether it's here or in the EU or

(L-R) John Hasselmann, Nilesh Naik and Bhawnesh Mathur on the terrace during a cocktail hour.

in China, the decisions really affect our members.

***Goldman:** We all know that from RoHS, right? Everybody got blindsided by that.*

Hasselmann: Yes. RoHS, conflict minerals, and there are others. When something is coming across a policymaker or regulator's desk, or there's some sort of proposed rule or a piece of legislation, I want someone to say, "Have we called IPC?" Because that's my goal: to be at the table. There are a lot of other trade associations, but we represent the whole supply chain, we're global, we do standards with a lot of professional development and educational training around these standards, and we do trade shows. That's very unique compared to some of the traditional trade associations that are based here in Washington, D.C. who just focus on advocacy. We've got a lot of real-world experience to back up our advocacy.

***Goldman:** A lot of people probably think, and I'm just trying to be the devil's advocate here, why not let IPC do all that? Why do I need to come to town? What is the importance of having companies or company executives come? What difference does it make?*

Hasselmann: The congressional staff see us all the time, but it's much different when a CEO takes the time to call a member of Congress, or to come to Washington or Brussels.

If a CEO comes in who's on the front line every day—making decisions that affect their employees, making decisions about whether they need to have a factory here or there, who is looking at their bottom line in terms of keeping revenues up and keeping their employees and customers happy—then members of Congress and policymakers are going to understand how important it is when they make a decision, and that they're going to impact us.

If CEOs take the time to be advocates, policymakers are going to listen because they have to make those decisions, too. We're always trying to get the decision makers together in the same room to come up with a plan and make it work. And that's the goal.

***Goldman:** And if your members are not there?*

Hasselmann: Then you can't complain.

***Goldman:** That is also true; how do you complain about something if you didn't engage or take part?*

Hasselmann: That is the beauty of this. I'm a firm believer in petitioning the government, and under our Constitution, we have the ability to do that. Without their trade associations, the nurses, firefighters, engineers and CEOs aren't going to come to D.C. all the time, so we try to bring that opportunity to our members.

Let me give you another example. We instituted a program called "Meet the Policymakers," in which we bring them into the factories and facilities and let them see all the innovation that is happening and to meet the workers. In the last few years, we have done over 30 or 40 site visits with members of Congress and our member companies around the U.S., and we have done some with our members in China with Chinese officials as well.

***Goldman:** So they get to meet the voters.*

Hasselmann: And the workers are their voters, exactly. It's a two-way street. When they see the factories they're just like, "Wow."

***Goldman:** They probably can't even conceive of what goes on in there either.*

Hasselmann: They're usually very interested, and when we get the employees together, they are thrilled to be able to engage with them. We'll set up meet-and-greets or town hall meetings and just let them ask whatever they want, and that brings the advocacy opportunity to the employees, too.

***Goldman:** If any IPC members called up and said, "Hey, can you arrange something with my congressman and help me get them into our factory?" That can actually happen?*

Hasselmann: Yes, we will do our best, no question. That's why we're here. But it's not all happening here, it's really happening out there—all the work, the innovation, the R&D, and the jobs. We're just trying to make sure that policymakers are educated and that the legal, regulatory and legislative environment is conducive to continuing to grow our industry.

We want to be at the table. We want to be able to debate the pros and cons of anything that may have a negative or positive impact on the industry. We want to be a stakeholder, and we're going to be that voice. We're making a lot of strides, and this event is really helping us to do that.

> "We're making a lot of strides, and this event is really helping us to do that."

***Goldman:** How many congressmen will you guys be seeing over the next few days?*

Hasselmann: It's a two-track approach. As a group, we have about ten key meetings with members of Congress and administration officials. But then we also arrange individual meetings for our attendees with their representatives where they have facilities. So they can go in and establish those relationships and we facilitate that, and that allows us to spread the word even more.

We probably have 30 to 40 of those individual meetings, where it's an opportunity for that CEO or that executive to talk about very specific issues of concern to them and whatever they want to raise. The policymaker wants to know, "What's keeping them up at night? What can I do to be helpful? What do you need? What can I do to cut some red tape for you?" They want to do that in a heartbeat for businesses in their district.

This is the opportunity for our members to talk about those very local issues, but also we're there to talk about some of the broader issues where we can come back and work with them

because it's an opportunity to help their constituents. That's where we're trying to connect dots.

***Goldman:** Connecting dots is a good way of saying it.*

Hasselmann: I get excited about this, as you can tell. [Laughs]

***Goldman:** Well, like you said, this event is only one part of your job overall, but it's going to make your job easier.*

John Vaughan of Zentech told me he's been coming for six years, and in the first few years he didn't really notice much, and then he started seeing stuff happen, like the things that you guys were advocating becoming some real success stories. Things don't happen right away though, it takes time. These guys can't just come here and then expect next month that something's going to happen as a direct result. It doesn't happen that fast.

> "We try to help facilitate that conversation and try to offer policy solutions that help the industry."

Hasselmann: I think they get it. For the most part it is like long-term planning, like when these CEOs have to look ahead with a five-year plan. There's a new Congress here every two years, they roll out an agenda, and we have an agenda, too. Where do they mesh? We try to help facilitate that conversation and try to offer policy solutions that help the industry.

***Goldman:** I was just thinking there are some very loud voices here in Washington these days. You probably can't get louder than them, so you just have to be that voice of reason in there.*

Hasselmann: Oh yes, and I think that's what we are. Our issues are bipartisan. We have great market research on the industry, we have data, and we have information that backs up what we are advocating for. For example, when we're talking about some of these environmental issues, we have data that explains why we think this approach is better than that approach. Policymakers and their staff thrive on all that. Instead of just rhetoric that people get so wrapped up in, we try to be that voice of reason because we can bring in the data that supports our message, and that's my role: To make sure it's bipartisan and that we have the meat behind our message.

***Goldman:** And does that get through?*

Hasselmann: It does. The serious policymakers appreciate it, and they'll come back to us. With all the engineers and the technical and compliance people involved with IPC, we can go and ask them, 'How would this regulation affect you?' Then we have the execs and CEOs to weigh in and say, 'Okay, we think this will be good for the industry, let's go.' I like that we can draw on the expertise of all segments of a company.

***Goldman:** I noticed, too, that a lot of the issues on your agenda, like conflict minerals, are not just challenges in our industry. It's not like we're saying do this for me, me, me. IPC's agenda is good for business, for the industry, and for the country.*

Hasselmann: Exactly, it's even more powerful than just us. We're working to be seen as a leader among the various industry groups. It's very important to form and be part of broader coalitions of industry groups to bring more voices to the table. Coalition building is critical, and we have a lot of organizations that we work with, here and abroad, depending on the issue. We have members of IPC that are members of other organizations, and they want us to work together and achieve more synergy. They understand that the collective voice among different industries is just as important for getting something done.

***Goldman:** John, I really appreciate you taking time to speak with me today. I wish you the best in the next 48 hours.*

Hasselmann: Thank you, Patty. **PCBDESIGN**

STI Electronics Participating in IPC's IMPACT Washington, D.C.

Dave Raby is president and CEO of STI Electronics and an eager participant at this year's IMPACT Washington, D.C. event. We talked early on the first day of the event, before a heavily scheduled day for the attendees.

Patty Goldman: *Dave, what are your thoughts about this year's IMPACT event, overall?*

Dave Raby: I'm excited. I came to this event for the first time last year and have been looking forward all year to coming back. It's really great to meet the other people in here. It's all senior executives from companies in our industry from all over the country. Whenever senior executives from the same industry get together, we usually find out we share many of the same concerns and that is true with this group. Washington, D.C. is also a foreign world to most of us, and it's great to get up here and see what's going on here and see what our representatives are thinking. What is even better is, through the efforts of IPC, we can actually have an influence on some things and can give them our opinions. As Americans, it is what we're supposed to do but I don't think most of us do anything unless there's a group like this supporting it and organizing it.

Dave Raby

Goldman: *Yeah, there's some reluctance.*

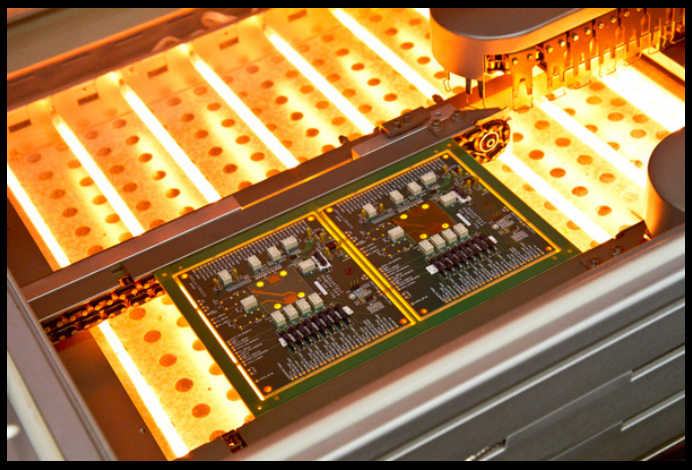
Raby: It's hard to come up here as a lone wolf and just say, "Hey, I want to support this bill."

Goldman: *First of all, how would you get to see anybody?*

Raby: That's a good question but also how would you even know what bill was out there? IPC does a great job of scheduling visits with key people from all over the country and also with the representatives from my states. STI has employees in four states and tomorrow I'll be visiting both senators from one of the states and our U.S. representative from two others. IPC staff does a great job of keeping us informed on the legislation that is at various levels of the process and gives us a very good idea on where the people we are meeting with stand. We can then express our views and let them know we appreciate their support and encourage more support, or let them know why we see their particular stance as a problem. My opinion may or may not change their opinion but I've been impressed with how they really will listen and consider how a certain piece of legislation will affect my company and their constituents. It was fascinating to me when I came last year just to see how the government works. We often complain about how it doesn't work, but in re-

Silicon Valley's LEADING Full Turnkey PCB Manufacturer

FABRICATE & ASSEMBLE – ALL IN HOUSE



FABRICATION

- Rigid PCBs 2-60 Layers
- Simple to Industry-leading Technology
- Quick Turn Prototypes and Production
- Proprietary Wrap Plating Process for Unsurpassed Reliability

ASSEMBLY

- Latest FUJI Equipment
- 100% SPI Standard
- 100% AOI Standard
- 100% 2D/3D X-ray Standard

To Request a Quote through our website and enjoy NO tooling, programming, or stencil costs, Click Here!

GORILLA circuits

www.gorillacircuits.com

ISO 9001:2008 – MIL PRF 55110/31032 - ITAR

ality, it has done a pretty good job over the last 240 years or so.

Most of the legislation starts for a good reason. I compare it to my shower thoughts at home. I've come up with what seemed like some of the world's greatest solution or idea as I'm getting ready to go to work in the morning. I'm so excited when I can get to the office and tell my staff about them. Most of the time, as soon as I start saying them out loud I realize that it may actually be the dumbest idea I've ever had. The legislation starts for a good reason and seems like a good idea at the time and, by the time you get through the process that they go through, there's hopefully people like us that have come in to say, "Yeah, this really is a good idea but could you add this to it?" Or, "We really don't like this. Have you thought of what this would do to business? While it seems like a good idea overall, maybe if you took out this line, then it really would do what you want." Right now, you've got the unintended consequences that are going to happen.

***Goldman:** You really have to think about the consequences.*

Raby: Exactly, and no offense to the lawmakers but they are trying to improve our lives and businesses but usually have no idea how they work and some of the unintended side effects some of their laws can have. Just making up an example here but there could be something that will save my company $10,000 but causes me to have to hire a full time person just to fill out the paperwork. That didn't save me $10,000. That put a productive person out of work. That's a big reason for being here this week.

Just as a citizen, it's frustrating to watch the democrats versus the republicans, republicans versus democrats. If I have an idea, no matter how good it is, you're not going to like it, and vice versa. It's nice to get up here and see they actually do talk to each other (or at least many of them do), and there are some things that they cooperate on.

***Goldman:** There are things behind the news that happen.*

Raby: Right. Apparently, talking to each other doesn't make the news.

***Goldman:** That's for sure; it's not as exciting. Do you think you'll come next year? Do you see this as a good event to attend?*

Raby: Yes I do. If it's anything like last year—and from the schedule we have, it looks like it will be—I'll be back next year. I also want to encourage other business leaders and owners to do the same. It costs two or three days of time plus your travel expenses and can have a direct benefit on your company's and your industry's future as well as your country's future. Plus, it is interesting, educational and we have some fun, too.

***Goldman:** There's a pretty heavy schedule this year, from what I understand.*

Raby: We're hearing from four different presidential campaigns this morning, which is something that of course we didn't have last year. It may not be a kind way of saying it, but it's a straight-from-the-horse's-mouth type of thing that you don't get sitting at home on the couch. It'll be interesting to hear how that goes and we get to question each candidate on their thoughts on keeping (or making) American manufacturing competitive with the rest of the world. Then we're also meeting with several congressmen and it will be great to get their perspective on things, and also give them our perspective.

One of the things I'm talking about is the NNMI, the National Network for Manufacturing Innovation. We are meeting today with Senator Orrin Hatch, who's the head of the Senate Committee on Finance. Senator Hatch is number three in line to be president, as far as a succession plan.

***Goldman:** That's not somebody you can just knock on his door and talk to.*

Raby: I'm from a little town in Alabama. That's not somebody I'm used to talking to. I'm a little nervous about that, but I also know from last year, IPC will get me through it.

Congressional meeting.

Goldman: *Right, they keep you well-informed of all the important stuff.*

Raby: They've educated me, but they're also going to be sitting next to me. If I start to stumble, they'll steer me in the right direction, or if the senator asks a question that I don't know the answer to, I just look over and they'll help to answer it.

Goldman: *And John Hasselmann said they have all the data ready as well. So they'll have a lot of answers.*

Raby: John and his entire team do a fantastic job.

Goldman: *Okay. It's probably about time to head in to the meeting. Well thanks so much for your time, Dave.*

Raby: Thanks Patty, I appreciate you being here, because publicizing this can do nothing but help it.

Goldman: *That's what I hope to see happen also.*

Raby: IPC does a great job on this. There's nobody else up here looking out for us in the electronics industry. **PCBDESIGN**

Creation Technologies on IMPACT Washington, D.C. 2016

Meeting with congressional leadership, peers, competitors—it's all valuable, according to Bhawnesh Mathur, president and CEO of Creation Technologies. Here are his thoughts on the benefits of the event after the close of the first day of IMPACT Washington D.C.

Patty Goldman: *Bhawnesh, tell me about your day here at IMPACT. How was it for you?*

Bhawnesh Mathur: There are so many good things that come out of these annual IMPACT events. First of all, I feel like it's always important to partner with our political leaders. Their interests and our interests are the same. We want to see the economy grow. We want to find jobs. We want to move technology forward. I think when people have aligned interests, they should meet, talk and find ways of working together. It's always good to do that.

The specific issues that we talked about today will help us get better aligned. It's not an overnight thing, but for the last several years that we've been coming here we've learned to speak with each other. We've developed credibility with each other. We've actually shared accomplishments with each other that we can celebrate and so on. Every time we come here I think we move the ball forward and that's a good thing.

I also enjoy meeting all of my peers, and suppliers and competitors. I think it's pretty cool that we can be competitors in the morning and come here and work together on behalf of the industry, and that benefits everybody. Obviously we're trying to win against each other on certain days, but if our industry doesn't win, we don't win. I feel a sense of urgency to do everything we can to help the electronics and manufacturing industries grow and get some of the benefits.

Goldman: *How many years have you been coming to IMPACT?*

Mathur: I think this is my fourth or fifth year.

Goldman: *Do you recall what prompted you to attend the first time?*

Mathur: I do remember. I was in Denver, Colorado then, and I thought if I met my politicians, my senators, and my congress people, I could do more with them in Denver. I came and advocated on behalf of my company at that time. I told them, if you help me I can create jobs here, and I asked them if they could help my company get economic relief in this or that way. And we started a dialogue, and it actually helped. It didn't solve all of our problems of course, but it was enough to get me started. I also began to network with their connections and that helped too. So I definitely benefited from attending the first time.

Goldman: *So you came back the second time?*

Mathur: I came back, and now I manage a company that's in 12 different locations around the world, with nine in the U.S. and Canada. So I find benefit. Representative Paul Ryan visited our Milwaukee manufacturing facility the week before he got named to Speaker of the House, and we told him we could use some help with an OEM that we wanted to partner with. He said, "I visited your facility, I spent the whole day here and I liked it all. I'll talk to those guys." And I don't know if that played a part, but we did end up winning the business in the end.

Goldman: *Have you had congressmen come and visit in any of your other locations?*

Mathur: Yes, we like to invite them—there are lots of benefits to that. First, they get to understand who we are and a little about our industry. One of the benefits, which is really at the top of our list now, is when people like Mr. Ryan visit, is giving our people a chance to meet local leadership and ask questions at a town hall meeting. They likely would not otherwise get this chance. We may have anywhere between 200–500 people in a Creation business unit and this is a way for us to bring the community to our team. That's another benefit of coming to work with our company.

Goldman: *So actually getting a congressman to your facility came about because of coming to IMPACT?*

Mathur: Yes, when we meet these fellows, we're always trying to figure out the next step, and one of the next steps is to say, "Why don't you come visit us?" That actually became an initiative that John Hasselmann and his team run now. We measure how many visits we have, we have targets and a process. We've invited Prime Minister Justin Trudeau of Canada, we've invited President Obama, and he wrote us a letter back saying "No thank you, but Penny Pritzker, the Secretary of Commerce, will be available to come meet with you."

Goldman: *Nice!*

Mathur: Yes. We're finding that we get some recognition, our people can meet the leadership, and that's a good thing.

Goldman: *And conversely, your congressman learns a little bit about you and about the industry.*

Mathur: Absolutely, that's right. Many of them completely understand what we're doing and some of them have no idea what we do. It becomes especially clear when they take a tour, though, because it's hard to explain exactly what we do in a conference setting like this.

Goldman: *A picture is worth a thousand words, but I'm sure a tour is worth a gazillion words.*

Mathur: For sure. When you see 200–300 people making state-of-the-art medical equipment, which is an area Creation specializes in, and you see an incubator we made where a baby is brought after she is born, or a CAT scan machine that we designed, or an ultrasound machine that we worked on, that's very powerful.

Goldman: *It sounds like lots of good things happened today then.*

Mathur: Absolutely.

Goldman: *How many congressmen did you meet with?*

Mathur: Formally, three.

Goldman: *Of course everybody has their own three that they meet, I guess.*

Mathur: Everyone has their own three. I feel like because I've been coming to IMPACT for a while, I try to not be as vocal. I think others need to participate and feel like they've been a part of it. But our message is getting across, and I certainly believe the view that it's the authentic person who makes the biggest difference. You can't just hire someone to come in your place and represent whatever it is you believe in.

Goldman: *Excellent! Any final thoughts?*

Mathur: I think politics are always dynamic. One of the things I'm learning is that while we've done all of this, you can't ever stop or let your foot off of the accelerator. If anything, I think we need to do more of it in more places. Creation Technologies is a global organization with global customers, so there's Europe, Asia and other places where we can get involved and make a difference.

Goldman: *Bhawnesh, thank you so much.*

Mathur: Thank you. **PCBDESIGN**

IPC is One Thing, but Constituents are Quite Another

As I was unable to attend the actual meetings with the representatives, I wanted to catch the thoughts of those who did. I've known Nilesh Naik, CEO of Eagle Circuits in Dallas, Texas, for many years. We sat down for a chat after the first day's events.

Patty Goldman: *Nilesh, tell me how things went on the first day of IMPACT.*

Nilesh Naik: It's been a great day. Unfortunately I've missed Capitol Hill Day, or IMPACT, for the last two years, and was excited and glad to make it this year. It's just a great opportunity to visit with senators and congressmen. The exciting thing is they passed the R&D tax bill last year, so it was a good chance to finally say thank you. Interestingly, we've been on the Hill for the last seven or eight years asking for it to become a permanent tax credit. It's neat to see the process come to a complete end, and actually have a permanent tax credit. So I'm excited from that point of view.

It's quite amazing; every time we come to Capitol Hill, whether it's the legislative system, the congressmen or the senators, they do genuinely want to listen and hear from you. It's always great to present your perspective and situation when you get a chance to talk to them. They say, "These are my constituents," and they do listen.

Goldman: *So it's been worthwhile for you to attend?*

Naik: Absolutely worthwhile.

Goldman: *You get to meet with your own congressman tomorrow, is that right?*

Naik: Yes, and I'm looking forward to that. We've been fortunate enough to have Congressman Johnson come to our factory, so that was just even more fun. The Congressman has actually seen the manufacturing process. He's touched the product and talked to our people.

Goldman: *They actually begin to understand…*

Naik: They absolutely understand. Surprisingly, they do know a lot of what's going on. The challenge for them is they've got numerous other issues that are also going on. They're looking at a bigger picture and still saying, "Hey, how do we solve your problem?"

Goldman: *Your congressman knows what's going on. He's been to your factory, but what about all those congressmen who represent those companies that are not here? What do they know about your business or circuit boards and our industry?*

Naik: It's a great question, and one of the things that always concerns me. It is disappointing—that we don't get more of our member companies out here. This is the one time of year where you get a chance to meet your congressman, and you get to talk about your industry. It's amazing, they're eager to listen and learn, but you've got to come tell them. If you don't, you miss out on it. You've got to build those relationships.

Goldman: *It's a great opportunity to get your voice heard.*

Naik: Absolutely. They hear you, and today there were very positive responses with all the congressmen and senators that we spoke with. They're all for it. They understand what we're talking about. Heck, they want manufacturing

back in the United States. It's not like they want to get rid of it. When you can point out certain things that are hurting U.S. manufacturers, they're going to listen and they're going to see what they can do to fix that.

Goldman: *That's good.*

Naik: We had a great talk about that, and we had a great talk on TSCA. One very concerning topic that will affect all the PWB guys is the changes in regulations that are being proposed by the EPA. They're trying to get us to document all recycled material. The way the EPA has proposed the regulation, they are actually disincentivizing the PCB shop to recycle product. It would be cheaper for us to go send our waste streams to the landfills. This is totally counterproductive.

Obviously, recycling is the best way to do things, but the way it's being written and the way it's being proposed by the regulators right now will actually be a disadvantage. Again, this is a great conversation to share with the senators and congressman and say, "Wait a minute, that doesn't make sense. Why are we doing this?"

Goldman: *How far along is that? How critical is it to get that message out right now?*

Naik: I believe it's something that is going to come up within the next few months. The EPA is close to the end of writing their regulations. It's something that all the board shops need to be aware of or else we will just be creating more work.

Goldman: *They all should at least be calling their congressmen right now.*

Naik: Without a doubt. The challenge is exactly that. If we don't bring it to their attention, then they don't know how to fix it. Then all of the sudden you've got the EPA who just goes and creates all this stuff. If you want change, you have to be a part of the change.

Goldman: *You've got to have the answers for them.*

Naik: Absolutely. From that point on, the IPC does a fantastic job, with Fern Abrams and the whole team that John Hasselmann has here. They've got a hand on all the issues affecting us as an industry, so they can give you that feedback and answers. But at the end of the day, the congressmen don't want to listen to IPC staff. They want to hear from their constituents. The constituents have to show up. Actually, they can even just make a call. Even if they call the congressman, that's a huge thing. One interesting thing we learned today was even just a hundred call-ins about a particular topic will actually change the needle and may even change the direction a congressman would go. We do have to get active if we want to have change.

Goldman: *I imagine face-to-faces are even more persuasive. If a number of people get in touch with the representative in their district and get a face to face with them, that's got to have even more sway.*

Naik: A face to face is totally worth it. If a congressman is having a town hall meeting in your district and you go see them, it makes a huge difference. Especially if they hear the same message a couple of times, they are going to come back to D.C. and talk to the legislative assistant and say, "Okay, this is what I heard back in the district, so what are we doing about this?" Your voice is heard, but you've got to speak up.

Goldman: *What are you expecting for tomorrow?*

Naik: I'm looking forward to seeing Congressman Johnson tomorrow and possibly Senator Cornyn. I'm looking forward to both of those meetings. Again, just see if we can further the message and continue asking for their support.

Goldman: *Excellent. Thank you so much, Nilesh.*

Naik: You're welcome. **PCBDESIGN**

The Many Reasons why People Attend IMPACT Washington D.C.

At the end of Day One, spirits were high within the IMPACT group. They had heard from the top presidential campaigns, met with key representatives on the top three issues, and were now relaxing on a private terrace with a great view of the U.S. Capitol building. Everyone was looking forward to a pleasant awards dinner. I had the opportunity to converse with Tom Edman, CEO of TTM; Ed Moll, VP of Viscom; and Tim Redfern of Redfern Associates, who was at the event representing Insulectro. Not able to convince them to give up their spots on the terrace, the four of us had a great chat on the spot.

Patty Goldman: *Tom, why don't you begin by telling me how your day was? What did you learn?*

Tom Edman: It was a good day. Number one, I thought the morning was very interesting. While we didn't get a chance to hear from the presidential candidates, we had a chance to hear from their proxies. I thought they did an excellent job of representing the candidates, and to hear it in person and hear some of the positions that they've taken. That was very interesting.

Tim Redfern: I also found that interesting. I have never experienced anything like that before, talking to proxies, and getting the experience of both the democratic and the republican campaigns was an interesting comparison. I think the importance of selecting the right proxies came through today.

Goldman: *That's good. How about your afternoon sessions?*

Redfern: In the afternoon we actually came over to Capitol Hill and had meetings with four different congressmen and senators. We got a

(L-R) Tom Edman, Tim Redfern, Rick Lies, and Ed Moll.

chance to really get up close and personal and actually have real dialogue, which I thought was a great opportunity and in some cases very timely for what's happening right now.

***Goldman:** Is this your first time to IMPACT?*

Redfern: Yes, it is my first experience here.

***Goldman:** Why did you decide to attend?*


Tim Redfern

Redfern: Mikel Williams inspired me, and he's not even here! He's been encouraging me for the last four or five years to participate and get involved in this event. I talked with him a couple weeks ago and told him I was coming and he said, "I'm not going to be there this year." He's always attended and has been very involved on the IPC Government Relations committee. But I'm really happy to be here. It's a great opportunity to see how this side of the business works.

***Goldman:** Ed, is this your first time here?*

Ed Moll: Yes, it is.

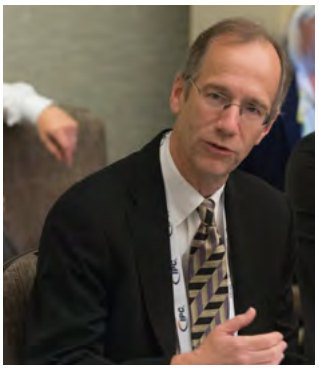
***Goldman:** And what are your impressions?*


Ed Moll

Moll: I've loved it. This has been very informative for me. I've never done anything like this and this is the first time I've had the opportunity. I'm here because my boss, Carsten Salewski, couldn't make it because of business travel, so he asked me to attend for him and I jumped at the opportunity.

***Goldman:** Even from the very start this morning there was some serious discussion about which topics to discuss and the agenda—and that was impressive in itself. Some of us don't always pay attention to this stuff, and there's a lot that goes on behind the scenes that affects everybody. That was surprising to me.*

Moll: The entire day was eye-opening for me. Meeting with Congressmen or their staff makes you feel as though you are making a contribution to our industry. I look forward to the meetings scheduled for tomorrow. It's been more than interesting being here.


Tom Edman

Edman: I was one of the spokespeople for the TSCA reform and I think the timing couldn't have been better. I've come to Capitol Hill not with IPC, but with other organizations before, and we never had this kind of timing. We have a bill that has been passed by the house and the senate but was then sent to conference. So we had a chance to directly impact some of the wording, we hope, that will go into the final bill. I think that is an unusual opportunity and definitely an opportunity on something that we feel is very important to the industry. From that standpoint I think we had a very good day.

***Goldman:** What do you plan to do tomorrow?*

Edman: Tomorrow we're off to the individual meetings with each of our local representatives where we expect to have more focused discussions about where we operate and more local issues. Today we were focused on broader industry issues that IPC had set up in advance for us to discuss.

***Goldman:** Thanks very much guys.*

Moll: See you next year. **PCBDESIGN**

Making Connections at IMPACT Washington, D.C. 2016

Rick Lies is CEO of Chemcut, an equipment supplier to the PCB industry. He's a veteran of the IMPACT Washington, D.C. events and shared his experiences at this year's gathering at the close of Day One.

Patty Goldman: *Rick, we're at end of Day One here at IMPACT. I've been busy getting everybody's thoughts on the day. How did things go for you today?*

Rick Lies: It was definitely a full day. As usual, IPC has done a great job of putting together a group of speakers and meetings for us. It's always interesting and informative to come to Washington D.C. and see what our legislators and our representatives are doing and what their thought patterns are.

I've been doing this for four or five years, but this year was a little bit different because we had the surrogates from the different candidates come in. I don't think they changed my mind on any of them, but they probably reinforced some things that I already was thinking. They definitely reinforced the fact that things are not going completely smooth for either party.

With the meetings that we had, IPC did a good job with the talking points, which are important. The companies here mainly come to support IPC and the customers—the people we sell to. A lot of the things that they were going after, like the Toxic Substance Control Act

Rick Lies

Congressman Bill Johnson (R-OH) addresses the IMPACT 2016 participants after dinner.

(TSCA) don't directly relate to us, but if it affects their business it ultimately affects our business. So for them to be successful we need to modernize it. Other issues like the R&D tax credits are important to us, where they need to increase the tax credit from 14 to 20%.

***Goldman:** What would you say to somebody who is thinking about coming to IMPACT next year—or not thinking about it?*

Lies: I definitely think it's worthwhile to come, if for nothing other than to support the industry. Again, IPC does a great job of identifying and developing an action plan for the issues that are important for the industry, our survival, and our growth in the future.

***Goldman:** Have you seen results over the five years you've been attending?*

Lies: Definitely, there have been good results. The R&D tax credit was made permanent, and TSCA is seeing legislation moving forward that will modernize it.

***Goldman:** Fantastic.*

Lies: What's interesting is I think people need to come up here to reinforce these things with their own representatives. It has helped me develop a relationship with our local Congressional Representatives and Senators.

***Goldman:** So you actually could invite them into your factory.*

Lies: In 2015, we had our Congressman Glenn Thompson, who represents our district, take a tour of our facility in State College. Since then he has actually called us to set up another visit.

***Goldman:** That's great. What are your expectations for Day 2?*

Lies: The first day is always sitting down and everything is very formal, you have your meetings you go to. Tomorrow is more the individuals to meet their own representatives and senators.

***Goldman:** Now will you meet anyone new tomorrow, like your senator?*

Lies: I'll meet with my representative, Congressman Thompson. We're not a large community so he knows many of the people that work for Chemcut.

***Goldman:** But still, there's nothing like having that relationship. Thanks so much, Rick.*

Lies: No problem. Thank you, Patty. **PCBDESIGN**

A First-Timer's Perspective on IMPACT Washington, D.C. 2016

I met with Faisal Pandit, president of Panasonic Factory Solutions Company in Illinois, for a quick chat about his experience at IMPACT Washington D.C. 2016.

Patty Goldman: *Faisal, how was your day? I'm curious about what you got out of it and what your experience was like.*

Faisal Pandit: This is my first year at IMPACT and I've got to tell you it has been very exciting and a great opportunity for me to take in a lot of valuable information. I've been in the electronics manufacturing industry for more than 25 years and there are certain serious impediments affecting the growth of this industry in North America. It's important for us to take a very proactive stance in trying to remove those impediments if we are to ever drive any meaningful organic growth.

So an opportunity to interact with our leaders who make decisions for us is a wonderful thing. IPC put some serious issues on the table and the congressmen listened. Ultimately, when you tie the growth of manufacturing—or the importance of manufacturing—to job creation, that resonates well with politicians.

Goldman: *Somehow they just don't get that until you tell them.*

Pandit: Right. They may not necessarily link it otherwise. I think that worked out quite well in terms of communicating the message and getting that going, but as somebody said earlier, in Washington things move at an incremental pace. There are no revolutions or anything major right away.

Goldman: *Right—you are not going to see anything tomorrow.*

Pandit: It's a matter of continuing to raise your voice and having these interactions from time to time, but overall it was a great day.

Goldman: *So what made you decide to come this year?*


Faisal Pandit

Pandit: I'm on the supply side of the industry and in the past I didn't really think about attending. But this year was different because I'm personally a big advocate of reviving manufacturing in North America, and we as a company are trying to work with some private and public partnerships to help enhance the manufacturing skillset in North America, which I consider to be a major impediment to the growth here.

I know a lot of people are focusing on STEM programs and things like that. We are in the early stages of trying to put a focus on the manufacturing skillset within community colleges, within high school programs and things like that. We are trying to see what we can do as a company, and I think it would require some level of support from various levels of the political establishment. By coming this year, I wanted to get a sense of what people are talking about in terms of political issues and get an understanding of the process and how we can leverage these contacts and build up on what IPC is doing.

Goldman: *Did you meet your objectives or your expectations?*

Pandit: Absolutely. I learned a lot about what IPC is doing on the regulatory side and on the political establishment contact point of view, and I think it's very positive. It did meet my objectives. I think IPC has strengthened its focus on becoming a very powerful voice for the industry.

Goldman: *That's excellent. Would you return next year?*

Pandit: I look forward to being here again.

Goldman: *Thank you so much, Faisal.*

Pandit: Thank you. **PCBDESIGN**

Shaping the Issues that Matter Most at IMPACT Washington, D.C. 2016

IMPACT Washington, D.C. 2016 encompassed two intensive days of meetings for the participants. I tried to catch their thoughts at various stages of the event. Optimum Design Associates' VP and General Manager Everett Frank was happy to provide some concluding thoughts.

Patty Goldman: *Everett, here on this final day of IMPACT Washington D.C., what are your overall impressions of your time here and how it's been for you?*

Everett Frank: It's been wonderful. I think this is my fourth year attending IMPACT.

Goldman: *It seems everybody comes back to IMPACT.*

Frank: Yes, most people do. There's a very high return rate. It's a great opportunity to connect with what's going on in the industry, particularly from the regulatory perspective. We spend a good portion of time advocating for industry issues, both with departments of the administrations and with members, and so it's a very good opportunity to impact those issues.

Goldman: *IMPACT is a good name for the whole thing.*

Frank: Exactly—you can't overuse that term.

Goldman: *How were your meetings yesterday?*

Frank: Really good. It's always interesting and fascinating to look at the different perspectives and where they're coming from. As an example, we were on the labor issue quite a bit yesterday, and trying to relate our issues differently versus republicans and democrats and how we present our business needs in a way that resonates with both sides.

Everett Frank

Goldman: Do you feel there were accomplishments yesterday?

Frank: Well, I guess the proof is in the pudding, but they were well received. We certainly consistently hear back from the members that these kinds of conversations are productive to them and that they're impacted by them. They listen, I think. You always kind of wonder how much, but I do think it moves the needle and that us being here reinforces the industry's messages.

Goldman: *How about this TSCA issue they keep talking about?*

Frank: You know, honestly, it's not a direct impact to my business.

Goldman: *Just to your customers or suppliers?*

Frank: It does impact my suppliers. In our industry, PCB manufacturers who buy the chemicals are the ones who are impacted. So my supply chain is impacted. I buy from those guys, but I don't sell to them.

Goldman: *That's important too.*

Frank: It's very foundational. I mean really everything in electronics rests on what the PCB fabrication guys do. I was joking with some of them yesterday that represent those interests that they needed to speak more highly of what their companies do when they introduce themselves. Because some of these companies in the room are just foundational to the technology in our country and in our world. The things that the chemical and PCB fabrication guys do are just so important. Like Tom's company, TTM, is the largest company in the world

of PCBs, which are literally the backbone of everything that happens. Then companies like Isola are the raw goods material suppliers to them. Isola is the most innovative company in PCB materials in the world. It's not just some company called Isola. These companies are just crucial to technology development.

Goldman: *You probably want to point to your congressman's little phone and point out they wouldn't have that without us.*

Frank: It's really true. To the congressmen and women's credit, the blur of issues and the range of issues that they face is just stunning. By and large they engage, they understand, and they know the issues we come in and bring up. It's not some obscure thing they have never heard of.

Goldman: *Anything else you'd like to add about what's been happening the last couple of days? My impression is that everybody here is very focused. This is no lark down here, this is serious business, and very important business at that.*

Frank: Yes, our government relations staff at IPC and the Prime Policy group just do an outstanding job of shaping the issues. Obviously the industry can sort of highlight the main issues, but if you were to look across the range of things, there are hundreds of things going on in the regulatory world that affect our industry, but they do a great job of shaping and funneling us to the critical things. There's an interplay there, too, between what's important to the industry and what we can actually have an impact on. For instance on TSCA, that's an element where we can actually make an impact on certain things.

Goldman: *Fight the battles that you can fight and win.*

Frank: These things are just the craziest things too, like in the past there's been language that gets inserted into various types of regulations that might take years to settle. For example, we spent a couple of years on very subtle language related to how PCBs are delineated on the U.S. Munitions List, which controls ITAR classifications. Two or three years over literally one sentence. But to our industry and to our PCB manufacturers in particular, that language was very, very important. But unless we tell Congress, how would they know?

Isola's president Jeff Waters (L) with Everett Frank, VP/general manager (R) at Optimum Design Associates.

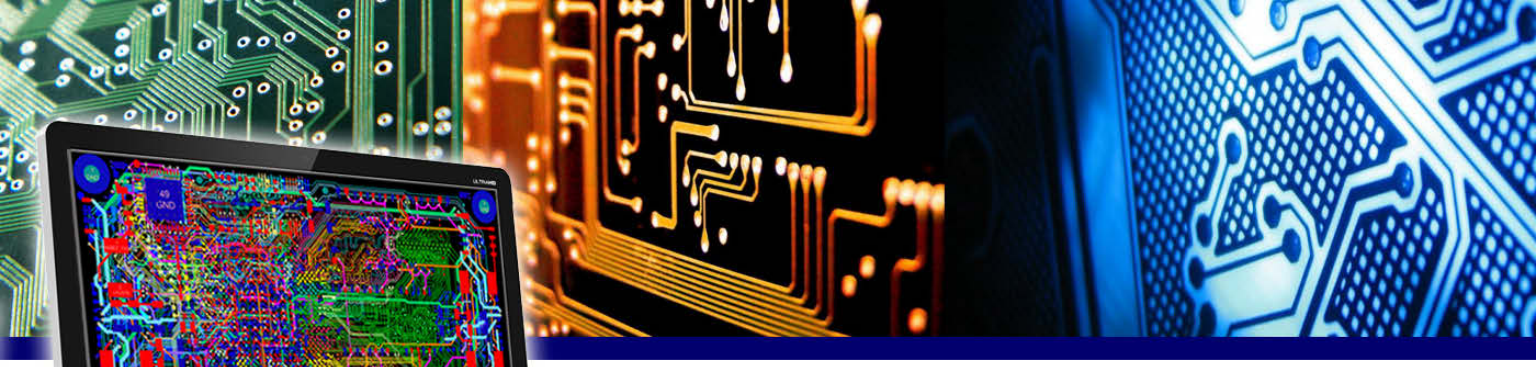
Again, our government relations team and Prime Policy put us in the right meetings at the right times. We were in meetings over the years with not only members and committee staff, but with staffers at DoD and in the White House who were specifically controlling that language. The opportunity to do that is impressive. If you step back and think about it, it's impressive that IPC is coordinating an effort to target us at issues that can really make a difference in our industry.

Goldman: *It impacts the everyday workings of your companies.*

Frank: They do a great job with that. This year they have us focused on the Department of Labor and the classifications of direct versus indirect employees. What the Department of Labor is talking about doing is just crazy talk and frankly I wasn't really that aware of it until IMPACT this year. They've pointed it out to us as something that we can go and be heard on.

Goldman: *Everett, thank you so much.*

Frank: Thank you. **PCBDESIGN**



BOARD LEVEL SIMULATION SPECIALISTS

ICD Stackup Planner - offers engineers/PCB designers unprecedented simulation speed, ease of use and accuracy at an affordable price

- 2D (BEM) field solver precision
- Characteristic impedance, edge-coupled & broadside-coupled differential impedance
- Unique field solver computation of multiple differential technologies per stackup
- Heads-up impedance plots of signal and dielectric layers
- User defined dielectric materials library - over 28,000 materials up to 100GHz

ICD PDN Planner - analyze multiple power supplies to maintain low impedance over entire frequency range dramatically improving product performance

- Fast AC impedance analysis with plane resonance
- Definition of plane size/shape, dielectric constant & plane separation for each on-board power supply
- Extraction of plane data from the integrated Stackup Planner
- Definition of voltage regulator, bypass/decoupling capacitors, mounting loop inductance
- PDN EMI Plot with EMC Limits. Frequency range up to 100GHz
- Extensive Capacitor Library – over 5,650 capacitors derived from SPICE models

iCD
In-Circuit Design Pty Ltd

LEARN MORE

www.icd.com.au

Mastering "Black Magic" with Howard Johnson's Seminars

by Barry Olney
IN-CIRCUIT DESIGN PTY LTD / AUSTRALIA

Dr. Howard Johnson, the world's foremost authority on signal integrity, has recently released his High-Speed Digital Design (HSDD) Collection. This includes professionally recorded seminars that he presented, for more than 20 years, at Oxford University and worldwide and is arguably the most practical and enlightening course on high-speed—black magic—ever delivered. Howard's unique, explicatory presentation style creates an unforgettable picture of signal propagation by practical example. If you want to gain some of his enthusiasm and master the art of high-speed design, then the collection is a must-have.

I recently had the opportunity to review all three of the seminars in this collection, a total of 36 hours of viewing time. When presented with a selection of three seminars, to watch, I guess it is only natural to want to start with the more advanced topic. But I am glad that I forced myself to start at the beginning to refresh the basics before moving on to the more complex issues. It is amazing how much I either did not know or had simply forgotten over the years. Or maybe I've just killed too many brain cells along the way!

I know that during my own courses, there is always one guy at the back who falls asleep. And

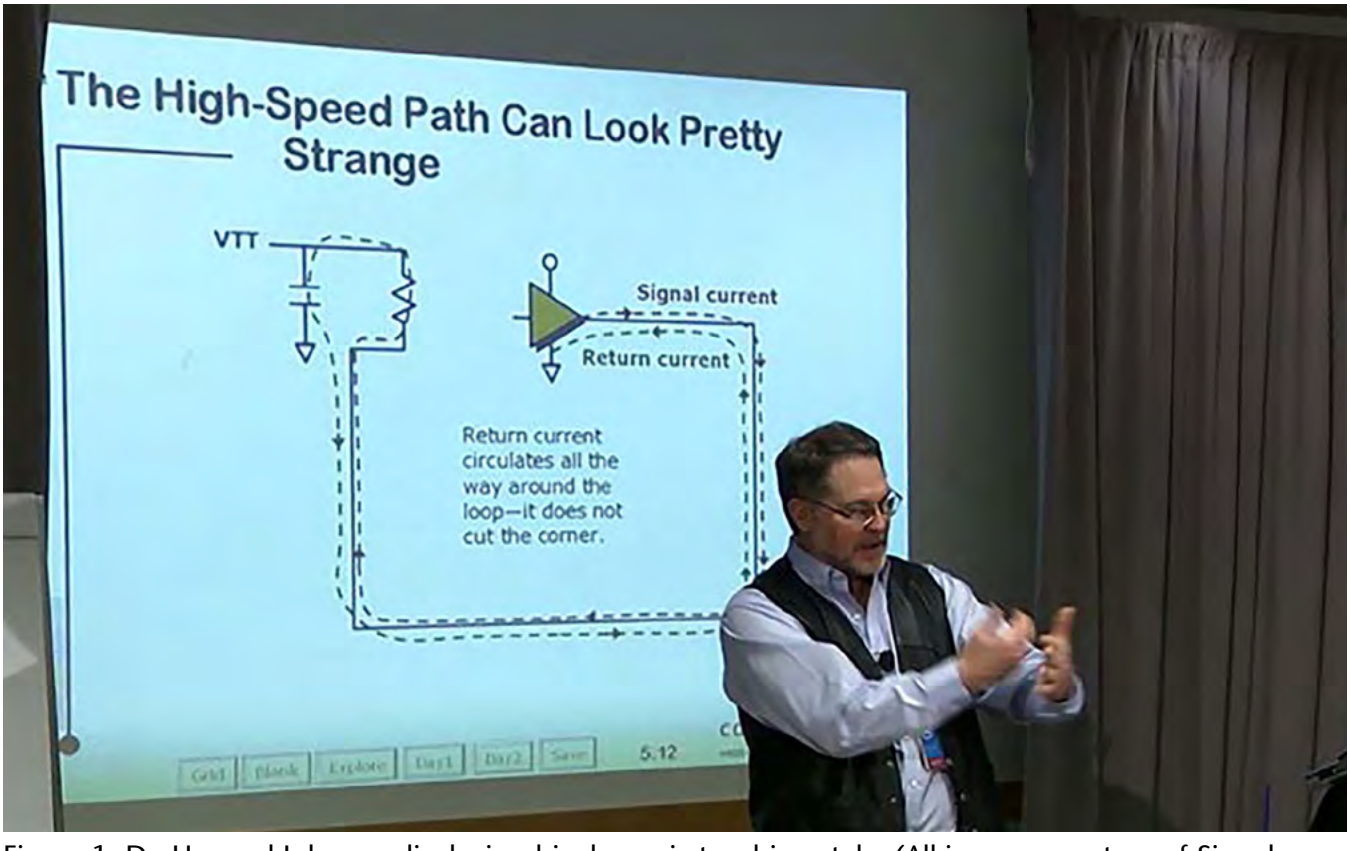


Figure 1: Dr. Howard Johnson displaying his dynamic teaching style. (All images courtesy of Signal Consulting Inc.)

strangely enough, he is always the one to give bad feedback, probably because he didn't learn anything. But I guarantee that you will be on the edge of your seat throughout Howard's entire seminar series. His dynamic teaching style ensures you feel like you are not just a part of the audience—you are actually participating in the demonstrations. The picture that Howard paints leaves a lasting impression on how electromagnetic fields propagate and how they induce voltages and current (crosstalk) into nearby signals. The following is a section-by-section discussion of the course contents.

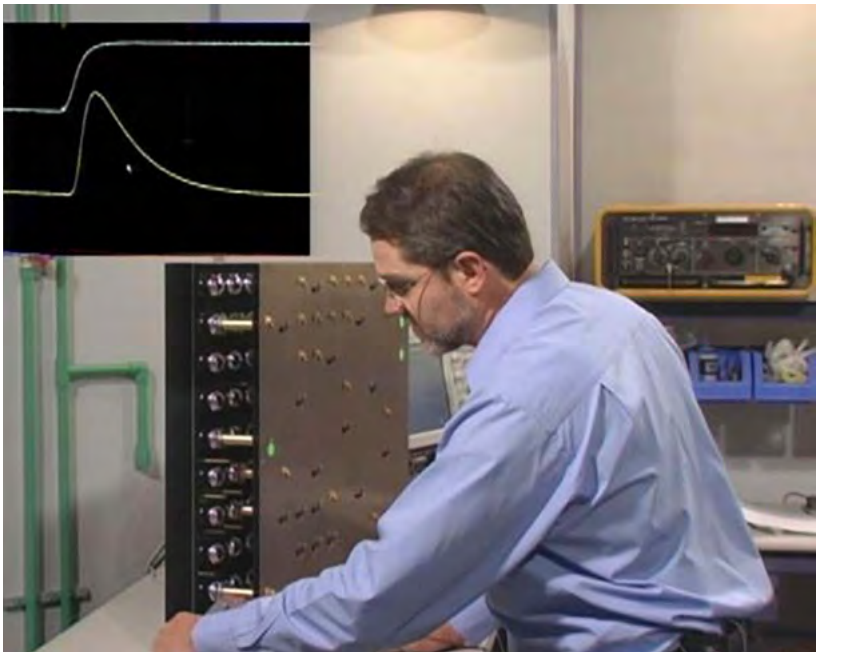


Figure 2: Dr. Johnson measuring crosstalk in his giant scale BGA model.

1. High-Speed Digital Design

Engineers and PCB designers need to understand electromagnetic theory, appreciate how coupling occurs and why energy moves to unintended, sensitive parts of the circuits. A logic schematic diagram masks details crucial to the operation of unintentional signal pathways vital to your understanding of signal performance, crosstalk and EMI. To realize these factors, one must uncover the hidden schematic, operating behind the logic diagram, to reveal the parasitic elements that affect the circuit. These parasitics are invisible to the uninitiated, but become very clear once skillfully explained in detail. You will gain new insight into what really happens in the circuitry.

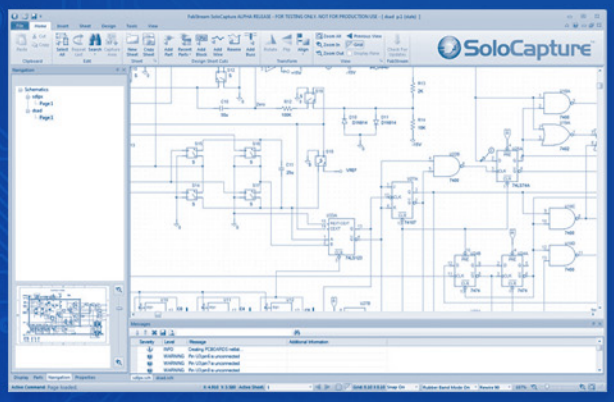
Also, understanding the frequency band that really matters for digital design is very important. Traditionally, we used 0.35/Tr (where Tr is the rise time in ps) for the upper bandwidth. However, Howard recommends using an upper knee frequency of 0.5/Tr, which forms a crude, but useful, translation between time and frequency domains. So for instance, if the rise time is 500ps, which is typical these days, then the upper bandwidth is actually 1GHz regardless of the clock frequency. Furthermore, the constant improvement in the IC process reduces die size which speeds-up the rising edge. This in turn pushes the knee frequency up, causing signal overshooting and ringing.

Howard also cites the difference between "lumped element" and "distributed systems." In Circuit Theory 101, we are taught using lumped element assumptions where the system delay is much less than the signal rise time. However, Howard points out that when the system delay is much larger than the signal rise time, a more complex distributed analysis is required. This system is characterized by distributed delay and reflections; this is the real world of high-speed design. Here, capacitance draws surge current causing reflections, inductance causes ground (supply) bounce and noise, mutual capacitance causes crosstalk between high impedance circuits and mutual inductance produces crosstalk in connectors especially where the layout is questionable.

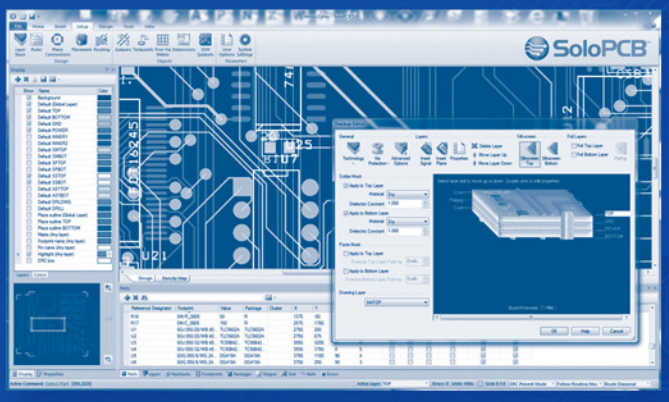
I was impressed by the way that Howard addresses bidirectional signal terminations. I have simulated the position of a series resistor on a bidirectional data trace and it doesn't make any difference whether the resistor is placed at either end or in the middle. But, having a resistor at both ends is an elegant solution, as the resistor and input capacitance, of the tri-state load, basically form an AC termination—I would not have thought of that!

FREE PCB Design Software

Project Tutorials, Resources and Community to Create PCB-based Electronics



Powerful Schematic Capture



PCB Editing

SoloPCB Designer has powerful Schematic Capture and PCB layout, providing all the features and functionality you need to design the PCBs for your electronic products. It supports a broad range of capabilities from single-sided PCBs to mulit-layer PCBs with advanced technologies.

When your PCB is designed, order the CAM files to manufacture your PCBs. Once you purchase these files, you can take them to any PCB manufacturer and have your boards made.

$49
2 Layer File Set

$99
4 Layer File Set

$149
6+ Layer File Set

Join the Community

Our online community is dedicated to all electronics enthusiasts and is a place where you can download tutorials, post your questions and ideas.

Solo-Labs.com

2. High-Speed Noise and Grounding

The second seminar in the series focuses on interactions in the real world: electromagnetic compliancy (EMC), which encompasses radiation and susceptibility. Howard looks at a subset of EMC effects, specifically, coupling between circuits within one system, or between large systems in a complex product. He further deliberates on the sort of things that cause intermittent, unreliable behavior of the product in the field and he advises on how to alleviate these issues before they arise.

In Figure 2, Dr. Johnson measures crosstalk in his really cool giant scale BGA model, and looks at supply bounce and ground ball placement on the BGA package. There are also many Signal Integrity Laboratory (SI Lab) experiments embedded in the videos that really help demonstrate the distribution of high-frequency current on reference planes and how crosstalk develops where current loops overlap. And, more importantly, how to avoid crosstalk.

Splitting planes and creating moat and drawbridge constructions are an effective audio frequency solution. But RF coupling (in Figure 3) is quite different, in that it takes place through magnetic and electric fields that can easily propagate through space, going right across a reference plane cut. A sensible solution here is to increase the spacing of coupled devices or add solid shielding. However, you can never completely isolate system components as parasitic capacitance links them whether you like it or not.

For low-impedance circuits, like high-speed computers, most of the issues are related to inductance. Whenever you deviate from a solid reference plane, returning signal current spreads far and wide forming large loop areas and creating radiation. And, mutual inductance connects nearby circuits that don't even touch each other.

If you do a lot of system testing and design verification, in the lab, then this seminar is for you. The second half focuses on system level grounding, connector issues and measurement. Howard also looks at clock related noise issues, the two separate modes of operation—common

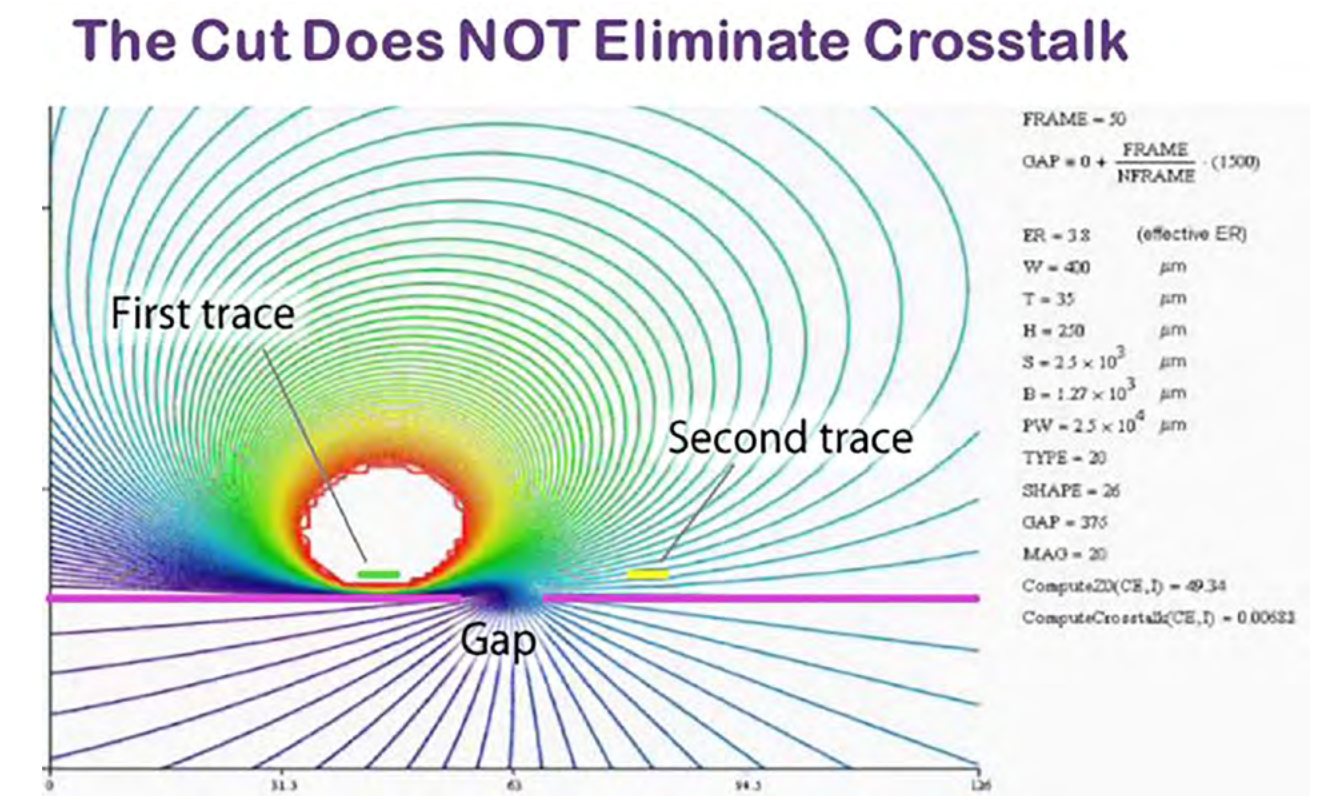
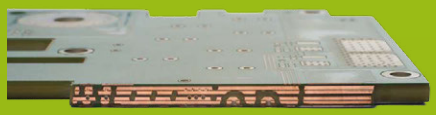


Figure 3: Split planes do not prevent RF coupling.

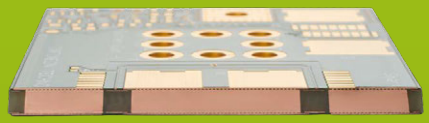
BEST-IN-CLASS PRINTED CIRCUIT BOARDS FOR POWER ELECTRONICS

SCHWEIZER ELECTRONIC

1. SOLUTIONS FOR HIGH CURRENTS



Heavy Copper Board
- e.g. for AC/DC and DC/DC converter



Inlay Board
- e.g. for start-stop systems

2. SOLUTIONS FOR HEAT DISSIPATION

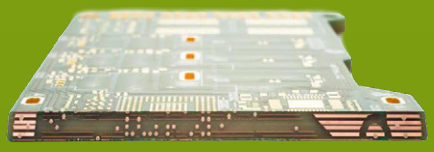
IMS Board
- e.g. for LED headlights

Cool Board
- e.g. for cell connectors and battery switches

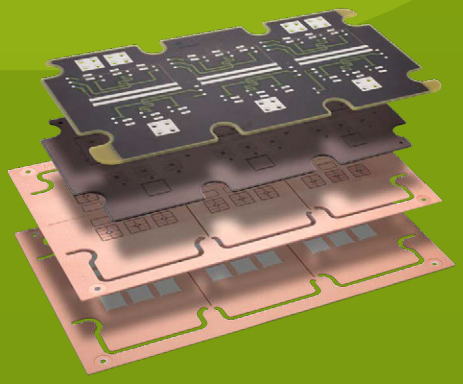
3. SOLUTIONS FOR HIGH CURRENTS AND LOGIC ON JUST ONE PCB

PROJECT HJ-LEVEL

GEFÖRDERT VOM Bundesministerium für Bildung und Forschung



Power Combi Board
- e.g. for electrical pumps



Smart p² Pack® (Power Embedding)
- e.g. for electrical motors
- low-inductance solutions
- improved electrical, thermal and EMC features
- miniaturisation due to new interconnect approach

Schweizer Electronic AG
Einsteinstrasse 10
78713 Schramberg, Germany
T +49 7422 512 0
info@schweizer.ag

www.morethanPCBs.com

and differential—plus he discusses clock jitter in detail.

3. Advanced High-Speed Signal Propagation

Now I am ready for the final seminar. I am so glad I did the first two seminars, rather than jumping straight into the advanced level. I am now fully prepared, with all the background knowledge, to move forward with the more complex issues of signal integrity. This seminar is for experienced digital designers, who need to drive their designs to the upper limits of speed and distance. Howard stated, "…without signal integrity tools, you do not know how close you are to that limit. It is our responsibility, as designers, to push the system as close as possible to the edge without ever failing."

As seasoned designers, we are used to looking at circuit parameters in the time domain, like a waveform on an oscilloscope. However, as clock frequencies and edge rates continue to accelerate, one needs to focus instead on scattering parameter (S-parameter) models in the frequency domain in order to effectively evaluate signal propagation in a lossy medium. A two port S-Parameter model of a transmission line is derived in both matrix and equation form. However, cascading networks cannot be evaluated from input to output but rather need to have combined S-parameters for multiple port analysis.

The power spectral density of a digital signal is typically below the knee frequency and if the parasitic impedances are not significant, then digital signals tend to pass undistorted. This is illustrated in Figure 4. Howard prefers to conservatively over-estimate the bandwidth, so that all effects, above the knee frequency, can be safety ignored.

Next is a detailed look at the transmission line model. Howard uses the transverse electro-

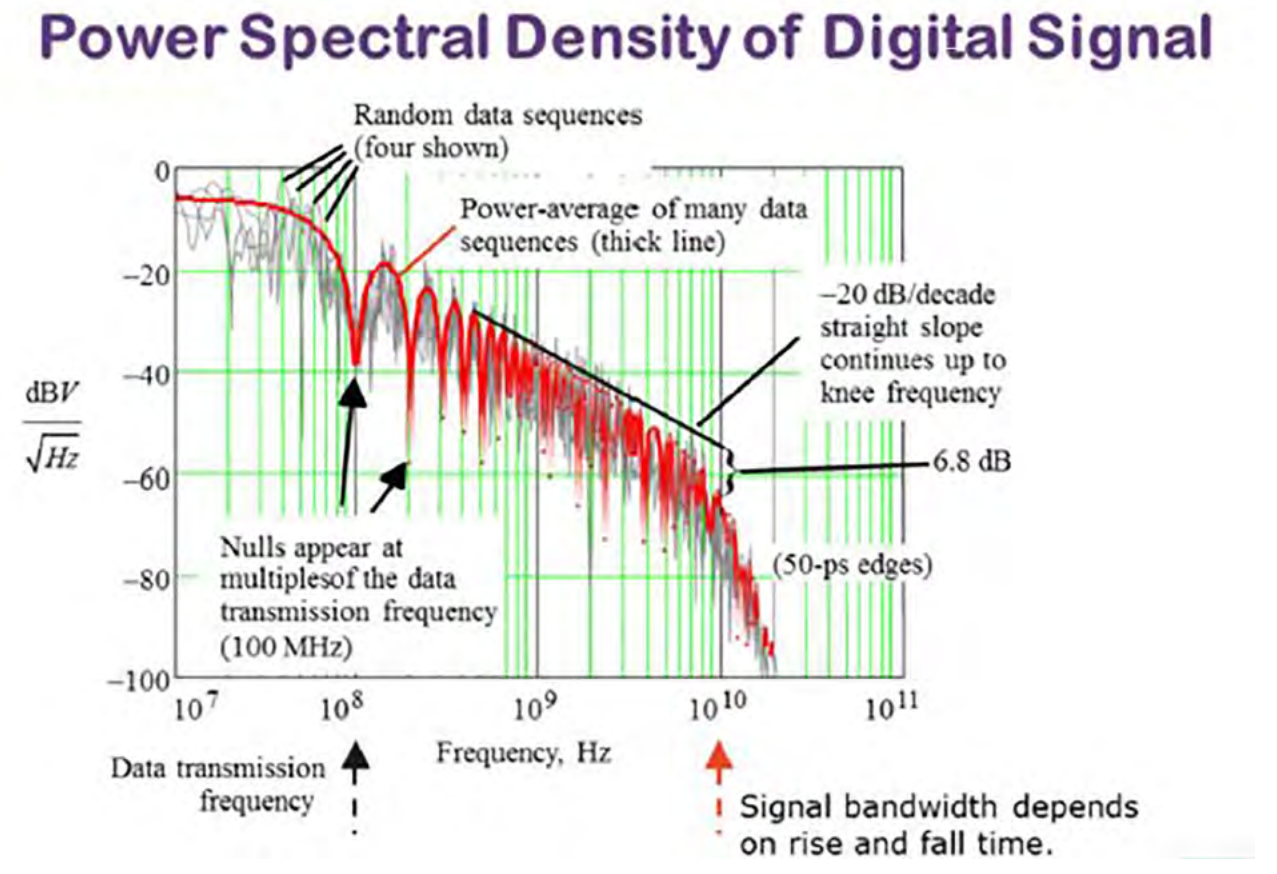


Figure 4: Power spectral density of a digital signal.

BRADY®
WHEN PERFORMANCE MATTERS MOST™

Solve Challenges & Add Value with Traceability

How does an effective traceability system impact manufacturing?

Find out with Brady's Traceability Whitepaper. This report explains:

- Cost, process and product quality improvements
- Why traceability is critical
- 4 key benefits of traceability
- Considerations for implementing a system

Visit BradyID.com/trackingPCB to download the Traceability Whitepaper.

magnetic (TEM) mode of propagation, where the electric and magnetic field lines are restricted to the direction of propagation (normal to the direction of propagation), to demonstrate where field lines are prevalent. Transmission lines can be broken down into stages, when the signal and return path are a short fraction of the rise time away from the reference plane.

The returning signal current path is the key to understanding mutual inductive problems in connectors, crosstalk between parallel transmission lines and also EMC. Signal current does not flow down to the end of the signal trace and then return back to the source. But rather, as the signal wave front propagates, the return current builds up simultaneously creating multiple paths back to the source. Howard demonstrates this very effectively with animation showing charged particles as little balls moving in slow motion down a signal trace, while corresponding particles on the returning signal trace move in the opposite direction. It is this type of demonstration that really sticks in the mind and creates a lasting impression. There are also many other animations that clearly demonstrate examples throughout the three seminars.

"Transmission lines can be broken down into stages, when the signal and return path are a short fraction of the rise time away from the reference plane."

A discussion of skin depth, where current flows only in a shallow band at high frequencies, and dielectric losses that need to be considered at high frequencies, follows. A non-uniform distribution of current, around the periphery of a conductor, creates a non-concentric field. At high frequencies, magnetic lines of force will not penetrate a conductor but rather flow tangential to the conducting surface. The trick is to ensure that the radiated field lines do not flow between the victim signal trace and reference planes, inducing current and hence crosstalk. Howard also points out that dielectric loss is not a function of the conductor geometry but depends on the loss tangent of the surrounding materials. This is why a low loss dielectric material (Df = 0.002) is more effective, at high frequencies, compared to standard FR-4 (Df = 0.02). Dielectric loss is also frequency dependent, so the maximum frequency of operation is also significant.

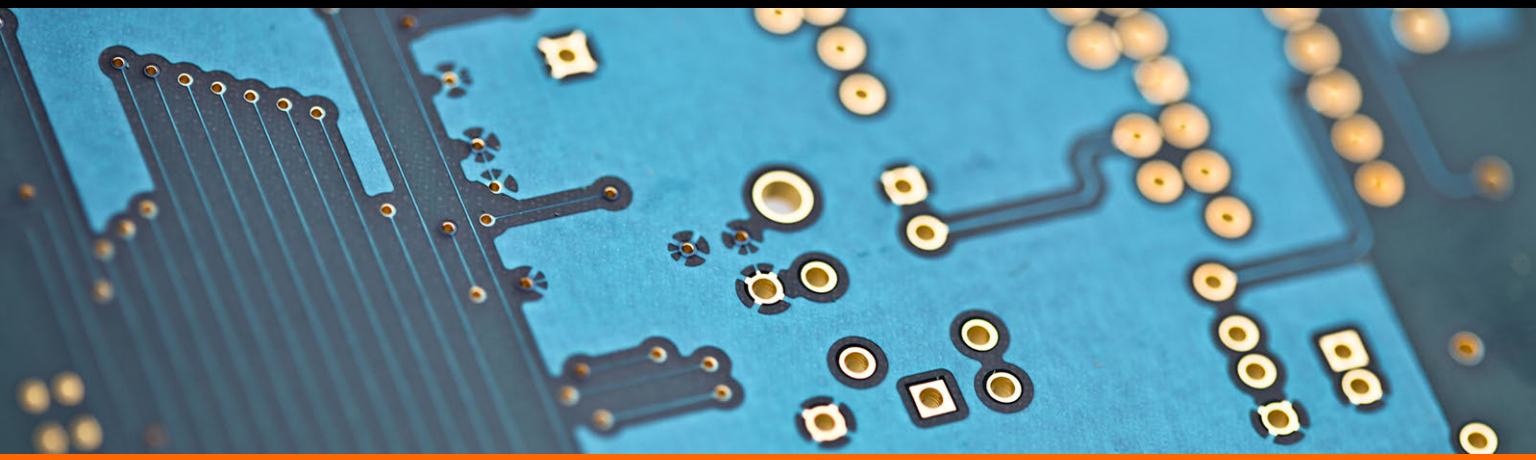
Finally, the section I have been waiting for: PCB traces, connectors, vias and differential signaling. Microstrip and stripline stackup configurations are discussed, in detail, with the effects of surface roughness, nickel plating and solder masks. This necessitates the use of a 2D field solver such as the ICD Stackup Planner.

The time-space diagrams of transmission line reflections are very enlightening. This is a simple way of analyzing reflections from downstream loads on the transmission line. A load at the end or multiple loads, along a multi-drop line, look like capacitors (IC input capacitance). These delay the rising edge and reflect back down the transmission line as near-end crosstalk (NEXT), creating "potholes" in the signal. Howard has a unique solution to this crosstalk by deliberately creating an equal and opposite reflection to neutralize the pothole. This technique is ideal for the fly-by routing of multi-drop loads. I will definitely implement this strategy on my next DDR3/4 design.

Types of PCB connectors and the use of proper grounding, to alleviate high inductance and EMI, are also discussed. Vias are basically a type of connector that transfers a signal from layer-to-layer in a multilayer PCB. So, ground stitching vias need to be utilized in order to reduce loop inductance. Removal of unused inner layer via pads reduces capacitance. Dangling vias create reflections above 1GHz but can be improved by back-drilling, using truncated vias or special antipad clearances. Howard's approach for differential vias is unique.

Differential signaling also has its challenges and both differential-mode and common-mode noise are discussed in detail. Reducing common mode noise is the key to good differential design. There are also special issues with crosstalk,

SUNSTONE CIRCUITS *from Prototype to Production*



PCB fabrication, assembly, and free award-winning CAD design software.

Leading the industry for over 40 years as the preferred ***prototype and production PCB solutions provider*** for design engineers.

WE ALSO OFFER:

- Best overall quality, value and competitive pricing
- Design for Manufacturability Tool: **DFMplus**
- Fully supported and free **PCB123®** design software
- Integrated assembly services
- FREE Ground Shipping
- PCBs as low as $28 with **ValueProto®**

100%

On-time Guarantee

QUOTE NOW ›

Questions? Comments? Suggestions?

We appreciate hearing from you!
Contact us at: 1-800-228-8198 or via e-mail at support@sunstone.com

SUNSTONE CIRCUITS

on differential signals, that can easily be avoided. Solutions for breaking a pair, termination, change of reference planes, trace skew and DC blocking layout techniques are described in detail.

Finally, one of the most important aspects of HSDD is revealed: clock distribution. All you ever wanted to know about effectively routing clocks, loaded delay, the forward crosstalk (FEXT) of serpentine traces, daisy-chain routing of multiple loads and the effects of jitter on clock eyes. Dr. Johnson's collection is a must-see for all digital design engineers and PCB designers who need to understand electromagnetic theory, appreciate how coupling occurs and why energy moves to unintended, sensitive parts of the circuits. And, more importantly, how to prevent electromagnetic coupling.

I read that when Albert Einstein was teaching at Princeton, he prepared an examination paper and handed it to his assistant. The assistant queried, "Albert, isn't this the same exam you gave this class last year?" Einstein replied, "Yes it is. The questions are the same, but the answers have changed!"

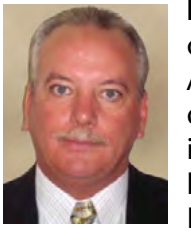
Digital designers need to keep up with the fast changing pace of technology. For all the latest solutions, to complex Signal Integrity issues, I recommend the High-Speed Digital Design Collection. **PCBDESIGN**

References

1. Dr. Howard Johnson, Signal Consulting: High-Speed Digital Design Collection

2. Howard Johnson, Martin Graham: High-Speed Digital Design – A Handbook of Black Magic

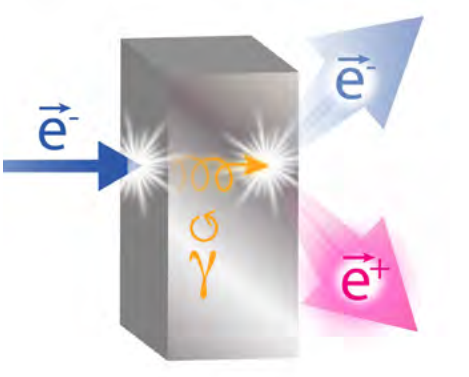
3. Howard Johnson, Martin Graham: High-Speed Signal Propagation – Advanced Black Magic



Barry Olney is managing director of In-Circuit Design Pty Ltd (ICD) Australia. The company is a PCB design service bureau that specializes in board-level simulation. ICD has developed the ICD Stackup Planner and ICD PDN Planner software, which is available here.

Spinning Electrons Yield Positrons for Research

Using the Continuous Electron Beam Accelerator Facility (CEBAF) at the Department of Energy's Jefferson Lab, a team of researchers has, for the first time, demonstrated a new technique for producing polarized positrons.



Jefferson Lab Injector Scientist Joe Grames says the idea for the method grew out of the many advances that have been made in understanding and controlling the electron beams used for research in CEBAF.

"We have a lot of experience here at Jefferson Lab in operating a world-leading electron accelerator," Grames said. "We are constantly improving the electron beam for the experiments, pushing the limits of what we can get the electrons to do."

Grames and his colleagues would like to take that finesse a step further and transform CEBAF's well-controlled polarized electron beams into well-controlled beams of polarized positrons to offer researchers at Jefferson Lab an additional probe of nuclear matter. They named the endeavor the Polarized Electrons for Polarized Positrons experiment, or PEPPo.

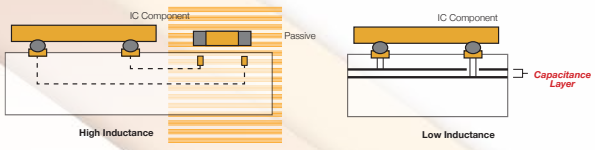
Throughout the process, the polarization of the original electron beam is passed along. The researchers use a magnet to siphon the positrons away from the other particles and direct them into a detector system that measures their energy and polarization.

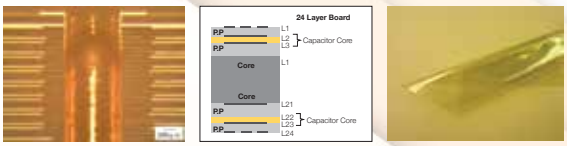
"We showed that there's a very efficient transfer of polarization from electrons to the positrons," said Grames.

THE SMART SOLUTION TO IMPROVE SYSTEM PERFORMANCE

FARADFLEX ULTRA THIN MATERIALS FOR HIGHER PERFORMANCE PCBS

Faradflex is a high performance, ultra-thin laminate or resin coated copper material designed as an integral part of the PCB, module or chip package.





FaradFlex® Improves System Performance, Price, Quality, and Reliability by:

- Reducing PCB size and/ or thickness
- Utilizing VSP Copper to provide the lowest loss signal for high speed designs
- Increasing functionality
- Improving power distribution (power integrity)
- Reducing the number of SMT Discrete Caps, Traces, Vias, Pads
- Eliminating hundreds (thousands) of SMT and etched and drilled features sites for possible failure on each PCB

For a free sample contact us at:
sales@oakmitsui.com or call us at 503-313-1905

FARADFLEX®

EMBEDDED CAPACITANCE

OAK-MITSUI TECHNOLOGIES
MITSUI KINZOKU CORPORATE GROUP

For more information, visit www.faradflex.com

The Dilemma: Soldermask for High-Frequency PCBs

by John Coonrod
ROGERS CORPORATION

High-frequency and high-speed digital PCBs may not have issues with soldermask. However, depending on their construction, other PCBs can have an issue with soldermask causing degraded electrical performance. PCBs with a stripline structure, in which the signal layer is buried within a multilayer, typically do not have an issue with electrical performance degradation due to soldermask. Soldermask can impact PCBs with RF circuitry on the outer layers, which can lessen high-frequency electrical performance.

Typically, PCBs with RF traces on the outer layers have minimal or no soldermask in the RF circuitry areas. Many times the soldermask is applied in areas where components are soldered to the PCB but the soldermask is developed away in the areas where conductors have critical RF performance. There are many reasons to avoid soldermask coverage on RF conductors, due to inherent soldermask properties. Most soldermask used in the PCB industry is liquid photoimageable (LPI), which is typically high in dissipation factor (Df) and high in moisture absorption, and the thickness can vary due to processing or design.

The typical soldermask has a dissipation factor of about 0.025 when tested at 1 GHz, and moisture absorption is about 1–2% depending on the formulation. For comparison, many high-frequency laminates have a Df value of about 0.005 or better and moisture absorption is typically no worse than 0.3%. The higher Df property of soldermask raises the circuit's dielectric loss, which causes an increase in insertion loss. The moisture absorption can cause differences in impedance and phase response, but it is typically more problematic for losses where it can cause increased insertion loss.

Another point to consider is that RF circuitry on the outer layer of a PCB will usually be a microstrip or grounded coplanar waveguide

COAST TO COAST CIRCUITS, INC.

TECHNOLOGY FROM COAST TO COAST

For over 40 years, we have been producing advanced solutions for our customers in many industries across the globe including Defense, Medical, RF/Microwave, Semiconductor and Down Hole Exploration, among others. How can we add value to your next project?

LEARN MORE **CHECK OUT OUR CAPABILITIES**

Our name has changed!
Click here for more info.

FORMERLY KNOWN AS
Speedy Circuits and Metro Circuits

Two Facilities.
One Integrated Solution.

NEW YORK OPERATIONS
585.254.2980

CALIFORNIA OPERATIONS
714.898.4901

www.c3circuits.com **sales@c3circuits.com**

(GCPW) structure. Both of these structures can have lower insertion loss and they get some loss benefit due to their fields using air. Air is the lowest-loss medium for electromagnetic waves, and these waves use electric and magnetic fields. When a microstrip or GCPW is covered with soldermask, some of the fields which were using air as the dielectric medium are now using soldermask instead.

Soldermask will never be as low-loss as air, so the soldermask will always increase the insertion loss. A thin layer of soldermask causes less insertion loss than a thick layer of soldermask. In the case of GCPW, there are fields between the ground-signal-ground conductors on the outer layer. When soldermask fills the gaps between these conductors, the insertion loss is increased significantly more than for a microstrip circuit covered in soldermask. The previous is an issue that can cause confusion where one type of circuit can be impacted by loss much more than another type of circuit.

When applying soldermask over RF conductors, proper design considerations must be applied. For example, if a multilayer PCB has two outer copper layers made of 20 mil RO4350B laminate, which has a Dk of about 3.73 at 3.5 GHz, the propagating wave on a microstrip transmission line circuit will have a wavelength that is about 1.85" (47 mm). The design can have patches of soldermask covering portions of the microstrip circuitry and are less than 1/10 wavelength or 0.185" (4.7 mm). When the soldermask is less than 1/10 wavelength, there is no significant impact on the RF circuit performance. This is often done for RF components that are soldered onto the PCB and the patches of soldermask act like "solder dams," which constrain the solder flow in a specific area during the reflow operation.

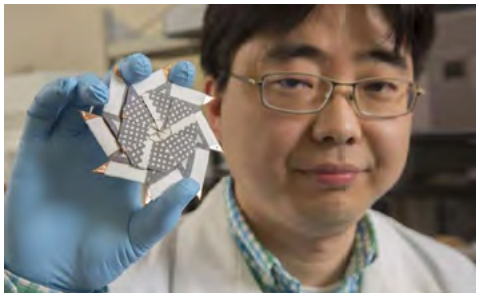
If the RF circuitry cannot be covered by soldermask, the designer often uses electroless nickel immersion gold (ENIG) as a final finish so the copper will not oxidize and cause problems later. The addition of ENIG will also increase insertion loss, but for a different reason than the soldermask. Nickel has about a quarter of copper's conductivity, so it will cause more conductor losses, which will cause an increase in insertion loss.

However, the insertion loss impact due to ENIG is frequency-dependent due to skin depth and at certain frequencies the nickel may not cause as much loss as other frequencies. Engineering is a game of tradeoffs and sometimes RF circuit designers will consider the difference of losses due to ENIG and soldermask. They may choose one over the other for their particular application due to the operating frequency or the working environment. **PCBDESIGN**

John Coonrod is a senior market development engineer for Rogers Corporation. To read past columns or to reach Coonrod, click here.

Origami Ninja Star Inspires New Battery Design

A new disposable battery that folds like an origami ninja star could power biosensors and other small devices for use in challenging field conditions. Seokheun "Sean" Choi, assistant professor of computer and electrical engineering at Binghamton University, along with two of his students, developed the device, a microbial fuel cell that runs on the bacteria available in a few drops of dirty water.



Choi previously developed a paper-based origami battery. The first design, shaped like a matchbook, stacked four modules together. The ninja star version, which measures about 2.5 inches wide, boasts increased power and voltage, with eight small batteries connected in series. Paper-based biosensors include pregnancy tests and HIV tests.

DIBBLE 4 LEADERS

THE FOUR NEW AGREEMENTS FOR SUSTAINABLE SUCCESS

Profitability is more about leadership and management than any other factor.

Any business can be transformed quickly, easily and sustainably with the proven effective systematic methods we employ. For over 25 years, our clients have averaged a 300% ROI on this work. Returns of 1000% are not unusual, especially in early stages of implementation. **Contact us today** and get started on the path of doing the right things that allow growth and profitability to happen.


BOOM!

The Four New Agreements to EXPLODE PROFITABILITY

click to read

Learn more about the roadmap used to build great companies with a high level of profitabiity in this article from the March 2016 issue of **The PCB Magazine.**

For 25 years we have been doing Four New Agreements consulting and training, significantly improving businesses. This stuff really works!
—David Dibble

4newagreements4leaders.com

858-775-0722 Pacific

MilAero007 Highlights

Beyond FR-4: High-Performance Materials for Advanced Designs, Part 1

In the past 40-plus years of PCB manufacturing, the primary material of choice has overwhelmingly been e-glass supported FR-4 resin laminates. This is due to the excellent dimensional stability and reasonably acceptable thermal performance (based on glass transition temperature and decomposition temperature).

IMPACT Washington, D.C. 2016: Industry Leaders Advocate for a Pro-Manufacturing Policy Agenda

IPC places a high priority on educating government officials about key policy issues of importance to the electronics industry. That's why top executives from leading electronics companies gathered in Washington, D.C. recently for "IMPACT Washington, D.C. 2016."

NASA Investigates 3D Printing for Building Densely Populated Electronic Assemblies

As detector assemblies get smaller and denser—packed with electronic components that all must be electrically connected to sense and read out signals—it's becoming increasingly more challenging to design and manufacture these all-important instrument devices.

Dragon Finalizing Departure Preps

The SpaceX Dragon is being packed with critical science today and tomorrow before its release and splashdown on Wednesday. The crew is also reviewing Dragon departure procedures and training for its release from the grip of the Canadarm2 robotic arm.

Beyond FR-4: High Performance Materials for Advanced Designs, Part 2

In Part 1, we covered basic FR-4 and variants that have been used in the commercial and military market for the past few decades, but in this column we will delve into the newer materials that target a specific application and/or market segment.

Industry Weighs in on Green Aviation Tech

Aviation's future is bright…green. That was the environmentally promising message delivered during the Green Aviation Technical Interchange Meeting (TIM), held recently at NASA Langley Research Center in Hampton, Va.

NASA, Virginia Tech Test Management Platform for Unmanned Aircraft Traffic

Efforts to protect air travelers are becoming essential as business leaders ramp up efforts to use unmanned aircraft for agriculture, real estate, inspections, and commercial purposes, officials from the Virginia Tech Mid-Atlantic Aviation Partnership said.

Raytheon to Start Production of First Multi-spectral Targeting System

The U.S. Air Force awarded Raytheon Company a $90 million first-lot production contract for the next-generation Multi-Spectral Targeting System. The AN/DAS-4, the latest variant of the MTS family of sensors, incorporates greater fire control and Target Location Accuracy technology for precise coordinates.

NASA Space Launch System's First Flight to Send Small Sci-Tech Satellites into Space

The first flight of NASA's new rocket, the Space Launch System (SLS), will carry 13 CubeSats to test innovative ideas along with an un-crewed Orion spacecraft in 2018.

Station Readies for BEAM Expansion

The Expedition 47 crew is getting a new module recently attached to the Tranquility module ready for expansion later this week. The International Space Station residents are also running experiments today exploring a wide variety of phenomena and checking station gear.

SILVER Lining...

Priced with the market

COOLSPAN® TECA

thermally & electrically conductive adhesive

COOLSPAN® TECA Film provides an easy solution to solving your thermal management challenges. Available directly from Rogers Corporation, without the middleman, COOLSPAN TECA is supported by Rogers' technical staff to help you gain the best result from this environmentally stable conductive adhesive. Keep your cool and maintain your budget.

- Epoxy-based Adhesive Film
- Market Priced to Silver Spot Price
- Excellent Thermal & Electrical Properties
- Predictable Flow
- Durable & Reliable

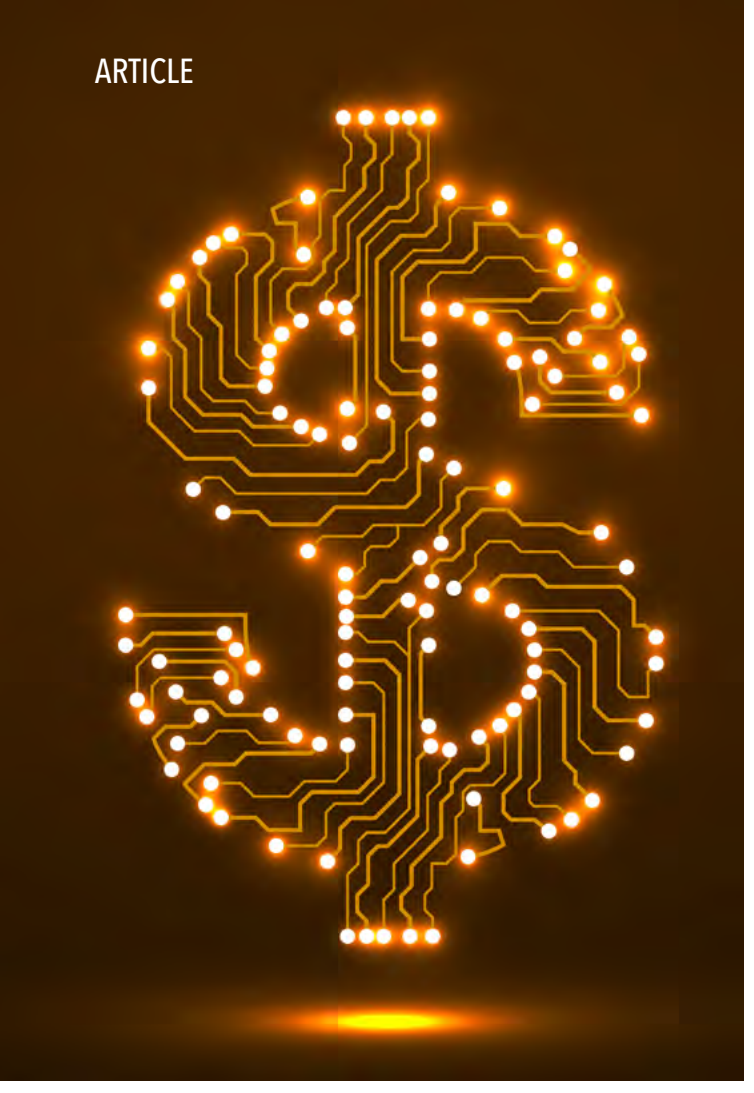
Contact Rogers Corporation for more information or to request a sample

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

ROGERS CORPORATION

Advanced Connectivity Solutions

www.rogerscorp.com/COOLSPAN



The Importance of DESIGN FOR PROFIT (DFP)

by Barry Matties

In this interview, Interconnect Design Solutions' Mike Brown and I took a few minutes during the recent Geek-A-Palooza event to discuss the importance of material selection and designing for profitability, how automation affects the design process, and the future of the design community.

Barry Matties: *Mike, tell me a little bit about IDS and what you do.*

Mike Brown: We provide PCB layout and mechanical engineering services as an extension for OEMs as well as bareboard fabricators and assemblers that need design support. Basically, we're in the trenches working with the electrical engineers to help realize our product and bring it to market.

Matties: *Are you doing actual design for customers or are you just assisting and educating?*

Brown: We're actually doing design at the rigid board level, the flex-rigid, and flex board level as well as mechanical engineering and enclosure design. We're helping engineers realize their project from concept through reality.

Matties: *So you start from the beginning?*

Brown: We start from the beginning with a functional spec. We're sitting there basically looking at a block diagram sometimes.

Matties: *It's a collaborative effort?*

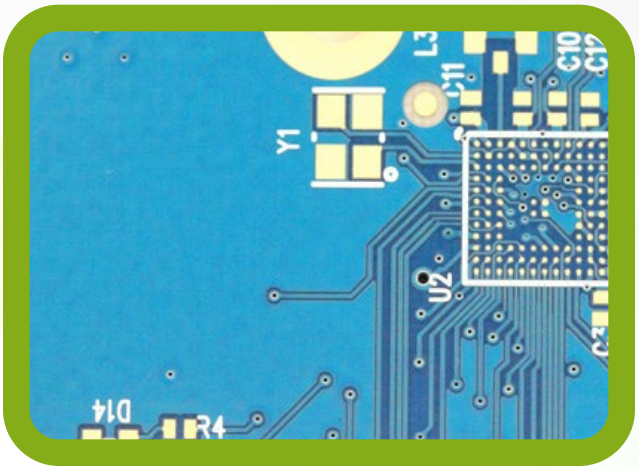
Brown: Yes, it is very collaborative.

Matties: *Isn't that the best way to go, really?*

Brown: It is. You have to be engaged in the beginning in order to be successful in the end.

Matties: *And yet many engineers don't take this track; what's the downside of that?*

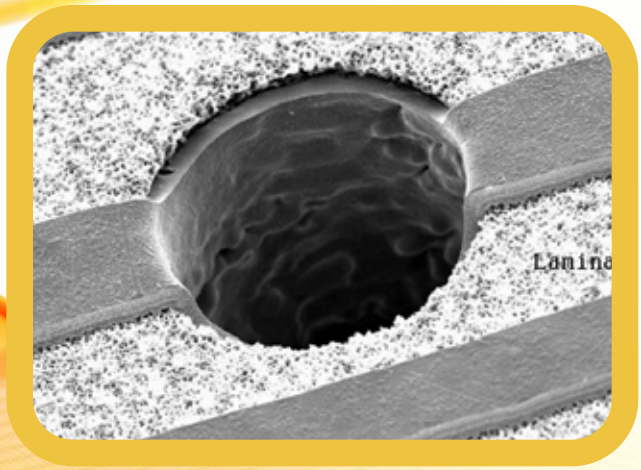
We deliver Advanced PCB Solutions!



Fine line PCB, 2 mil line /space

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

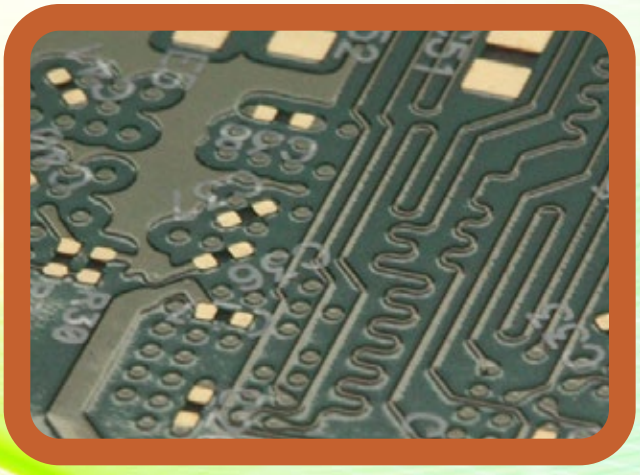
CLICK TO LEARN MORE



Landless Via Technology

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

CLICK TO LEARN MORE



Impedance Tolerance <5%!

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

CLICK TO LEARN MORE

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

CANDOR INDUSTRIES, INC.

ADVANCED PCB SOLUTIONS

www.candorind.com | sales@candorind.com | (416) 736-6306

Brown: The problem is there's a lack of communication and in a lot of cases, multiple spins of your product can occur; which in turn costs more because you typically don't have your pwb layout designer collaborating up front with your primary engineering team. The layout designer really needs to understand the product from the beginning. They need to understand the design intent and what the end goal of the product is. By having them involved in the beginning up front at the functional spec level and then at the detail spec level, they can help make conscious decisions that ultimately will affect the overall cost, time line and schedule of the product.

"The layout designer really needs to understand the product from the beginning. They need to understand the design intent and what the end goal of the product is."

Matties: ***So we talked a bit about DFM, but one of the things I'd like to talk about is DFP, or design for profitability. I think profit in any project begins with design.***

Brown: It does. It comes down to choice of materials and component selection. Let's just talk about at the bareboard level, for example. Let's just say if I have an RF board that has a lot of high-speed digital on it, am I going to make the entire board out of Rogers? No, it wouldn't make sense. If I did, next thing you know it's going to be a very costly board, because the bareboard material has a cost driver built in that's more expensive from the beginning. Then there's also the issue of processing. Processing a 16-layer board that's all Rogers is much costlier than just a Rogers end cap with a multi-layer board that's HR 370. You can marry those materials together just fine, but you just have to take into account some processing in the beginning.

Matties: ***Because over materialization is an extremely expensive proposition in your end game, isn't it?***

Brown: Right. It depends on what your design intent is. If I'm designing just a power control module for a thermostat, there's really not a whole lot to that. Why would I pick a material as exotic as Megtron 7, that has really tight weaved fiber and is a much more costly alternative than say a simple FR-4, when it doesn't need it?

Matties: ***How does a designer get this education?***

Brown: It's through magazine articles, like those at iConnect007, trade publications, mentorship and working directly with the engineering teams. Usually that's just something that's gained over time through experience with the various customers and OEMs.

Matties: ***The other thing I see in the design community is a bunch of older guys. There are not a lot of young guys coming in.***

Brown: You know this is the third time I've had that conversation today alone. I'm also seeing a trend where the baby boomer generation are retiring, so now there's going to be a big void in the whole industry. I've been in the industry going on 27 years. I do see a lack of mentorship from the older generations and a lack of incoming designers. When they go to engineering school they start as an electronic technician. There is no career path to be a PCB designer—it doesn't exist. They're either going to be an electrical engineer, a mechanical engineer, a systems engineer or what the hot topic is today, an application engineer. When they start to learn design, they don't necessarily want to go into PCB design. If you ask any PCB designer that's been around, I guarantee you'll never get the same answer twice on how they became a PCB designer.

Superior Quality Begins Here.

Since we first began in 1985, delivering the highest quality products possible has been our first priority. Our quality control systems are certified and we renew our commitment to quality each and every day.

Delivering quality products on time, combined with timely and accurate communication from our customer service team is how we create lasting relationships with our customers.

Take a look at our quality »

COMMERCIAL • MILITARY • MEDICAL • BAREBOARD

U.S. Circuit, Incorporated
2071 Wineridge Place
Escondido, CA 92029
+1 (760) 489-1413
www.uscircuit.com

Mike Brown

***Matties:** That makes sense, especially back in those days. One of the things that I see happening is a lot of automation. Is automation replacing the need for designers?*

Brown: I've been hearing it for the last 20 years. Especially when you talk about auto routers, for example. The old theory was that the great Cooper and Chan Specctra autorouter that came out back in the late '80s to early '90s was supposed to be the end-all, be-all for designers. The problem is that technology keeps changing, the chip scale keeps changing, and the type of materials keep changing. The frequencies and speeds keep getting higher and higher, so things keep getting harder to design. The packaging is smaller to have the system on, and you have integrated embedded passives and components at that point. The board becomes an integrative part of the overall solution.

Automation is great, but you have to have somebody that knows how to drive the automation. You have to have an understanding of what it takes to design some of these products, and the problem is some of the guys who are writing the software for the automation aren't necessarily designers. As far as the automation that occurs there, it still takes that human intervention to drive it the right way. Anybody can get a license to drive a car, but can anybody go on the Daytona 500 or run some road rally race and win? The whole process of design is an evolution. Throughout the course of design from start to finish, when you lay down the first track, when you import the first net list, the design is going to be slightly different than when you started because it's an evolutional process. Whether it be through that one spin or multiple spins, there's an evolution that occurs and power consumption.

It may be designed initially to handle a certain amount of amps or watts for that particular product. Well next thing you know, as you actually design the product you might be able to reduce the power consumption because you're simulating at maximum power. When you do that it is definite overkill. Do you need to design it to that maximum power curve or can you design it for less? There might be a cost driver on the board, maybe fewer layers or less copper. Your component selection might be derated at a different level than the higher rated components. Again, it comes back to the original question about designing out costs or DFP.

***Matties:** Yes, you can design to increase profit and performance. Just to shift gears a little, we're here at Geek-A-Palooza. What do you think?*

Brown: It's actually a pretty good event. It's allowing everyone to network in a more casual atmosphere than a traditional trade show type environment, which is kind of nice. It lets people let their hair down per se, and relax. This is nice.

***Matties:** Well, thanks for being here. It's been great talking with you.*

Brown: Thank you. It's been a pleasure. **PCBDESIGN**

OML

OPEN MANUFACTURING LANGUAGE

The standard for the Internet of Manufacturing (IoM) has arrived!

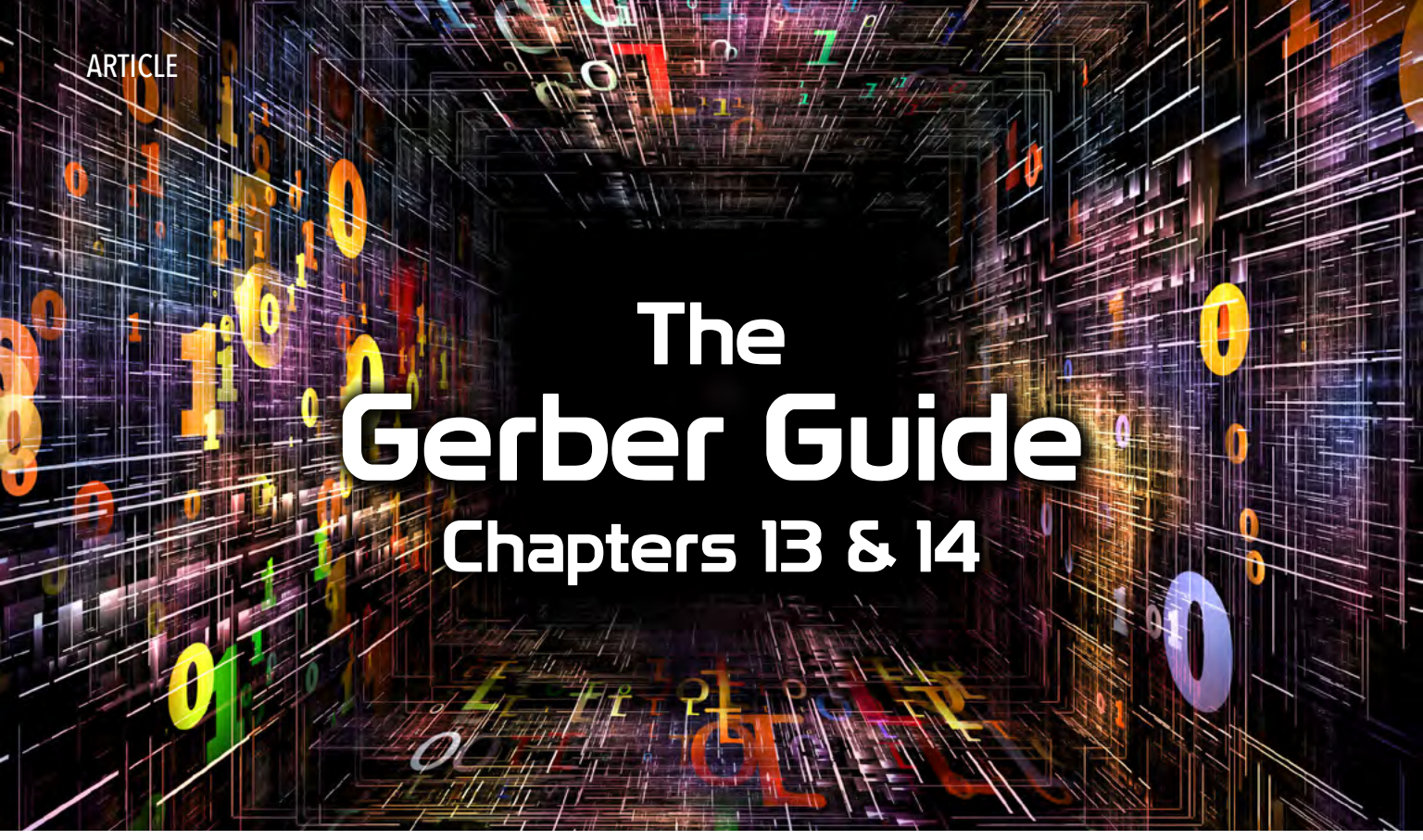
The Open Manufacturing Language (OML) is a real-time communication standard for PCBA manufacturing that defines the interconnectivity of assembly production processes and enterprise IT systems.

For the first time, IT teams, solution providers, and equipment providers can easily integrate shop-floor data to create manufacturing execution solutions based on a single, normalized, vendor-neutral communication interface.

Take part in shaping the future!

Become a member of the OML Community where PCB Assembly industry professionals have FREE full access to the OML specification, white papers written by industry experts, and a place to share ideas in our community forum.

Visit http://www.omlcommunity.com and join the community!



The Gerber Guide
Chapters 13 & 14

by Karel Tavernier
UCAMCO

It is possible to fabricate PCBs from the fabrication data sets currently being used; it's being done innumerable times every day, all over the globe. But is it being done in an efficient, reliable, automated and standardized manner? At this moment in time, the honest answer is no, because there is plenty of room for improvement in the way in which PCB fabrication data is currently transferred from design to fabrication.

This is not about the Gerber format, which is used for more than 90% of the world's PCB production.There are very rarely problems with Gerber files themselves; they allow images to be transferred without a hitch. In fact, the Gerber format is part of the solution, given that it is the most reliable option in this field. The problems actually lie in which images are transferred, how the format is used and, more often, in how it is not used.

Each month we look at a different aspect of the design to fabrication data transfer process. In this monthly column, Karel Tavernier explains in detail how to use the newly revised Gerber data format to communicate with your fabrication partners clearly and simply, using an unequivocal yet versatile language that enables you and them to get the very best out of your design data.

Chapter 13: The File Extension

Wikipedia states, "A filename extension is a suffix (separated from the base filename by a dot or space) to the name of a computer file applied to indicate the encoding (file format) of its contents or usage. Examples of filename extensions are .png, .jpeg, .exe, .dmg and .txt."

Microsoft states, "A file name extension is a set of characters added to the end of a file name that determine which program should open it." The advantage of this rule is that the file format is clear without first opening the file, and, consequently, so is the preferred application for the file.

To quote from the Gerber format specification:

The Gerber Format has a standard file name extension a registered mime type and a UTI definition.

Standard file extension: .gbr or .GBR.

Use the standard file extension .gbr or .GBR on all Gerber files.

All too often, names such as pn674847.top are used, meaning the top layer of job PN674846. This idea dates from the 1980s, the days of MS-DOS, the first Microsoft operating

MIRACO

Engineering And Providing Balanced Interconnect Solutions

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

...connecting technology

system. The file names in MS-DOS were restricted to a measly eight characters. Abusing the file extension to gain a few extra characters may have made sense in those days, but it makes no sense today.

File extensions such as .ger, .pho, etc. are also sometimes used. They do indeed express the format, but in a proprietary, non-standard way. Always use the standard extension .gbr or .GBR.

One may object that the supply chain is used to a file name convention such as in pn674847.top and it is not possible to simply throw it overboard. Fine: give this file its full name, in this case, pn674847.top.gbr. This is valid and the old file name is still prominent.

Remember: Always use the standard file extension ".gbr" for Gerber files.

Chapter 14: Negative Copper Layers

Negative layers are a relic of the 1960s and 1970s and the age of the vector photoplotter, which are now as obsolete as the mechanical typewriter. The vector photoplotter was similar to a pen plotter, but instead of using ink and paper, it wrote onto photosensitive film using a stationary light "pen." The film was held firm on a flat table that moved in the X-Y plane under the pen's light beam which was switched on and off as the image dictated. Every movement was governed by commands in input Gerber files.

This was fine for drawing tracks. The problem started with planes, or anything with large copper pours such as that shown in Figure 2.

This is because vector plotters created these large copper pours using a technique called "vector-fill," "painting," or "stroking." This involved repeatedly moving the table back and forth under the pen, just as a child moves a crayon back and forth over an area until it is completely filled, as in Figure 3.

In principle, this worked. But in practice, the input Gerber file, containing the zillions of draws that were needed for the vector fill, was huge. More importantly, it took ages—easily an entire shift—to plot a plane, so it was highly impractical.

The solution was to plot in negative, in other words, creating the clearances rather than the copper pours, as shown in Figure 4.

This eliminated the need for painting, and the Gerber file size and plotting times were kept to a minimum. The negative film thus created was then used in the photolab as a phototool, to generate the positive film that was necessary for downstream production processes. Although this added a step and involved manual work, it was infinitely better than blocking the expensive photoplotter for a whole shift with impatient customers breathing down one's neck and demanding their plots. Negative films made a lot of sense in the days of vector photoplotters.

But time has moved on since then, and so

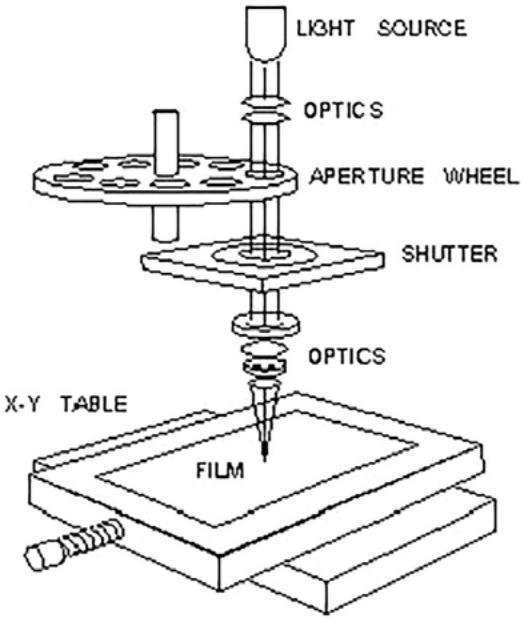


Figure 1: Vector photo plotter.

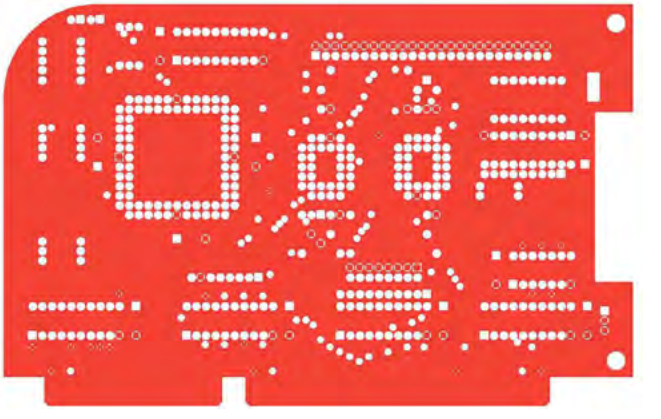


Figure 2: A plane layer.

2016 Programs

www.ipc.org/2016-events

June 6

ITI & IPC Conference on Emerging & Critical Environmental Product Requirements — Conference

Boston, MA USA

June 8

ITI & IPC Conference on Emerging & Critical Environmental Product Requirements — Conference

Chicago, IL USA

June 10

ITI & IPC Conference on Emerging & Critical Environmental Product Requirements — Conference

Silicon Valley, CA USA

June 29

IPC Education Online — Webinar

Wisdom Wednesday — for IPC Members ONLY
30 minutes of FREE technical insight
from industry experts

July 19–20

IPC Technical Education — Workshop

Chicago, IL USA

July 27

IPC Education Online — Webinar

Wisdom Wednesday — for IPC Members ONLY
30 minutes of FREE technical insight
from industry experts

August

IPC Education Online: Summer School — Webinar

August 31

IPC Education Online — Webinar

Wisdom Wednesday — for IPC Members ONLY
30 minutes of FREE technical insight
from industry experts

September 21

IPC Education Online — Webinar

Wisdom Wednesday — for IPC Members ONLY
30 minutes of FREE technical insight
from industry experts

September 21–23

IPC India 2016 — Conference & Exhibition

Co-located at electronica India and productronica India 2016

September 24–30

IPC Fall Committee Meetings — Meeting

Held in conjunction with SMTA International

Rosemont, IL USA

September 26

EMS Management Meeting — Meeting

Rosemont, IL USA

October 19

IPC Education Online — Webinar

Wisdom Wednesday — for IPC Members ONLY
30 minutes of FREE technical insight
from industry experts

October 25–27

IPC-SMTA Cleaning and Conformal Coating Conference — Conference

Chicago, IL USA

November

IMPACT Europe 2016 — Meeting

Brussels, Belgium

November 2

PCB Carolina: Regional Trade Show — Conference and Exhibition

Presented by the RTP Chapter of the IPC Designers Council

Raleigh, NC USA

November 2–3

IPC Technical Education — Workshop

Held in conjunction with PCB Carolina

Raleigh, NC USA

November 7–11

IPC EMS Program Management Training and Certification — Certification

Chicago, IL USA

November 16

IPC Education Online — Webinar

Wisdom Wednesday — for IPC Members ONLY
A Vision for the Industry

December

IPC Education Online: Winter Semester — Webinar

December 7–9

HKPCA International Printed Circuit & IPC APEX South China Fair — Conference and Exhibition

Shenzhen, China

December 14

IPC Education Online — Webinar

Wisdom Wednesday — for IPC Members ONLY
Be a Resource for Your Customers, Suppliers, and Team Members: making the Most of IPC Services

February 14–16, 2017

IPC APEX EXPO 2017 — Conference and Exhibition

San Diego, CA USA

Questions? Contact IPC registration staff at registration@ipc.org or +1 847-597-2861

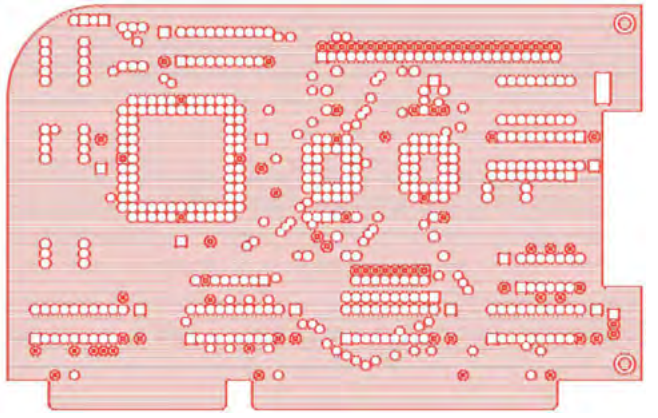


Figure 3: A stroked plane layer.

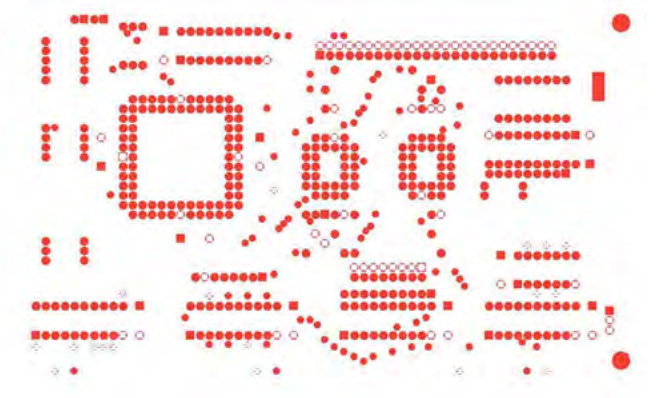


Figure 4: A plane layer in negative.

have printing devices. For decades, plotters and direct imagers have used raster-scan technology, whose speed depends solely on image area and resolution. Image content no longer dictates throughput so same-size planes and signal layers take exactly the same time to plot.

So, negative files no longer offer benefits. On the contrary, they introduce some serious disadvantages:

• While positive layers have clearly defined limits, negative layers do not, so arbitrary limits must be imposed.

• In a mixed data set, there is no standardized method by which to define which layers are negative, so manual reverse engineering is necessary. When all layers are positive there is no problem.

• Most importantly, negative layers do not contain copper pour outlines, so these must be created. This can be done by maintaining specific clearances from the profile, for example, but this is still guesswork and reverse engineering. In positive layers, on the other hand, copper outline is clearly defined.

How to define a positive copper plane:

```
G04 We define the antipads
%TF.AperFunction,AntiPad*%
%AD11C....*%
....
G04 We now define the extent of the copper
pour*
LPD*
G36*
X...Y...D02*
X...Y...D01*
...
G37*
G04 And now we flash clearances
%LPC*%
D11*
X...Y...D03*
....
```

This gives CAM clear and unequivocal information which is robust, numerically accurate, and the anti-pads will register with the drill files. And it's as compact as a negative file, and clearly defines the extent of the copper. Perfect.

There is no longer any benefit in transferring the data in negative. On the contrary: It is a relic from the bad old days that adds confusion, manual work, and risk.

Always use positive copper layers. **PCBDESIGN**

This column has been excerpted from the Guide to PCB Fabrication Data: Design to Fabrication Data Transfer.

Karel Tavernier is the managing director of Ucamco.

The ANS team is devoted to technological innovation as well as providing our customers with a high rate of satisfaction and speedy service. With our strengths in R&D, we produce PCBs that meet your specific requirements promptly, and at competitive prices.

The ANS mission is to pursue product and service excellence through continuous improvement in a quest for Total Customer Satisfaction.

ANS

REQUEST A QUOTE TODAY!

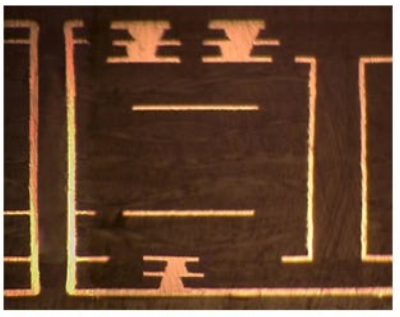
Head office:
ANS Industrial Co. Ltd.
Kowloon, Hong Kong
Tel: (852) 2757 1882

Factory:
Remapp PCB (Foshan) Co.,Ltd.
Guangdong Province, China
Tel: (86) 757-8862 3888

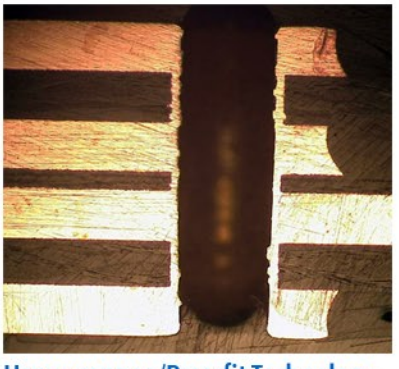
www.anspcb.hk

Email: sales@anspcb.hk

PRODUCTS

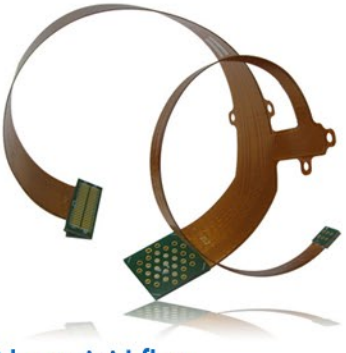


Blind/Buried/Stack via technology



Heavy copper /Pressfit Technology

Multi-layered aluminum board
lead-free HAL



8 layer rigid-flex
30mm flexible finished thickness

UL IPC RoHS

TOP TEN

Recent Highlights from PCBDesign007

1 Douglas G. Brooks Co-Authors "Trace and Via Currents and Temperatures"

Douglas G. Brooks, PhD and Dr. Johannes Adam, CID have teamed up to write PCB Trace and Via Currents and Temperatures: The Complete Analysis. Brooks has been looking at trace current and temperature relationships since the mid-1990s. Now, he and Adam, of the consulting group Adam Research, have assembled decades of knowledge into these pages.

2 Designers Notebook: Flexible and Rigid-Flex Circuit Design Principles, Part 6

The designer is generally under pressure to release the documentation and get the flexible circuit into production. There is, however, a great deal at risk. Setting up for medium-to-high volume manufacturing requires significant physical and monetary resources. To avoid potential heat from management, the designer must insist on prototyping the product and a thorough design review prior to release.

3 Webinar: Leveraging SerDes Design Flows for IBIS-AMI Model Development

In this webinar attendees will learn about a new workflow that has been developed jointly between MathWorks and SiSoft, which enables semiconductor companies to reuse their SerDes designs directly for IBIS-AMI model creation, simulation, and validation.

4 Design Automation Tools, Today and in the Future

Kelly Dack has been designing PCBs for over three decades, at OEMs of all kinds. Now a PCB designer with a Washington state contract manufacturer and a certified trainer with EPTAC, Kelly enjoys waxing philosophic about PCB design and design automation in general. I asked Kelly to answer a few questions about the direction EDA tools are headed, and whether he'd like to see more control, or more automation in his PCB design tools.

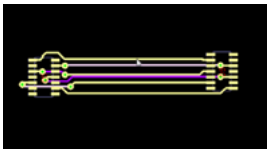
5 Cadence Paper: Automating Inter-Layer In-Design Checks in Rigid-Flex PCBs

Flexible PCBs (flex/rigid-flex) make it possible to create a variety of products that require small, lightweight form factors such as wearable, mobile, military, and medical devices. This paper discusses some of the key challenges to address and also introduces a new PCB design approach that enhances productivity.

6 The Partnership: Design Engineers and PCB Designers

Randy Faucette is founder, president and director of engineering at Better Boards Inc. in Cary, North Carolina. Founded in 2003, Better Boards provides electrical engineering, PCB design, signal and power integrity analysis, and a variety of other services. I asked Randy to talk about some of the occasional tension between PCB designers and design engineers, and what he thinks can be done to help open the lines of communication.

7 Zuken Introduces Perfect Springback Routing in CADSTAR 17



Zuken has rolled out routing enhancements in the latest version of its CADSTAR desktop PCB design software. Other productivity enhancements include improved routing patterns for differential pairs, and etch factor support. Jeroen Leinders, CADSTAR worldwide sales manager, said, "CADSTAR 17 challenges the view that PCB Desktop software has to be complicated."

8 Read the DesignCon Award-Winning Paper by Mentor Graphics and Wild River Technology

This paper, "BER- and COM-Way of Channel-Compliance Evaluation: What Are the Sources of Differences?" won the DesignCon 2016 Best Paper Award. It was written by Vladimir Dmitriev-Zdorov, Chuck Ferry, and Christian Filip of Mentor Graphics, and Alfred P. Neves of Wild River Technology.

9 Beyond Design: DDR3/4 Fly-by vs. T-topology Routing

JEDEC introduced fly-by topology in the DDR3 specification for the differential clock, address, command and control signals. Fly-by topology supports higher-frequency operation, reduces the quantity and length of stubs and consequently improves signal integrity and timing on heavily loaded signals. Fly-by topology also reduces SSN by deliberately causing flight-time skew, between the address group and the point-to-point topology signals, of the data groups. Barry Olney explains.

10 SiSoft: Optimizing the State of the Art

In the 20 years since its founding, SiSoft has been at the forefront of signal integrity analysis tool development. Now, the company is leading the way with a new technology called OptimEye and tools for creating accurate IBIS-AMI models. At DesignCon, I caught up with Todd Westerhoff, VP of semiconductor relations, and asked him to give us an update on the company's newest technologies.

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.

iNEMI Asia Roadmap Workshop/Webinar
June 23, 2016
China

Symposium on Counterfeit Parts and Materials 2016
June 28–30, 2016
College Park, Maryland, USA

IPCA EXPO 2016
August 18–20, 2016
Delhi, India

Medical Electronics Symposium
September 14–15, 2016
Marylhurst, Oregon, USA

IPC India/electronica India 2016/ productronica India 2016
September 21–23, 2016
Bengaluru, India

IPC Fall Meetings
September 24–30, 2016
Rosemont, Illinois, USA

SMTA International 2016
September 25–29, 2016
Rosemont, Illinois, USA

electronicAsia
October 13–16, 2016
Hong Kong

TPCA Show 2016
October 26–28, 2016
Taipei Nangang Exhibition Center
Taipei, Taiwan

electronica
November 8–11, 2016
Munich, Germany

FUTURECAR: New Era of Automotive Electronics Workshop
November 9–10, 2016
Atlanta, Georgia, USA

International Printed Circuit & Apex South China Fair (HKPCA)
December 7–9, 2016
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**

THE pcb design MAGAZINE

June 2016, Volume 5, Number 6 • The PCB Design Magazine© is published monthly, by BR Publishing, Inc.

ADVERTISER INDEX

I-Connect007

GOOD FOR THE INDUSTRY

FREE SUBSCRIPTION

Subscribe

007 Media Kit

subscribe.iconnect007.com

mediakit.iconnect007.com

EDITORIAL CONTACT

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

SALES CONTACT

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

pcb design 007 | pcb 007 | SMT 007 | mil aero 007 | flex 007 | ein 007 | RTW

www.iconnect007.com