

I·Connect007

JULY 2021

DESIGN007

M A G A Z I N E



**“A must-read for anyone
new to system analysis.”**



Lee Ritchey
Founder and President
Speeding Edge

**THE
SYSTEM DESIGNER'S GUIDE TO...™**

SYSTEM ANALYSIS

Electromagnetic Interference and
Thermal Analysis of Electronic Systems



Brad Griffin
Cadence

**100%
Books**



Download Now



Distinctly different.

Our books are written by recognized industry experts.
At around 8,000 words, they are unique in that they are able
to be incredibly focused on a specific slice of technology.

“I-007ebooks are like water in the desert
...it’s up to you to drink it in order to survive!”

Stephen V. Chavez
PCEA Chairman, MIT, CID+



VISIT OUR LIBRARY



DESIGN007

M A G A Z I N E

ASK THE EXPERTS

This month, we put your burning questions to our team of PCB designers and design engineers: Lee Ritchey, Carl Schattke, Rick Hartley, Heidi Barnes, Stephen Chavez, Eric Bogatin, Cherie Litson, Chris Young, and Happy Holden. The entire issue is packed with answers to questions that designers like you are wrestling with right now. Enjoy!

FEATURE ARTICLE

12 Meet the Experts

▼ EXPERT ANSWER TOPICS

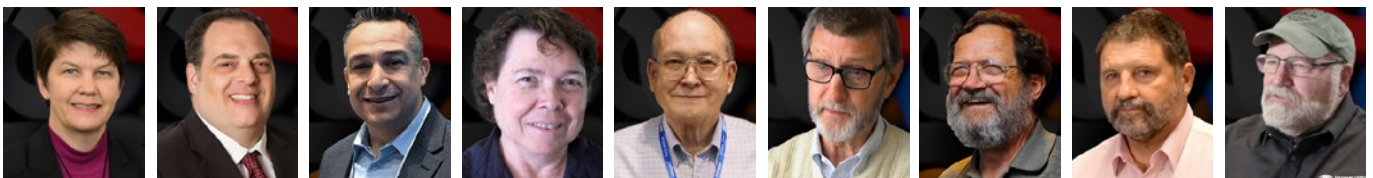
- 14 Skip Vias
- 20 Tying Vias to Ground and Power Planes
- 26 Class 3 and Annular Rings for DFM Copper
- 28 Flooding
- 34 Crosstalk
- 36 Stackup Options
- 37 Electrical Engineering Curriculum With Fabrication Process
- 48 Solid Reference Planes or Not
- 50 Skew from Fiber Weave Effect

▼ MORE EXPERT ANSWER TOPICS

- 62 Dimensions on Drawing Required
 - 70 Designing 10–12 Layer Stackups
 - 72 Fabricators Not Sharing Capabilities
 - 75 Weak Microvia Workarounds; Capacitor Lead Time
 - 76 Retiring Engineers and Attracting New PCB Designers
 - 78 Interpreting Data Sheets for Heat Dissipation
-

FEATURE COLUMN

- 82 Ask the Experts—PDN Filters
by Istvan Novak



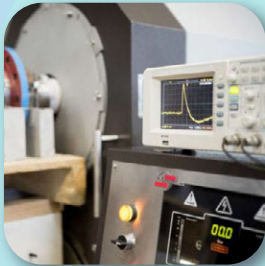


Candor

Circuit Boards

Innovative solutions for complex circuit board manufacturing

Click to see what we bring to each sector of the industry:



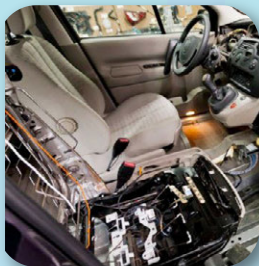
RF/Microwave
PCBs



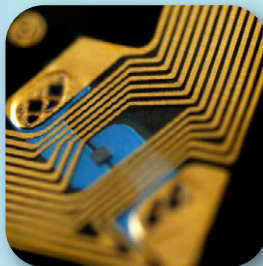
Power
Electronics



High-Speed
Data



Automotive
Electronics



Lamination



Startups

Our technology produces what our customers dream!

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

DESIGN007

M A G A Z I N E



38

SHORTS

- 33** Electronic Paper on an Industrial Scale
- 60** Researchers to Develop Made-to-Measure 3D Laser Beams
- 71** Inkjet Printing Shows Promise as New Strategy for Making E-Textiles
- 74** New I-007 eBook Highlights System Analysis
- 77** Intelligent Carpet Gives Insight into Human Poses

DEPARTMENTS

- 105** Career Opportunities
- 120** Educational Resource Center
- 121** Advertiser Index & Masthead



HIGHLIGHTS

- 80** PCB007
- 90** MilAero007
- 102** Top 10 PCBDesign007



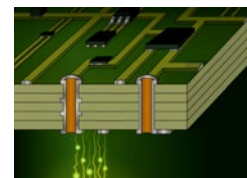
ARTICLES

- 38** Whizz Systems: The Silicon Valley CM
by the I-Connect007 Editorial Team
- 52** DFM 101: PCB Controlled Impedance
by Anaya Vardya



COLUMNS

- 10** Just Ask the Design Experts
by Andy Shaughnessy
- 16** Switchbacks in Tuned Routing
by Barry Olney
- 22** Some Timely Advice
by Tim Haag
- 30** The Board Thickness Challenge
by Matt Stevenson
- 56** Resins—Fundamental Considerations for Circuit Designers
by Beth Turner



- 64** Say What You Do—Do What You Say
by Kelly Dack





Model Your High-Speed Connector and PCB as One in True 3D

While others struggle with the modeling and simulation of just one complex connector, we enable you to model and simulate the connector with a PCB in one go.

LEARN HOW

FLEX007

This Month in Flex

This month, our Flex007 contributors delve into ways to reduce respins in rigid-flex designs. At rigid-flex's price point, you definitely don't want to pay for respins if they can be avoided.

FLEX007 FEATURE COLUMNS

- 92** **Rigid-Flex Design Without Respins—
A Webinar Review**
by Andy Shaughnessy

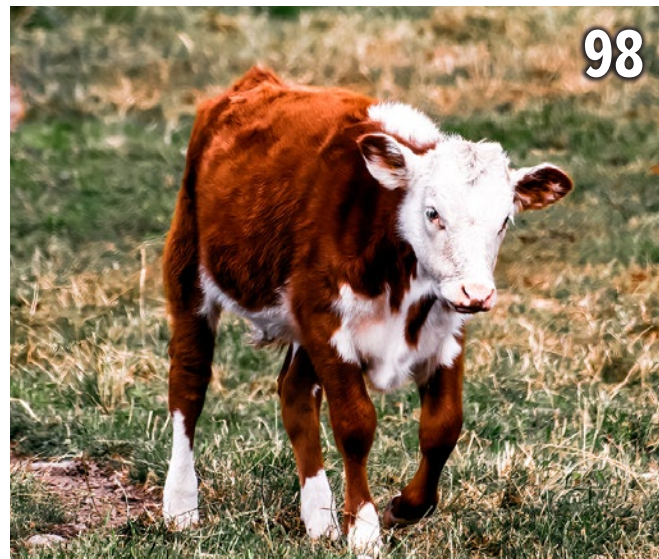
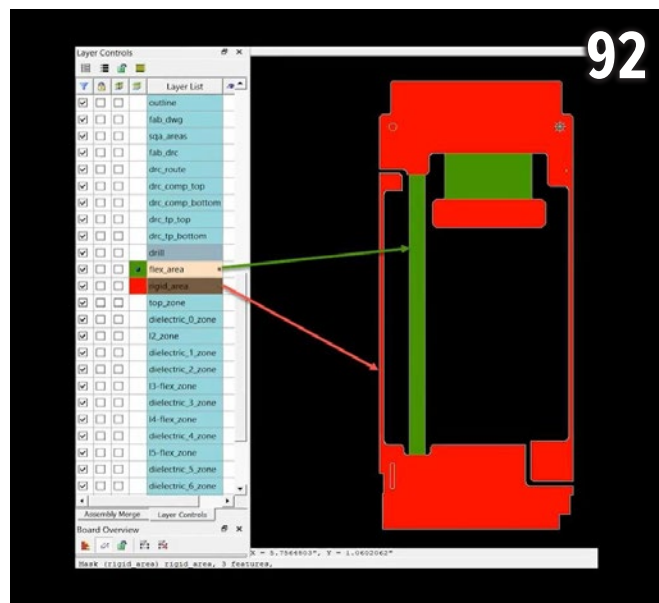
- 98** **The Calf Path—Redux**
by Joe Fjelstad

HIGHLIGHTS

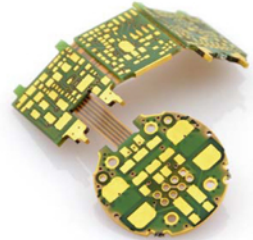
- 96** **Flex007**

FLEX007 SHORT

- 94** **Hire or Be Hired at jobConnect007**



Focused on Flex

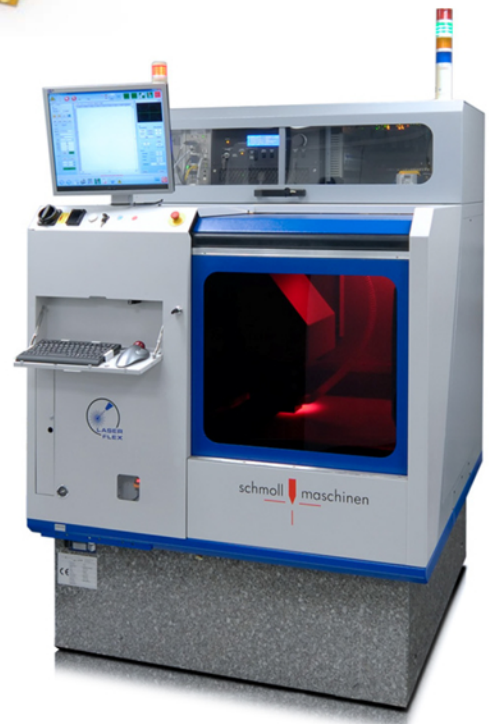


Laser μ Flex series

The versatility and precision of laser processing makes it the ideal tool for machining a wide range of materials.

Typical applications:

- Polymer cover foil
- Covered polymer laminates
- Thin, rigid-flex materials
- Inner layers and prepregs
- Separation or decap from tracks
- Structuring cavities
- Micro-via and through-hole drilling



For more information call us: +1 (336) 660-2701

burkleamerica.com

Just Ask the Design Experts

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

We want to help PCB designers do their jobs better, faster, and easier. When you get right down to it, that's really the whole point of this magazine's existence—to give you design information that you can use right away.

Sure, we try to make the magazine fun to read. We plan great covers months in advance; the June *Star Trek* cover is a great example. We have entertaining columnists and contributors who make perusing through the magazine an enjoyable experience so that you'll look forward to coming back each month.

We also focus on the theory behind PCB design. But all the electronics theory and background information in the world isn't as

useful as that one little high-speed design tidbit that helps you put your current project to bed. Ideally, you will learn something in these pages each month that you can use on your job immediately, so that you can get on to the next job. And the design jobs keep coming, like Lucy Ricardo and Ethel Mertz working on the conveyor belt at the chocolate factory.

I can't think of a better way to help our readers get the info they need right now than by going right to the "horse's mouth," as the old axiom goes. What are your big problems today?

We decided to let the readers pose their questions to our team of industry experts.



We asked our readers to send in their questions, and they responded with queries from across the spectrum: DFM, stackups, supply chain issue, education, signal integrity—you name it.

We put these questions to our team of PCB designers and design engineers: Lee Ritchey, Carl Schattke, Rick Hartley, Heidi Barnes, Stephen Chavez, Eric Bogatin, Cherie Litson, Chris Young, and Happy Holden. As you might expect, these experts often had slightly different takes on the same question, and occasionally they contradicted each another entirely.

Often, there are no hard, fast answers to a particular PCB design problem. Ask 20 design instructors how they would solve a particular problem, and you might get 20 possible scenarios. There's a lot of nuance in this segment of the industry, which is what makes this such a fascinating beat for me to cover. (In my previous life as a crime reporter, there wasn't much nuance. I usually focused on whether a suspect was dead, in jail, or out on bond, and not much else.)

So, this month, we bring you an entire issue devoted to Q&A. Check it out; there's so much here that I bet you'll learn something, even if you started designing boards before the Water-

gate break-in. The supply chain question drew some of the most varied responses: what are you doing to alleviate the component shortages we're facing today?

We also bring you our regular contributors: Barry Olney, Istvan Novak, Tim Haag, Matt Stevenson, Anaya Vardya, Kelly Dack, and Joe Fjelstad. And we have a great interview with Muhammad Irfan and Dan Williams of Whizz Systems.

The world is starting to open again, hopefully for good. We'll be covering the live, in-person trade shows and conferences coming up in the next few months, such as DesignCon, PCB West, SMTA International, and IPC APEX EXPO. Hard to believe that I last flew on a plane in February 2020, flying back from two weeks in the Golden State covering DesignCon and IPC APEX EXPO 2020.

That seems like a lifetime ago. Virtual shows are all well and good, but I'm looking forward to seeing you all in person again. **DESIGN007**



Andy Shaughnessy is managing editor of *Design007 Magazine*. He has been covering PCB design for 20 years. He can be reached by [clicking here](#).



MEET THE EXPERTS

For our “Ask the Experts” issue, we asked our readers to send in their questions for our team of veteran PCB designers and design engineers. There were no real rules here; questions could be about any topic related to PCB design.

Readers responded with a list of questions that ran the entire gamut, from DFM through signal integrity and much, much more. Most were technical design questions, but a few readers asked about issues such as the supply chain, the future of design education, and replacing those designers and engineers who are busily planning their retirement.

We’d like to thank our experts for sharing their thoughts, insights, and opinions with the industry. We would also like to thank the designers and design engineers who sent in their questions. We depend on input from readers like you to help keep us on top of the topics and trends in PCB design.

All the experts featured in this issue are listed here for easy reference. We hope you enjoy our “Ask the Experts” special feature. Designers are hungry for this knowledge, and we’re happy that we could help them get the answers they need to do their jobs better.

Heidi Barnes is a senior applications engineer for high-speed digital applications in the Design Engineering Software Group at Keysight Technologies. She holds five patents, and she’s authored over 20 papers on signal and power integrity. Among other accolades, Heidi has been awarded NASA’s Silver Snoopy and named DesignCon’s 2017 Engineer of the Year.

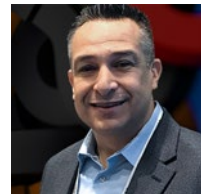


Eric Bogatin is a high-speed design instructor and an adjunct professor at the University of Colorado–Boulder in the ECEE department. Known as a “signal integrity evangelist,” Eric is



the dean of the Teledyne LeCroy Signal Integrity Academy. He’s written six books on signal integrity and interconnect design and over 300 papers. An engaging speaker, Eric is known to toss candy bars to class attendees who answer questions correctly.

Stephen V. Chavez is a senior-level CID+ PCB designer with 30 years of experience, and president of the Printed Circuit Engineering Association (PCEA). He is also an IPC Master Instructor Trainer (MIT) for IPC’s CID and CID+ programs. A veteran of the Marine Corps, Stephen has spent the last 10 years as the lead PCB designer with a major aerospace company.

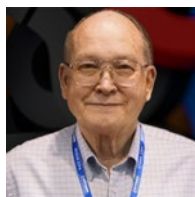




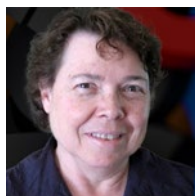
Rick Hartley is a PCB design instructor and founder of RHartley Enterprises. He is a PCB designer and design engineer who has spent over 50 years in the electronics industry in a variety of roles. Prior to opening his own consulting firm, Rick was a senior principal engineer with L-3 Avionics Systems. He teaches high-speed design classes at trade shows and conferences around the world.



Happy Holden is known as the “Father of HDI” and author of *The HDI Handbook*. Happy was a pioneer in fabrication and design processes at Hewlett-Packard in the early 1970s. He retired from H-P after 28 years; he later served as a senior PCB technologist for Mentor Graphics’ System Design Division, and CTO for Foxconn’s MIP Division, which manufactured millions of iPhones. Happy is currently a technical editor with I-Connect007.



Cherie Litson is the founder of Litson1 Consulting and an instructor with EPTAC and Everett Community College. She is a Master Instructor for the IPC CID and CID+ programs, and she has over 30 years of experience in PCB design. Cherie holds a BSEE. She is also an avid golfer, taking LPGA teams from local to national competitions in the past five years.



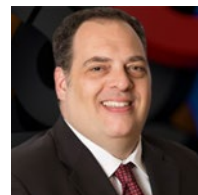
Lee Ritchey is a PCB design instructor and founder and president of Speeding Edge, a high-speed design consulting firm. He served as program manager for 3Com Corporation, where he was responsible for overseeing the signal integrity aspects of hardware design and product packaging for the company’s router, switch, hub and NIC products. Lee is the co-author of *Right The First Time, A Practical Handbook on High-Speed PCB and System Design*.



Carl Schattke is a lead PCB design engineer with a North American automobile manufacturer, and a PCB design instructor. He previously served as senior hardware engineer in PCB design at Intel. He learned PCB layout by working as an apprentice to his father, hand-taping hundreds of designs. He enjoys discussing PCB design rules, and he’s been known to quote the Dalai Lama: “Know the rules well, so you can break them effectively.”



Chris Young has over 20 years of experience as an electrical engineer, working predominantly in the avionics/aerospace industry, supplemented by experience in the semiconductor and medical fields. He is the owner/lead engineer of Young Engineering Services. Chris is also a columnist with Milaero007.



ASK THE EXPERTS

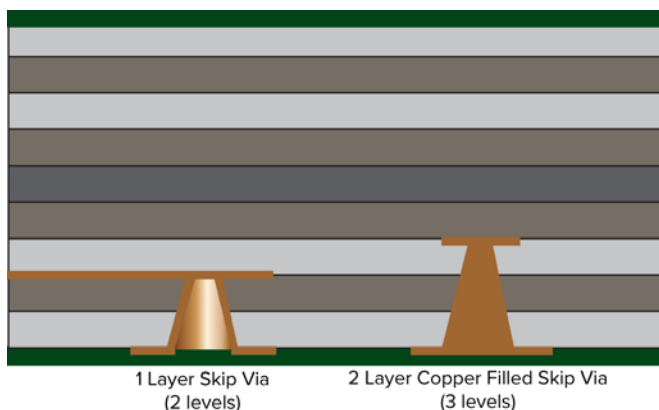
YOUR BURNING QUESTIONS... ANSWERED

Q *Are skip vias common? Do they cost more than regular microvias? Our company thinks it might be worth it if it saves us even one lamination cycle.*

Heidi Barnes: The RF/microwave world has been using skip vias for at least 20 years, and yes, it did avoid the requirement for sequential lamination with the ability to go from layers 1 to 2 or layers 1 to 3 without having a via stub. This also has the advantage of increasing the routing area below. The challenge with skip vias is that the deeper they go, the wider the diameter needs to be so that the plating can reach down into the via. Typically, one tries to not go less than a 1:1 aspect ratio. A lot depends on the fabricator; if a shop specializes in high-volume sequential lamination with microvias, then they may charge more for the additional laser tuning and plating issues that come with skip vias vs. just doing more of what they do

best. As always, it depends. I would consider both electrical performance/reliability and cost before making the decision.

Carl Schattke: Skip vias are not common; they add cost to a standard through-hole board because they add an additional drilling step for each side where they are used. However, they will be lower cost than adding an additional lamination cycle. I do find them in use on RF boards where you have GND on top, then the RF, then a GND layer again. You can bring the RF trace to L1 to L2 with a regular microvia, then shield vias around it with the L1 to L3 skip vias. This can save cost on an RF board. The other area where all laser drill vias make sense is when you have no mechanically drilled vias. Laser drilling is seconds per panel vs. mechanical drilling in the minutes or high minutes on larger boards. Skip vias do not cost more than regular microvias; they just use a little more energy to ablate the material away a bit deeper, but far less than buried vias on an extra lam cycle. They will cost a slightly larger pad size, but I usually just use that size when designing so I don't have to keep track of it, and it is a minimal difference. **DESIGN007**





Serious About Speed

For over 30 years Prototron
Circuits has led the pack
when it comes to
providing quality
circuit boards FAST.

Be it Class 3, AS9100,
ITAR or MIL-PRF-31032,
Prototron has the speed
you need.



Prototron Circuits
America's Board Source

Switchbacks in Tuned Routing

Beyond Design

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

A switchback is a 180° bend in a road, rail, or path, especially one leading up the side of a mountain. I used to enjoy driving the Great Ocean Road (Figure 1), which is a National Heritage-listed 243-km stretch of windy road along the southeastern coast of Australia. The tight serpentine bends certainly delay your journey but also make it a more enjoyable trip.

Switchback also refers to a long trombone bend in a tuned serpentine trace. But rather than increase the delay of the signal, the switchback actually speeds it up due to the near (NEXT) and far-end (FEXT) crosstalk effects. In this month's column, I will look at why long, parallel switchbacks should be avoided.

Designing a memory interface is all about timing closure. Each signal's timing needs to be

compared to the related clock or strobe signal in such a way that the data can be captured on both the rising and falling edge of the strobe, hence the term double data rate (DDR). However, the constant increase in data rate has made the timing margin associated with each rising and falling edge much tighter. To match the delay of critical signal timing, adjustments are required to the length of the individual signals within a group. This is accomplished by adding serpentine (accordion) bends in the traces to decrease the velocity of the signal to match the longest delayed signal. However, the opposite occurs—the velocity of the signal is sped up by the serpentine.

When an electromagnetic (EM) wave is guided by a serpentine trace, with coupling between the bends, there is an increase in the speed of the signal. That is, the EM wave negotiates the serpentine section faster than that of a straight trace of the same length. This acceleration is caused by crosstalk coupling (NEXT and FEXT) between the parallel trace segments of the serpentine traces. The amount of acceleration is directly proportional to the coupling strength between the bends and to the rise time of the signal. For long, coupled lengths (those longer than the critical length), signals may become distorted as they pass the serpentine section.



Figure 1: The Great Ocean Road in southern Australia.

Integrated Tools to Process PCB Designs into Physical PCBs



Visualize

Use manufacturing data to generate a 3D facsimile of the finished product.



Verify

Ensure that manufacturing data is accurate for PCB construction.



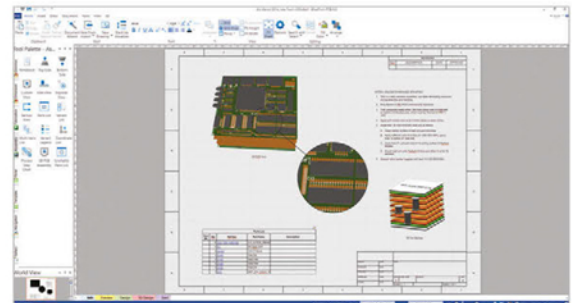
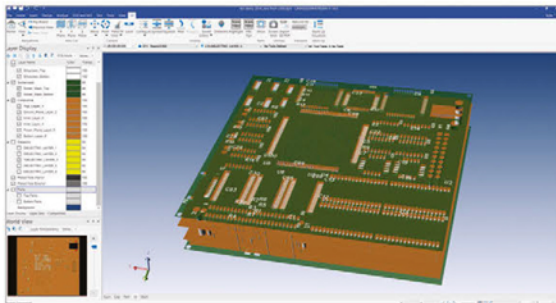
Panelize

Minimize costs in both PCB fabrication and assembly through maximum panel usage.



Document

Quickly and easily specify the PCBs form, fit and function.



CAM350®

Verify and Optimize PCB Designs
for Successful Manufacturing



BluePrint-PCB®

Create Comprehensive Documentation to
Drive Fabrication, Assembly & Inspection

DownStream Technologies, offers an integrated manufacturing data preparation solution for PCB post processing including manufacturing data visualization, stack-up definition, PCB panel visualization and support for 3D PCB documentation. This environment allows PCB designers and engineers to visualize, verify and document a printed circuit board before manufacturing.



800-535-3226 | downstreamtech.com

©2019 DownStream Technologies, Inc. All rights reserved.

NOW
WITH **3D**

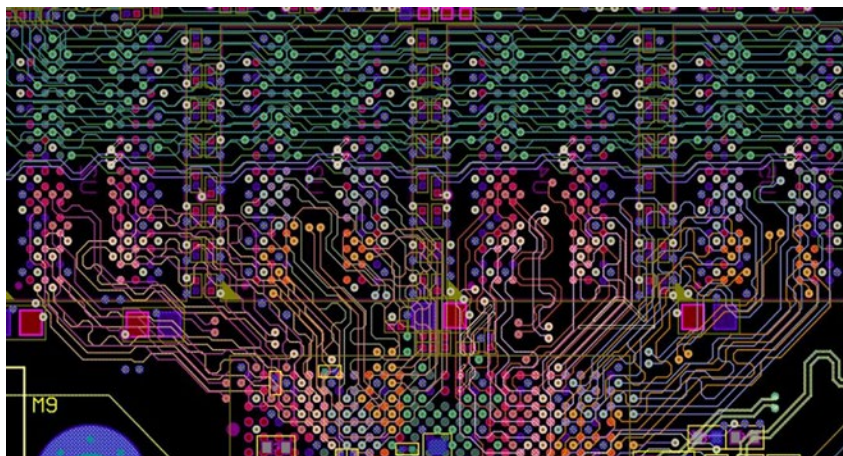


Figure 2: DDR4 fly-by routing.

Signals pass relatively undistorted along short, coupled serpentine sections, but distortion begins to occur when the parallel trombone length approaches one-third the signal wavelength. Figure 2 depicts a properly routed DDR4 fly-by design using short serpentes.

A tight design calls out explicit tolerances on signal timing. The timing requirements for DDR memory are outlined in the JEDEC standards and should be used as a metric for accuracy. The meandering traces must be compacted into an extremely tight space and so one is often tempted to use any method available to complete the design. The boundary between short and long coupled switchbacks is fuzzy. In general, when the round-trip delay of a heavily coupled switchback far exceeds one-third of the rise time, you get seriously distorted signals; when it's much less than one-third, you get advanced timing—and that's what we typically see. A 1 ns rise time used in an FR-4 dielectric limits the maximum useful coupled switchback length to about 1 inch (2 inches, round trip). A 100 ps rise time limits the maximum coupled switchback length to about 100 mil.

In an outer layer microstrip configuration, the mutual capacitive coupling between adjacent traces is generally weaker than the mutually inductive coupling, driving the FEXT coefficient negative. However, forward crosstalk does not exist in the stripline configuration.

The fine balance between inductive and capacitive coupled cross-talk produces almost no observable forward crosstalk. Also, the peak amplitude of the crosstalk is considerably reduced. So, all other factors being equal, here is just another good reason why one should always route high-speed signals on the inner layers of a multilayer PCB. Stripline edge-coupled signals can also be placed closer to each other as compared to the microstrip equivalent,

which leaves more space for routing and is always welcomed.

When selecting a serpentine routing method, one should avoid long, coupled switchbacks as highlighted in violet in Figure 3. This was taken from a DDR4 reference design that I came across recently. Don't try this at home! The dark blue highlighted serpentine has an ideal configuration.

Figure 4 plots the comparison of a straight trace vs. a serpentine trace routed on the outer microstrip layer. Green is the driver; red is the straight (reference) trace; and blue is the serpentine trace with short, coupled segments. As can clearly be seen, the blue serpentine trace leads the red reference trace by 15 ps despite being the same length. As the trombone parallel sections increase, so does the velocity of the signal. The dip in the blue serpentine trace (around 5 ns) is the forward crosstalk which would not be present on an inner stripline layer.

If the switchback delay is much less than the signal rise time, the NEXT distortion blends into the overall shape of the rising edge. The NEXT distortion for short switchbacks doesn't impact the shape of the rising edge, but it advances the time of arrival. That is, short, coupled switchbacks produce smaller delays than the total trace length would indicate. Long, coupled switchbacks also distort the signals and are not recommended. The key is to route the clock and strobes first as straight as pos-

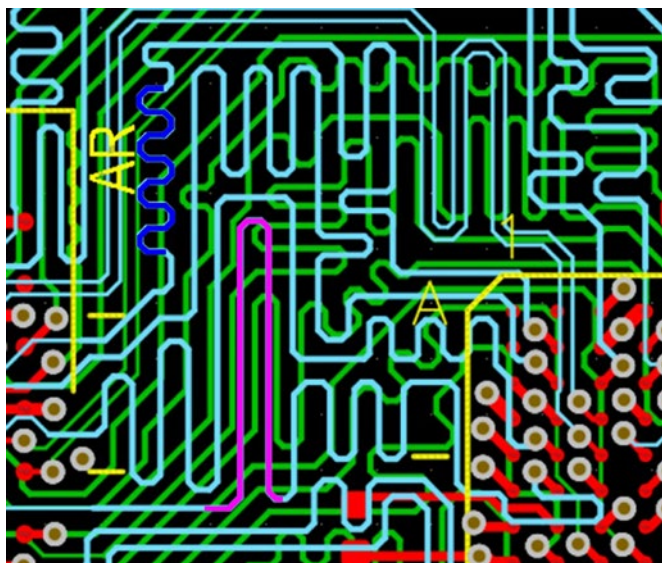


Figure 3: Another poor DDR4 reference design with long switchbacks.

sible, then tune the delay of each signal group to its reference clock. This will ensure that the signals have settled before the data is captured.

Key Points

- Designing a memory interface is all about timing closure
- The velocity of the signal is sped up by the serpentine. The EM wave passes the serpentine section faster than that of a straight trace of the same length
- This acceleration is caused by crosstalk coupling (NEXT and FEXT) between the parallel trace segments of the serpentine traces
- For long coupled lengths, signals may become distorted as they pass the serpentine section
- Signals pass relatively undistorted along short, coupled serpentine sections but distortion begins to occur when the parallel trombone length approaches one-third the signal wavelength
- When the round-trip delay of a heavily coupled switchback far exceeds one-third of the rise time, you get seriously distorted signals; when it's much less than one-third, you get advanced timing

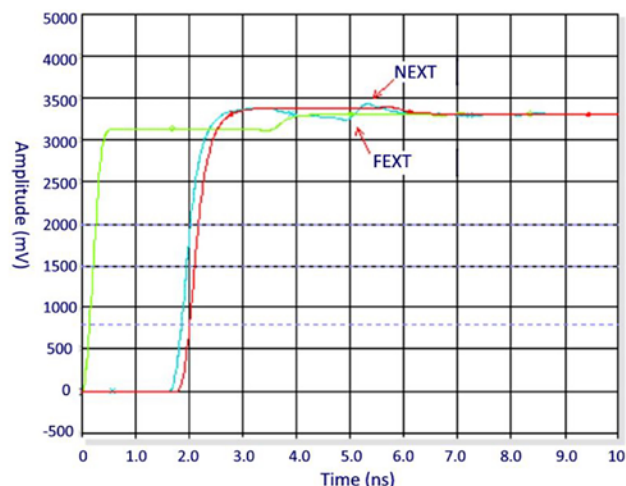


Figure 4: Serpentine trace vs. straight trace (microstrip).

- Forward crosstalk does not exist in the stripline configuration
- Stripline edge coupled signals can also be placed closer to each other compared to the microstrip equivalent
- The NEXT distortion for short switchbacks doesn't impact the shape of the rising edge, but it advances the time of arrival
- Long, coupled switchbacks also distort the signals and are not recommended

DESIGN007

Resources

1. "Beyond Design: Stackup Configurations to Mitigate Crosstalk," Barry Olney, *Design007 Magazine*, February 2021.

"Serpentine Delays," by Howard Johnson, *Signal Consulting Inc. Originally published in EDN Magazine, Feb. 5, 2001.*



Barry Olney is managing director of In-Circuit Design Pty Ltd (iCD), Australia, a PCB design service bureau that specializes in board-level simulation. The company developed the iCD Design Integrity

software incorporating the iCD Stackup, PDN, and CPW Planner. The software can be downloaded at www.icd.com.au. To read past columns or contact Olney, [click here](#).

ASK THE EXPERTS

YOUR BURNING QUESTIONS... ANSWERED

Q *What is the best method for tying vias to ground and power planes for bypass capacitors when using multiple caps of different values for one component power pin?*

Rick Hartley: When connecting bypass/decoupling capacitors to power/ground planes, each capacitor should be connected with its own set of vias. Sharing vias between caps should be avoided unless absolutely necessary (in cases where component density is extremely high). The best connection arrangement is the one with the lowest via pair inductance. Low via pair inductance is achieved when the power and the ground vias are very close together and very near the capacitor pins. Large vias that are far apart have higher inductance than small vias that are close together. Close spacing, not large size, is the key to low via inductance.

Using multiple values of bypass/decoupling capacitors is generally not a great idea. Doing so can, and often does, lead to problems with anti-resonant peaks in the power bus impedance, caused by the parallel resonance of the capacitance of one device relative to the inductance of another device of a different value. Multiple values can be implemented successfully, but will require extensive simulation with a high-end power bus simulator, to minimize anti-resonant peaks. High power bus impedance at any frequency can cause SI and EMI issues.

Heidi Barnes: First, capacitors should be selected for flat matched impedance and not just “factor

of 10” old-school leveraging. If a large cap is placed next to a small cap one can have a Pi resonator between the ESL of the large cap and the C of the small cap. Selecting the right ESR to achieve a target Z will help reduce the number of caps by maximizing the bandwidth of power delivery for each capacitor before the next capacitor takes over. Next, as your question asks, is the importance of ground and power vias to have low inductance to the component power pin. The best way to do this is to start with a thin dielectric between the power and ground planes to form a parallel plate with small loop inductance. Three mils is typically sufficient and can significantly reduce both the via loop inductance as well as the path inductance to the load. Finally, using capacitor footprint topology that gets the power and ground return vias as close as possible will help reduce the inductance further. To maximize the high frequency bandwidth, the smallest capacitor should be placed as close as possible to the power pin. In the case of BGA devices this is typically on the opposite side of the part directly across the power and ground vias to the BGA.

Eric Bogatin: Why use different value capacitors on the same power pin? As a general rule, never share power or return vias.

Cherie Litson: Put the smallest value cap closest to the power pin. Where the via goes depends upon the type of signal. RF signals prefer to have the via to the plane after the pin connects to the cap. Digital signals prefer to have the via go directly to the plane and then to the cap.

Lee Ritchey: Since virtually all capacitors are surface mount, there is no need for thermal ties between the power vias and the planes. All a designer needs to do is drill a hole in the appropriate plane and make a full contact with the plated through-hole.

Carl Schattke: The smallest inductive loop is going to be the least parasitic. In order for the energy to flow in a circuit, the inductive force must be overcome before the voltage will rise. Basically, the spin happens before the push. To reduce the time and energy this takes, we want to minimize the area that has to be energized. This area is going to be the 3D gap between the power trace and the return path. Closer is always better electrically, but if it's too close, though, you start to get into the manufacturing concerns of copper-to-copper plating limitations based on copper weights and etch processing, not to mention the hole-to-hole limitations of cracked laminate, and potential CAF problems. Ideally, we place the vias at the edge of the manufacturing tolerances that will not add cost or reduce reliability.

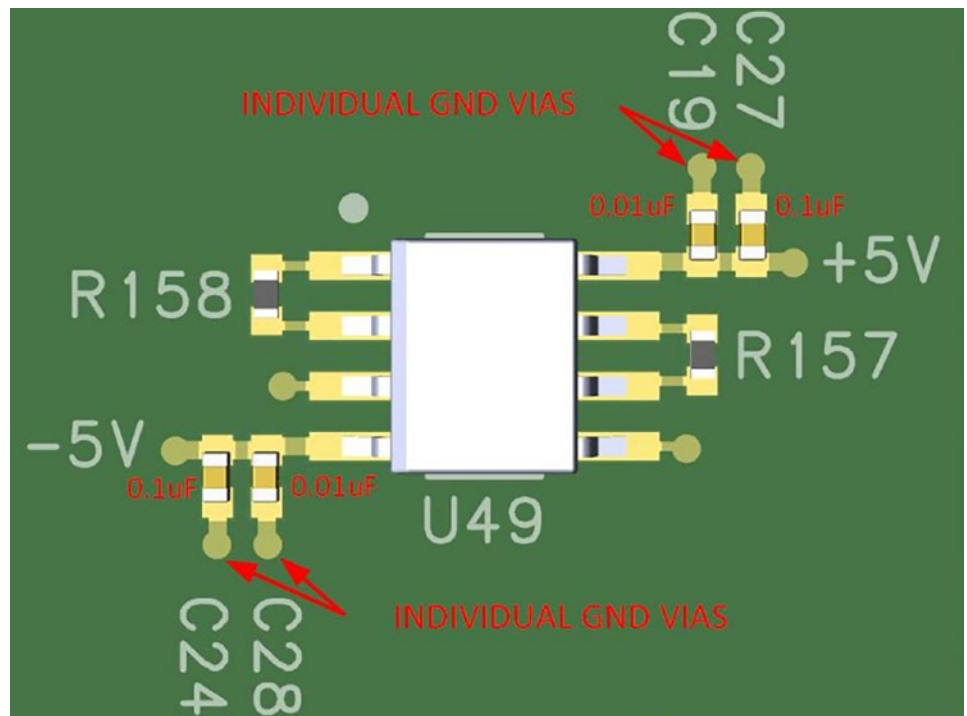


Figure 1: Typical placement of a decoupling capacitor near a device.
(Source: Chris Young)

Chris Young: In general, when dealing with multiple decoupling capacitors (0.1 mF, 0.01 mF, 0.001 mF values are common) per power pin, a typical approach is to put the smallest-value capacitor closest to the power pin and the larger value as close to the smaller capacitor as possible. This is usually done to shunt noise or interference signals away from the power input of a device. Power vias are usually placed at the largest value capacitor and a trace drawn directly (as possible) from the via to the power pin through the pads of the capacitors. Each decoupling capacitor should have an independent ground via placed as close as possible to the pads. Keep in mind that traces and space between components should be as small as possible as well while maintaining distances allow for manufacturing capabilities. See Figure 1 for a typical placement of decoupling capacitors near a device. **DESIGN007**

Some **Timely** Advice

Tim's Takeaways

by Tim Haag, FIRST PAGE SAGE

Have you ever heard of Bert Christman^[1-2]? Bert was a storyteller, a graphic illustrator to be exact, and in 1936 took over as the lead writer and illustrator of the nationally syndicated comic “Scorchy Smith.” As a daring pilot/adventurer, Scorchy’s stories fed into the public’s interest in aviation that had been fueled by the exploits of Charles Lindbergh and other notable pilots of the day. Under Bert’s guidance, Scorchy soared to new heights in aerial adventures, and the comic’s popularity continued to grow. But that wasn’t enough for Bert Christman, who wanted to find new ways to connect to his readership with realistic stories and illustrations of Scorchy’s flying adventures. To accomplish this, he took his research to the next level with flying lessons and quickly earned his wings.

In the late 1930s, Scorchy Smith’s adventures expanded beyond the domestic problems that readers were used to, and he faced new levels of international threats coinciding with real-world turbulences. At the same time, Bert decided to put his writing on hold as he joined the U.S. Navy as a dive-bomber pilot. Not only did he want to defend his country, but he wanted to broaden his understanding of aerial combat for future stories with the real-life experiences of being a combat pilot. Three years later he left the Navy to join the American Volunteer Group, otherwise known as the Flying Tigers, in the defense of China. It was here that Bert’s life literally blended with his writing as he lived the same type of adventures that he had penned for Scorchy Smith and other heroes in the comics. Bert commented on this dangerous life in a letter home,

“Things are getting hot here. Even Scorchy Smith would be satisfied.” Sadly, this is also how he met his fate when shortly thereafter he was shot down during a dogfight in January 1942.

I realize that this story doesn’t have anything to do with the printed circuit design and manufacturing industry, but it is a real professional motivator for me, so let me



3rd Squadron Hell's Angels, Flying Tigers, over China, photographed in 1942 by AVG pilot Robert T. Smith (Source: Flickr, The Commons)

We are dedicated
to excellence through

innovation, technology and most importantly, service.



COMMERCIAL • MILITARY • MEDICAL • BAREBOARD



U.S.CIRCUIT

www.uscircuit.com



tie it all together. In this edition of *Design007 Magazine*, we are exploring advice from the experts. As I was pondering some of the things that I've learned or wished that I had learned along the way, the story of Bert Christman came to mind. What I find so captivating about him is that he was willing to put everything on the line in order to improve his abilities in producing a better product. I am not at all suggesting volunteering for hazardous duty pay to improve our DDR routing skills, but his story does serve as an inspiration as to how committed a person can be in improving their skills and abilities. I just wish that I had learned this lesson a lot earlier in my career so I would have had more time to put it into practice. Maybe I might have learned a few other skills in my professional journey as well. In fact, if I had a time machine, here are a few other pieces of advice that as a designer I would have liked to have known earlier too.

I just wish that I had learned this lesson a lot earlier in my career so I would have had more time to put it into practice.

It's 'Circuit Board Design,' Not 'Place and Route Design'

When I was first starting out in PCB layout, I was fascinated by placing parts and routing nets on the screen as any new designer is. Soon I came to understand many of the nuances of PCB layout, and how the different spacing rules applied in the art of place and route. But I also made a lot of mistakes that could have been avoided if I had only grasped the concept that I was actually designing a

"circuit board" as opposed to simply performing place and route. I remember not understanding—and therefore ignoring—some of the early signal integrity problems that were being discussed around me. These included topics such as broadside coupling and signal return paths, all of which were important to what I was doing. Thankfully, my work was monitored by senior designers, and I don't believe that I caused any serious damage. But if I could, I would sit the younger version of me down and explain that layout is an all-encompassing process that begins with the schematic, and not when you start throwing parts on the screen.

Board Integrity Starts With Library Integrity

Like a lot of designers eager to get rolling on their first layouts, I was happy to grab any old part out of a library and throw it into the design. What I didn't fully realize at the time, though, was how essential a properly built PCB footprint is to the manufacturing process, and that real-life components would eventually have to be soldered to these land patterns. I shudder to think of how many times I may have swapped an anode for a cathode in some of my first layouts. Thankfully, this lack of attention to detail didn't seem to result in too many actual problems, although I do remember having to go back and make a few corrections here and there. But again, with the Wayback Machine dialed into my earliest days of PCB layout, I would explain to my younger self the entire concept of circuit board manufacturing from start to finish. I would then explain how manufacturing relates to design and the importance of choosing the right footprints for the parts in the BOM. I would make sure my younger self understood the critical importance of dimensionally accurate footprints and land patterns, emphasizing that even one incorrect pad size could put the whole design in jeopardy of failing.

PCB Design is Not Just About Me

I'm really embarrassed to even admit to this one, but sadly it's true. As a junior designer, I tended to look upon the input of other members of the design team as being more of an annoyance than the collaborative partnership it should have been. Now, it is true that the working relationships between team members need to be managed appropriately, and not everyone can get everything in a design to be the way they want it to be, but I was completely out of line in my earlier days when it came to design change requests. "Move this part closer to that one? No way, that will mess up my beautiful routing." "What do you mean that will cause a manufacturing problem? Who cares? The design looks great." Once again, if we could just run the DeLorean up to 88 mph, I would give my younger self a good slap and convince him that the sun doesn't rise and set with how the design looks. The priority is that the circuit board must actually work and be manufacturable above all. Great Scott!

The priority is that the circuit board must actually work and be manufacturable above all. Great Scott!

Getting Your Ducks in a Row Avoids Getting All 'Fowled' Up

It is amazing when I think back to how many times I rushed through important parts of a project, just so I could get into the "fun stuff." Here's one example that caused me grief on more than one occasion as a junior layout designer: "Oh I don't need to worry about all of these separate design rules, let's just set the default and start hooking up traces." Are you

wincing right now just as much as I am with that memory? If I could just convince the Guardian of Forever to send me back to the first occurrence of haste in my career, hopefully, I could change the course of history. I would tell myself that taking an extra 30 minutes up front is much preferable than the hours of re-work that I was headed for later.

Relax, Have Fun, and Don't Take Yourself So Seriously

And lastly, I would borrow the phone booth from Bill and Ted long enough to advise my younger self to lighten up just a little. When I first started laying out boards, I could be kind of intense, which didn't always work out very well. I would get down on myself for mistakes that I had made, while at the same time trying to manipulate people into doing what I thought was best. There is so much joy to be found in what we do and who we work with that such a high level of intensity just isn't necessary. Plus, it isn't very healthy either. Please don't misunderstand me. We still need to be diligent in our work, and always aim to improve ourselves just as Bert Christman did. But we don't need to kill ourselves and those around us doing it.

And speaking of killing ourselves while working, my wife is in the other room watching basketball all by herself, and I think that I am going to join her for a while. Keep on designing everyone, and I'll see you next time. **DESIGN007**

References

1. Life Imitates Art, *Colorado State University Magazine*.
2. Cartoonist Bert Christmas Did Not Survive War, *The Daily Cartoonist*, May 31, 2021.



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

ASK THE EXPERTS

YOUR BURNING QUESTIONS... ANSWERED

Q *Do you have any ideas on interpreting IPC-6012 Class 3 annular ring standards, especially for DFM? It seems very loosey-goosey, re: hole placement, etc.*

Cherie Litson: There is no “loosey-goosey” hole placement for Class 3 annular rings. There must be 1-mil minimum annular ring around the hole after drilling. No breakout is allowed. Holes get drilled before the etching of the copper. Thus, there is no “centering the drill” on a hole. That’s a typical misconception from anyone who thinks it’s a mechanical issue. It’s not. It’s about having enough annular ring to account for the fabrication tolerances. If you’re not specifying them in your notes, then the “standard” tolerance is taken from the drawing format title block and that will usually be ± 5 mils. Most fabricators can hold a tolerance of ± 3 mils on material movement during fabrication.

Heidi Barnes: I am not directly familiar with this standard, but from a high level there is always a challenge of over-specifying the engineering details at the expense of over-constraining the

fabrication process. Building PCBs does take a lot of coordination between the design engineer and the fabricator to understand what tolerances need to be specified without escalating fabrication costs. It is exciting to see how far PCB fabrication has advanced with the push to higher densities, higher power, and faster speeds. However, I believe that there is still many an engineer out there who feels like it is magic that a PCB works. Statistically it works out, but if one were to add up all the worst-case fabrication tolerances, it would not.

Carl Schattke: Class 2 allows some breakout; Class 3 does not. Obviously, there is drill drift and registration tolerances that come into play. Work with your manufacturer to see what pad ring sizes will be needed for your application. They will give you that based on their process capability. **DESIGN007**



SIGRITY
AURORA

ANALYSIS

Resolve PI and SI issues at any stage of your design with Sigrity Aurora. Real-time, overlaid simulation results allow you to quickly analyze, identify issues, and verify fixes from within a shared PCB design and analysis platform.

Experience true, electrically-aware design from start to finish with best-in-class Sigrity and Allegro engines.

See How >>>

ASK THE EXPERTS

YOUR BURNING QUESTIONS... ANSWERED

Q *Is it best practice to flood with copper (tying all areas to GND as needed) all unused portions of all layers?*

Rick Hartley: Copper pours can be a double-edged sword. From a manufacturing point of view, copper pours in unused areas will help balance the construction of the PC board, helping to prevent bow and twist and ensuring even copper plating on the outer layer features (traces, pads, etc.). If the only concern is that of even plating on the outer surfaces, thief copper (in the form of copper dots) can be used instead of copper pours. That said, copper dots do not help with bow and twist caused by uneven copper on inner layers. Talk to your fabricator, who will explain why these statements are generally true and how to best handle your particular designs.

Copper pours can have great value to the circuit, but only when implemented correctly. Randomly placing ground pours on all signal layers generally offers no value to the performance of the circuit, nor do these pours help control EMI. Worse, randomly placed copper pours can cause problems if the segments are not properly attached to a plane with adequate vias to keep copper elements from resonating. Resonating copper elements can cause coupling of energy into other features on the PC board.

One technique that can be very helpful is to alternate layers of power and ground in the board stackup. For example, in a six-layer board, with a ground plane on layer 2 and a

power plane on layer 5, placing power pours on layers 1 and 3 and placing ground pours on layers 4 and 6 will create a PC board of the following arrangement:

- L1: Signal/Power
- L2: Ground Plane
- L3: Signal/Power
- L4: Ground/Signal
- L5: Power Plane
- L6: Ground/Signal

The above stackup, due to the added power pours on L1 and L3 and ground pours of L4 and L6, will improve power delivery to ICs on the board. This can and usually will improve both signal integrity and EMI. The EMI improvement is *not* caused by the shielding effect of the copper pours, but rather by the improved power delivery due to lower power bus impedance at high frequencies (the harmonics of the square waves).

To avoid unwanted resonances with copper pours, tie all power pours to the power plane with vias placed every one-tenth wavelength of the max harmonic frequency of the fastest transmission lines in the circuit. Also tie all ground pours to the ground plane every one-tenth wavelength as well. Frequency in GHz=0.5/signal rise time.

This six-layer stack-up is simply one example. This technique can be used on all PC boards: four-layer boards, or 8, 10, 12, 20, or 40 layers.

Chris Young: There is no one answer that fits all situations when it comes to flooding routing layers with copper (usually GND). There are a lot of factors that should be considered, and I suggest taking a needs-based approach. Here are some reasons that I have flooded a layer or a region with copper:

- To increase the heat capacity of a PCB to improve its heat-sinking capability
- To provide copper balance in a PCB where thieving is not allowed
- To reduce the difficulty for a PCB fabricator to meet multiple controlled impedance requirements within a single PCB
- To strengthen a PCB to make it more resilient to mechanical stress
- To shield a signal from unwanted noise or other aggressor signals

Heidi Barnes: Flooding all unused areas with copper is kind of a funny way to put it, since the copper is already there, and it comes for free. CAD tools start with a blank PCB and put copper down. A real PCB starts with copper and then etches away what is not needed. The RF/microwave world has been doing solid fill with vias for a long time to provide shielding and very low impedance ground. One can even add edge plating to take the shielding and low impedance a step further. Maximizing the copper ground fill can also add mechanical strength. However, one does need to watch the stackup symmetry to avoid warping, and etch uniformity often leads to cross-hatched ground fill instead of solid fill.

Carl Schattke: This will vary from project to project and intended application. Generally speaking, more ground is better. More con-

nection points from layer to layer is better. Floating copper will have some resonance frequency to it. You will not know what that is as you design your board. Smaller pieces of copper will resonate at a higher frequency than larger pieces due to the propagation times that bounce at the edges. So generally, we try to tie any larger sections down to other layers with a via and not have long meandering sections just creating mini antennas all over the board.

The decision to use ground flooding or not should be made on a case-by-case basis, based on signal frequency and noise margins associated with that. In my experience it usually does not hurt if done properly. Be mindful of the increased risk of shorts in the manufacturing process if the polygon to other copper value is set too low. Just because a manufacturer can do a minimal gap does not mean that minimal gap will give you the best yield. Always be designing for the best costs and yields.

Eric Bogatin: It is never a good practice to flood copper on unused regions. Use a dot pattern to get more uniform copper plating and to balance the copper, but do not flood copper. It provides no value and will often create more problems than it could ever, in the best case, potentially solve.

Lee Ritchey: There is no good reason to flood unused space in signal layers with ground. It just makes the Gerber files much larger, and it might actually hurt signal integrity if done incorrectly.

Cherie Litson: No, not always. If you only have a few traces on a layer, then flooding that layer with copper and tying it to GND or PWR will balance the fabrication process and decrease your bow and twist. If I have an RF board, I have to be careful where I'm creating capacitance between the layers or where I need to have *no* copper as it will interfere with Bluetooth signals. **DESIGN007**

The Board Thickness Challenge

Connect the Dots

by Matt Stevenson, SUNSTONE CIRCUITS

Size constraints, functional requirements, and environmental factors can make selecting PCB thickness difficult. Here we will examine best practices for choosing board thickness that results in quality, highly functional PCBs.

It's Not Always as Easy as 1.57 mm

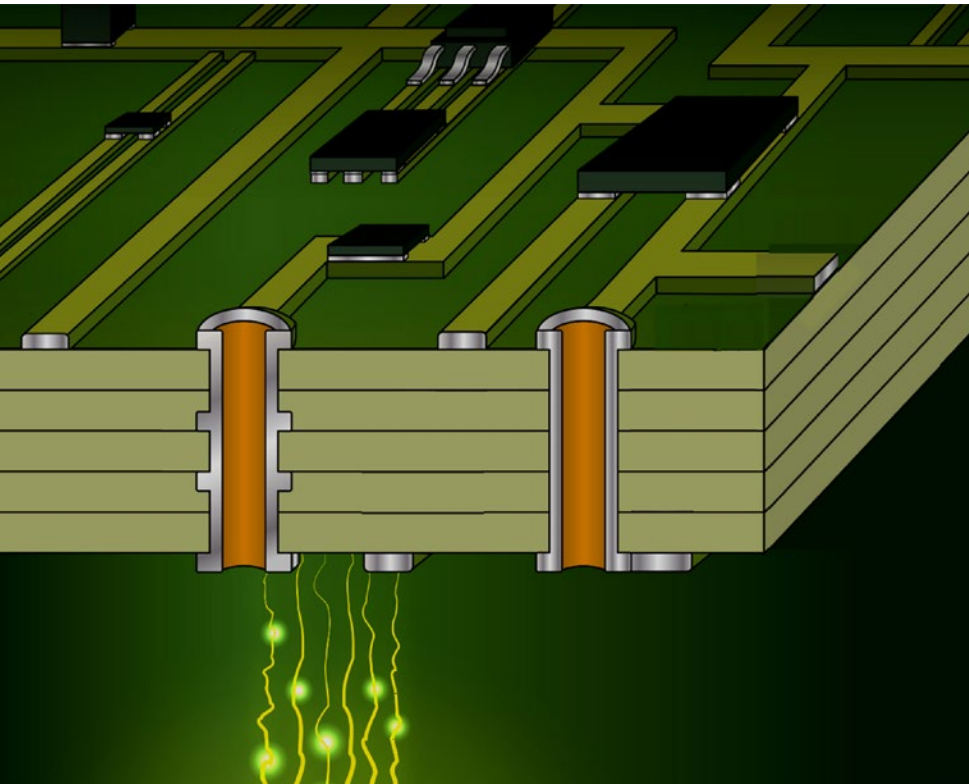
Industry standard or your design tool's default board thickness are great places to start, but designers increasingly must consider using a custom board thickness. There are several factors to consider as you determine if a commonly used thickness will work. How thick does the board need to be to work?

How thin must it be to fit into the device? What are its function and operating environment?

Thinner boards offer more flexibility and are preferable if keeping the device weight lower is a priority. Thicker is better if durability is an issue, because thick PCBs are less brittle and won't break as easily. They also weigh more, run hotter, and will not work for many applications or assembly types. To determine thickness, you must consider how functional requirements will impact the number of layers you'll need, as well as what signal and via types are needed.

The board's operating environment is an equally important consideration. Is your board going into a wearable or handheld device that operates in a regulated environment or is it part of a Mars rover?

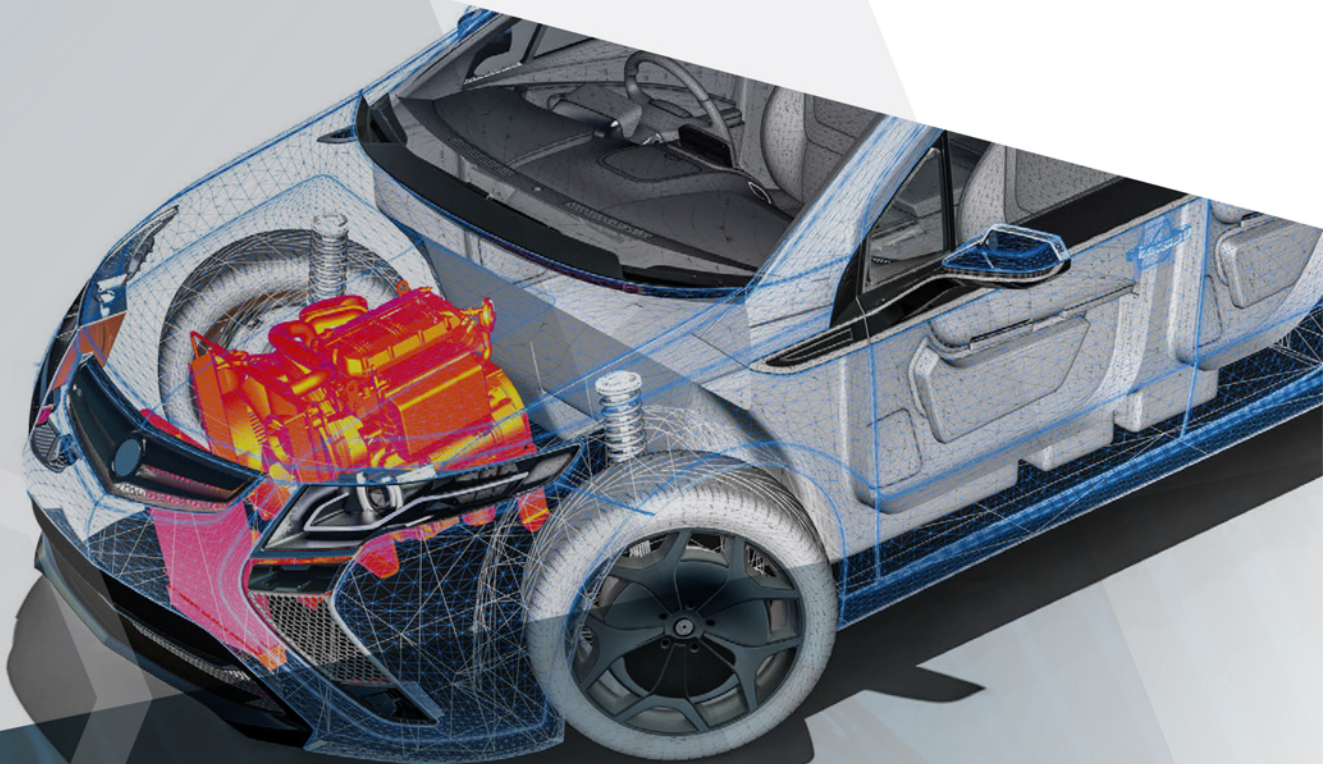
When your manufacturer receives a design, they can evaluate it before production to ensure its manufacturability and functionality, but very seldom are manufacturers made aware of how the board will eventually be used. It's unlikely they'll be able to judge whether a board, as designed, is too thick or too thin for the device it goes into or the environment in which it operates.





ventec
INTERNATIONAL GROUP
騰輝電子

autolam: Base-Material Solutions for Automotive Electronics



autolam 

Automotive electronics technologies are evolving at an increasing rate. Paying attention to the properties of materials at the substrate level is the first step towards achieving the most stringent performance targets of today's automotive manufacturers. autolam offers the solutions demanded by the diverse and unique requirements of automotive applications today and in the future.



ventecclaminates.com

If, after evaluating these factors, you're still not sure if a standard or common thickness will work for your design, here are some guidelines to help you choose what will work best for your board.

So, You Think You Need a Custom Thickness?

If you have read any of our other *Design007 Magazine* articles, you're probably aware that we consider PCB design as much an art form as a science. As devices demand more and more from boards, PCB design becomes less straightforward. So, if determining thickness is proving difficult during the design phase, we recommend also considering these relevant factors.

First, can your PCB be manufactured as designed? Your manufacturer will have production tolerances. If your board is too thick or too thin, you'll find yourself back at the drawing board.

**If your board is too thick
or too thin, you'll find
yourself back at the
drawing board.**

PCB design is a game of millimeters. Thinner PCBs take up less space, so if you are building a device that will fit in your hand, go as thin as necessary to accommodate the space. And once inside the assembly, will it be adequately protected to function properly?

Protection from the operating environment is largely about extreme cold or heat. Heat that can threaten functionality doesn't always come from operating in an environment like a car engine or industrial grade oven. Components themselves can generate enough heat to cause an issue, so be sure to determine if you need to prioritize thermal transfer or

insulation as you contemplate board thickness.

PCB weight can be an issue more often than you might think. Is your board part of a cell-phone, Bluetooth headset, or tablet device? Thinner boards are lighter and can help give your device that trim figure we all appreciate.

Pay careful attention to your connections. PCB edge connectors or PCB trace-based connectors, for example, require a board with thickness matched to fit in the mating portion of the connector. Some through-hole parts have pin lengths that limit board thinness.

Keep your ear to the ground for frequency concerns. Layer-to-layer thickness will affect signal integrity, crosstalk, impedance calculation, and signal loss. For high frequency signals, impedance control is a factor in determining the right thickness of your board.

There are trade-offs with board flexibility. Will your product be subjected to excessive shock and vibration? Thin boards that flex can cause broken connections or components, so be sure it's properly supported in your assembly.

Decision Time: Thick or Thin?

Though the list of design considerations can theoretically go on forever, drawing conclusions about these factors will help with peace of mind about PCB thickness during the transition from design to prototyping. Following are our general recommendations about when to prioritize thicker or thinner boards.

Thicker Boards

- Your board needs to be durable or isn't well protected in the assembly—or both
- Because of its size, the PCB needs to be stiff
- Your components require a certain amount of thickness
- You have a high layer count, more than six

Thinner Boards

- Your board is going in a tight space
- PCB weight is an issue
- If heat is a concern, thinner boards are

better at dissipating heat. That's because vias are shorter and pass through a smaller amount of insulation

- If your vias are very small and the board is too thick, drill bits will break before they make it through the board, making manufacture slower and more costly

Since cost is always a factor in our industry, we should note that standard thicknesses are less expensive. But if a standard thickness is not right for your design, we highly recommend customizing. Initial cost savings can go up in smoke if the boards aren't durable enough because they're too thin or prone to overheating because they are too thick.

Remember What You Learn About Board Thickness

And write it down. As you consider thickness in your design more often, we encourage crafting an easily replicated checklist approach to make this challenge easier in the future.

Group what you learn into categories for easy reference. We recommend:

- Manufacturer tolerances
- Operating environment
- Thermal issues
- Size and weight constraints
- Frequency considerations
- The relationship between connections and board thickness

Your design can be spot on in every respect, but if it's the wrong thickness it will likely fail and create costly rework. Before you send your design to your manufacturer, be sure your board is the right thickness for your needs. **DESIGN007**



Matt Stevenson is the VP of sales and marketing at Sunstone Circuits. To read past columns or contact Stevenson, [click here](#).

Electronic Paper on an Industrial Scale

Researchers at Linköping University, together with colleagues from Rise and the Royal Institute of Technology, have developed an electronic paper that can be manufactured on an industrial scale. In the pilot project, rolls of length 10 metres and width 20 centimetres have been successfully manufactured, which means that full-scale production is fully possible. The paper has high conductivity, and long-term uses include energy storage and the construction of electronic components.

"We have been trying to take printed electronics, in which paper is used to support electronic components, to the next development stage, which is to incorporate the components into the paper. This pilot project shows that it is possible, on a scale that transcends the slow and small-scale experiments in the research lab," says Isak Engquist, associate professor and head of research in the Laboratory of Organic Electronics (LOE) at Linköping University.



The paper is made from a mixture of cellulose, conducting polymers and charcoal from coconut, which makes the material flexible, durable and electrically conductive. In addition, the production is cheap and environmentally sustainable, which is a great advantage over traditional electronic components. These often contain scarce metals and toxic substances.

Two projects are behind the discovery: OD+1D+2D=3D, led by Linköping University, and the Digital Cellulose Center, DCC, led by Rise.

"It will be interesting to see the new applications that our creative researchers can come up with when they get their hands on this material. The paper is functional through its complete

thickness, which means that it can conduct much more current and store much more energy than thin layers printed onto conventional paper," says Engquist.

(Source: Linköping University)

ASK THE EXPERTS

YOUR BURNING QUESTIONS... ANSWERED

Q *Crosstalk is getting worse, after we had it under control. It's not just transmission lines anymore. It's almost laughable, but it's costing us money. What can be done?*

Rick Hartley: Crosstalk is getting worse, due to the many ICs of today with very fast rise and fall time outputs. Crosstalk, like many problems, is not related to clock frequency, rather to the rate of change of energy in transmission lines, due to most of today's ICs having rise times faster than 1 nsec, with some even faster than 500 psec. It only takes an inch of parallelism between two transmission lines to cause maximum crosstalk coupling between the lines when signals have 500 psec edges.

There are several techniques that can vastly reduce crosstalk:

1. Always route signal lines directly above a plane, preferably a ground plane.
2. Try to space critical lines (ones with really fast edges) at least 1X the distance to the nearest plane for inner layer routes, and 2X distance to the plane on outer layers.
3. Whenever possible, series terminate transmission lines that are fast enough to be an aggressor. Series termination vastly improves crosstalk, as well as ground bounce, EMI, power bus switching noise and ringing (vswr/return loss).
4. Only use "guard traces" when they can be attached to a ground plane with vias spaced every one-tenth wavelength of the highest frequency ($0.5/\text{rise time}$). Do not

overdo this. Vias closer than one-tenth wavelength simply take up space and make routing on other layers harder. When signal traces can be spaced far enough apart to allow for a guard trace that is properly attached to a ground plane, you can likely eliminate the guard trace, since the aggressor and victim lines will be very far apart, typically six to eight times the distance to the nearest plane.

To ensure that the above concepts have achieved the intended goals, use a 3D field solver or other high-end simulator to estimate crosstalk prior to building the board. The techniques mentioned here will not eliminate crosstalk, rather will reduce it to more acceptable levels.

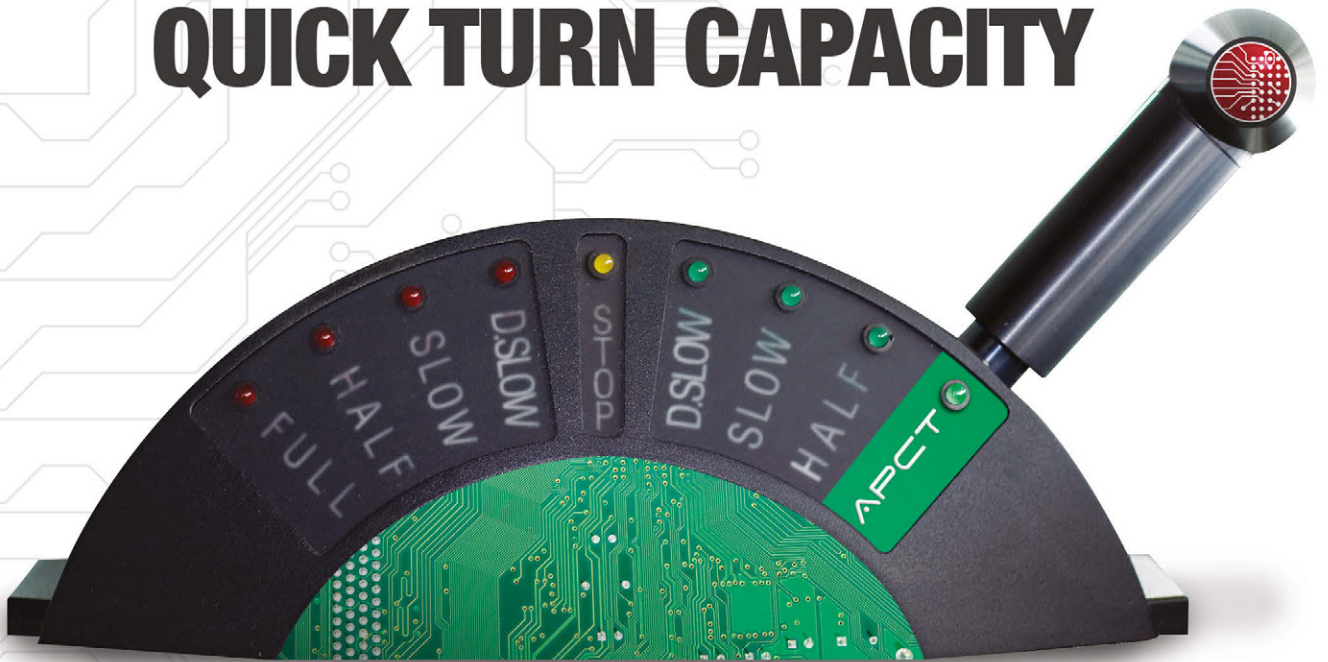
Carl Schattke: It's hard to know exactly what this question is asking and commenting on, but in general, smaller geometries in silicon are shrinking spaces and isolating circuits at the silicon level. This can reduce the noise margin at the PCB level and create problems as silicon processing can make a previously working PC board nonfunctional as the silicon shrinks and takes more of that overall margin away. Re-designs will cost money, but sometimes that's your only option. Hopefully, the re-design can save money in other ways.



APCT

Passion | Commitment | Trust

QUICK TURN CAPABILITY QUICK TURN CAPACITY



**A Broad Range of Technologies Including Rigid Flex...
Manufactured In Days Not Weeks!**

Technology Expertise | Detailed Engineering Support | Empowered People

APCT.com | 512.548.7435

Printed Circuit Board Solutions

APCT.com

APCT Santa Clara HQ

408.727.6442

APCT Anaheim

714.921.0860

APCT Orange County

714.993.0270

APCT Wallingford

203.269.3311

APCT Global

203.284.1215

Lee Ritchey: It is not at all difficult to control crosstalk. Methods for doing this have been in place for four decades, at least.

Eric Bogatin: Yes, crosstalk is a major problem in many designs. It is manageable when there is a continuous return plane adjacent to signals. It is hardest to control in the via field of a BGA breakout field. Using a 3D full wave solver is essential in evaluating different design options. The general guideline is using return vias when possible, and back-drilling via stubs.

Heidi Barnes: Not a lot of information in your question, but yes, crosstalk only gets worse as

edge speeds go up and power delivery voltage margins decrease with low voltage power delivery. EM simulation is often the best way to see what is going on with the PCB parasitics. One possible solution would be to reduce the dielectric thickness between the power and ground layers and check the impedance of the power rails.

Cherie Litson: Crosstalk can happen for many reasons. It's difficult to say what exactly is creating the issue without looking at your layout. Hire any one of us and we can help pinpoint it with you. **DESIGN007**

ASK THE EXPERTS

YOUR BURNING QUESTIONS... ANSWERED

Q *With a 1+N+N+1 stackup, how should we connect the two stacks? We're having an office disagreement. Is ALIVH our only option? Are there any low-cost fab processes that will work here?*

Carl Schattke: As you start to use less conventional technology, costs tend to go up. That's why it's less conventional. After all, if it was easy everyone would be doing it.

Lee Ritchey: To connect the two stacks in a build-up design such as that described in this question, it is necessary to laminate them together first. Electrical connections will be made between them using the conventional through-hole drilled and plated holes. **DESIGN007**

ASK THE EXPERTS

YOUR BURNING QUESTIONS... ANSWERED

Q *Do any colleges offer classes on PCB manufacturing and chemical processes as part of their EE programs?*

Eric Bogatin: Yes, the University of Colorado-Boulder does. I teach a class there called “Practical PCB Design and Manufacture.”

Cherie Litson: These are few and far between. There is no official curriculum in any EE program that addresses these issues. Most colleges are struggling with being able to fit in all the information that is required by most EEs in our industry now. Plus, they’re being pushed to “teach them faster” from all sides. Thus, the education of PCB manufacturing has gone to the professional industry. PCEA has a course for EEs that will fill this gap. IPC has the CID and CID+ courses that mostly target those who have already done some designs and have found out that they need to know more. Both courses cover similar subjects and taking both is sometimes a good idea.

Carl Schattke: Not any that I’m aware of. Many programs will have cool projects to work on that require some PCB design and the new EEs will get a chance to do some of this kind

of design. It’s rare to have teaching staff knowledgeable in this area, and most grads I interview seem lacking in the basic skills and lack any advanced skills in this area as well. There are exceptions, where a project needs boards, and the students learn what not to do as they have to deal with the problems they make for themselves in the design process. That can be a very rewarding experience.

Happy Holden: For the last two years, Michigan Technology University has taught a course on PCB fabrication that includes a lab where the students fabricate a 4-layer multilayer board as an elective in their EE program. This next year, they will also offer a PCB design course with a lab. This follows other universities like Harvey Mudd College in California that offered a PCB design course with labs to test the boards they designed for signal integrity. An ad-hoc group has been formed by Marc Carter to assist other universities in starting PCB programs as part of their EE programs. **DESIGN007**



Whizz Systems: The Silicon Valley CM

Interview by the I-Connect007 Editorial Team

The I-Connect007 Editorial Team spoke with Whizz Systems in the months prior to COVID. As we come out of lockdown, we decided to check in with Muhammad Irfan and Dan Williams of Whizz Systems.

In this part of the interview, they discuss Whizz Systems' unique methods for navigating the many handoffs involved in the design and manufacturing process, as well as their drive to educate their customers about this process. They also share their thoughts on the trends they've seen as a CM working out of Silicon Valley.

Barry Matties: Let's start with a quick update on where you are now and how things are going.

Muhammad Irfan: Last year, because of COVID, our Q2 had some impact, but right after that it's been recovering very steadily and growing. As far as general business, we're very blessed that we're doing very well. In terms of our plans and updates, we have replaced some of the equipment. Generally, we try to keep up

with the new capability or equipment when we see something out there that we should have. From a setup point of view, we're in good shape. We've added some new equipment, but generally, we are staffed and equipped to deal with any new projects that come in. I'll repaint the oral context of who we're trying to sell and why we are positioned the way we are positioned and how it helps us solve customers' problems. It's not something we have to sell, but customers have problems, and we are tailoring our setup to solve their challenges.

Generally, in our business, in product development, you have some level of a spec that customers start with, and they have a product architecture, some level of architecture in mind. From then onward, they're either using internal resources to do some level of design, some parts of the design, and then they go out to companies to help with various aspects during the design, including the hardware design itself, the FPGA design, the mechanical design, running thermal simulations, electrical simulations, signal integrity, and power integrity simulations. And possibly, if there is an FPGA on the board, they might engage an FPGA design



Making DFM Accessible For Any Business Size

**From home garage to enterprise -
PCBflow makes it a breeze to certify designs
for manufacturing**

Siemens solution enable companies to optimize the entire Electronics Manufacturing process in a smart and efficient way. Our lean manufacturing experts will walk you through the design-to-manufacturing flow and show you how to leverage the digital twin to improve your processes.

www.pcbflow.com

Learn
More





Muhammad Irfan

company to develop RTL code to write the program for the FPGAs. That's all included within the design phase itself.

Traditionally, in this model in the market, customers approach you with some in-house resources, and they go to multiple companies to help them with different aspects of this. Very few companies have all this insight internally. Tier ones, yes, but they are modeling around 70% workload to be handled internally, with 30% peaks and valleys through outsourcing. Mid-sized and onward companies would go out to somebody for some aspects of this. And then comes the part where they are handing off to a board layout shop, and that board layout shop might be a manufacturing company that has board layout capability internally, or they may work with somebody outside like a manufacturing house to build their prototypes.

Then once the prototypes are done, the customer's engineers bring the board to the lab or the system and they start debugging, import their software on it, and start bringing it up. When they have issues with the board, they go back to the manufacturing house and say, "Can you rework this part or that part? We need to replace these values of resistance and caps. The disconnecter is screwed up, we need to reverse the planarity on these or pin up on this."

This is normal when they're doing the debug. Also during the debug is a combination of hardware, firmware, and software resources together during the bring-up of the board, when the customer launches their software on it. It becomes a collaborative effort. We saw all that, and we see multiple handoffs. Number one, the one hand does not necessarily talk to the other and the customer becomes a translator between these. Number two, when there is a hiccup in one area, it's a linear hit on the schedule because the customer is in the mid-

dle. The customer feels a schedule delay. The fab shop is going to say, "You gave me the Gerber on this day, and it's a three-day turn. My day one starts today, not two days ago when you're supposed to give me the Gerber." Or, "By the way, my line is busy now, I'm on to something else." Things like that.

Also, there's the material coordination. Between design house, the mechanical design house, and the prototype manufacturer, and the customer in the middle of it, the BOM is constantly evolving. There are changes in the BOM. It becomes the customers' responsibility, or if it's left to the design hub to coordinate directly with the manufacturing shop; there are a lot of things that fall to them there. Especially with the given market, the horrendously long lead time on components, and in any market that we have built for, there are always parts that have long lead times. Right now, it's just a very crazy time, because everything becomes a lead time.

Knowing all of this, we said, "What are we going to be for the customer?" If we're just another manufacturing shop, yes, there's plenty of them around. If you're another design group, yes, there are a few good ones around. If you're a good layout shop, yes, there are a few good ones around or some manufacturing shops that have good layout capability. What's our value to the customer? So that is how we designed and set up from day one, and we've been growing around that mindset. Our DNA is built around that we are the customer's extension of their design team, their ops team, and their manufacturing shop. We allow integration of all of those under one roof, and we truly are our customers' extension.

They pick and choose what level of services, so our customers say, "You claim you do a lot of things. What is really your value-add?" Our answer is that our true value-add is the true integration of all of these, in-house, with our own internal resources. We give you the capability to engage us very early on, spec-level definition, to helping you launch a production-



worthy product, with everything in between, including certifications. There are some things we don't do internally like certifications or board fabrication, but we are well-established with several suppliers with good processes internally, good documentation, and know how to work with those and get that done.

Matties: How important is that to your customers? Because we still hear that communication is lacking throughout the supply chain.

Irfan: Unfortunately, yes, most customers still underestimate these handoff issues and the miscommunication. Part of our job is to actually educate and walk the customer through what happens if this is not done properly, and once customers understand, they become very loyal, long-term customers. So that's why we have very long-term retention of employees and customers, and by the way, we have the best retention of employees in our industry. Our design team, our manufacturing staff, our office staff, our program management, they're 95+% consistent for the last 15 years. We have added more but have not lost some.

So, we position to be that offering of a complete solution to customers, but not hounding them into giving us a complete outsource model. We give them the flexibility to pick and choose and engage us as they see fit for

every project, so project-to-project engagement could be different. But to the customer, they think, "There's a very viable partner that's experienced, has the design processes, has the know-how and a 20-year track record. They can take it from here to here, but I can choose. I may not need them for all the projects in the full capacity." So that's really our value, the flexibility to engage at any level. We have so many products that were successfully launched in our 20 years of experience that we're in pretty good shape, we have a lot to offer, and we're very proud of what we have and very confident of what we have.

The other thing is the mindset. We are constantly on the lookout. We have a substantial amount of money in our budget every year that we spend on non-customer projects that's just R&D. When we look at new upcoming trends, most shops can't afford it; they are waiting for a customer P.O. and they jump into that. We actually go out ahead of time and we may build a board on our own, put several technologies on a board, run some integration, and run some simulations so we have some collateral to actually show the customer. It gives customers the confidence that these guys have an R&D mindset and can do what most shops cannot.

Matties: At your facility a couple of years ago, I was impressed with some of the projects that



you were funding. I think at the time you had a lighting project of some sort going on, and we don't hear much about assemblers and EMS companies putting money into R&D.

Irfan: Thank you. That ties into how we run the operation. We don't sell ourselves as a cheap shop. We are not selling on price; we're not competing with the small assembly jobs on a price difference of a few hundred dollars. Once the customers understand, then our job is a lot easier to sell to those customers. In some areas, our rates are better than the given local market because of our offshore setup that's integrated into our local presence. But overall, we are a profitable business, so we could stay in business and stay ahead of the curve for our customers.

We run a viable, healthy, long-term profitable business that can sustain itself through ups and downs. We didn't have layoffs in 2001-2003, or in the 2008 downturn. We didn't lay off any manufacturing workers or any office workers.

Matties: Moving to what the customers are bringing to you, what sort of trends are you seeing in the actual products that they're asking you to build? Anything significant in the last couple of years?

Irfan: We've had a major program with one of the big chip makers creating a 5G program for O-RAN, the Open Radio Access Network. Traditionally, the big guys were the suppliers of these radio boards that customers use to access the cellular network. There is a trend for white-label boxes. Intel partnered with us to do a reference design platform, to white label an O-RAN box. This box is to enable a lot of 5G radio access network customers, radio providers, who would sublicense the design or buy boards

from us. We shied away from labeling it as our product because we're a service company, but we put the reference design together for our partners. This reinforces that recognition with the tier ones. When they're pitching your name to a lot of customers, instantly there's credibility.

The challenge comes when you're individually taking a lot of designs and putting them together as a system, but it's a very compact, smaller form factor. More complex than the logic of that board is how to physically make it work, and we've become really good in that space because of our in-house signal integrity, power integrity, thermal simulation, and mechanical design—that has become a very crucial part of electronics design. There are a lot of electromechanical design challenges with different products because of the form factor, cost reduction requirements, and very high speed that causes thermal issues and other noise.

Matties: Dan, you're in the marketing area. What trends are you seeing in the marketing space?

Dan Williams: I think clients are getting very choosy and it's kind of two worlds. It's a very closed ecosystem, and in some cases, it's tough

BUILD A BETTER WORKFORCE

- Electronics Assembly for Operators
 - IPC-A-610 for Operators
 - IPC-J-STD-001 for Operators
- Wire Harness Assembly for Operators



2/3 of electronic industry companies have difficulty finding production workers.¹

vs



Upskill Your Employees with IPC Electronics Workforce Training

IPC Electronics Workforce Training helps you bridge the skills gap. If you need to train assembly operators or wire harness assembly operators, these courses deliver consistent training and are easily scalable.

See our current course listing on training.ipc.org.

Courses can be offered directly to employees or integrated into your training programs.

¹ IPC. (2017). Findings on the Skills Gap in U.S. Electronics Manufacturing.



Dan Williams

to get in. On the other hand, when companies announce layoffs, that is generally good news for us. Depending on where those layoffs are, that's generally a tightening of their resources, which is good for us. It may take a while for us to see that,

but those are some of the trends we're seeing. Personally, I think as people are working from home more, companies are realizing they don't need as many people. But businesses like ours tend to do well in up-and-down markets because you can hire us and fire us any time you want. We don't take it personally.

Matties: Are you seeing more startup business coming through with the supply chain shift and layoffs? Frankly, I would think that there may be some entrepreneurship going on, if you will.

Williams: We're pretty picky about what we go after. What I would say is that the types of projects that are out there tend to be smaller, the new ones, the IoT stuff. We chatted with a guy this morning who is doing weather thermometers, and he's trying to string the whole thing together with some cryptocurrency, so there are some weird models floating around out there. All I said was that as long as we get paid in U.S. dollars, we'll take a look at it because we don't understand cryptocurrency or whatever they're talking about. There are a lot of science projects out there, and on that note, if we get involved in too many of those it can bog us down.

Irfan: Just a couple of comments. One, Dan joined us on February 15. Just before joining, we had a couple of conversations, and I came to find out that Dan had heard good things about Whizz. It was a good validation for us, somebody coming in and bringing that fever from outside.

Matties: What you're saying is, it's going to be really easy for Dan to make sales (laughs).

Irfan: No, no, no (laughs), but it adds to Whizz's credibility. Traditionally, we didn't have any salespeople here. This is actually the first time we've had two in sales and a third helper to do some analysis. Traditionally, we had not gone out to approach any customers. It was mostly word of mouth.

Matties: Dan, you've seen a lot of trends and changes over the years. What really sticks out to you as a significant change over the last 30 years?

Williams: The thing that sticks out to me the most is that there are still so many people doing it. It's absolutely mind-blowing that more board shops haven't just closed their doors. I started at Sigma Circuits. In the '90s, it was Sanmina buying everything up, but all these people, in one form or another, are still around.

Matties: There's a lot of thinking that the supply chain needs to shift, and there's a growing demand on electronics. Let's face it; automobiles back in the '80s used a fraction of the boards that they're using today. With IoT 4.0, it seems we have sensors in our shoes and we're all wearing smart watches and such.

Williams: Boards are shrinking. I had a conversation with a large integrator, and she said, "We're interested in your DFX, your DFT, but we don't do a lot of volumes of boards." Boards have shrunk over the years, so that was the first time I realized that's really happening. Boards are getting smaller, and they can do more.

Matties: We may not be doing as many panels, but there are a lot more boards on those panels.

Irfan: While the boards are shrinking because of a lot of IoT and consumer applications, at

the same time, people know there's so much computing power and they can grow a lot more. Another trend we are seeing is a slow migration toward a lot of supercomputing-type horsepower and to form factors that you could not conceive as possible 10 years ago. For example, there's a customer we are doing a product design for right now, a large package. I've not seen that big a package—8,000 pins on that. I remember almost 15 years ago we were really pounding our chests for doing 3,000 pin-count parts as a special.

Andy Shaughnessy: I noticed that you have a big design team and it's primarily made up of EEs. Could you tell us about that?

Irfan: Yes. It's a combination of EEs, and part of that offshore team is the material procurement team. For every group here at Whizz, we have an offshore group for them to basically enhance our coverage of those services. And we can do it 24/7. Generally, we can get about six working days out of a week and multiple shifts when needed.

Shaughnessy: And what led you to get into FPGA design?

Irfan: That's because of our relationship with Xilinx. We have had a 20-year relationship with Xilinx, so we do a lot of projects with them. They refer their customers to us, so we just grew in that space and that's why.

Shaughnessy: Are you seeing an expansion of OEMs using FPGAs?

Irfan: Yes, the FPGA market is very healthy and it's growing. So we're talking about both things. One is about designing systems and boards using FPGA, the other is designing and writing RTL software code for the FPGAs. We do both. In a very typical scenario, Xilinx customers would come to us with a current reference design, and they will want to add a bunch more features integrated to some other existing system or designs, and those are the things we're very efficient with design. And then we are rewriting the FPGA codes to re-target for the newer platform or developing ground-up new code for FPGAs.

Shaughnessy: And you said you'll work with people at any stage, right? Whether they come to you with the design started or whatever, and say, take care of this?

Irfan: Yes. About five years ago we had a design from a company that had shelved the design for a couple of years. It was a very large telecom switch, and they came to us and said the people who were working on this project were gone. It had four large Xilinx FPGAs on it, a lot of RTL code that had to be re-looked, understood, and redeveloped, and the whole system to redesign. We took on their project, it took about a year and a



half, and we completed it. We've taken very complex, large-scale projects as well and completed those.

Shaughnessy: You came to the rescue.

Irfan: Exactly.

Matties: What advice would you have for an OEM that wants to streamline their process?

Irfan: For one, do not underestimate the handoffs between different outsourcing vendors if they have multiple projects in the development phase, because these are the typical "gotchas" in this whole process. The second is to leverage partners who can offer more integration internally. It reduces the risk for them and minimizes product delays. Their time to market is a significant advantage, and they have one neck to choke: "This is the schedule; this is what I need. If in between your fab is delayed two days, make it up on the assembly side, I don't care, but I need boards on this day." That's a huge advantage we offer because we're able to recover from those nuances that happen in between a process.

Matties: Right. You mentioned the handoff several times in this conversation, and that's something we actually look at, even in our own company. The handoff between one department and the next, or one supplier to one customer. Really, that first 15% is the most critical because if you get that first 15% wrong, it's doomed for failure. When you say for an OEM to focus more on the handoff, what do you mean? Explain that just a little so there's clarity, please.

Irfan: If they're outsourcing the handoffs, they should look at minimizing the handoffs between multiple vendors. For example, in a typical handoff, there's a separate mechanical design shop, a separate design group, or they may be internally doing design, maybe a layout

subcontractor, a CM, and sometimes they have a separate kitting house. It's fielding the handoffs in between those, running simulations, and keeping the schedule moving in parallel. Those are the handoffs we're talking about, the schedule ownership with one party.

Matties: Right. And when we look at the handoffs, we're also looking at data, right? It's really the data. They have to show the data, make sure it's complete and understood, even coming into a BOM, because that's a handoff point of requirements to the vendor or to you in this case. That's a critical handoff as we see it.

Irfan: And faster decision-making because of the component lead time issues.

Matties: When they come to you, they need to be ready to move on it. It's not just kicking tires and trying to budget something for a year down the road because that doesn't play in this market; that's what I'm hearing.

Irfan: Yes. And there's another important aspect. When they're planning their product development and they're planning their resources, sometimes under schedule pressure they want to run very fast, and they do a lot of multiple things in parallel. There are stages where you can run fast and do things in parallel, and then there are stages where you actually become very counterproductive, and that's the part we try to educate customers on up front: "We have no problem running as fast as you like. But if you give us the go-ahead, we start running at 100 miles an hour where you have not actually made up your mind on something." Let's say there are five design blocks in that system, and they give us a green light and we have resources, we're running fast because they're on a schedule crunch.

But if the customer has not really made up their mind on block three and block four, and they've not communicated that to us, they feel that when they turn us on, they can try to

solidify that in parallel. What happens is we go faster than they thought, and then it becomes unpleasant for them. “How come you got all this work done? I wasn’t ready for block three and block four. I had to do it this way.” If they communicated that early on, we would not have gone to those. It’s running fast but in a very defined way. So that is very important to really know which prints you are running in which box and to communicate up front. It goes back to very classic communication, and people say we know it, but we see that happening repeatedly most of the time.

When they get an additional tab of ours, it’s because of changes they made, and that unpleasantness always comes up, “Why did you go so far ahead of this? I had to redo this.” “Well, you asked us, you gave us a green light, and we started running fast.” Now, we have a lot of experience, so we quickly try to gauge where the customer may still be making up their mind, and we try to not exercise the horsepower in some of those areas. But still, if you’re not communicating, that may happen. That’s another thing for OEMs to watch out for.

Matties: Dan, do you have any closing thoughts?

Williams: I think that paints a proper picture of what we have going on here, and it’s a complete one-stop: get your design done, get your prototypes done, brought up, and tested. The only part that we don’t produce is the printed circuit board. Kind of like the Rouge plant in Detroit that made the whole automobile except for the rubber tires. That’s the Whizz story.

Irfan: Also, we can start them here in the Valley, protect their IP, and give them paths to cost reduction to our own factory in Malaysia. It’s nice for customers to know that, even though we’re a small company, we have a sizeable operation here in Silicon Valley, and we can build decent volumes out of our Malaysia facility. So, they know that it’s our own factory.



Matties: What sort of volumes are you building in Malaysia?

Irfan: In Malaysia, we’re building a new factory there. We will have another 80,000 square foot facility there in another four months, so the process has already started. Product that requires maybe 50,000 to a 100,000 boards or systems a year is a good fit for Malaysia. Anywhere from a few hundred a month to tens of thousands a month, that’s Malaysia.

Matties: That’s great to have that as an option for your customers. It just furthers their confidence.

Williams: Thank you. We’re excited about where we are today.

Matties: Gentlemen, we greatly appreciate your time today and hearing what you’re doing and your insights.

Irfan: Sounds good. Thank you. DESIGN007

Visit Whizz Systems online at:
whizzsystems.com

ASK THE EXPERTS

YOUR BURNING QUESTIONS... ANSWERED

Q *Is there any instance in which a ground ref plane should not be solid? Certain datasheets used to recommend removing the ground ref plane in some situations.*

Lee Ritchey: There is no defensible reason to ever cut a ground plane. Sure, there are datasheets that say to do this. They are simply wrong.

Eric Bogatin: The only case for this practice would be if you are dealing with uV analog signals on a board that is also carrying 10A transient signals. Otherwise, no.

Heidi Barnes: Typically, one can get in a lot more trouble removing ground planes. Just consider what the field lines are doing when the ground reference is removed. The field lines will spread out and have a bigger chance of interfering with something else or radiating. Removing ground planes can reduce capacitance and in some rare cases provide isolation, but it is always best to use an EM simulator to make sure the performance benefit is understood.

Carl Schattke: Sometimes we purposely want a large area for a return path. We may want to do this on the area below an antenna, for instance, because we want that energy to propagate outward and not have a direct return path. Another reason: if we want to change the reference layer (with voided-out copper) to make a wider RF trace with a corresponding lower tolerance to manufacturing variations, giving us lower overall variation in the trace imped-

ance. Sometimes we remove the area under a high-speed pad to reduce the impedance at the same place it increases due to the component package above the pad.

Cherie Litson: Only two instances come to mind: in a flexible circuit where you need to bend the board, and when weight is an issue, such as with items going into space. When the ground ref plane is not solid, the grid should be such that it will still support a return path for the signals above and below it. Also, it should not create an issue for fabrication. It can cause acid traps on the outer layers. With inner layers, this is not as much of an issue.

Rick Hartley: There are some unusual filtering techniques available (i.e., defected ground structures) that remove small areas of ground plane under some traces in RF circuits. But these techniques should only be used if completely understood.

In general, splitting or removing ground is considered a really bad idea. If you consider where the energy is located in a transmission line—in the E&H fields, in the space between the trace and plane—removing any portion of the ground under or near the trace will simply cause the fields to spread out from their normal position in the PCB dielectric. When fields spread, it almost always leads to trouble.

The engineers who write app notes and data



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

While others struggle just to keep pace, we stay ahead of the curve by constantly investing in the equipment, people and resources needed to be the best total PCB solutions provider in the industry.

VIEW OUR CAPABILITIES



www.eagle-elec.com

sheets usually have a solid knowledge of “circuit theory” (the behavior of voltage and current). Unfortunately, PC boards’ behavior does not revolve around circuit theory. PC boards function using field theory, the movement of energy through the dielectric of the board. When that is understood, it becomes obvious that splitting ground generally leads to trouble.

In the witty words of James Pawson, a UK-based EMI consultant:

Q: What do you call an engineer who splits the ground plane?

A: A customer! **DESIGN007**

ASK THE EXPERTS

YOUR BURNING QUESTIONS... ANSWERED

Q *The skew from fiber weave effect is cutting into our jitter unit interval budget. Aside from using much more expensive laminates, what are our options?*

Eric Bogatin: Route your differential pair pitch on the same pitch as the glass weave, between 16-20 mils. Use an L-glass that matches its Dk closer to the resin. Use a low glass content laminate.

Heidi Barnes: If you can handle a little more loss, there is the option of zig-zagging the traces so that they end up with an average skew rather than bi-modal. Another option is if you actually know the spacing of the fiber weave, then one can pick a trace width such that a bus of parallel traces will always land on the same percentage of fiber weave and thus have the same relative skew.

Lee Ritchey: With the very high data rate in modern designs, controlling skew is often harder than controlling loss. There are no cheap solutions that are also safe. Rigid control over the type of glass weave is the only solution

I have found that works. Just like with cars, if you want to go fast, it costs more money.

Carl Schattke: Obviously, you are talking about a high frequency of operations and a desire to reduce costs. Sometimes designers will rotate designs on a production panel by 10 or 20 degrees to not have the traces ride over an all-glass area or an all-resin area as it makes its way across the substrate. This can help, but will it help you in your application? I don’t know; you’ll have to run some tests.

Rick Hartley: If your boards are sized such that you can rotate them slightly (10 to 15 degrees) in the fabrication panel, and still get the same number of boards per panel, you will likely lower the fiber weave skew issues dramatically. There is no guarantee that this will solve the problem, but it usually does. **DESIGN007**

Need help finding the right people for your team?

DB Management knows that finding the right people can be like looking for a needle in a haystack. We can streamline your hiring process with pre-qualified candidates.

Engineering • Management • Sales & Marketing



[Click here to get started](#)

☎ 207-649-0879 ✉ danbbeaulieu@aol.com



DFM 101: PCB Controlled Impedance

Article by Anaya Vardya

AMERICAN STANDARD CIRCUITS

Introduction

One of the biggest challenges facing PCB designers is not understanding the cost drivers in the PCB manufacturing process. This article is the latest in a series that will discuss these cost drivers (from the PCB manufacturer's perspective) and the design decisions that will impact product reliability.

DFM

Design for manufacturing (DFM) is defined as the practice of designing printed circuit boards that meet not only the capabilities of the customer's assembly manufacturing process, but also the capabilities of the board fabrication process at the lowest possible cost. While not a substitute to early design engagement with the PCB fabricator, this series of articles will provide guidelines that will help to "design for success."

Controlled Impedance

PCBs that contain controlled impedance lines require specific constructions and tighter

manufacturing process controls. The fabricator needs to tailor the construction for PCBs requiring impedance to precisely match the required nominal impedance values. The fabrication drawing should specify the required nominal impedance and tolerance and the fabricator will create a construction to meet the intended design impedance requirements.

Primary Impedance Factors

- Trace width
- Copper thickness
- Dielectric spacing
- Indicated "Reference Only."
See Impedance Requirements
- Overall PCB thickness
- Material requirements

Top fabricators use some form of impedance modeling software to determine the specific PCB construction required to produce the specified impedance. The PCB drawing should only specify the nominal impedance, tolerance, and nominal line width. This will allow for the creation of the most cost-effective PCB material construction.

Support For Flex, Rigid Flex and Embedded Component Designs Now Available.

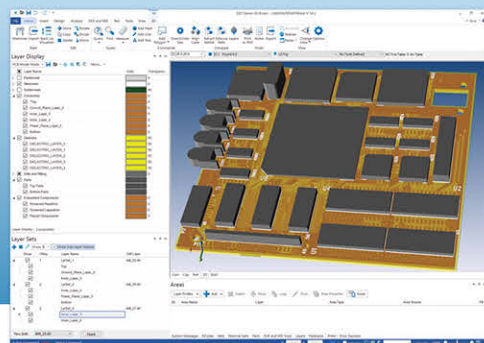
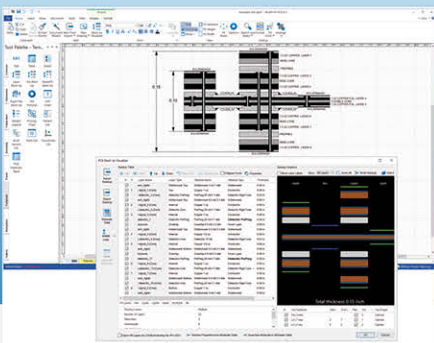
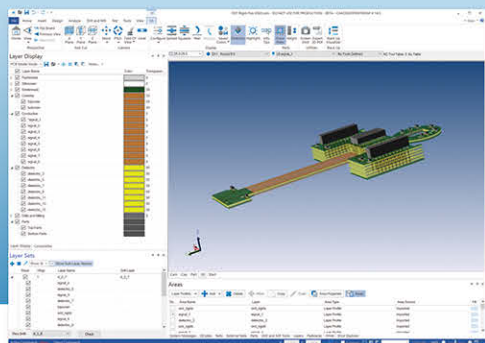


BluePrint-PCB®



CAM350®

- Import and Visualize Flex, Rigid-Flex and Embedded Component Designs
- 3D Visualization to Validate PCB Construction and Component Assembly
- Manage Variable Stackup Zones for Rigid-Flex Designs
- Easily Create Custom Flex or Rigid-Flex Fabrication and Assembly Documentation



For more information visit downstreamtech.com or call (508) 970-0670

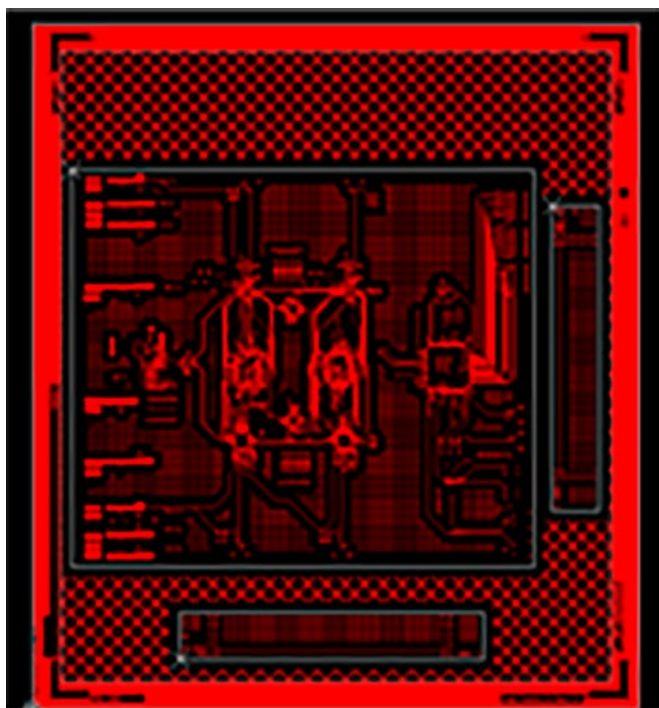


Figure 1: PCBs that contain controlled impedance lines require specific constructions and tighter manufacturing process controls.

Multiple Impedance Considerations

Some PCBs require multiple impedance values on the same signal layers. It is critical that your PCB fabricator can create impedance coupons to reflect the appropriate model for each impedance requirement. However, testing multiple impedance values on a given signal layer will cause the coupon to be wider than normal. These wider coupons take up additional valuable panel real estate, and because of this, designate one target impedance value to be tested per layer whenever possible.

There are two types of impedance classifications that are generally specified: single-ended and differential.

Single-Ended Impedance

Single-ended impedance is established by the interaction of a single trace and its reference plane(s). There are four basic impedance classifications:

1. **Microstrip:** A trace on an outer layer with a single reference plane below it.

2. **Embedded microstrip:** A microstrip line that has a dielectric over the top of it; solder mask will change a microstrip into an embedded microstrip line.
3. **Stripline:** A trace on an internal layer that has a reference plane above and below it.
4. **Dual stripline or offset stripline:** A stripline which is offset between the two reference planes; it generally is used when two adjacent signal layers are routed orthogonally and have reference planes outside of them.

Differential Impedance

Differential impedance is established by the interaction of two traces and their reference plane(s). There are five basic impedance classifications:

1. **Edge coupled microstrip:** Comprised of two adjacent traces on an outer layer with a single reference plane below it.
2. **Edge coupled embedded microstrip:** An edge coupled microstrip line that has a dielectric over the top of it; solder mask will change a microstrip into an embedded microstrip line.
3. **Edge coupled stripline:** A configuration with two adjacent traces on an internal layer, which is centered between a reference plane above and below it.
4. **Edge coupled dual stripline or offset stripline:** An edge-coupled stripline which is offset between the two reference planes; it is generally used when two adjacent signal layers are routed orthogonally and have reference planes outside of them.
5. **Broadside coupled stripline:** A configuration with the two differential lines on adjacent layers directly one above the other; these are offset striplines centered between their two reference planes.

Controlled Impedance Design Guidelines

- Standard impedance tolerance: $\pm 10\%$
 - Tighter tolerances are available but need special care

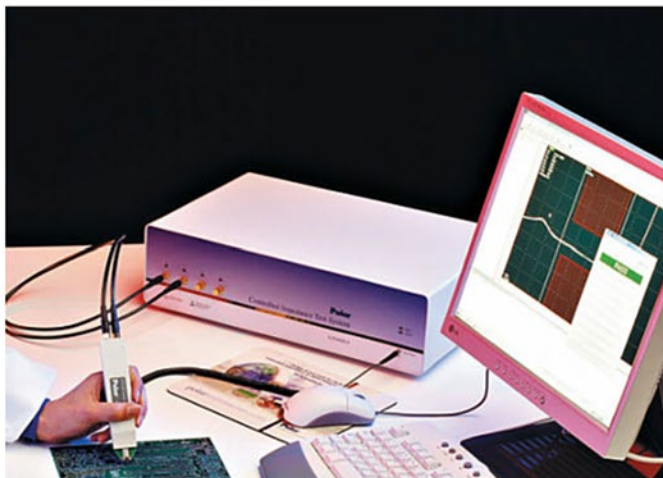


Figure 2: Controlled impedance test systems such as this Polar Instruments example can perform a series of impedance tests automatically. (Source: Polar Instruments)

- Broadside-coupled striplines should ideally be used on a core
 - Try to avoid having prepreg between them. This is required to control the Z-axis alignment between the two signal layers
- Specify the design trace-to-trace spacing for correct modeling
- Give the impedance lines a different aperture (usually 0.0001" difference) than non-impedance traces
 - Makes it easier to adjust line widths if needed
- Specify reference planes

Minimizing Impedance Costs

Using the following guidelines will help to minimize impedance costs:

- Specify impedance only on layers that really require this
- Route all controlled impedance traces onto the same layer
- Specify a $\pm 10\%$ tolerance when possible
- Designate one target impedance value to be tested per layer
- Couple power/ground on adjacent layers when possible
 - Allow for modifying the construction to meet overall tolerance

Conformance Tests

100% Testing

All impedance coupons are 100% tested when specified by the customer. After comparisons against specified values, the measurements are electronically stored.

Serialization

Serialization is a traceability process for controlled impedance jobs. Impedance is typically measured with a TDR (time domain reflectometer) by adding test coupons to the panel. Before the impedance has been tested, the coupon and PCB have a serial number marked on them.

Understanding the cost drivers in PCB fabrication and early engagement between the designer and the fabricator are crucial elements that lead to cost-effective design success. Following your fabricator's DFM guidelines is the first place to start. **DESIGN007**



Anaya Vardya is president and CEO of American Standard Circuits; co-author of *The Printed Circuit Designer's Guide to... Fundamentals of RF/Microwave PCBs* and *Flex and Rigid-Flex Fundamentals*; and author of *Thermal Management: A Fabricator's Perspective*. Visit I-007eBooks.com to download these and other free, educational titles. He also co-authored "Fundamentals of Printed Circuit Board Technologies."

Resins—Fundamental Considerations for Circuit Designers

Sensible Design

by Beth Turner, ELECTROLUBE

Welcome to the first of my new series on encapsulation resins. I am going to focus on the all-important subject of resin selection, types of chemistries available and their properties, application, mix and cure. First, though, let me introduce myself. I'm Beth Turner and have worked for Electrolube for five years, starting as a member of the Global Technical Support Team. I've spent my career as a research and development chemist working on adhesives, resins, and conformal coatings at Apollo Chemicals before moving to Electrolube, and have previously worked on specialist projects, including bio-functional nano-coatings and coatings affording protection to electronics in harsh environments.


When it comes to the choice and applications of resins, there's a great deal to talk about, and over the forthcoming months, I

hope to provide some useful tips and design advice that will help in your quest for reliable circuit protection. For this resins series, I will start by reviewing some fundamentals that will help circuit designers understand some of the essential properties of resins before deciding which is most suitable for the task at hand.

Selecting the Correct Resin For Your Application

Choosing the correct encapsulation resin is possibly the most critical aspect of the design process. It is imperative to understand where and how the finished unit is going to be used and what performance criteria are expected of it. It is best practice to draw up a list of the ambient or standard operating conditions that the unit will be exposed to, and then list what the extremes of those operating conditions are likely to be.





Hmm, what is the
difference between
base and finished
copper weights?

PCBs are complex products which demand a significant amount of time, knowledge and effort to become reliable. As it should be, because they are used in products that we all rely on in our daily life. And we expect them to work. But how do they become reliable? And what determines reliability? Is it the copper thickness, or the IPC Class that decides?

Reliable answers. **Reliable** PCBs.

Every day we get questions like those. And we love it. We have more than 490 PCB experts on 3 continents speaking 19 languages at your service. **Regardless where you are or whenever you have a question**, contact us!

What's your PCB question?
www.ncabgroup.com/PCBquestion



Another important factor to consider is the duration of time that the unit will be exposed to in the most extreme of operating conditions. For example, there is a vast difference between specifying a chemically resistant resin that can withstand fully permanent immersion in anti-freeze, and one that only requires resistance to occasional splashes of antifreeze that are wiped clear after short periods of exposure.

Another important factor to consider is the duration of time that the unit will be exposed to in the most extreme of operating conditions.

Similarly, if an application reaches a maximum temperature of 150°C, but this only occurs once a day for a couple of minutes and the rest of the time the normal operating maximum is 90°C, then it is sufficient to specify a resin with a maximum operating temperature of, say, 100°C. Most polymeric materials will tolerate wider temperature excursions than originally quoted for continuous operation over short periods, similar to those seen in typical reflow profiles.

Other application requirements for consideration might include flame retardancy. Is UL certification essential as part of material qualification or is it customary to accept a manufacturer's internal test report? For LED lighting assemblies, optical clarity and UV resistance is important. Information on changes to correlated colour temperature (CCT), colour space ($L^*a^*b^*$) and total colour difference (ΔE^*) can be useful when qualifying a resin to pot directly over a PCB with LEDs and lenses. Or perhaps a design engineer wants to protect their circuit design from potential IP theft, in

which case it is better to select an opaque resin. RF communication is used in many industries; these communication devices can benefit from encapsulation resins with low loss tangent and low dielectric constant properties to provide electrical insulation.

Electrification in the transport industry requires high power density electrical components to sustain long ranges and high horsepower. To manage the heat generated by small, high power devices, thermally conductive encapsulation resins can effectively dissipate heat away from hot components to a heat sink to sustain long-term performance. There are several different thermal characterisation techniques including steady state and transient methods; each method gives some variety in results so it's worth bearing that in mind when comparing literature from different manufacturers.

What Types of Resin Chemistries are Available?

Resin chemistries fall within three major classes: epoxy, polyurethane, and silicone. Epoxy is the strongest and most chemically resistant of the three, but it is brittle, challenging if not impossible to remove for rework, and is typically limited to operating temperatures between -40°C and +150°C. Epoxy resins offer excellent adhesion to a wide range of substrates.

The tough and flexible polyurethanes are suitable for applications operating at lower temperatures. Typically, polyurethane resins are only suitable for applications reaching maximum temperatures of 110°C for long periods (though some can go to 130°C). The chemical resistance of a polyurethane resin is generally lower than that of an epoxy, but polyurethanes outperform epoxies in water and high humidity environments. Polyurethane resins are typically used in marine applications, where water penetration resistance is critical, and for applications subject to a high level of physical stress, such as the potting of accelerometers,

or sensors embedded in road surfaces. If large temperature swings are expected over a short timeframe, then the flexible nature of the resin is advantageous as there is a low probability of stress cracking occurring.

Silicone resins have the widest operating temperature range (-50°C to +220°C) and when cured are the most flexible of the three resin chemistries. Their adhesion to certain substrates is poor, as is their chemical resistance, particularly to chemicals more commonly encountered in everyday use, such as alcohols, aromatics, and ketones.

While they do tend to differ widely in terms of performance characteristics, all resin types have excellent electrical insulation properties across their respective temperature ranges. Resin chemistries have advanced considerably in recent years and there are now resins available that exhibit properties and performance criteria which is often well beyond the normal boundaries expected for that resin type.

How Much Resin Should You Apply?

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

What Are the Key Factors for a Successful Mix Ratio?

The mix ratio is quite possibly the most critical aspect of resin mixing. Brace yourselves to face long-term adverse repercussions if you get it wrong. There are two methods of mixing a resin with its associated hardener, either by hand or by using specialist dispensing equipment. If mixing by hand, then the ratio of the

weight of the two components is the more useful method to employ. If mixing via dispensing equipment, then the volume ratio is used.

If the job is reasonably small, then you are likely to use a resin pack, which provides the resin and hardener in precise quantities, in separate compartments of the pack. When you are ready to use the product, you simply remove the clip or other separating device between the compartments and “massage” the resulting pouch, ensuring that both components are completely mixed. For further support on mixing techniques, there is a how-to video in the Knowledge Base section of the Electrolube website. For larger production jobs, the resin and hardener are supplied in separate bulk containers, suitable for use with two-component metering and mixing systems. It is important to check the shot sizes for each component regularly to ensure that the correct amounts of resin and hardener are being dispensed.

It is important to check the shot sizes for each component regularly to ensure that the correct amounts of resin and hardener are being dispensed.

When mixing bulk resin and hardener, it is important to avoid introducing excessive amounts of air, which will form micro-bubbles within the cured resin. The liquid resin and liquid hardener used for polyurethanes are particularly sensitive to moisture, so it is important to avoid humid conditions when potting to avoid micro-bubble formation. These micro-bubbles may expand when hot and create voids. If you are not happy with

bulk materials mixing and incurring the potential problems of introducing too much air (and moisture) into the mix, then it might be more appropriate to use automated metering and mixing equipment, which will accurately mix resin and hardener in the correct proportions, and usually in an inert atmosphere.

Remember, incorrect ratios will lead to a poor cure and the physical properties, tensile strength, elongation, and modulus will differ from those specified in the manufacturer's data sheet.

Important Considerations for Cure Temperature

Once a resin and its associated hardener are mixed, the reaction can be very fast but also very exothermic, which can lead to the possibility of a runaway reaction; remember that the exotherm will also increase as potting volume increases. Not all reactions have a high exotherm; the exotherm temperature can be controlled by adjusting the chemistry or by using a filled system, as the fillers absorb some of the heat as well as reduce the concentration

of the active component within the hardener that promotes this rapid cure. Controlling the temperature during the cure profile is important because the components requiring potting or encapsulation might be adversely affected by raised temperatures.

It is important to follow the recommended cure profile to ensure resins are fully cured and the physical properties achieved match those specified on the manufacturer's datasheet.

I hope these points have been useful and informative. Please contact our technical support team if you need any advice with your resin application requirements. In the meantime, please watch for my next column, where I will be examining resins in more depth. **DESIGN007**

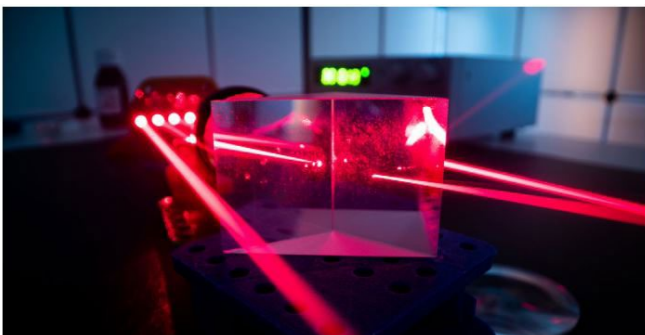


Beth Turner is head of encapsulation resins at Electrolube. To read past columns from Electrolube, [click here](#). Download your free copy of Electrolube's book, *The Printed Circuit Assembler's Guide to... Conformal Coatings for Harsh Environments*, and watch the micro webinar series "Coatings Uncoated!"

Researchers to Develop Made-to-Measure 3D Laser Beams

Researchers at the National Robotarium, hosted by Heriot-Watt University, have secured £586,000 to develop 3D laser beams whose shape can be changed.

The innovation is set to transform the manufacturing and healthcare technology industries, making it easier and cheaper to produce products that require highly-precise manufacturing, such as medical equipment and mobile devices.



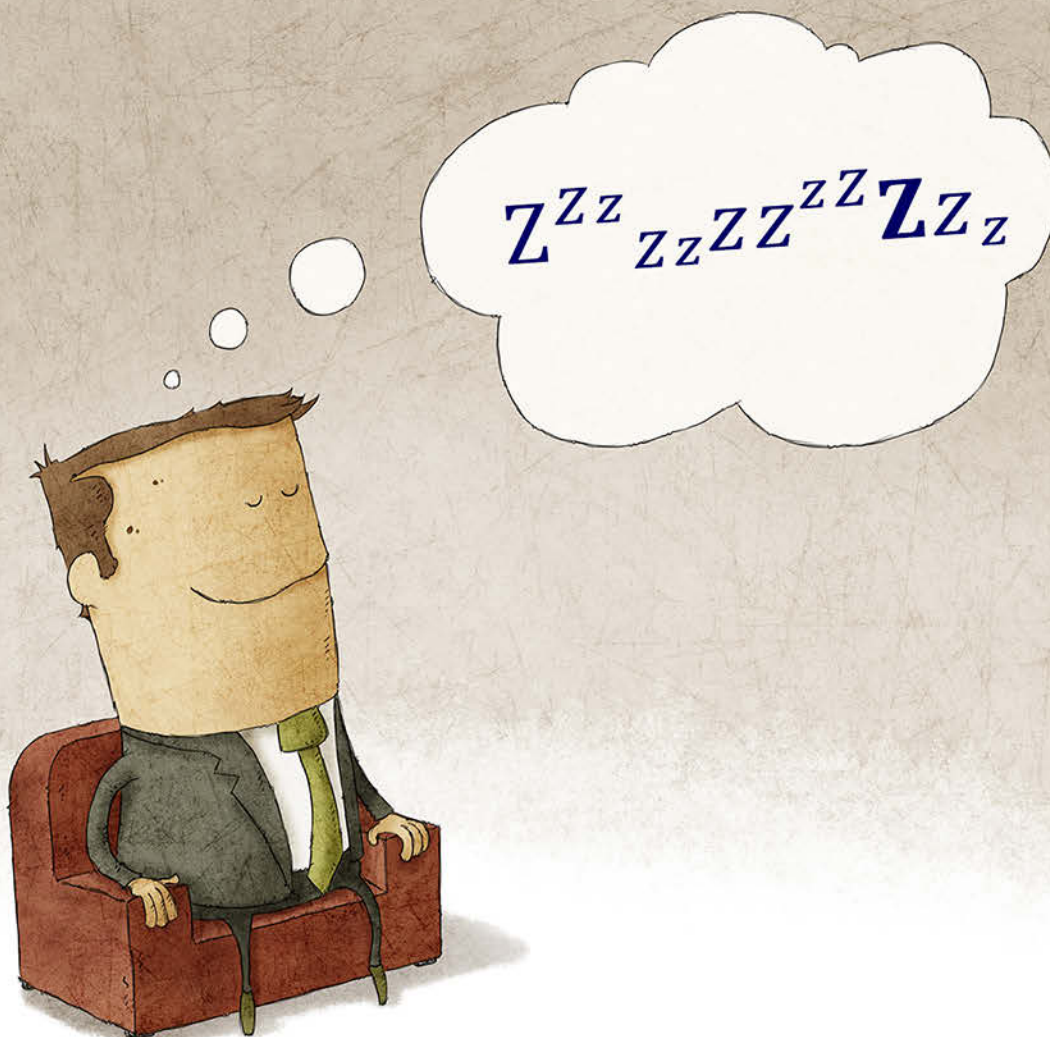
Lasers are a crucial component of modern manufacturing, with the global laser processing market projected to grow from £2.8 billion in 2020 to £4.1 billion by 2025. They are used widely by industry to produce precise incisions and mould materials into specific shapes.

The research to be undertaken at the National Robotarium will develop laser beams which have been specifically designed to meet the exact manufacturing requirements of products, improving efficiency and precision.

UK Government Minister for Scotland Iain Stewart said: "This is cutting-edge technology in every sense of the phrase. "This exciting research is being supported both by a £586,000 UK Research and Innovation grant, and our £21 million investment in the National Robotarium through the Edinburgh City Region Deal."

(Source: Heriot-Watt University)

We **DREAM** Impedance!



Did you know that two seemingly unrelated concepts are the foundation of a product's performance and reliability?

- Transmission line impedance and
- Power Distribution Network impedance

DISCOVER MORE

iCD software quickly and accurately analyzes impedance so you can sleep at night.

iCD Design Integrity: Intuitive software for high-speed PCB design.



"iCD Design Integrity software features a myriad of functionality specifically developed for PCB designers."

– Barry Olney



www.icd.com.au

ASK THE EXPERTS

YOUR BURNING QUESTIONS... ANSWERED

Q *Are dimensions still required on board outline drawings when you have a Gerber or other profile available and very intricate shapes and curves? Would a note be more appropriate with a tolerance?*

Stephen Chavez: Ah... this a debatable topic indeed, especially within our current technology and tools, which have improved greatly over time. As the integration between design teams and industry suppliers continues to evolve when we talk about the “digital thread” and the collaboration and integration during the hand-off stage from design to fabrication, quality content and quality data exchange are paramount. Within today’s industry evolution, we now can generate intelligent data that far exceeds the use of simple non-intelligent Gerber data from our EDA tools.

Today’s intelligent data formats such as ODB++ and IPC-2581 provide the design team the ability to seamlessly integrate and exchange data content with our suppliers so that we can be more efficient and better optimized. Communications between the design team and the supplier are key. And designers must understand what is minimally required to be conveyed in order to produce the designed intent of the PCB/PCBA.

Do we need to dimension everything? Can we simply add specific notes/instructions on our drawings? It’s not a simple “yes or no” response. It depends on the complexity along with criticality of the feature of the design in question. If it’s critical, then it should be docu-

mented and conveyed to the supplier during the data exchange.

This data exchange would include some form of documentation, especially regarding fabrication drawings that convey the engineering intent to the supplier. This is extremely important and attention to detail is critical. In general, one of the first lines in our fabrication drawings states the IPC class to which the board is to be fabricated, as well as what call-outs it is to be inspected to. This ties in the following industry specification that are recommend as industry guidelines and best practices:

- IPC-352A Documentation Requirements for Printed Boards, Assemblies and Support Drawings
- IPC-2614 Sectional Requirements for Board Fabrication Documentation
- IPC-2615 Printed Board Dimensions and Tolerances
- ASME Y14.5 Geometric Dimensioning and Tolerances

With that said, every company has its own internal standards and guidelines. Typically, dimensioning a PCB within a fabrication drawing would show dimensions of critical features such as holes, slots, board outline/edges, and

locations, etc. It's up to the engineering team to confirm with the supplier what is expected, what is acceptable, and what is agreed upon to produce and deliver a PCB at the required level and quality. We don't want the supplier to have to make assumptions or guess, and a good supplier will not do so anyway. Instead, a good supplier will hit you with a "technical query," while your job is stopped and placed in a holding pattern, until you clarify any concerns from the supplier.

Cherie Litson: You obviously think that the fabricator can follow your outline somehow. Only if you send them a route file will they be able to do that. You need to have dimensions and tolerances on those dimensions and notes so that the board can be built and checked properly. If not, then you get what you get, which will probably not be quite what you wanted. Gerbers are stupid camera "open, draw, close" files, and they are only for exposing what copper to keep. They are effective for the copper, but not for a board edge. (There's no copper there, or should I say, there shouldn't be.) Having a dimensioned drawing helps to clarify any misinterpretation of the files you are sending, even if you're sending ODB++ or IPC-2581 files. There have been many times where I missed checking a box in the output software and things got crossed up. When fabricators had a picture of what my inten-

tions were, they could contact me if there was a mismatch.

Carl Schattke: The short answer is no, they are not required. Long answer: There are a few reasons to be more specific in our documentation. I like to have a layer reserved for the board routing profile. This can be imported and used by the manufacturer. Adding dimensions lets the manufacturer know that they are doing this right, and they can check the value. Also, adding dimensions lets incoming inspection at the PCBA vendor check to make sure the routing was done correctly and that the image is registered to the board outline properly. You have to add the dimensions needed so that they can be checked if you don't want to have that be a possible failure mode.

Rick Hartley: When dimensions on the drawing match the information in the electronic files, this tells the fab and assembly people that we are paying attention and gives them a better sense of confidence that all the dimensions are correct. Are they strictly needed? No, but they can be helpful.

Lee Ritchey: Yes, it is vital to dimension a PCB on the fab drawing. How else will a fabricator determine how to bid on it? Gerber data is rarely given to a fabricator until a P.O. is issued. **DESIGN007**

I.007e
WORKSHOP

American Standard
Circuits

On Demand: Free Training Video Series

**FLEXIBLE
CIRCUIT
TECHNOLOGY**

with Expert Joe Fjelstad

WATCH NOW

Say What You **Do**— Do What You **Say**

The Digital Layout

by Kelly Dack, CIT, CID+, PCEA

Introduction

In this month's column, I'll pose some questions and give you something to consider regarding joining an electronics industry organization. Next, I'll hand it off to PCEA Chairman Stephen Chavez, who conveys excitement and anticipation of events to come and how the PCEA will be involved. Again, I am happy to provide you with a growing list of events for 2021.

PCEA Updates

Are you finding your place in the PCB industry? If you are feeling a bit disconnected from the rest of the world as you perform your PCB engineering job, maybe you should consider

joining a trade organization. Aligning and participating with a trade organization that is well matched to your areas of expertise can be monumentally beneficial to your career. It can put you in touch with others who can help you discover what you don't know—and which you may find out you needed to know. There are many professional development groups and trade organizations in our PCB engineering industry. Finding the right ones to join may not be as easy as you think. On the other hand, by following some of the criteria our PCB engineering quality assurance stakeholders use to find good products and services for your company, it will be easier to narrow your list because you know some valuable things to look for.

Many of us in the PCB engineering industry do not necessarily specify or qualify materials ourselves. But we might work closely with





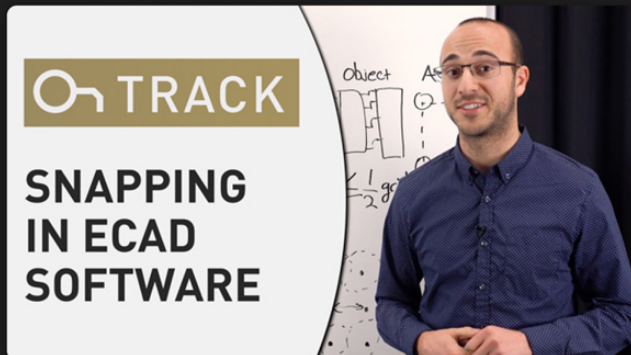
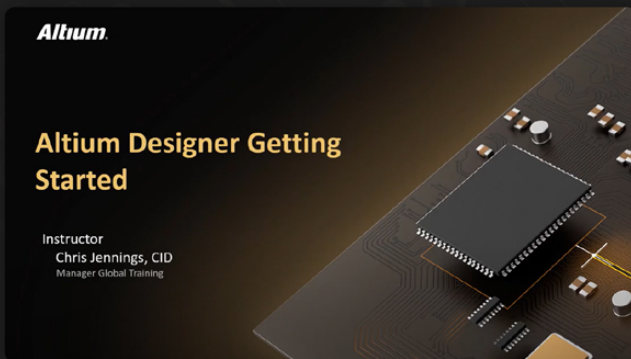
ALTIUM
ACADEMY

Online Education for PCB Designers and Electrical Engineers

Access over 350 tutorial videos

- ✓ On-demand Webinars
- ✓ What's New with Altium Designer
- ✓ Getting Started with Altium 365
- ✓ Whiteboard Series

Watch Now



Altium.



those who take part in qualifying the materials and manufacturers who are responsible for checking out and making important decisions regarding the quality of the materials or processes offered. It could be valuable to understand how these industry stakeholders set up business relationships and roadmaps for moving forward.

All these folks—sometimes referred to as quality auditors—have something in common when they search for resources and providers: They often belong to or are associated with industry organizations and look for conformance to quality standards.

Let's reflect on our own resource requirements for a moment. As mentioned before, we may not be tasked in our day-to-day jobs to find sourcing for the next super-computer chip or specially formulated material to route its circuitry on. But regardless of our professional part in the printed circuit engineering industry, we can benefit from using some of the same audit criteria to qualify a professional organization we choose to associate with.

Quality auditors look for several important things when evaluating a new supplier with which to engage in successful, mutually beneficial, long-term business relationships. Let's think for a few moments about how some of these criteria can be applied to evaluating an industry trade organization. When searching for a trade organization to join, wouldn't you want to "audit" and check for some of the following?

- A culture of operational compliance: Are their standards and operating procedures documented? Do they say what they do and do what they say? Do they provide open book operations with clearly defined stakeholders and areas of responsibility and traceability?
- Social compliance: Are the management personnel or board of trustees well-known members of the community or relevant industry? Are they diversely represented? Are they transparent and communicative,

free of any potential conflicts of interest? Are their business dealings and the mission of the organization clear and consistent?

- Does the organization offer education and training for the goods they provide? If there is a charge, is it of justifiable value?
- Can the organization readily provide documentation and explanation of its business ethics and value to the stakeholders it serves?

Maybe you have some criteria of your own. It is important that the organizations you reach out to for information are responsive to your interests and respond quickly with details on how they not only can connect you with information which can help your career, but demonstrate ways you can become vested in the organization's membership. At the very least, a potential trade organization should be able to offer you a tangible description of what it stands for. Its purpose and mission must be aligned with your need for understanding and desire for engagement in order to engage in a successful, mutually beneficial, long-term relationship.

Message from the Chairman *by Stephen Chavez, MIT, CID+*

Wow, it's hard to believe the year is half over and summer is in full swing. I live in Arizona; our monsoon season has started and we're feeling those triple digit temperatures. As these days/months are flying by, it just seems as if there is never enough time, and the never-ending workloads keep piling up. Don't even get me started on those "honey-do" lists. As PCEA evolves and grows, the original vision that has become our mission statement (Collaborate, Educate, and Inspire) is ever more a reality. Personally, it is about making a positive difference in our industry by helping others to be successful and succeed in life. We do this by networking, sharing experiences both good



Stephen Chavez

and bad, passing on knowledge while inspiring others, or to be inspired by others. That's PCEA at our core.

The PCEA vision is unfolding as our individual chapter activities gain momentum and our chapter growth flourishes. As new members continue to join the PCEA collective daily, along with our sponsorships and affiliations, the PCEA synergy is spreading as we envisioned. It's truly amazing to see the positive energy through true collaboration in each chapter, and each member.

Now that we are coming out of the pandemic darkness and COVID-19 restrictions loosen (if not going away), we are finally getting back to in-person interactions, and without those face masks. I think our hands have aged faster than the rest of our body due to the constant washing and sanitizing. Industry trade shows and conferences are coming back to life with a few of them already having taken place with much success. Yahoo and thank God.

Because we are back to in-person interactions, the second half of this year will be awesome regarding industry events such as Design-Con 2021, SMTA international, PCB Carolina 2021, and PCB West 2021, to name a few. I simply cannot wait to attend my first industry face-to-face conference or trade show. It is so long overdue, in my opinion. When I speak with other industry friends and colleagues who have already attend a live industry event in person, I am amused to see that they are filled with excitement and a kind of "happy conference euphoria," yet exhausted from being out of conference "shape."

So, I look forward to the hustle and bustle of traversing through crowded airports and congested rental car parking lots or even taking Ubers and taxis just to get to my first live conference. I simply cannot wait; it's similar to when we were kids just a few days before Christmas morning. The excitement is building inside me, that's for sure. You can also be assured you will see a PCEA presence at many of these conference and trade shows one way

or another. Whether greeting visitors at a conference booth on the showroom floor, or at an informal meeting with other fellow members, PCEA will be there. So, look for us, join in on the conversations, and integrate as you see fit. We would love to engage and interact with you face-to-face.

As always, if you have anything to do with printed circuit engineering, I highly recommend you get involved with and join the PCEA collective—your percentage of long-term professional development increases significantly.

Refer to our column and the PCEA website to stay up to date on up-and-coming industry events. There are many free webinars out there, so take advantage of these opportunities. Visit us at pce-a.org.

I wish health and safety to you and your family. Best of success to all as 2021 unfolds.

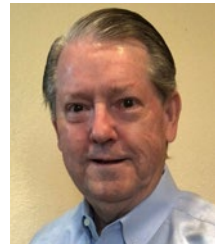
Warmest Regards,
—Steph

Chapter Activity **by Scott McCurdy**

PCEA-Orange County Chapter President

The PCEA-Orange County Chapter in southern California held a virtual "Lunch 'n Learn" event on May 19. Hosted by EMA Design Automation, Senior Field Applications Engineer Orlen Bates gave an educational presentation on "Designing for RF—Tips and Tricks from the PCB Pros."

The topic of RF design must have struck a chord, as this meeting drew an audience of 98 printed circuit professionals, making it one of the largest events in our chapter history. Orlen's many decades of manufacturing and design experience fueled his talk and the images and examples he presented provided a valuable learning opportunity of the best practices and how to avoid design pitfalls to those in the audience. There was also a lengthy question and answer period at the end which added



Scott McCurdy

great value with audience participation in this webinar format.

As the pandemic wanes and California starts to open, we hope that later this year our chapter will be able to have in-person events again. We all miss that great feeling of being in a large group of our fellow colleagues, interacting in these educational and interesting events.

Next Month

August is sure to be a time for “doing what we say” as far as the PCEA’s commitments for engaging in activities to collaborate, educate, and inspire those both inside and outside our organization. We hope to bring you more information regarding PCEA’s role in certifying those who have completed the Printed Circuit Engineering Designer program, authored by several renowned members of the PCEA and offered through EPTAC Corporation.

Upcoming Events

Below is our list of upcoming events. Hope to see you there.

DesignCon 2021

August 16-18, 2021
San Jose, CA

PCB West 2021

October 5–8, 2021
Santa Clara Convention Center
Santa Clara, CA

SMTA International 2021

Nov. 1–4, 2021
Minneapolis, Minnesota

PCB Carolina 2021

Nov. 10, 2021
Raleigh, North Carolina

productronica

Nov. 16–19, 2021
Munich, Germany

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

Conclusion

We hope you will look at all the information and resources on our PCEA website, broadcast through our online chapter presentations, and written about in this column. Indeed, we want to say what we do and do what we say when it comes to being a trade organization that will pass your audit for value, integrity, and making a difference in your PCB engineering career.

See you next month or sooner! **DESIGN007**



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

jobCONNECT007

Companies seeking talent with circuit board industry experience post their jobs with us.

jobconnect007.com

Gain knowledge, save time and
improve your bottom line with ASC's

FREE RESOURCES



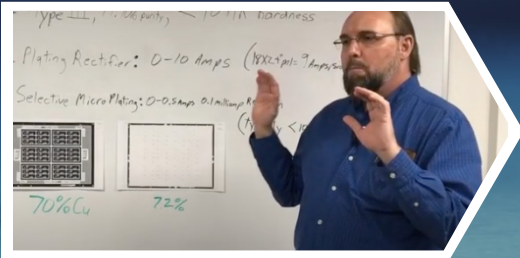
WEBINARS

Attend weekly free webinars
focused on different PCB topics



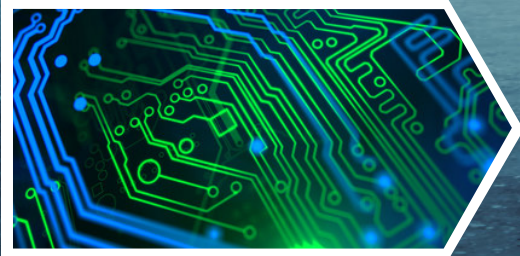
eBOOKS

Download concise and relevant
books written by the ASC Experts



EXPERTS BLOG

The ASC Experts help you learn more
about advanced PCB technologies



ONLINE DFM

Upload Gerber files and get results
back in just 60 minutes



American Standard Circuits

Creative Innovations In Flex, Digital & Microwave Circuits

www.asc-i.com | (630) 639-5444

ASK THE EXPERTS

YOUR BURNING QUESTIONS... ANSWERED

Q *We're having lots of problems designing good stackups for 6-, 8-, 10- and 12-layer boards with multiple powers and grounds. Any suggestions?*

Stephen Chavez: When it comes to stackups, getting it right makes all the difference from the very beginning. The stackup is the foundation of a PCB design. It sets up the structure for any power and impedance requirements. Using multiple powers (PWR) is the norm and usually required in today's complex PCBs, especially when implementing a complex FPGA. As for multiple GND (reference) planes in a design, you need to truly understand how to design a quality stackup to set yourself up for overall success. It is suggested to only have "one" GND reference utilized on multiple planes throughout your stackup. It's also recommended to have this GND reference plane adjacent to a PWR plane for better coupling when it comes to controlling the energy within your stackup design. The general goal with this PWR and GND pair setup is to increase your capacitance and reduce your inductance. This will help in mitigating SI and EMC issues that may present themselves. Keep in mind that good signal trace-to-reference plane setup is also required to mitigate similar issues related to SI and EMC.

Most SI and EMC issues can be traced to design errors such as bad PCB stackup, poor PWR and GND utilization, and poor signal routing. Placing PWR and GND planes adjacent to one another within a PCB stackup, and routing signals adjacent to their appropri-

ate reference plane, without routing the signal over any interruptions (gaps) in that reference plane, is the key. It amazes me how often designers simply make this type of mistake. So, the key to success is to get the PCB stackup correct early in the design phase. This would include working with your PCB supplier in creating and verifying the stackup. Then, implement industry best practices for both GND and PWR utilization along with quality signal routing. Take the time to do the due diligence to get it right the first time. It will be worth it.

Rick Hartley: Placement is everything. I've said this for at least 40 years. I have put as many as 20 different voltages on a single power plane of a high-speed board, and made the circuit function perfectly. Is this easy? No. It requires perfect part placement. On a very dense board with 4,000 to 6,000 components, placement this good can take weeks. If you have taken the time to achieve placement that good and you still cannot get the job done with one or two power planes, plus some small areas of localized power (IC cores) on signal layers, then you will likely need to increase layer count. I know you have already figured that out.

One other comment: If you have more than one ground in the design, it's likely that someone does not understand the basic concepts of energy movement and the reasons why

you almost never need more than one ground plane.

Carl Schattke: Engineer a return path for the best results. Quite a few in-depth articles are available on this topic.

Cherie Litson: Number one, take Rick Hartley's class. Number two, make sure you group components over the appropriate power and ground references and treat power as a fat signal trace, not a plane. Number three, don't run traces over gaps in your planes.

Lee Ritchey: Yes, with the multiple voltages found in most modern designs, it is a tough job assigning power and ground layers. There is no simple answer to how to do this. My latest design has 29 power rails in a 22-layer stackup. It took almost a month to get the power part of the design right, and it only took

two days to develop the design rules. Welcome to the 21st century.

Heidi Barnes: I would take a look at the advantage of thin dielectrics to lower the path inductance and the loop inductance of power and ground vias. This should make it possible to do more of a wide trace power rail routing instead of trying to use a whole plane layer. The lower path inductance also allows all but the smallest capacitors to be placed further from the load so that there is less congestion when routing multiple rails into a BGA-type device.

Eric Bogatin: Keep the same return plan adjacent to signal layers so you can add a return via when the signal transitions. Keep power and ground on adjacent layers with as thin a dielectric as possible. Use a power-power plane pair for dielectric fill. **DESIGN007**

Inkjet Printing Shows Promise as New Strategy for Making E-Textiles

In a new study, North Carolina State University researchers demonstrated they could print layers of electrically conductive ink on polyester fabric to make an e-textile that could be used in the design of future wearable devices. In addition, researchers said the findings suggest they could extend techniques common in the flexible electronic industry to textile manufacturing. They reported their findings in the journal *ACS Applied Materials & Interfaces*.

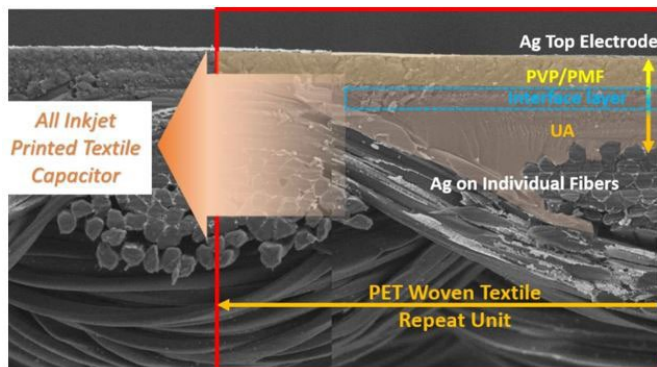
In the study, researchers described how they used a FUJIFILM Dimatix inkjet printer to create a durable and flexible e-textile material, what they did to reliably create the e-textile, and its properties. Part of their challenge was to find the right composition of materials so the liquid ink would not seep through the porous surface of the textile materials and lose its ability to conduct electricity.

They created the e-textile by printing layers of electrically conductive silver ink like a sandwich around layers of two liquid materials, which acted as insulators. They found that the chemical properties of the insulating materials, as well as of the tex-

tile yarns, were important to maintaining the ability of the liquid silver ink to conduct electricity, and prevent it from penetrating through the porous fabric.

"We wanted a robust insulation layer in the middle, but we wanted to keep it as thin as possible to have the entire structure thin, and have the electric performance as high as possible," Kim said. "Also, if they are too bulky, people will not want to wear them."

(Source: North Carolina State University)



ASK THE EXPERTS

YOUR BURNING QUESTIONS... ANSWERED

Q *No offense to your experts; I love your publication. But wouldn't designers be better off posing questions to fabricators' CAM departments? But they won't answer them anyway! Which begs a question: Why don't most fabricators share their capabilities, Valor settings, etc., with designers—and then complain?*

Lee Ritchey: Getting a fabricator's CAM department to talk to you is a tall order. Most won't do it.

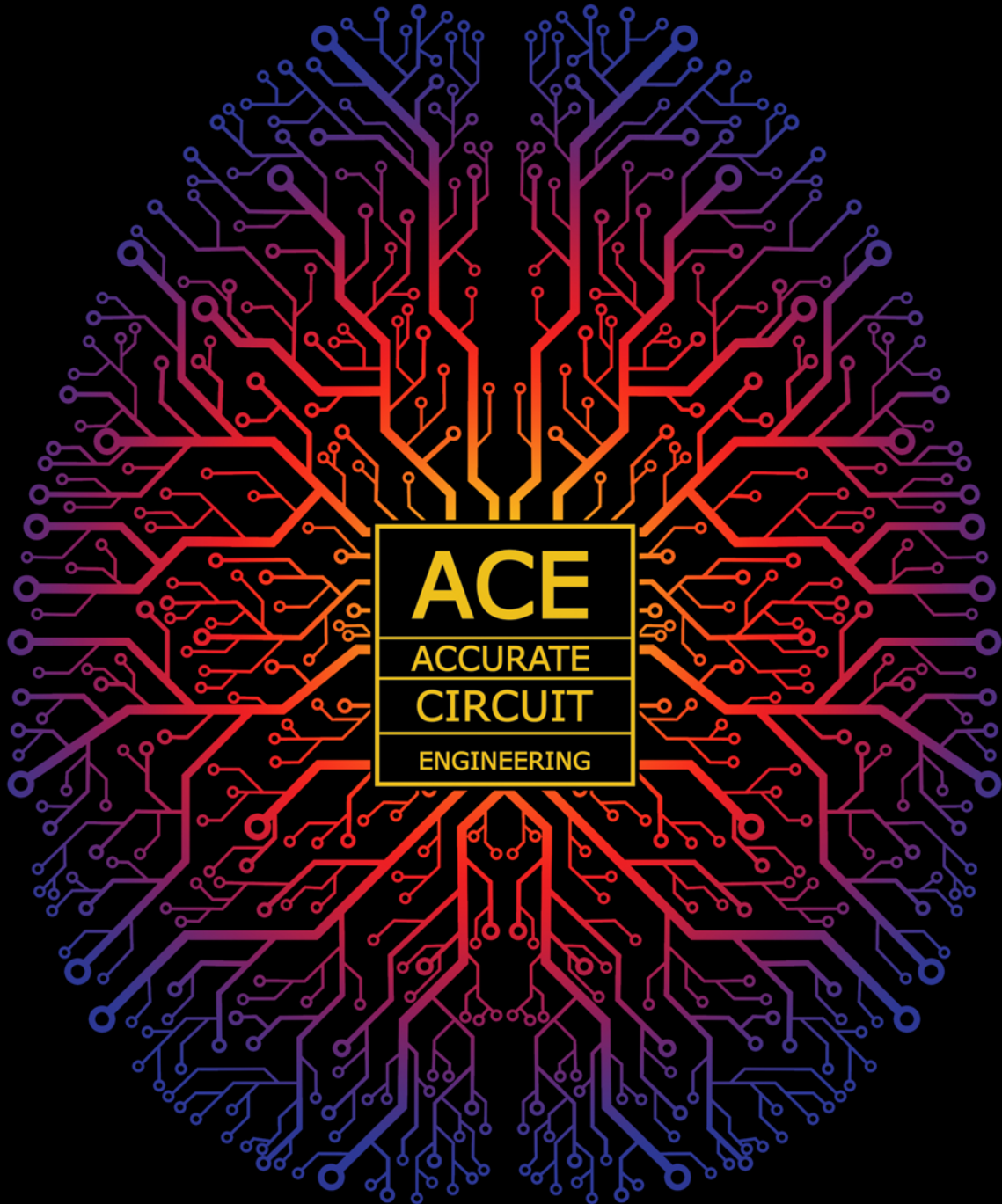
Cherie Litson: Ha! Totally feel your pain. That's why you ask us. We've been down this road before. You do have to pose some questions to your fabricators. And don't just talk to the salespeople; it's not their job. Talk to the fabrication engineer. Know that they are very busy people and don't have much time to train you. Also, the fabricator doesn't want to tell you that what you did was basically stupid, from their point of view. It helps if you know the right questions to ask. Take a good DFM class or earn your Certified Interconnect Designer (CID) from IPC. This will help you find out how to ask the question so that they will give you the answer you need to hear. It may not always be the answer that you want, though.

Stephen Chavez: No offense taken. I'll just ask a rhetorical question. It all comes down to relationships. How good is yours with your suppliers? If you have a very good relationship with your supplier, then this is not an issue you would run into. If you don't have a good or established relationship with your supplier, then why not? What are you doing to improve this?

You should be engaging with your fabricator's CAM department regularly to ensure overall success. Good designers and engineering teams do this regularly and have well-established relationships in place with their supplier(s). I would agree that some suppliers don't openly share their Valor settings, which many consider their IP. Most suppliers will share their capabilities, though. You simply need to ask for it. Many suppliers share this on their websites. If your supplier will not share their capabilities with you, then maybe you should rethink utilizing that supplier. As for the Valor (DFM/DFA) settings, I would think that once you have an NDA in place with that supplier, getting that information would not be too difficult. Again, this comes down to relationships and simply asking for this data for the mutual benefit of a long-term relationship between your company and the supplier.

If the supplier is good and in it for the long haul with you, chances are that you can obtain that information. You can blame industry competition between suppliers. Suppliers don't openly share such content simply because every supplier has set up unique internal processes and structures for their long-term success. It all comes down to what is considered internal IP and the competitiveness among

Don't Overthink Your Prototype Builds



Go with the team that can make your ideas a reality...

RF • MICROWAVE • HIGH-SPEED • QUICK TURN EXPERTS



Accurate Circuit Engineering

3019 Kilson Dr
Santa Ana, CA 92707

(714) 546-2162 • sales@ace-pcb.com

For technical assistance, call:

James Hofer
(714) 227-9540

www.ace-pcb.com

companies eager to get ahead and survive in today's industry.

Heidi Barnes: The correct answer is that you need to ask both the engineer and the fabricator. There are always conflicting trade-offs and it would be a rare case where someone is an expert at both.

Chris Young: This is an opportunity to ask a broad range of questions to a group of people that collectively have a massive breadth and depth of experience. These people have business relationships spanning decades that have generated millions, if not billions, of dollars in PCB fabrication and assembly revenues. If there is an answer to be had, these people can find it.

Rick Hartley: I don't know who your fabricators are, but I've never had trouble getting answers from the fabricators we used when I was employed full time. Maybe you need to encourage your company to switch fabricators. Good fabricators (and assembly CMs) know that working closely with the customer benefits everyone.

Carl Schattke: In my experience, fabricators will share their Valor setups if the business case presents an opportunity for them. Almost all vendors will answer questions about improving yields and what can make that happen. Overwhelmingly, I find PCB vendors really are willing partners to work with us on making a better product. As we succeed, they succeed. **DESIGN007**

New I-007 eBook Highlights System Analysis

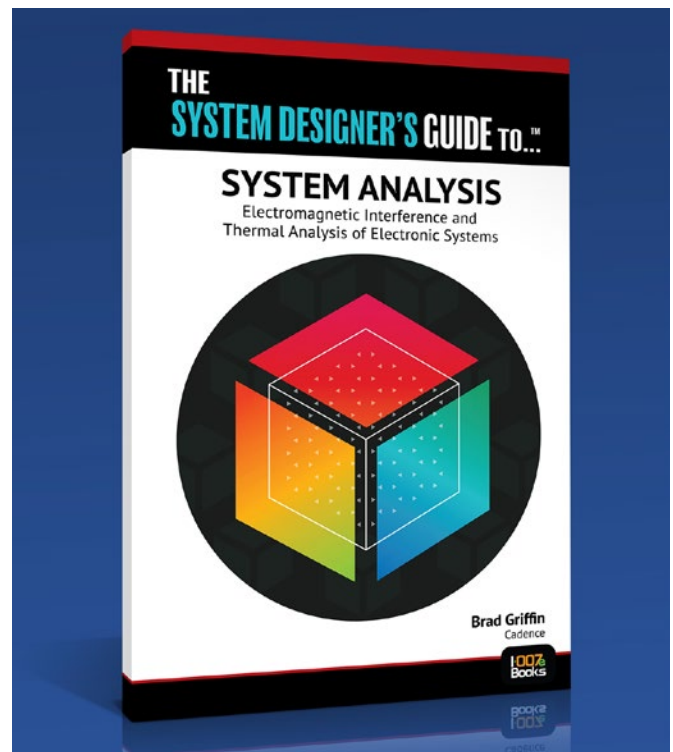
In this latest title from I-007eBooks, readers will learn how system-level analysis of complex and high-speed electronic designs is critical to solve electromagnetic, electrothermal, and electromechanical simulation challenges and to ensure that the system works under wide-ranging operating conditions.

According to signal integrity expert Lee Ritchey, founder and president of Speeding Edge and author of *Right the First Time: A Practical Handbook on High-Speed PCB and System Design*: "Author Brad Griffin offers readers a very good explanation of system-level analysis of complex and high-speed electronic designs. I highly recommend this book for those who are getting started with the design of PCBs and systems that are used in data centers."

Download your free copy today!

You can also view other titles in our *full library*. *The System Designer's Guide to...* series is specifically dedicated to educating the printed circuit and systems design sectors and serves as a valuable resource for people seeking the most relevant information available.

We hope you enjoy *The System Designer's Guide to... System Analysis: Electromagnetic Interference and Thermal Analysis of Electronic Systems*.



For more information, contact:

Barb Hockaday

I-Connect007 eBooks

barb@iconnect007.com

+1-916-365-1727 (GMT-7)

ASK THE EXPERTS

YOUR BURNING QUESTIONS... ANSWERED

Q *The IPC reliability committee has been studying the “weak microvia” problem in defense PCBs. They recommend using staggered microvias vs. stacked as a temporary workaround. What do you think is causing this defect?*

Carl Schattke: CTE mismatch is the root cause of these defects. Metal expands at a different rate than fiberglass and resin. A shorter via stack will have less contraction and expansion than a longer one. Here are a few more reasons to use staggered vias: There is a far wider pool of vendors that can make those boards. Also, it's less risky, as the staggered via processing does not rely on the same sintering process as

the stacked via, so it is generally considered a bit more reliable. Results may vary.

Lee Ritchey: The so-called “weak stacked via” is caused by the expansion of the resin in a PCB during soldering. It breaks the bond between the bottom of a via and the copper layer below it. It is a very real problem and is only solved by not stacking vias. **DESIGN007**

Q *Where can we buy capacitors with less than 60 weeks lead time?*

Heidi Barnes: It might be time to get out the EM simulation software and start looking at reducing capacitors on your power delivery networks. This would also help with finding alternative less popular values with shorter lead times.

Lee Ritchey: It depends on the capacitor type and size. When I select capacitors, I start with what is the most abundant in the supply chain and make my design work with them.

Carl Schattke: Some distributors charge higher prices, so they maintain some stock. Part of the engineering process of choosing parts is to look at lead times and availability for those parts. The cost of ignoring this, is potentially going to be some redesign effort later to address it. You are certainly not alone in having sourcing issues for critical parts. **DESIGN007**

ASK THE EXPERTS

YOUR BURNING QUESTIONS... ANSWERED

Q *Our company's senior designers and EEs, including me, are heading for retirement and pickleball. How can we attract new/young people into PCB design?*

Heidi Barnes: Give young people the tools and resources they need to be creative and push them to do something new and innovative. Access to a variety of EE CAD tools and a well-equipped lab can be a game changer.

Rick Hartley: Most future designers will be EEs who do their own layout. Everyone I have talked to who is both a circuit engineer and a PC board designer has told me that PCB design (except in the case of very simple boards) is the harder of the two disciplines. That being the case, maybe companies need to get off the dime and offer a much higher salary to engineers who are willing to actively take on both disciplines. The days of "designers" who are not degreed EEs are fading fast. The future designer will be an EE and PCB designer in one package. Those folks deserve a higher wage.

Eric Bogatin: By funding scholarships for engineers who want to study PCB and high-speed digital design.

Cherie Litson: Go to your local colleges and the continuing education department. Get involved. I'm partially retired and I love teaching basic electronics and doing special presentations on DFM to those who want to improve their career options. Some of these people will eventually become PCB design engineers.

Schools love this. Be on an advisory board. You get great perks and only have to attend one to four meetings a year.

Lee Ritchey: Attracting people into PCB design is a very tough sell. There are too many other offers out there that promise big pay and stock options.

Stephen Chavez: Let's face it: Major universities today are in the business of making money and that includes getting research grants both from the private and government sectors. PCB design education alone is simply not a big enough draw for such funding. Those of us who have been in the industry for several decades remember when a career in PCB design came with very little respect. That mentality had gotten us where we are today. The industry is lacking the next generation of engineers for printed circuit engineering. Just look how many job postings there are nowadays. There are more design jobs open than I have ever seen in my career.

Today's PCB designer is not like yesteryear's "draftsman." Today, that individual is so much more and requires an engineer-level education, and a broader understanding of design, fabrication, and assembly—along with the unique skillset of PCB design. A career in PCB design can be both fulfilling and lucrative, dependent

on one's education, skill set, and experience.

My advice is to simply get involved within industry associations such as the Printed Circuit Engineering Association (PCEA), IPC, SMTA, IEEE, etc. I'd even go one step further; I believe senior-level engineers and designers need to step up and volunteer some time in local STEM programs at local colleges or high schools. Because most major universities lack PCB design curriculum or refuse to add it to their existing EE curriculum, the industry is now paying for it and will continue to pay for it as designers head for retirement. The younger generation simply does not know what they don't know, and a career in PCB design is one of those things they simply are not aware of.

IPC is heavily involved in many STEM programs throughout the industry. PCEA, a newer organization, has made "Collaborate, Educate, and Inspire" the core of its existence. PCEA has had its printed circuit engineering curriculum and certification program established within the industry, and in several major universities to date. PCEA hopes to get this curriculum implemented worldwide in every major col-

lege and university within the respective EE curriculum.

Finding those next generation PCB designers will be a challenge, especially if major universities and colleges will not implement PCB design as part of their engineering curriculum. So, I feel it's up to each of us senior-level engineers and designers to do our part to pass on this knowledge to the younger generations any way we can.

Carl Schattke: Pay more. Actually, it's not so much about that as it is about inspiring young people with the joy of solving hard problems that bring useful products to the marketplace. Few people get to go home at the end of the day and say, "I did that." It's one of the key reasons that engineers are some of the hardest working of all professions, but also one of the happiest of all professions. My core belief is that the more we help others, the more rewarded we are. Engineers can really help a lot of people because we design it once and many can use it, and many can profit from it. **DESIGN007**

Intelligent Carpet Gives Insight into Human Poses

The sentient Magic Carpet from Aladdin might have a new competitor. While it can't fly or speak, a new tactile sensing carpet from MIT's Computer Science and Artificial Intelligence Laboratory (CSAIL) can estimate human poses without using cameras, in a step toward improving self-powered personalized health care, smart homes, and gaming.

Many of our daily activities involve physical contact with the ground: walking, exercising, or resting. These embedded interactions contain a wealth of information that help us better understand people's movements.

Previous research has leveraged use of single RGB cameras, (think Microsoft Kinect), wearable omnidirectional cameras, and even plain old off-the-shelf webcams, but with the inevitable byproducts of camera occlusions and privacy concerns.

To infer the 3D pose, a person would simply have to get on the

carpet, perform an action, and then the team's deep neural network, using just the tactile information, could determine if the person was doing situps, stretching, or doing another action.

"You could envision using the carpet for workout purposes. Based solely on tactile information, it can recognize the activity, count the number of reps, and calculate the amount of burned calories," says MIT CSAIL PhD student Yunzhu Li, a co-author on the paper.

Since much of the pressure distributions were prompted by movement of the lower body and torso, that information was more accurate than the upper-body data. Also, the model was unable to predict poses without more explicit floor contact, like free-floating legs during situps, or a twisted torso while standing up.

(Source: MIT)



ASK THE EXPERTS

YOUR BURNING QUESTIONS... ANSWERED

Q *When a datasheet calls out an area of copper for heat dissipation, how do I interpret that requirement for my board?*

Cherie Litson: Look for the watts, or calculate it yourself—remember Ohm’s law and Watt’s Law? What kind of heat do you need to get rid of? How much copper do you have on your board? Then go to the online calculators (Saturn PCB Toolkit comes to mind) and crunch the numbers. If you still have questions, call the component manufacturer and talk with the electrical engineers there. Figure 1 is a great chart for you to keep around.

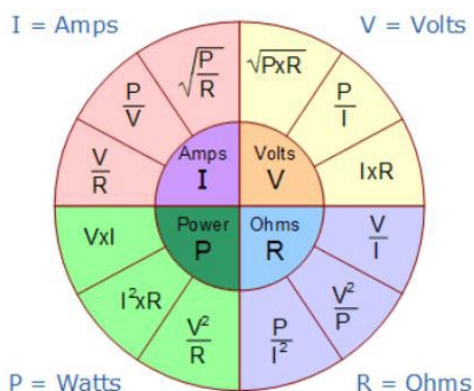


Figure 1: This handy chart allows users to calculate watts, volts, amps or ohms.

Carl Schattke: The first thing I want to know is how many watts the part is going to need to dissipate. Then, how am I going to be able to do that? A copper land area is not the only way to get heat away from a part. What’s the environment? We can get heat away with convection and conduction, or some combination of that. If you do not have thermal modeling tools available, the lowest risk is going to be following the guidelines on the data sheet. Usually, you need less area than what is called out on a datasheet,

but you may have a better method of dissipating the heat than they do. You may have a large ground plane, or some structure that acts as a conductor away from the part, or some active cooling components that draw heat away.

Lee Ritchey: Deciding on how much copper area is needed for heat dissipation requires a good thermal modeling tool. Any other approach is simply guessing.

Chris Young: I suggest reading the following application report from Texas Instruments: *AN-1520A Guide to Board Layout for Best Thermal Resistance for Exposed Packages*. This report details an approach to making low thermal resistance connections to copper regions within a PCB. A datasheet will typically give you a copper area and a copper weight. Rarely will datasheets give details about the environment where the test/characterization measurements were conducted. This leaves you in a bind because there is not enough data present to work your way back to any specific and “safe” requirements for your board. I have found that many IC manufacturers will do whatever is possible to show the best possible performance of the part in the datasheet. I have also found that this “whatever is possible” is not a reasonable approach to manufacturable designs. The most reasonable course of action is to build a prototype and take measurements in the environment(s) you expect your design to be operating in. DESIGN007



Meetings & Courses

January 22-27

Conference & Exhibition

January 25-27

San Diego Convention Center | CA

Achieve **Digital Transcendence**

Get ready to transcend the norm as you transform through education, innovation and collaboration at North America's largest gathering of electronics industry professionals.

See you in San Diego at IPC APEX EXPO 2022!

[IPCAPEXEXPO.ORG](https://ipcapepexpo.org) | [#IPCAPEXEXPO](https://twitter.com/IPCAPEXEXPO)

Thank you to all who have contributed to creating and supporting an excellent IPC APEX EXPO 2021 experience.



PCB007 Highlights



Punching Out! How to Stand Out Among Other Sellers ►

It is a very busy time in the M&A world. Many owners in the electronics industry, especially in the PCB and EMS sectors, are of retirement age. Meanwhile, there is a lot of interest from both competitors and private equity firms in making acquisitions in the space. Because there are a lot of deals happening, it is important to know how to stand out compared to other sellers.

Summit Interconnect Acquires Eagle Electronics ►

Summit Interconnect, Inc. is pleased to announce the acquisition of Eagle Electronics, Inc. Established in 1979 and located in Schaumburg, Illinois, Eagle is one of North America's leading providers of advanced prototype printed circuit boards.

Catching up With Kusu's Pascal Delloue ►

There are other countries in Asia besides China and Pascal Delloue intends to promote them. He has many years of experience in the global marketplace and his new company, Kusu Corporation, is poised to introduce SE Asian companies and the electronics products they can provide for the rest of the world.

Trouble in Your Tank: Training Your Team and Tools for Success ►

Columnists were asked to consider subjects such as optimizing business processes and strategy, process optimization, and training your team. From my view, developing critical thinking skills will help engineers trouble-

shoot technical issues and bring the issue to quick resolution, as this is certainly a good lead in training your team.

The PCB Norsemen: Building an Ability to Expect the Unexpected ►

How do you find techniques and strategies to transform a business plan developed in the boardroom, into a living strategy implemented into the company core and mindset of every colleague? And how do you do it when times are hard, as we experienced in 2020?

Atotech, Nabet India Launch COVID-19 Awareness Campaign 'Atotech Cares' ►

Atotech, a leading specialty chemicals technology company and a market leader in advanced electroplating solutions, is joining together with National Association for the Blind (Employment & Training) (NABET INDIA), a non-governmental organization (NGO), based in Gurgaon, India, in order to increase the impact of our COVID-19 relief activities.

Increasing Productivity Through Training ►

Too often when we think of training our manufacturing workforce, we focus on the shop floor. We look for opportunities mainly with those who are doing the making. We need to cast a wider gaze as the manufacturing process is not limited to the manufacturing floor. Quality, engineering, sales and marketing, logistics, and even accounting/finance are all part of the process that starts with a business plan and culminates with your trinket in the hands of your happy customer. All departments make up that "well-oiled machine."

*Welcome to the Summit
Interconnect Family!*

WELCOME TO THE SUMMIT

SUMMIT ACQUIRES EAGLE ELECTRONICS

READ THE PRESS RELEASE HERE



summit-pcb.com

eagle-elec.com

Ask The Experts— PDN Filters

Quiet Power

Feature Column by Istvan Novak, SAMTEC

In recent years I have been getting many questions about PDN filters from my course participants and from friends, colleagues, and even from strangers. Long gone are the days when the essence of power distribution design recommendation was “place a 0.1 μF bypass capacitor next to each power pin.” Power distribution networks used to primarily contain wires, traces, planes, and parallel bypass capacitors, but very few had systematically designed filters. Just for clarifying the nomenclature, Figure 1 defines what I call a parallel PDN and a PDN filter. For the purposes of this article, the differentiation between a parallel PDN and a PDN filter is the intentionally included series resistive and/or inductive element in the PDN filter.

The figure shows a very simple implementation of a PDN filter. In actual circuits we may have much more complicated circuits. We may have higher order filters with cascaded sections, filter sections connected in branching topologies, and we may also have filters that have to filter in “reverse direction,” preventing the noise from spilling out from the load connected to the output of the filter. Or maybe we need filtering in both ways.

In filter circuits like the ones in Figure 1, the usual question I get is: How do you describe the filter? Should we use transfer impedance (Z_{21}) or scattering transfer parameter (S_{21})? As I will show, in a lot of applications, none of these two would serve us well, we will need something else: a voltage transfer ratio.

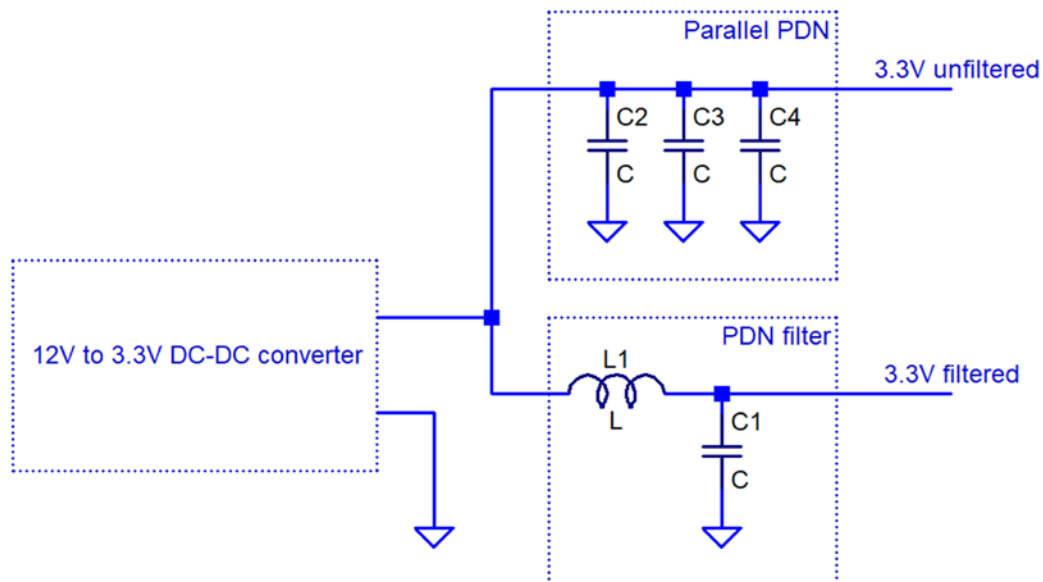


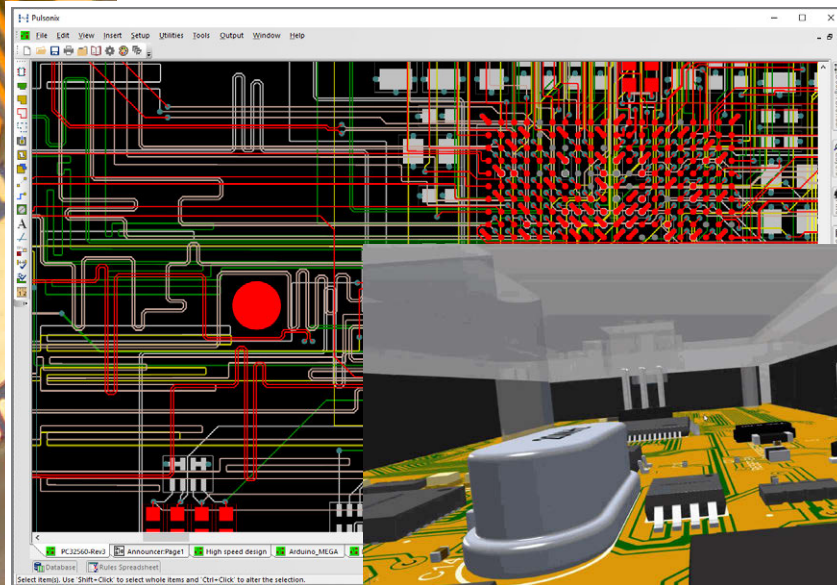
Figure 1: Definition of PDN filters.

Pulsonix PCB Design Software

PCB design done right

New Version 11.0 Increases Speed up to 80%

Multi-threading and GPU processing in this new release create a gigantic leap in speed, dramatically reducing the time it takes to perform many PCB design operations, such as Design Rule Checking (DRC), Copper Pouring, and Gerber generation. That's PCB done right!



Advanced PCB Design Features

- Intuitive placement and routing
- Dynamic copper pour
- Full rules engine for automatic DRC
- High-speed design rules
- Heads-up display length matching
- Rigid-Flex, embedded components, and chip-on-board
- Revision control and PLM integration

Try Pulsonix yourself at pulsonix.com/trial



To systematically design a PDN filter, first we must establish the requirements. In PDN design work, the biggest pain for a board designer is that input requirements that would guide the designs are very rare, many times almost non-existent. Establishing PDN requirements is often left to the PDN and board designer. There is one situation, though, when the board designer should be in the position to establish the filter requirements properly: when we need to attenuate the output ripple of a switching regulator for a sensitive pin of a chip that has a specified maximum allowed noise. To quote some simple numbers, the typical peak-to-peak regulator ripple may be around 10 mVpp, which is usually too much for a sensitive analog circuit, such as a reference clock or clock buffer circuit, PLL, SerDes supply, or sensitive analog circuit, more likely requiring around 1 mVpp maximum noise. In such cases, relying on the allowed ripple voltage limit of our sensitive circuit and having an estimate of the regulator's switching frequency and output ripple, we can determine an attenuation requirement, at least at the switching frequency of the regulator.

As an illustration, let's take a filter circuit that is supposed to attenuate 10 times the switching ripple of a DC-DC converter running at 500 kHz for an oscillator circuit that takes just a

few milliamperes of current. In addition to the series inductive element and the output capacitor, the circuit in Figure 2 also includes components that model the source side of the filter. R1 and L1 represent the impedance of the DC source, and C2 and C3 with their parasitics model the board capacitors. The L5-R5 circuit is the model for a small inductor. As opposed to a ferrite bead, where we would anticipate a large increase of series resistance and a substantial drop of inductance at high frequencies, this simple inductor model has frequency-independent series resistance and inductance.

The filter output is a single capacitor, modeled by C4-R4-L4. Note that this model intentionally has a very high ESR for C4 that we can get either from a small electrolytic capacitor (though it will likely have a much higher ESL) or we can use a regular ceramic capacitor with low ESR and add a 0.91-ohm series resistor. We also knowingly ignore the printed circuit board details and therefore we limit the AC simulation to below 100 MHz. The filter behavior at higher frequencies will be influenced and probably dominated by component placement, the PCB layout and stackup—something that is beyond the scope of this article.

We simulate the filter circuit in the frequency domain in the 100 Hz to 100 MHz frequency

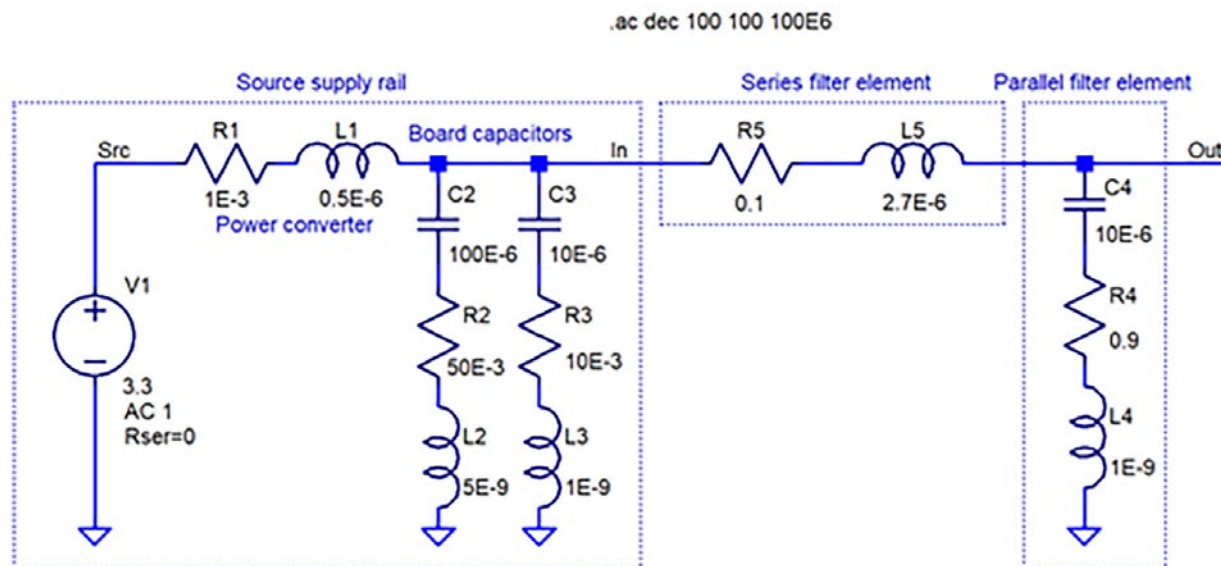


Figure 2: Schematics of a simple one-stage PDN filter.

range with a 1V ideal AC source with zero source resistance behind the R1-L1 elements modeling the DC source. Figure 3 shows the simulated voltages at various nodes. The V(src) source voltage shows a flat line at 0 dB, since this is the constant source voltage we enforce. V(in) and V(out) are the input and output voltages of the filter. And finally, V(out)/V(in) is the voltage transfer ratio from input to output. Note that while V(in) and V(out) both have some minor peaking around 20 kHz, the light-blue line of V(out)/V(in) shows no peaking, but it also has just 20 dB/decade slope, whereas the absolute output voltage drops more steeply.

The various impedances can be simulated by attaching a 1A AC current source to the pieces, as shown in Figure 4. The AC simulation uses linearized models and therefore the 1A AC test current that may otherwise damage some of the real components, can be used here with no concern.

In Figure 5 we see that the minor peaking is related to the impedance of the main supply rail that serves as a source and provides input to the filter. The impedance of the series inductor starts out at the 0.1-ohm DC resistance and at about 5 kHz it goes inductive. Due to the large series resistance, the impedance of the output capacitor of the filter becomes resistive

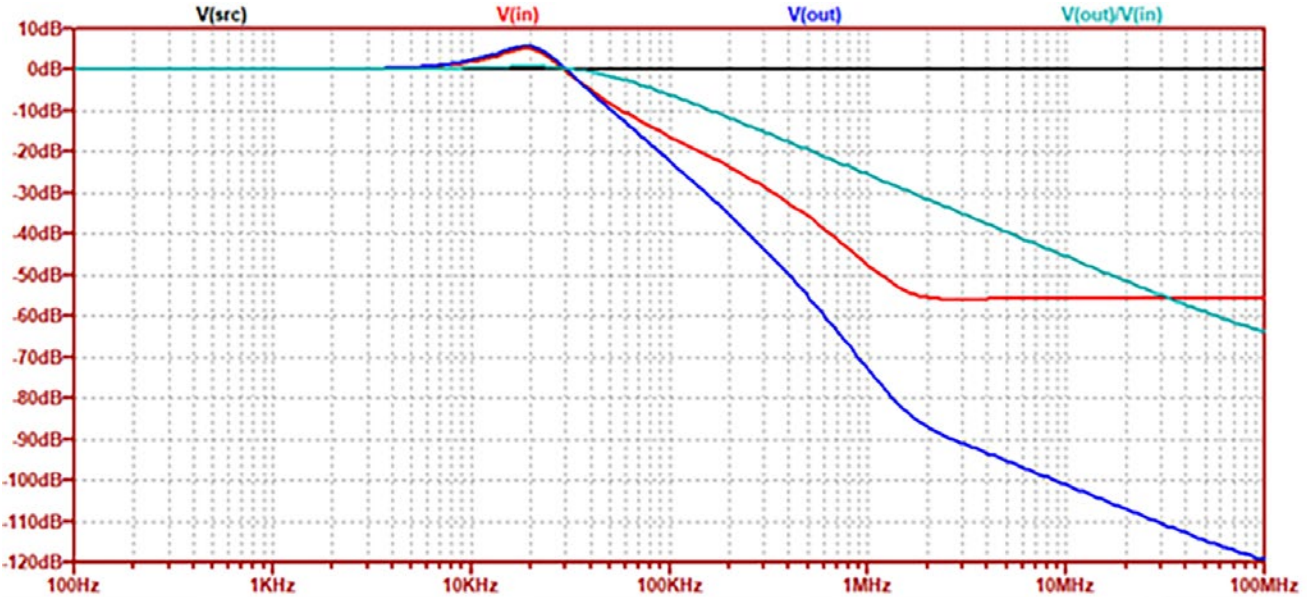


Figure 3: AC simulation results of the filter from Figure 2.

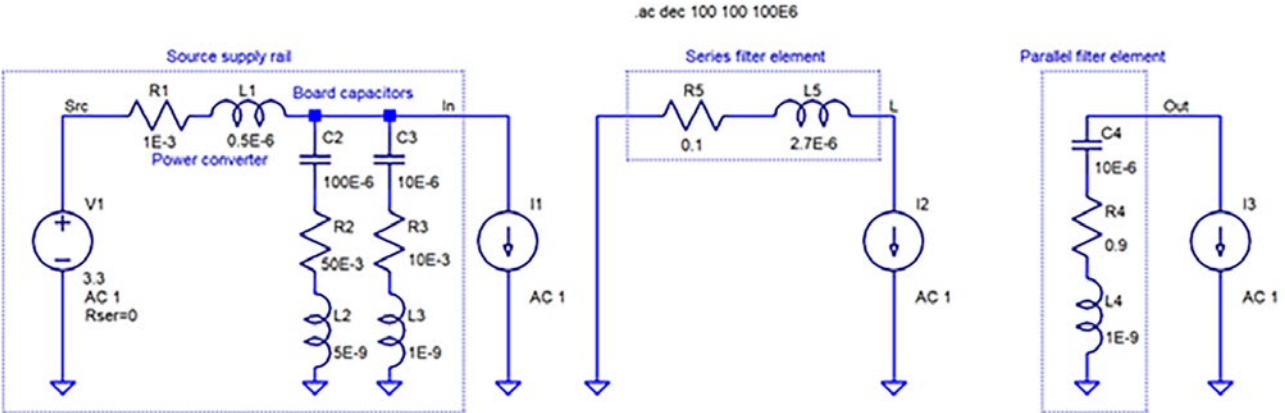


Figure 4: Simulating the impedance of blocks in the PDN filter.

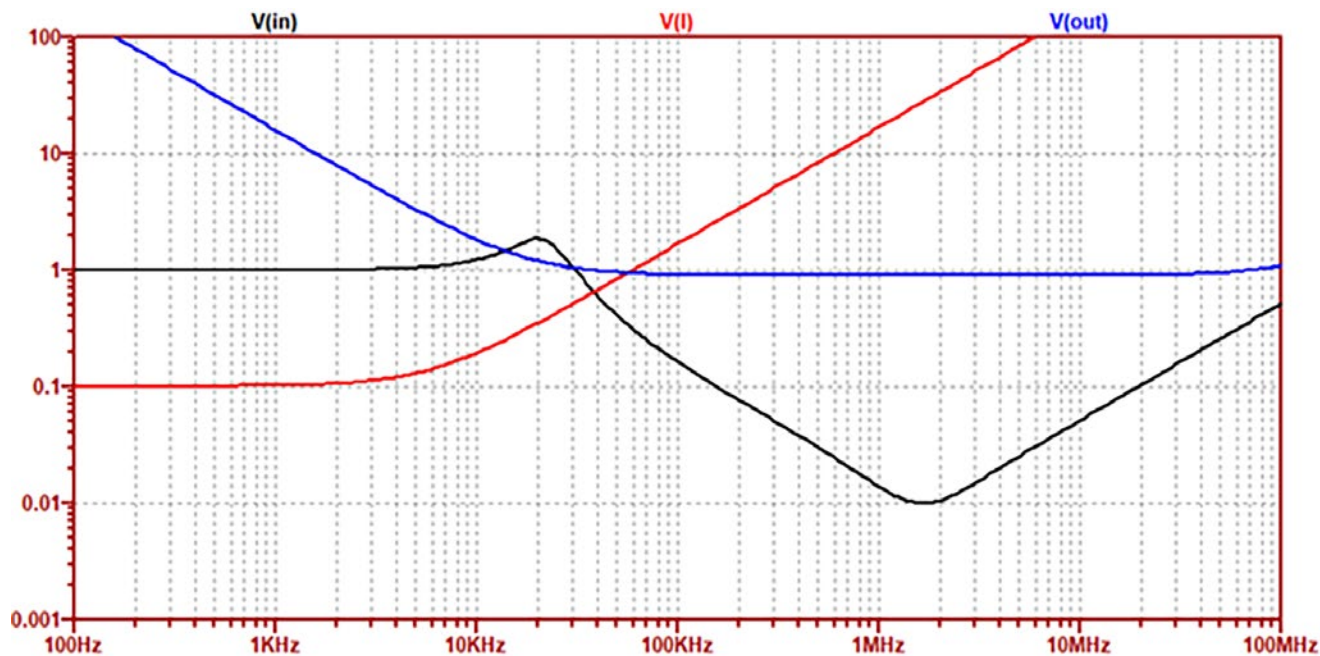


Figure 5: Impedance magnitude of the source rail, the inductor, and the output capacitor of the filter. Because we used a 1A AC source, the voltages at these nodes directly represent impedance.

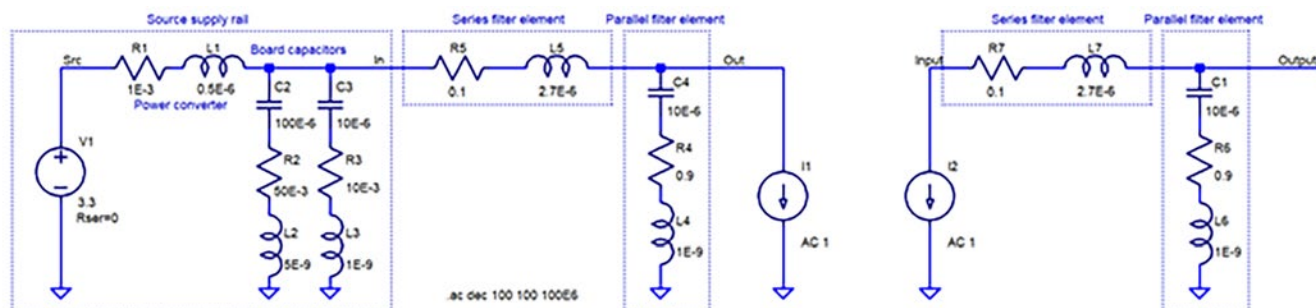


Figure 6: Simulating the full input and output impedance of the filter.

at about the same frequency where the source impedance has the peaking.

We can also simulate the input and output impedance of the full filter with its actual terminations. As shown in Figure 6, we can simulate the output impedance of the filter with the source-side impedance connected.

In the reverse direction, the input impedance is simulated with the assumption that the load has very high impedance and therefore we can leave the output of the filter open. On the input side of the filter, we drive the series inductive element without a shunt capacitor included. This is an acceptable approximation

for unidirectional filters when the impedance of the source rail is much lower than the input impedance of our filter. When we need to work with multi-stage and/or bi-directional filters, where the driving source impedance is not much lower than the input impedance of our filter, we need to identify and include, both at the input and at the output, the capacitors that belong to our filter circuit.

The input and output impedance curves of the full filter are shown in Figure 7. The input impedance curve compares to the blue line in Figure 5 with the difference that now we have a 2.7 uH inductance in series. This creates an

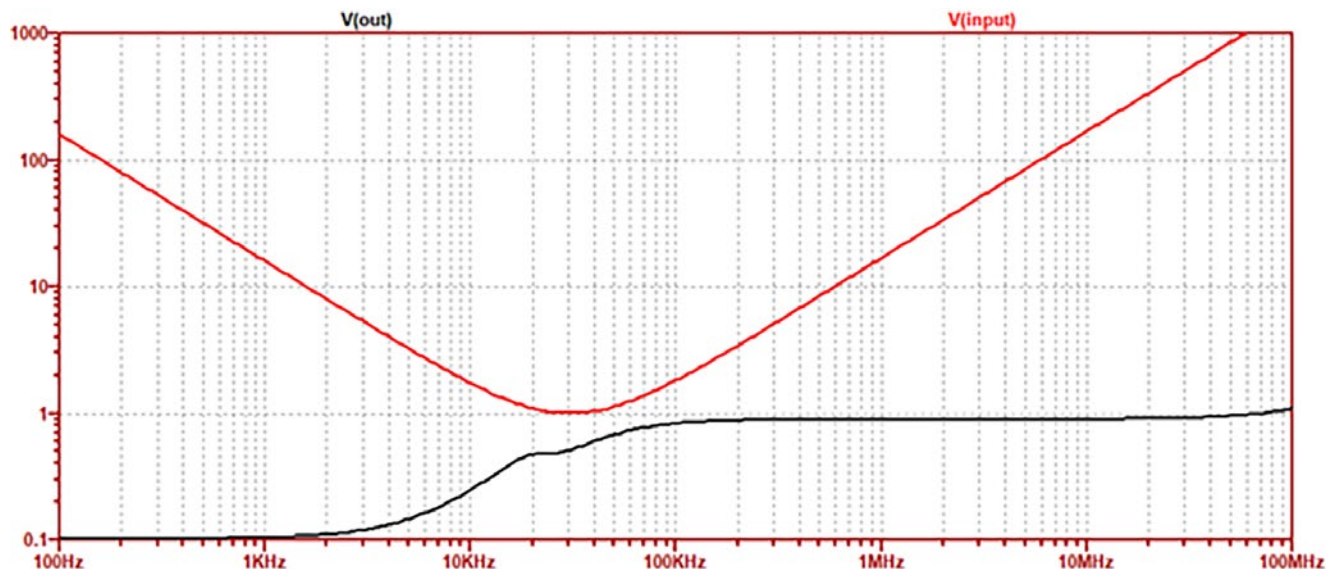


Figure 7: Input and output impedance of the full filter.

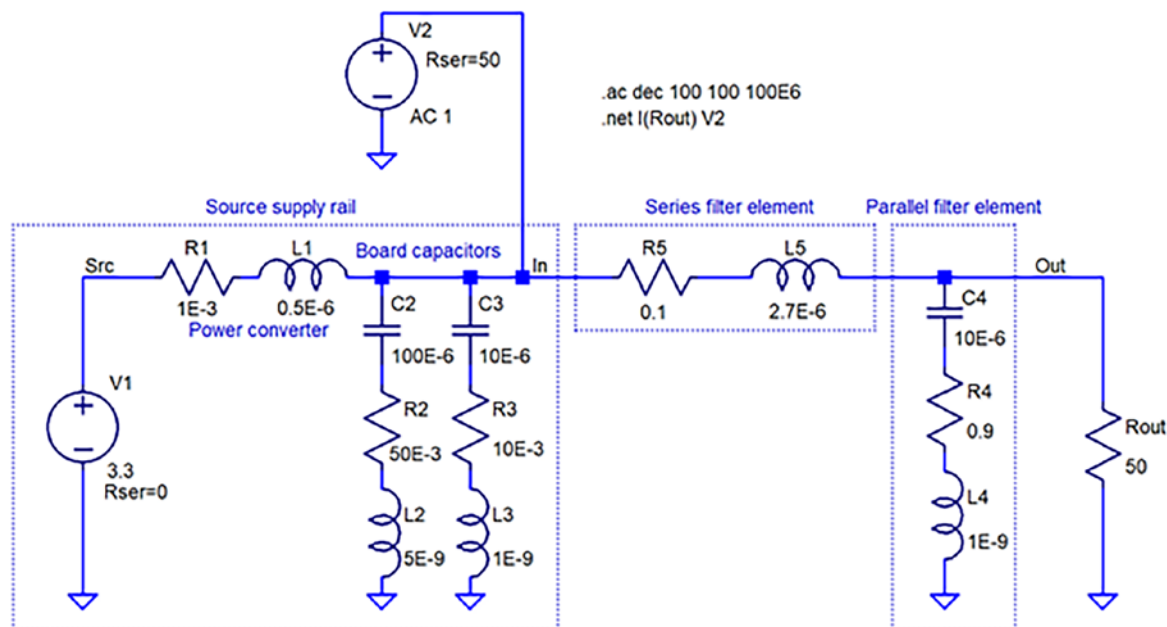


Figure 8: Simulation deck for network parameters.

impedance profile as if the capacitor on the output had a 2.7 uH ESL, which effectively removes the flat impedance plateau that the 0.91-ohm ESR created. The output impedance of the filter above 100 kHz matches the 0.91-ohm ESR of the output capacitor. At lower frequencies, the output impedance approaches the 0.1-ohm DC resistance of the inductor.

We can also simulate the network parameters that we may otherwise consider using

to describe the filter: Z21 and S21. To do this, we need to modify the simulation circuit by specifying the ports, the termination impedance and add one line that links the input and output: “.net I(Rout) V2.” The SPICE deck is shown in Figure 8, the result is in Figure 9. The three curves in the plot clearly illustrate the difference among them and show us why in our chosen case the voltage transfer function is a better metric.

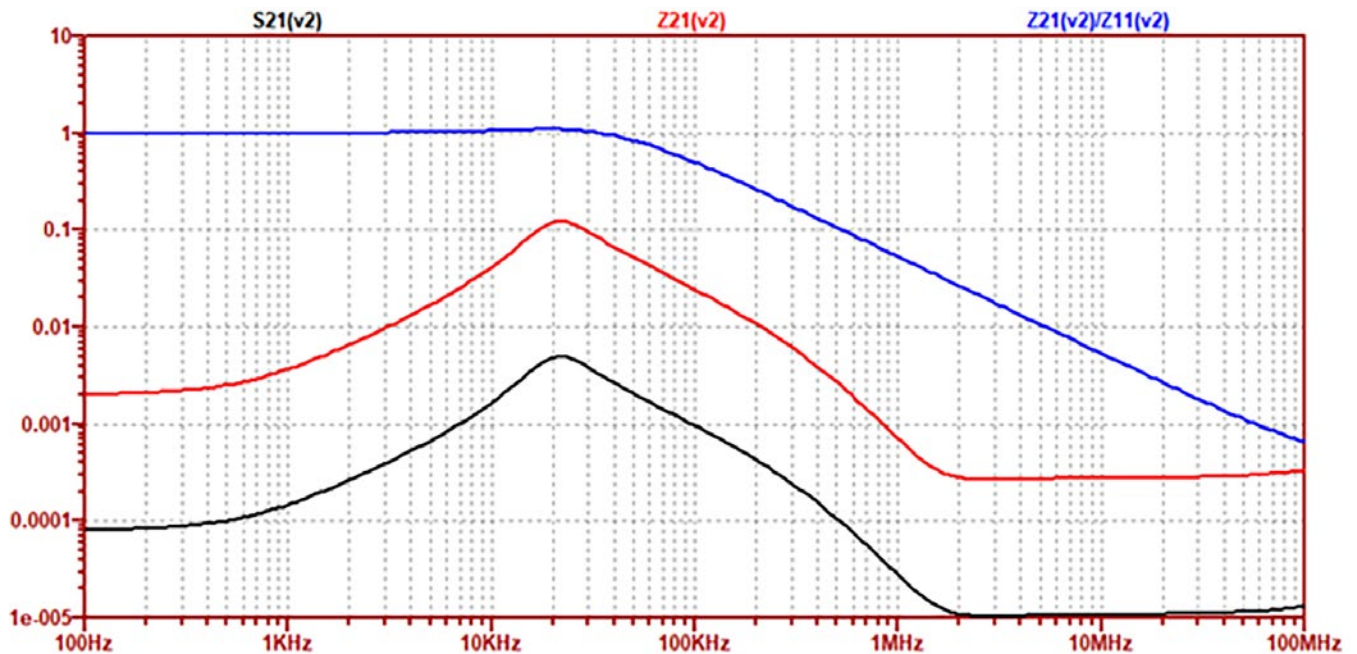


Figure 9: Z21, S21 and voltage transfer ratio.

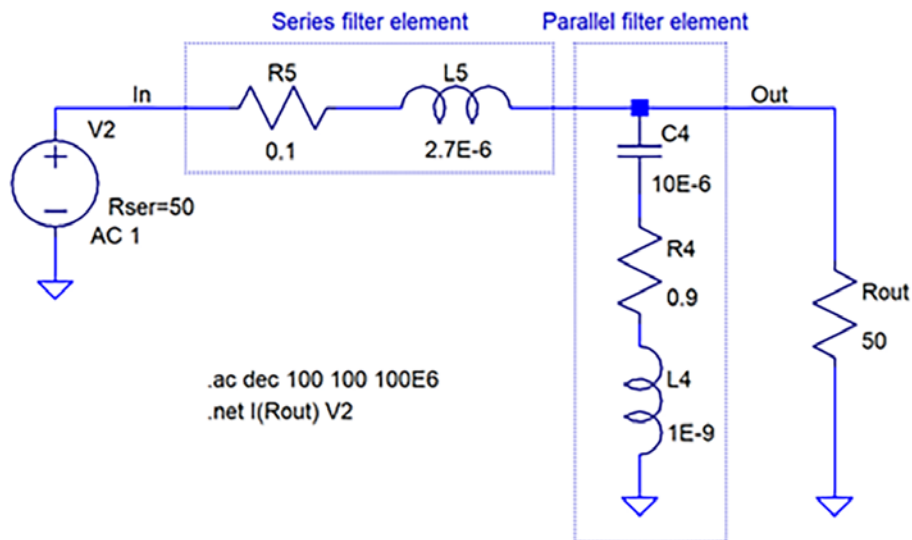


Figure 10: Simulation deck for filter transfer function without including the impedance of the input-side supply rail.

First, we can notice that the S21 and Z21 curves run in parallel. The vertical difference between them is 25x, which comes from the parallel equivalent of the two 50-ohm terminations we need to calculate S21. We also notice that both curves start at very low values at 100 Hz; this may seem to be good news, but in reality, it is misleading. In this test we assume that the input side of the filter is driven by a high-

current low-impedance supply rail. We get a low S21 or Z21 value simply because the low source-side impedance shunts out our test signal. In reality, whatever noise appears across the source rail will be forced across the input of the filter, which represents a much higher impedance and as such it has very little influence on the input noise.

When we calculate the noise transfer with

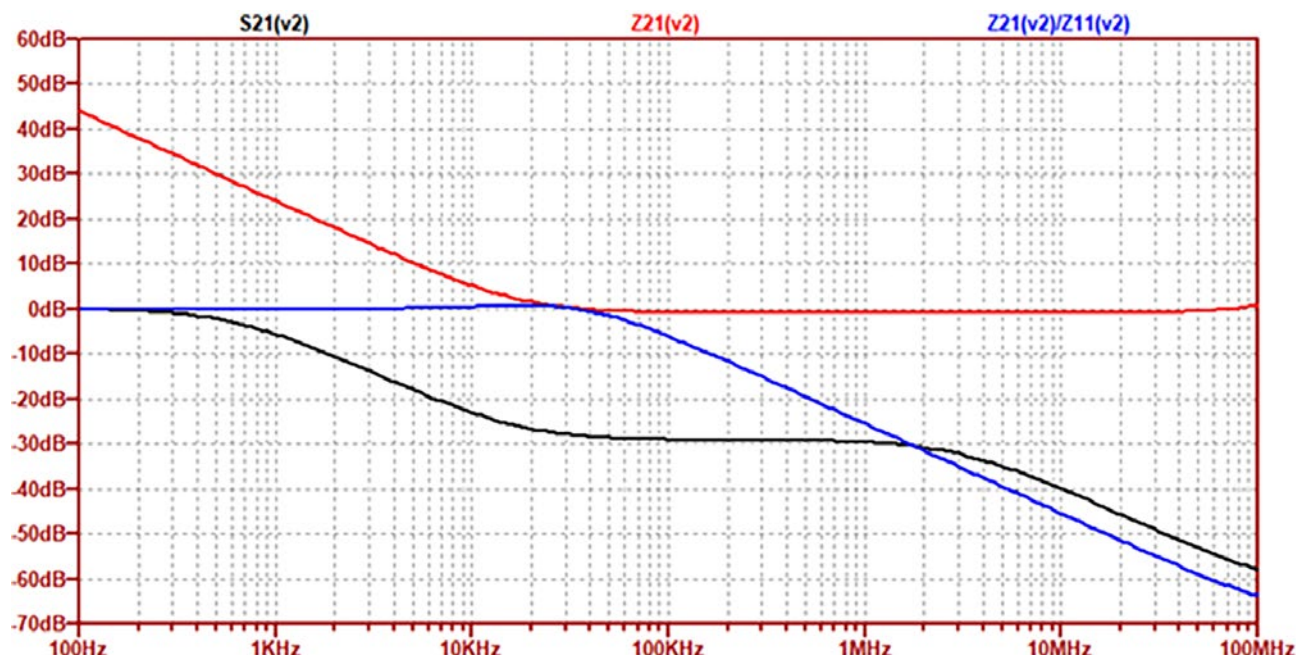


Figure 11: Transfer parameters for the circuit of Figure 10.

the source rail's impedance in place, the finite source impedance associated with S21 and the infinite source impedance associated with the current source for Z21 will produce this large attenuation at low frequencies where the low impedance of the DC source dominates the impedance of the source rail. The other extreme case, when we leave out the source impedance altogether, would produce equally unrealistic results. We show this in Figures 10 and 11.

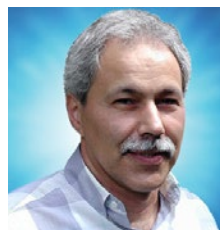
S21 starts out at 0 dB at 100 Hz but then it drops unrealistically due to the 50-ohm source impedance, which would be very unexpected from a high-current rail impedance. Z21, on the other hand, starts with a large gain and then it settles at the ESR of C4. Figures 9 and 11 illustrate that both S21 and Z21 hugely depend on the impedance of the source rail, whereas the voltage transfer ratio, by its definition, does not.

We need to remember that details matter; the illustration and conclusions are valid for the stated category of cases when we connect a filter to a main rail, which represents a source-side impedance much lower than the load impedance of the filter output. If

you are interested in further details, want to learn about filter measurements and are curious about correlations between measurement and simulation, check out the further reading listed below. **DESIGN007**

Resources

1. "Frequency-Domain Characterization of Power Distribution Networks," Artech House, 2007, Section 5.4.
2. "Using Ferrites and Inductors in Power Distribution Networks (PDN)," Samtec gEEK spEEk Webinar, December 3, 2020.
3. "Do You Really Need that Ferrite Bead in the PDN?" by Istvan Novak, *Design007 Magazine*, June 2020.



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



MilAero007 Highlights



Survey: Reasons U.S. Electronics Manufacturers May Exit Defense Market ►

In a new IPC industry survey and report, one-quarter (24 percent) of electronic manufacturers say the costs and burdens of compliance with the Cybersecurity Maturity Model Certification (CMMC) may force them out of the U.S. Department of Defense's (DoD) supply chain.

Defense Speak Interpreted: The U.S. Has a Space Force—JEDI Knights Next? ►

Does the U.S. Department of Defense's JEDI contract mean it's going into a Star Wars production? Sorry, no Stephen Spielberg this time. Sorting out the good guys and bad guys in this cloud computing scenario.

EIPC Technical Snapshot Review: Microvia Reliability Issues ►

Since the mid-1990s, when they were developed for mass production in the mobile phone industry, microvias have become principal enablers for high-density designs, and have evolved from single-level to complex stacked and staggered structures. They are fundamentally robust interconnects, although some aspects of their reliability are still under investigation.

Fortify, Rogers to Develop 3D Printed Dielectric Material Systems for RF Devices ►

Fortify, a Boston-based 3D printing startup, and Rogers Corporation, a global leader in engineered materials for advanced connectivity and power electronics, announced their

partnership to enable additive manufacturing of low-loss dielectric materials for radio frequency (RF) devices and electronics.

Industry CEOs Urge Action to Improve Electronics Manufacturing Ecosystem ►

More than 50 CEOs urged Commerce Secretary Gina Raimondo to take concrete steps to address challenges confronting the entire U.S. electronics supply chain.

The Government Circuit: Advocating for the Entire Electronics Supply Chain ►

Here in Washington, we are encouraged by recent policy discussions that indicate a bipartisan commitment to U.S. manufacturing that is long overdue. European officials are also promoting a policy agenda that could be very positive for electronics. But there is still more work to be done to bolster the entire electronics ecosystem.

Adventures in Engineering: The Ecosystem of Autonomous Flight ►

To be clear, autonomous flight is here, it has been done, the achievement has been made. Now, how do we create an ecosystem for this newly proven technology and how do we incorporate it into our lives?

Curtiss-Wright Selected by Lockheed Martin to Upgrade U.S. Navy Computers ►

Curtiss-Wright's technology leadership in Modular Open-Systems Approach-based rugged Commercial Off-The-Shelf modules tapped for U.S. Navy MH-60R/S Seahawk helicopter in response to cyber-attacks.

Design problems solved!



SUBSCRIBE NOW

FLEX007

A SPECIAL DESIGN007 MAGAZINE SECTION

Rigid-Flex Design Without Respins—A Webinar Review

Flex007 Feature by Andy Shaughnessy

I-CONNECT007

Rigid-flex circuits have become almost ubiquitous over the past decade; most personal electronic devices contain at least one rigid-flex circuit. But there's a downside to rigid-flex: Respins are almost a given with rigid-flex designs. And rigid-flex re-spins can be quite expensive, since this technology costs quite a bit more, on average, than flex or rigid boards.

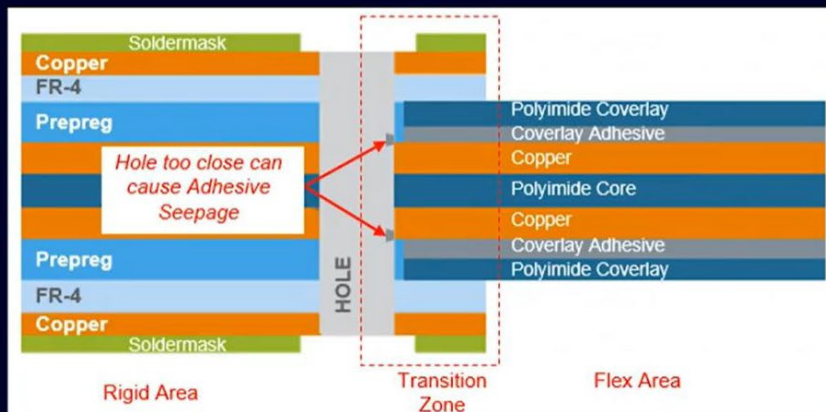
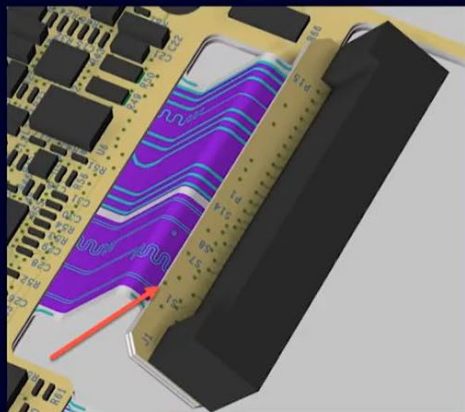
In this free 22-minute on-demand webinar, *Flex & Rigid-flex DFM Analysis*, Kevin Webb,

an NPI technical marketing engineer with Siemens EDA, explains how using Valor NPI early in your rigid-flex design flow can help you reduce your rigid-flex re-spins, or possibly even eliminate them completely.

Webb discusses how Siemens' drive to "left-shift" manufacturing information into the designer's hands allows rigid-flex designers to make more informed decisions early in the design cycle, where mistakes are less costly. Webb walks the user through a typical rigid-flex design, explaining how Valor NPI is optimized to work with rigid-flex circuits. The

Review and Release Process with Valor NPI

Holes too close to an Interface Area can cause adhesive seepage and can lead to stress on the plated hole barrel resulting in barrel cracks.



ENEPIG *UPGRADED.*

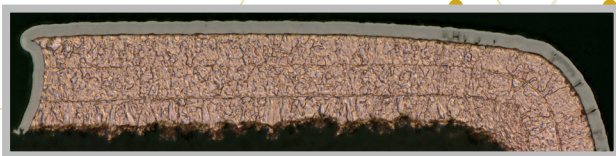
AFFINITY 3.0

HYBRID GOLD TECHNOLOGY

Traditional ENEPIG is limited by gold thickness capability and underlayer corrosion due to total reliance on galvanic displacement of nickel to fund gold deposition.

Affinity Gold 3.0 is a Hybrid Gold System - balancing galvanic displacement and chemically reduced deposition of gold to produce ENEPIG deposits with higher deposit thickness capabilities and elimination of corrosion.

TRADITIONAL IMMERSION GOLD



AFFINITY GOLD 3.0

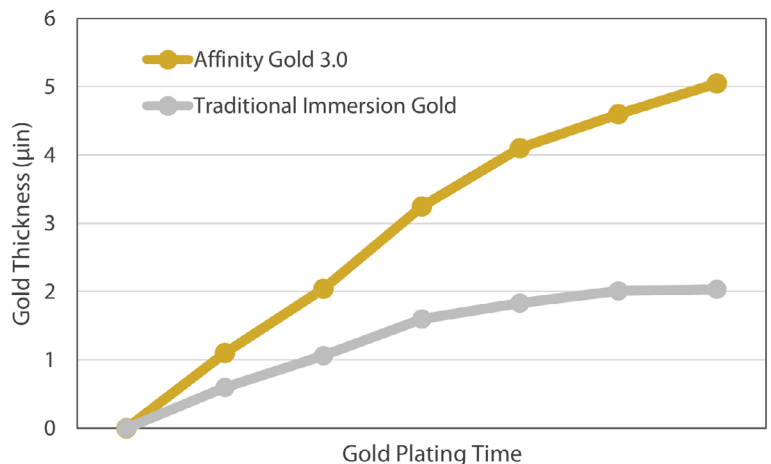


Enables an ENEPIG coating not limited by palladium and gold thickness

Relieves end-user concerns of corrosion from gold plating step

High chemical stability without the need for potassium cyanide replenishment

Gold Plating Rate Over Thick (20µin) Palladium



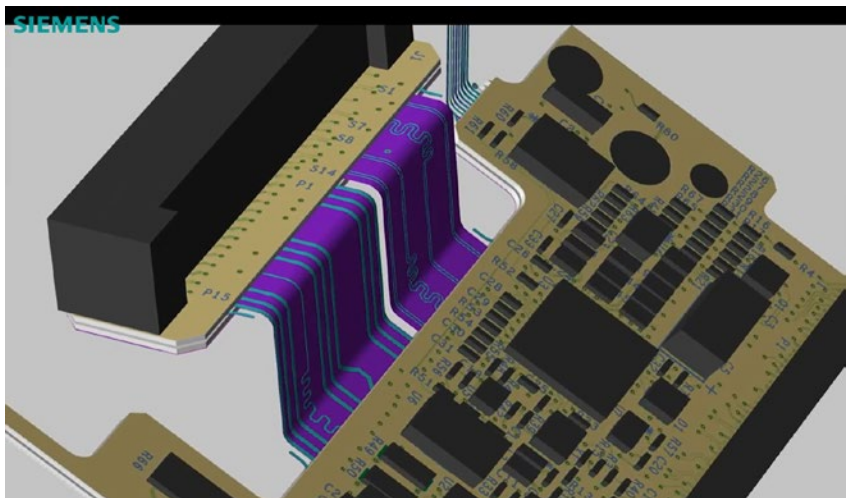
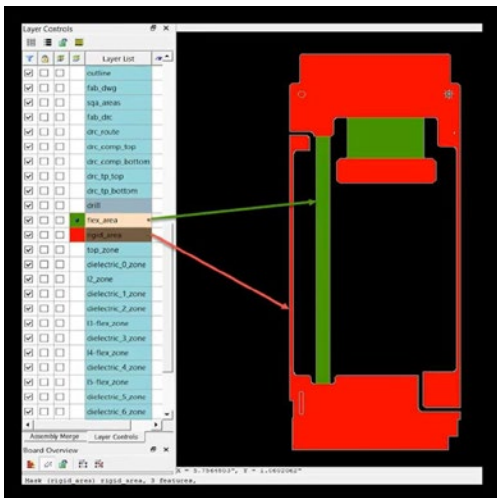


Figure 1: The webinar explains how Valor NPI detects errors in rigid-flex designs. The software includes 3D view, a handy feature for rigid-flex.

system helps the designer perform manufacturing checks that are unique to rigid-flex, such as overlapping traces that cause an undesirable “I-Beam effect,” or plated through-holes and vias in bend areas (keep-out areas), which could lead to barrel fracturing and open circuits. Webb also demonstrates how to use Valor’s Manufacturing Risk Analysis to identify potential problem areas in the circuit;

designers can set the tool to report all errors, or only errors in critical areas.

Webb manages to pack a lot of information into a webinar that rigid-flex designers can watch during their lunch break and still have time left over. If you design rigid-flex circuits, no matter what your skill level is, this is a must-watch webinar. **FLEX007**

[Access webinar here.](#)



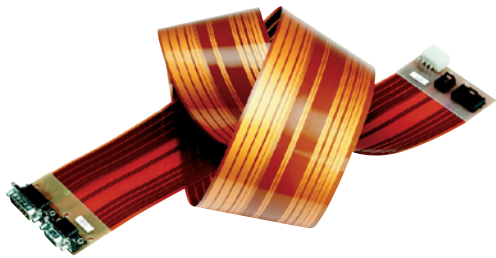
Hire or Be Hired at jobCONNECT007

Looking to make a job change, earn more money, or relocate? JobConnect007 is a fast-growing resource for industry professionals seeking new opportunities. Companies with names you will recognize are looking for you!

Looking for talent? I-Connect007’s Career Opportunities section is also great place to find industry-experienced talent for your organization.

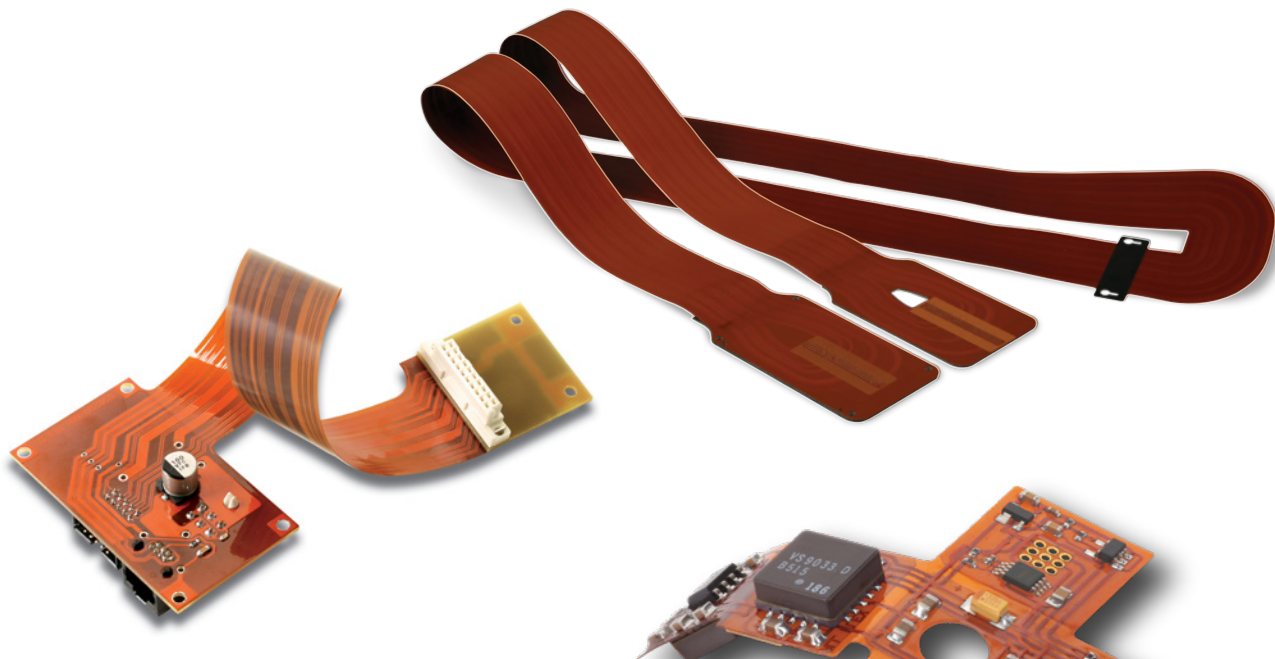
Download this month’s edition of the *jobConnect007 Career Guide* which has all our current listings in one spot for handy reference and for sharing with your friends and family.

Yes! I want one!

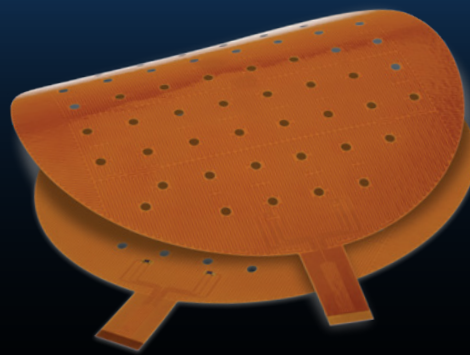


all flex

Flexible Circuits • Heaters • Assemblies



- Design application assistance
- Component assembly
- Fast response
- Quick delivery



All Flex Flexible Circuits & Heaters
1705 Cannon Lane
Northfield, MN 55057
(800) 959-0865
(507) 663-7162

allflexinc.com





Flex007 Highlights



Consider This: Designing Via-in-Pad for Higher Density Flexible Circuits ►

For flex circuit designs with high layer counts utilizing high-density outer layers, the extra area utilized for separate pads and SMT components severely limits the available space for trace fan out. By designing via-in-pad in flex and rigid flex circuits, it can significantly increase density, utilizing vias as mounting pads.

Fralock Holdings Acquires Lenthor ►

Fralock Holdings, LLC, a leading developer and manufacturer of engineered materials solutions for critical applications, announced the acquisition of Lenthor Engineering, Inc., one of the largest privately owned companies in North America that designs, manufactures and assembles flex and rigid-flex printed circuits. Fralock is a portfolio company of Arsenal Capital Partners.

Nan Ya PCB Posts 40% YoY Revenue Growth in May ►

Taiwan-based PCB manufacturer Nan Ya Printed Circuit Board Corp. posted sales of NT\$4.1 billion (\$148.1 million at \$1=NT\$27.68) in May 2021, slightly down by 0.4% from the previous month, but up by 40% from May of last year.

Flexible Thinking: Star Trek Memories ►

Columnist Joe Fjelstad not only watched *Star Trek* with fascination, he grew to become his own inventor, thanks to his father—an aerospace engineer. “The passion for flight, especially rocketry, entered my veins early,” he writes.

American Standard Circuits Acquires Orbotech Ultra Dimension 800 ►

West Chicago-based PCB fabricator American Standard Circuits has recently invested in Orbotech’s Ultra Dimension 800 4-in-1 AOI solution.

Flexium Posts 7% YoY Growth in May 2021 Sales ►

Taiwan-based flexible printed circuit (FPC) maker Flexium Interconnect Inc. has reported sales of NT\$2.4 billion (\$86.55 million at \$1:NT\$27.72) for May 2021, up by 7% compared to May last year, but almost flat from the previous month.

The Powerhouse Future Is Flexoelectric ►

Researchers have demonstrated “giant flexoelectricity” in soft elastomers that could improve robot movement range and make self-powered pacemakers a real possibility.

Future Washable Smart Clothes Powered by Wi-Fi Will Monitor Your Health ►

Purdue University engineers have developed a method to transform existing cloth items into battery-free wearables resistant to laundry. These smart clothes are powered wirelessly through a flexible, silk-based coil sewn on the textile.

Altix Receives Repeat Order for Direct Imaging Equipment ►

Xiamen Guangpu Electronics Co., Ltd., a leading Chinese manufacturer of LED lighting, photoelectric sensors, IoT based hardware and FPC products, has expressed its trust in Altix by ordering multiple direct imaging machines.

TAKING IT TO THE EXTREME

HIGH HEAT

BY TAIYO



We created our new liquid photoimageable extreme solder mask, PSR-4000 HH01XR, to withstand temperatures of up to 200°C for up to 2000 hours. Designed specifically for DI equipment, it is the premier choice solder mask for ground and air vehicles as well as heavy industrial machinery. PSR-4000 HH01XR is user friendly with wide processing latitudes, low odor, fast developing, and good resistance to alternate meal finishes while maintaining dams of 3 mils or less.

- Screen Print & Spray Application
- RoHS Compliant
- Meets Aerospace Outgassing Requirements
- Resistance to extreme temperature of 200°C for 2000 hours
- Fine Dam Resolution
- Halogen-Free

Contact your local sales representative today!

Visit www.taiyo-america.com for more information.



 **TAIYO
AMERICA**
UNMATCHED INNOVATION



Follow us on social media



The Calf Path— Redux

Flexible Thinking

by Joe Fjelstad, VERDANT ELECTRONICS

Regular readers might recognize that I have been writing for I-Connect007 since its inception and prior to that wrote for *CircuiTree Magazine*, which is its direct lineage ancestor founded in the 1980s. The staff has always been tolerant of my sometimes-off-the-wall digressions from the strictly technical commentaries to provide a different perspective, to hopefully make the technical a bit more understandable and appreciated.

I am taking liberty here to do so once again and repeat the presentation of a delightful poem shared with readers many years ago. The poem was written by Sam Walter Foss, a late 19th to early 20th century New Hampshire librarian and poet with great ability to bestow

life lessons with humor, humanity, and humility. One such poem is “The Calf Path.”

When I first read the poem many decades ago, it immediately struck me with its simple yet profound wisdom, which I trust the reader will appreciate as well. Since that fortunate discovery, the poem has often informed my conscious thinking (and unconscious thoughts as well, I am sure) especially when it comes to inventions and inventing. We are all creatures of habit and convention. Habits can be good things, making life more predictable and easier to navigate. However, our habits can also blind us, constrain our thinking, cause us to miss opportunities, and even lead us astray. It is important to look up and look around from



PREMIER GLOBAL SUPPLIER to the **MEDICAL DEVICE INDUSTRY**

FLEXIBLE CIRCUITS | EMS/ASSEMBLY | CONTRACT MANUFACTURING

Industry leading Flex Design Support

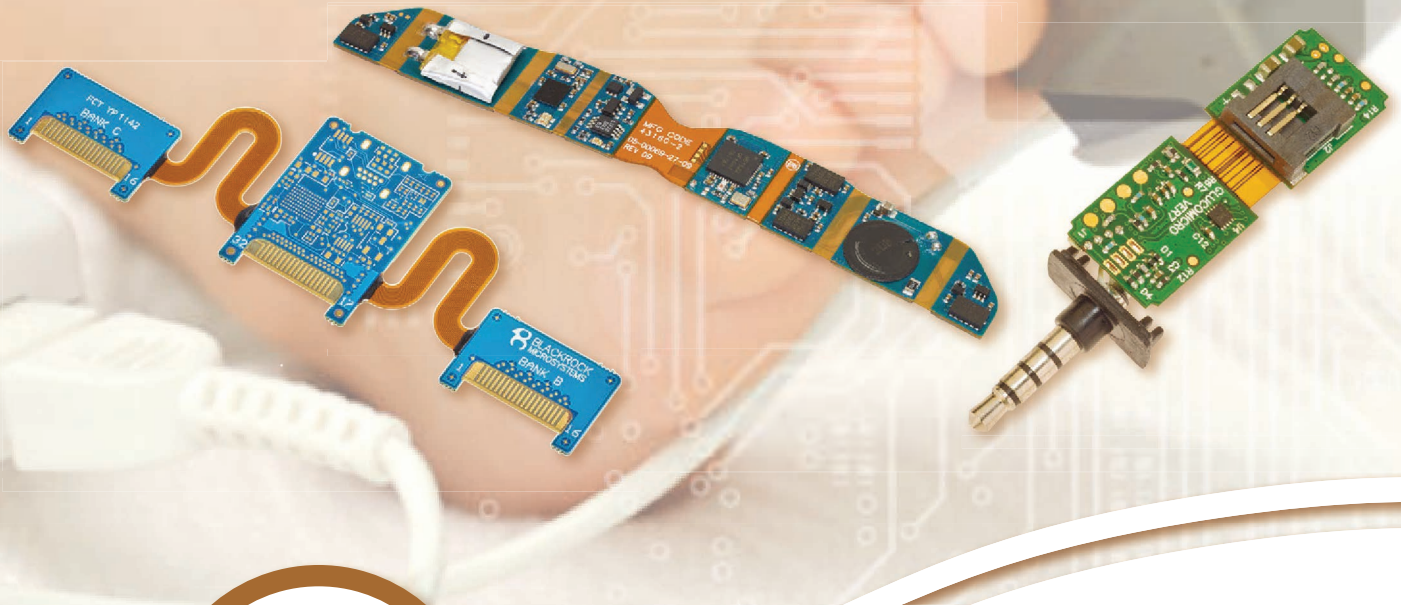
**Flexible Circuits, Rigid Flex, Flexible Heaters
Membrane Switches, Plastic Moldings**

Specialized EMS/Assembly Services

**Product Module to Complete
Product Box Builds**

ISO 13485 – FDA registered Box Builds

**EXPLORE the
POSSIBILITIES!**



Flexible Circuit Technologies
9850 51st Ave. N. | Plymouth, MN 55442
www.flexiblecircuit.com | +1-763-545-3333

time to time to make sure we are not missing out because of our habits, which should be tested every so often.

Without further comment, I invite the reader to read, enjoy, and hopefully remember “The Calf Path” and its timeless message.

The Calf Path

A Poem by Sam Walter Foss

One day through the primeval wood
A calf walked home as good calves should;
But made a trail all bent askew,
A crooked trail as all calves do.

Since then three hundred years have fled,
And I infer the calf is dead.
But still he left behind his trail,
And thereby hangs my moral tale.

The trail was taken up next day,
By a lone dog that passed that way;
And then a wise bell-weather sheep
Pursued the trail o’er vale and steep,
And drew the flock behind him, too,
As good bell-weathers always do.

And from that day, o’er hill and glade,
Through those old woods a path was made.

And many men wound in and out,
And dodged, and turned, and bent about,
And uttered words of righteous wrath,
Because ‘twas such a crooked path;
But still they followed—do not laugh—
The first migrations of that calf,
And through this winding wood-way stalked
Because he wobbled when he walked.

This forest path became a lane,
that bent and turned and turned again;
This crooked lane became a road,
Where many a poor horse with his load
Toiled on beneath the burning sun,
And traveled some three miles in one.

And thus a century and a half
They trod the footsteps of that calf.
The years passed on in swiftness fleet,
The road became a village street;

And this, before men were aware,
A city’s crowded thoroughfare.

And soon the central street was this
Of a renowned metropolis;
And men two centuries and a half,
Trod in the footsteps of that calf.

Each day a hundred thousand rout
Followed the zigzag calf about
And o’er his crooked journey went
The traffic of a continent.

A hundred thousand men were led,
By one calf near three centuries dead.

They followed still his crooked way,
And lost one hundred years a day;
For thus such reverence is lent,
To well established precedent.

A moral lesson this might teach
Were I ordained and called to preach;
For men are prone to go it blind
Along the calf-paths of the mind,
And work away from sun to sun,
To do what other men have done.

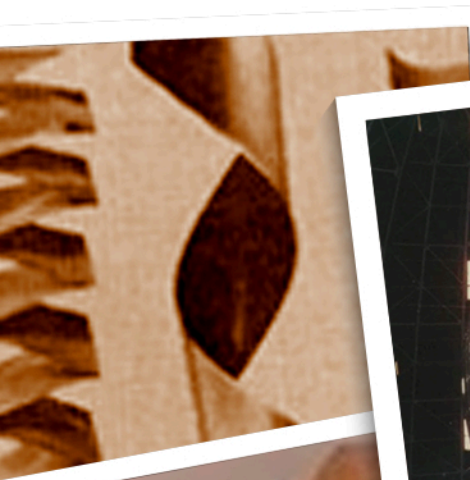
They follow in the beaten track,
And out and in, and forth and back,
And still their devious course pursue,
To keep the path that others do.

They keep the path a sacred groove,
Along which all their lives they move.
But how the wise old wood gods laugh,
Who saw the first primeval calf.
Ah, many things this tale might teach—
But I am not ordained to preach.



Joe Fjelstad is founder and CEO of Verdant Electronics and an international authority and innovator in the field of electronic interconnection and packaging technologies with more than 185 patents issued or pending. To read past columns or contact Fjelstad, [click here](#). Download your free copy of Fjelstad’s book *Flexible Circuit Technology, 4th Edition*, and watch his in-depth workshop series “Flexible Circuit Technology.”

OUR ADVANCED TECHNOLOGY HAS A LONG HISTORY OF MAKING HISTORY.



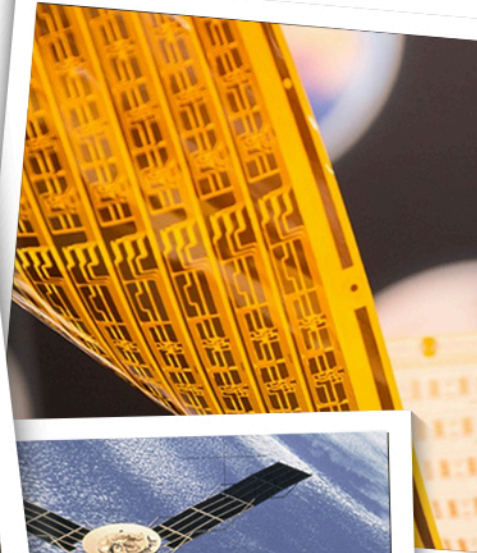
ECHO II 1961



NOVALINK 2015



VIKING LANDER



SKYLAB REPAIR 1965

For more than 65 years, we have been a pioneer in the electronics industry. With our long standing history of providing design and solutions, we invite you to be a part of making tomorrow's history!

Contact us today! 

www.sheldahl.com

 **Sheldahl**[®]

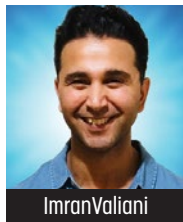
A Flex Company

TOP 10

Editor picks from PCBDesign007.com

1 Time to Market: Security is Key ▶

The tsunami of products coming to market means that confidentiality and security is more important than ever. But are they getting the attention they need?



Imran Valiani

3 Elementary, Mr. Watson: Trust but Verify ▶

Over many years, I have seen some elaborate PCB library systems. However, the best ones were those not based on the size but rather the quality of the information. That old axiom is definitely “not quantity but rather quality.”



John Watson

2 ExpressPCB Plus 3.0 Adds Functionality, SnapEDA Integration ▶

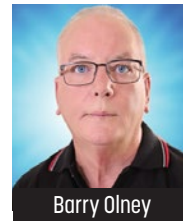
ExpressPCB has recently released version 3.0 of its ExpressPCB and Express PCB Plus PCB design tools. The I-Connect007 Editorial Team recently spoke with Michael Hebda, product manager for ExpressPCB. He offered a walk-through of the new Plus version 3.0 and explained how the new enhancements enable PCB designers to potentially cut hours or days off their design cycle.



Mike Hebda

4 Beyond Design: High-Speed Serial Link PCB Design ▶

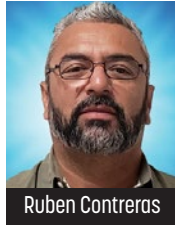
Serial communication has been used long before computers ever existed. The telegraph system using Morse code is one of the first digital modes of communication. All you need is two connections, which makes it simple and relatively robust. Columnist Barry Olney explains how this relates to PCB design.



Barry Olney

5 Fresh PCB Concepts: HDI Microvia Features in Illustrations ►

Ruben Contreras explains microvias and discusses aspect ratios with microvias. This is important to know when designing an HDI PCB because the different types vary in complexity. And the more complex, the more this affects the cost.



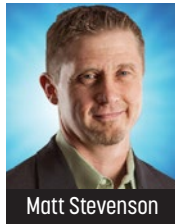
8 Altium Declines Autodesk's \$3.9 Billion Offer ►

According to the overnight news-wires, software giant Autodesk announced its intentions to acquire PCB design software developer Altium. In a recent press release from PRNewswire, Autodesk "confirmed that it has submitted a non-binding proposal to acquire all outstanding shares of common stock of Altium Limited."



6 Connect the Dots: There is No 'Final' Frontier for PCB Design ►

Our ongoing mission: To explore more manufacturable designs, to seek out higher-quality boards and enhanced functionality, to boldly design PCBs that no one has designed before.



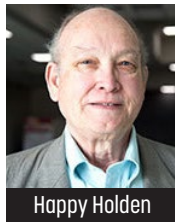
9 The Digital Layout: PCB Engineering on the Move ►

In this month's column, Kelly Dack examines the PCB engineering job outlook and evaluates the career moves we are making, while PCEA Chairman Steph Chavez lists some important attributes in our personal development that will keep our edges sharp.



7 Star Trek, the Original Series: An Homage From a Fan ►

It's fun to think back to the days when I first saw *Star Trek* on TV. In September 1966, I was a sophomore in college in chemical engineering. Being a science fiction fan for many years, I was looking forward to this new show, so I would go over to the student union building early in order to get a seat in the TV room. I got interested in science fiction from a few good movies like *20,000 Leagues Under the Sea* (1954) and *Forbidden Planet* (1956).



10 Planning and Communication—Key to Optimizing Your Design Time ►

How many times in our careers have we been asked, "Can we pull in the schedule?" and we can feel the hair on the back of our neck standing up on end. This type of question can be hard to hear simply because it is the wrong type of question.



PCBDesign007.com for the latest circuit design news and information.
Flex007.com focuses on the rapidly growing flexible and rigid-flex circuit market.

Career Opportunities



Is your team growing?

Find industry-experienced candidates at I-Connect007.

For just \$750, your 200-word, full-column ad will appear in the "career opportunities" section of all three of our monthly magazines, reaching circuit board designers, fabricators, assemblers, OEMs, and suppliers.

In addition, your ad will be featured in at least one of our newsletters, and your posting will appear on our jobConnect007.com board, which is also promoted in every newsletter.

Potential candidates can click on your ad and submit a resume directly to the email address you provide or be directed to the URL of your choice. If you wish to continue beyond the first month, the price is the same per month.

No contract required. We even include your logo in the ad, which is great branding!

To get your ad into the next issue, contact:

Barb Hockaday at barb@iconnect007.com or +1 916.365.1727 (-8 GMT PST)

I-Connect007
GOOD FOR THE INDUSTRY



Career Opportunities



Rewarding Careers

Take advantage of the opportunities we are offering for careers with a growing test engineering firm. We currently have several openings at every stage of our operation.

The Test Connection, Inc. is a test engineering firm. We are family owned and operated with solid growth goals and strategies. We have an established workforce with seasoned professionals who are committed to meeting the demands of high-quality, low-cost and fast delivery.

TTCI is an Equal Opportunity Employer. We offer careers that include skills-based compensation. We are always looking for talented, experienced test engineers, test technicians, quote technicians, electronics interns, and front office staff to further our customer-oriented mission.

Associate Electronics Technician/Engineer (ATE-MD)

TTCI is adding electronics technician/engineer to our team for production test support.

- Candidates would operate the test systems and inspect circuit card assemblies (CCA) and will work under the direction of engineering staff, following established procedures to accomplish assigned tasks.
- Test, troubleshoot, repair, and modify developmental and production electronics.
- Working knowledge of theories of electronics, electrical circuitry, engineering mathematics, electronic and electrical testing desired.
- Advancement opportunities available.
- Must be a US citizen or resident.

[apply now](#)

Test Engineer (TE-MD)

In this role, you will specialize in the development of in-circuit test (ICT) sets for Keysight 3070 (formerly HP) and/or Teradyne (formerly GenRad) TestStation/228X test systems.

- Candidates must have at least three years of experience with in-circuit test equipment. A candidate would develop and debug our test systems and install in-circuit test sets remotely online or at customer's manufactur-

ing locations nationwide.

- Candidates would also help support production testing and implement Engineering Change Orders and program enhancements, library model generation, perform testing and failure analysis of assembled boards, and other related tasks.
- Some travel required and these positions are available in the Hunt Valley, Md., office.

[apply now](#)

Sr. Test Engineer (STE-MD)

- Candidate would specialize in the development of in-circuit test (ICT) sets for Keysight 3070 (formerly Agilent & HP), Teradyne/GenRad, and Flying Probe test systems.
- Strong candidates will have more than five years of experience with in-circuit test equipment. Some experience with flying probe test equipment is preferred. A candidate would develop, and debug on our test systems and install in-circuit test sets remotely online or at customer's manufacturing locations nationwide.
- Proficient working knowledge of Flash/ISP programming, MAC Address and Boundary Scan required. The candidate would also help support production testing implementing Engineering Change Orders and program enhancements, library model generation, perform testing and failure analysis of assembled boards, and other related tasks. An understanding of stand-alone boundary scan and flying probe desired.
- Some travel required. Positions are available in the Hunt Valley, Md., office.

Contact us today to learn about the rewarding careers we are offering. Please email resumes with a short message describing your relevant experience and any questions to careers@ttci.com. Please, no phone calls.

We proudly serve customers nationwide and around the world.

TTCI is an ITAR registered and JCP DD2345 certified company that is NIST 800-171 compliant.

[apply now](#)

Career Opportunities



Logistics Assistant

Koh Young America is looking for a Logistics Assistant to assist and oversee our supply chain operations. Working alongside a Logistics Specialist, you will coordinate processes to ensure smooth operations using a variety of channels to maximize efficiency. You must be an excellent communicator and negotiator well-versed in supply chain management principles and practices. Also, you should be meticulous with a focus on customer satisfaction. These attributes are ideally complemented by a Bachelor's in Supply Chain Management or equivalent professional experience in the manufacturing industry.

This position is in our Duluth, Georgia, headquarters, where we serve our customers within North and South America. We offer health, dental, vision, and life Insurance with no employee premiums, including dependent coverage. Additionally, we provide a 401K retirement plan with company matching, plus a generous PTO policy with paid holidays.

Koh Young Technology, founded in 2002 in Seoul, South Korea, is the world leader in 3D measurement and inspection technology used in the production of micro-electronics assemblies. Using patented 3D technology, Koh Young provides best-in-class products in Solder Paste Inspection (SPI) and Automated Optical Inspection (AOI) for electronics manufacturers worldwide.

[apply now](#)



American Standard Circuits
Creative Innovations In Flex, Digital & Microwave Circuits

CAD/CAM Engineer

Summary of Functions

The CAD/CAM engineer is responsible for reviewing customer supplied data and drawings, performing design rule checks and creating manufacturing data, programs, and tools required for the manufacture of PCB.

Essential Duties and Responsibilities

- Import customer data into various CAM systems.
- Perform design rule checks and edit data to comply with manufacturing guidelines.
- Create array configurations, route, and test programs, penalization and output data for production use.
- Work with process engineers to evaluate and provide strategy for advanced processing as needed.
- Itemize and correspond to design issues with customers.
- Other duties as assigned.

Organizational Relationship

Reports to the engineering manager. Coordinates activities with all departments, especially manufacturing.

Qualifications

- A college degree or 5 years' experience is required. Good communication skills and the ability to work well with people is essential.
- Printed circuit board manufacturing knowledge.
- Experience using CAM tooling software, Orbotech GenFlex®.

Physical Demands

Ability to communicate verbally with management and coworkers is crucial. Regular use of the telephone and e-mail for communication is essential. Sitting for extended periods is common. Hearing and vision within normal ranges is helpful for normal conversations, to receive ordinary information and to prepare documents.

[apply now](#)

Career Opportunities



Arlon EMD, located in Rancho Cucamonga, California, is currently interviewing candidates for open positions in:

- Engineering
- Quality
- Various Manufacturing

All interested candidates should contact Arlon's HR department at 909-987-9533 or email resumes to careers.ranch@arlonemd.com.

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

Our facility employs state of the art production equipment engineered to provide cost-effective and flexible manufacturing capacity allowing us to respond quickly to customer requirements while meeting the most stringent quality and tolerance demands. Our manufacturing site is ISO 9001: 2015 registered, and through rigorous quality control practices and commitment to continual improvement, we are dedicated to meeting and exceeding our customers' requirements.

For additional information please visit our website at www.arlonemd.com

[apply now](#)



Marketing Coordinator/Writing Strategist: Embedded Software

Location: Portland, Oregon or USA (remote)

Job Number: 242982

Seeking a technology communications change maker! Siemens Digital Industries Software is looking for a content creator for its embedded software group. The ideal candidate for the Brand Marketing coordinator/writing strategist position will work closely with engineers and managers to write, edit and produce compelling technology marketing content (magazine articles, blogs, technology papers, multi-media, customer success stories and promotional materials). Do you possess creative energy and enjoy storytelling with an energetic team?

Requirements:

- Strong writing and editing skills
- Education and/or experience in technology, science, journalism and/or English
- A technical background or experience (such as a BS or an associate's degree in engineering or computer science) is preferred
- 1-3 years of experience in writing about technology solutions
- Basic knowledge of online publications, digital platforms and social media is useful to meet project specifications in a fast-paced environment
- Ability to research and collect data, repurpose existing materials, collaborate with subject matter experts, and translate technical information into compelling marketing communications content that engage audiences

Creative materials will be used globally, in a high-energy environment, supporting the world's leading industrial software company.

[apply now](#)

Career Opportunities



MANUFACTURERS OF QUALITY PRINTED CIRCUIT BOARDS

Maintenance Technician

Inspects work-related conditions to determine compliance with prescribed operating and safety standards. Operates power-driven machinery and uses equipment and tools commonly used to maintain facilities and equipment. Replace filters, belts, and additional parts for repairs and preventive maintenance. Moves objects weighing up to 150 lbs. using a hand truck or pulley. Cleans work area and equipment. Works with cleaning fluids, agents, chemicals, and paints using protective gear. Works at elevations greater than ten feet, climbing ladders, while repairing or maintaining building structures and equipment. Assists skilled maintenance technicians/workers in more complex tasks and possible after-hours emergency repairs. Must meet scheduling and attendance requirements.

[apply now](#)

Plating Operator

Plating operator for printed circuit boards. No experience necessary, will train. Must be able to work with chemicals, lift up to 50 pounds, and have good math skills. Minimum high school/GED or equivalent. All shifts (1st, 2nd, 3rd), 8 hours per day minimum, Monday thru Friday. Saturday and Sunday work is common allowing for steady overtime pay.

[apply now](#)



MANUFACTURERS OF QUALITY PRINTED CIRCUIT BOARDS

Water Treatment Operator

Responsible for operating waste treatment plant, our operation that converts wastewater in drains and sewers into a form that's metal free to release into the environment.

Control equipment and monitor processes that remove metals from wastewater. Run tests to make sure that the processes are working correctly. Keep records of water quality and pH. Operate and maintain the pumps and motors that move water and wastewater through filtration systems. Read meters and gauges to make sure plant equipment is working properly. Take samples and run tests to determine the quality of the water being produced. Adjust the amount of chemicals being added to the water and keep records that document compliance.

[apply now](#)

Drilling Operator

Drilling operator for printed circuit boards. Minimum 2 years of experience. Minimum high school/GED or equivalent.

All Shifts (1st, 2nd, 3rd), 8 hours per day minimum, Monday thru Friday. Saturday and Sunday work is common allowing for overtime pay.

[apply now](#)

Career Opportunities



Technical Writer (Full-time, Remote)

SEO company seeking a technical writer in the area of PCB design & manufacturing. We provide Search Engine Optimization and Thought Leadership services for well-managed, ethical companies. Our team produces high-level content for companies who are leaders in their industry. We are seeking writers who are well-rounded researchers with a particular interest in electrical engineering and impeccable writing skills.

Ideal candidates are seasoned writers with practical experience in electronic systems development in one or more of the following areas:

- Circuit simulation
- Schematic capture
- PCB layout
- Design analysis
- PCB fabrication
- PCB assembly
- PCB testing

Qualifications:

- Technical writing experience with a proven track record of independent research and content development. Experience in data sheet, report, or white-paper writing preferred.
- Electronic technician or engineering degree, ideally in electrical engineering, computer science, or mechanical engineering.
- Industry experience in PCB design, testing, or manufacturing.
- Punctuality, professionalism, and excellent time management skills.
- A reliable internet connection and computer

Salary & Benefits:

This is a full-time telecommuting position with a starting salary range of \$62,000 to \$68,000 annually. Benefits PTO, sick time, 401K, health and dental coverage, and more!

To see the full job description and apply, please click the link below.

[apply now](#)



Multiple Positions

Innovative Circuits, a quick-turn, high mix, low-volume PCB manufacturer located in Alpharetta, Georgia, is growing and looking for talented individuals to join the team.

Front End Engineering Manager

Oversee CAM, programming/production engineering and quoting departments. Ideal candidates will have 15 years' experience working in a printed circuit board front-end department with flex and rigid flex circuit board construction.

Process Engineer

Responsible for the implementation and maintenance of chemical and/or mechanical processes used to produce flex circuits, rigid flex and rigid printed circuit boards.

Third Shift Production Manager

Oversee third shift productions workers, product schedule and reporting.

Wet Lab Tech

Perform all lab analysis using burettes, pipettes, pH/ion meters, atomic absorption spectrophotometer, laboratory balance, hydrometers, hull cells, CVS, and all other lab-related equipment.

CAM Operator

Inspect, modify, and contribute to the initial development of producing flex circuits, rigid flex and rigid printed circuit boards based upon customer requirements and data files.

Quality Inspector

Responsible for verifying that the product meets customer requirements prior to shipping.

Wastewater Technician

Operate, monitor, maintain and troubleshoot the wastewater treatment facility and its processes.

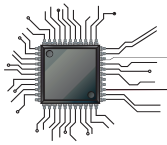
Production Worker

Machine operator and light chemistry in a PCB manufacturing environment.

Please visit the link below to view our opportunities and apply.

[apply now](#)

Career Opportunities



MivaTek

Global

Product Manager

MivaTek Global is preparing for a major market and product offering expansion. Miva's new NG3 and DART technologies have been released to expand the capabilities of Miva's industry-leading LED DMD direct write systems in PCB and Microelectronics. MivaTek Global is looking for a technology leader that can be involved guiding this major development.

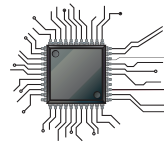
The product manager role will serve as liaison between the external market and the internal design team. Leadership level involvement in the direction of new and existing products will require a diverse skill set. Key role functions include:

- **Sales Support:** Recommend customer solutions through adaptations to Miva products
- **Design:** Be the voice of the customer for new product development
- **Quality:** Verify and standardize product performance testing and implementation
- **Training:** Conduct virtual and on-site training
- **Travel:** Product testing at customer and factory locations

Use your 8 plus years of experience in either the PCB or Microelectronic industry to make a difference with the leader in LED DMD direct imaging technology. Direct imaging, CAM, AOI, or drilling experience is a plus but not required.

For consideration, send your resume to N.Hogan@MivaTek.Global. For more information on the company see www.MivaTek.Global or www.Mivatec.com.

apply now



MivaTek

Global

Field Service Technician

MivaTek Global is focused on providing a quality customer service experience to our current and future customers in the printed circuit board and microelectronic industries. We are looking for bright and talented people who share that mindset and are energized by hard work who are looking to be part of our continued growth.

Do you enjoy diagnosing machines and processes to determine how to solve our customers' challenges? Your 5 years working with direct imaging machinery, capital equipment, or PCBs will be leveraged as you support our customers in the field and from your home office. Each day is different, you may be:

- Installing a direct imaging machine
- Diagnosing customer issues from both your home office and customer site
- Upgrading a used machine
- Performing preventive maintenance
- Providing virtual and on-site training
- Updating documentation

Do you have 3 years' experience working with direct imaging or capital equipment? Enjoy travel? Want to make a difference to our customers? Send your resume to N.Hogan@MivaTek.Global for consideration.

More About Us

MivaTek Global is a distributor of Miva Technologies' imaging systems. We currently have 55 installations in the Americas and have machine installations in China, Singapore, Korea, and India.

apply now

Career Opportunities



A Flex Company

Sheldahl, a leading provider of flexible interconnect products and electronic materials, is seeking candidates to join their diverse and skilled team.

We are looking for people who demonstrate:

- Intense collaboration
- Passionate customer focus
- Thoughtful, fast, disciplined execution
- Tenacious commitment to continuous improvement
- Relentless drive to win

Positions in America include:

Project Manager – Northfield, MN

Candidate will provide timely cost estimation and project budget definition, be responsible for maintaining customer relations, participate in meetings, etc.

[apply now](#)

Program Manager – Specialty Films

Candidate will work with our Specialty Films in the Aerospace, Medical, and Commercial Aviation markets providing timely cost estimation and project budget definition, maintaining customer relations, participate in meetings, etc.

[apply now](#)

Business Development Manager – North America

Candidate will provide leadership in the planning, design and implementation of customers' specific business plans and will provide vision, penetration strategies and tactics to executive managers in order to develop and drive external and internal senior-level relationships.

[apply now](#)



A Flex Company

Sheldahl, a leading provider of flexible interconnect products and electronic materials, is seeking candidates to join their diverse and skilled team.

We are looking for people who demonstrate:

- Intense collaboration
- Passionate customer focus
- Thoughtful, fast, disciplined execution
- Tenacious commitment to continuous improvement
- Relentless drive to win

Positions in Europe include:

Business Development Manager – France

Seeking out-of-the-box thinkers to help us take the ordinary to the extraordinary by cultivating current customer relationships and developing new business opportunities with our European team, based in France.

[apply now](#)

Business Development Manager – Germany

Seeking out-of-the-box thinkers to help us take the ordinary to the extraordinary by cultivating current customer relationships and developing new business opportunities with our European team, based in Germany.

[apply now](#)

Career Opportunities



ventec
INTERNATIONAL GROUP
騰輝電子

Technical Support/ Sales Engineer, UK

We are looking to expand our UK technical & sales support team. As a technical support/sales engineer (home office/Leamington Spa) you will assist potential and current customers in appreciating the benefits of using--and optimizing the use of--Ventec materials in their printed circuit board manufacturing processes, and so enhance customer loyalty and satisfaction, spread the use of Ventec materials, and grow sales. You will provide a two-way channel of technical communication between Ventec's production facilities and UK/European customers.

Skills and abilities required for the role

- HNC, HND, degree or equivalent in a technical/scientific discipline
- Sales experience/negotiating skills
- Printed circuit board industry experience an advantage
- Good written & verbal communications skills
- Ability to work in an organized, proactive and enthusiastic way
- Ability to work well both in a team and independently
- Good user knowledge of common Microsoft Office programs
- Full driving license essential

What's on Offer

- Excellent salary and benefits commensurate with experience

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

Please forward your resume to
anthony.jackson@ventec-europe.com

[apply now](#)



U.S. CIRCUIT

Plating Supervisor

Escondido, California-based PCB fabricator U.S. Circuit is now hiring for the position of plating supervisor. Candidate must have a minimum of five years' experience working in a wet process environment. Must have good communication skills, bilingual is a plus. Must have working knowledge of a plating lab and hands-on experience running an electrolytic plating line. Responsibilities include, but are not limited to, scheduling work, enforcing safety rules, scheduling/maintaining equipment and maintenance of records.

Competitive benefits package. Pay will be commensurate with experience.

Mail to:
mfariba@uscircuit.com

[apply now](#)

Career Opportunities

SIEMENS

Siemens EDA Sr. Applications Engineer

Support consultative sales efforts at world's leading semiconductor and electronic equipment manufacturers. You will be responsible for securing EM Analysis & Simulation technical wins with the industry-leading HyperLynx Analysis product family as part of the Xpedition Enterprise design flow.

Will deliver technical presentations, conduct product demonstrations and benchmarks, and participate in the development of account sales strategies leading to market share gains.

- PCB design competency required
- BEE, MSEE preferred
- Prior experience with Signal Integrity, Power Integrity, EM & SPICE circuit analysis tools
- Experience with HyperLynx, Ansys, Keysight and/or Sigrity
- A minimum of 5 years' hands-on experience with EM Analysis & Simulation, printed circuit board design, engineering technology or similar field
- Moderate domestic travel required
- Possess passion to learn and perform at the cutting edge of technology
- Desire to broaden exposure to the business aspects of the technical design world
- Possess a demonstrated ability to build strong rapport and credibility with customer organizations while maintaining an internal network of contacts
- Enjoy contributing to the success of a phenomenal team

***Qualified applicants will not require employer-sponsored work authorization now or in the future for employment in the United States. Qualified Applicants must be legally authorized for employment in the United States.*

apply now

INSULECTRO



Are You Our Next Superstar?!

Insulectro, the largest national distributor of printed circuit board materials, is looking to add superstars to our dynamic technical and sales teams. We are always looking for good talent to enhance our service level to our customers and drive our purpose to enable our customers build better boards faster. Our nationwide network provides many opportunities for a rewarding career within our company.

We are looking for talent with solid background in the PCB or PE industry and proven sales experience with a drive and attitude that match our company culture. This is a great opportunity to join an industry leader in the PCB and PE world and work with a terrific team driven to be vital in the design and manufacture of future circuits.

View our opportunities at
Insulectro Careers ([jobvite.com](https://insulectro.com/careers))

apply now

Career Opportunities

Now Hiring

Director of Process Engineering

A successful and growing printed circuit board manufacturer in Orange County, CA, has an opening for a director of process engineering.

Job Summary:

The director of process engineering leads all engineering activities to produce quality products and meet cost objectives. Responsible for the overall management, direction, and coordination of the engineering processes within the plant.

Duties and Responsibilities:

- Ensures that process engineering meets the business needs of the company as they relate to capabilities, processes, technologies, and capacity.
- Stays current with related manufacturing trends. Develops and enforces a culture of strong engineering discipline, including robust process definition, testing prior to production implementation, change management processes, clear manufacturing instructions, statistical process monitoring and control, proactive error proofing, etc.
- Provides guidance to process engineers in the development of process control plans and the application of advanced quality tools.
- Ensures metrics are in place to monitor performance against the goals and takes appropriate corrective actions as required. Ensures that structured problem-solving techniques are used and that adequate validation is performed for any issues being address or changes being made. Develops and validates new processes prior to incorporating them into the manufacturing operations.
- Strong communication skills to establish priorities, work schedules, allocate resources, complete required information to customers, support quality system, enforce company policies and procedures, and utilize resources to provide the greatest efficiency to meet production objectives.

Education and Experience:

- Master's degree in chemical engineering or engineering is preferred.
- 10+ years process engineering experience in an electronics manufacturing environment, including 5 years in the PCB or similar manufacturing environment.
- 7+ years of process engineering management experience, including 5 years of experience with direct responsibility for meeting production throughput and quality goals.

[apply now](#)

Now Hiring

Process Engineering Manager

A successful and growing printed circuit board manufacturer in Orange County, CA, has an opening for a process engineering manager.

Job Summary:

The process engineering manager coordinates all engineering activities to produce quality products and meet cost objectives. Responsible for the overall management, direction, and coordination of the engineering team and leading this team to meet product requirements in support of the production plan.

Duties and Responsibilities:

- Ensures that process engineering meets the business needs of the company as they relate to capabilities, processes, technologies, and capacity.
- Stays current with related manufacturing trends. Develops and enforces a culture of strong engineering discipline, including robust process definition, testing prior to production implementation, change management processes, clear manufacturing instructions, statistical process monitoring and control, proactive error proofing, etc.
- Ensures metrics are in place to monitor performance against the goals and takes appropriate corrective actions as required. Ensures that structured problem-solving techniques are used and that adequate validation is performed for any issues being address or changes being made. Develops and validates new processes prior to incorporating into the manufacturing operations

Education and Experience:

- Bachelor's degree in chemical engineering or engineering is preferred.
- 7+ years process engineering experience in an electronics manufacturing environment, including 3 years in the PCB or similar manufacturing environment.
- 5+ years of process engineering management experience, including 3 years of experience with direct responsibility for meeting production throughput and quality goals.

[apply now](#)

Career Opportunities



Sales Account Manager

Sales Account Management at Lenthor Engineering is a direct sales position responsible for creating and growing a base of customers that purchase flexible and rigid flexible printed circuits. The account manager is in charge of finding customers, qualifying the customer to Lenthor Engineering and promoting Lenthor Engineering's capabilities to the customer. Leads are sometimes referred to the account manager from marketing resources including trade shows, advertising, industry referrals and website hits. Experience with military printed circuit boards (PCBs) is a definite plus.

Responsibilities

- Marketing research to identify target customers
- Identifying the person(s) responsible for purchasing flexible circuits
- Exploring the customer's needs that fit our capabilities in terms of:
 - Market and product
 - Circuit types used
 - Competitive influences
 - Philosophies and finance
 - Quoting and closing orders
 - Providing ongoing service to the customer
 - Develop long-term customer strategies to increase business

Qualifications

- 5-10 years of proven work experience
- Excellent technical skills

Salary negotiable and dependent on experience. Full range of benefits.

Lenthor Engineering, Inc. is a leader in flex and rigid-flex PWB design, fabrication and assembly with over 30 years of experience meeting and exceeding our customers' expectations.

Contact Oscar Akbar at: hr@lenthor.com

[apply now](#)



Senior Process Engineer

Job Description

Responsible for developing and optimizing Lenthor's manufacturing processes from start up to implementation, reducing cost, improving sustainability and continuous improvement.

Position Duties

- Senior process engineer's role is to monitor process performance through tracking and enhance through continuous improvement initiatives. Process engineer implements continuous improvement programs to drive up yields.
- Participate in the evaluation of processes, new equipment, facility improvements and procedures.
- Improve process capability, yields, costs and production volume while maintaining safety and improving quality standards.
- Work with customers in developing cost-effective production processes.
- Engage suppliers in quality improvements and process control issues as required.
- Generate process control plan for manufacturing processes, and identify opportunities for capability or process improvement.
- Participate in FMEA activities as required.
- Create detailed plans for IQ, OQ, PQ and maintain validated status as required.
- Participate in existing change control mechanisms such as ECOs and PCRs.
- Perform defect reduction analysis and activities.

Qualifications

- BS degree in engineering
- 5-10 years of proven work experience
- Excellent technical skills

Salary negotiable and dependent on experience. Full range of benefits.

Lenthor Engineering, Inc. is the leader in Flex and Rigid-Flex PWB design, fabrication and assembly with over 30 years of experience meeting and exceeding our customers' expectations.

Contact Oscar Akbar at: hr@lenthor.com

[apply now](#)

Career Opportunities



SMT Operator Hatboro, PA

Manncorp, a leader in the electronics assembly industry, is looking for a **surface-mount technology (SMT) operator** to join their growing team in Hatboro, PA!

The **SMT operator** will be part of a collaborative team and operate the latest Manncorp equipment in our brand-new demonstration center.

Duties and Responsibilities:

- Set up and operate automated SMT assembly equipment
- Prepare component kits for manufacturing
- Perform visual inspection of SMT assembly
- Participate in directing the expansion and further development of our SMT capabilities
- Some mechanical assembly of lighting fixtures
- Assist Manncorp sales with customer demos

Requirements and Qualifications:

- Prior experience with SMT equipment or equivalent technical degree preferred; will consider recent graduates or those new to the industry
- Windows computer knowledge required
- Strong mechanical and electrical troubleshooting skills
- Experience programming machinery or demonstrated willingness to learn
- Positive self-starter attitude with a good work ethic
- Ability to work with minimal supervision
- Ability to lift up to 50 lbs. repetitively

We Offer:

- Competitive pay
- Medical and dental insurance
- Retirement fund matching
- Continued training as the industry develops

[apply now](#)



SMT Field Technician Hatboro, PA

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

Duties and Responsibilities:

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

Requirements and Qualifications:

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

We Offer:

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

[apply now](#)

Career Opportunities



Pre-CAM Engineer

Illinois-based PCB fabricator Eagle Electronics is seeking a pre-CAM engineer specific to the printed circuit board manufacturing industry. The pre-CAM Engineer will facilitate creation of the job shop travelers used in the manufacturing process. Candidate will have a minimum of two years of pre-CAM experience and have a minimum education level of an associate degree. This is a first-shift position at our Schaumburg, Illinois, facility. This is not a remote or offsite position.

If interested, please submit your resume to HR@eagle-elec.com indicating 'Pre-CAM Engineer' in the subject line.

[apply now](#)

Process Engineer

We are also seeking a process engineer with experience specific to the printed circuit board manufacturing industry. The process engineer will be assigned to specific processes within the manufacturing plant and be given ownership of those processes. The expectation is to make improvements, track and quantify process data, and add new capabilities where applicable. The right candidate will have a minimum of two years of process engineering experience, and a minimum education of bachelor's degree in an engineering field (chemical engineering preferred but not required). This is a first shift position at our Schaumburg, Illinois, facility. This is not a remote or offsite position.

If interested, please submit your resume to HR@eagle-elec.com indicating 'Process Engineer' in the subject line.

[apply now](#)



eptac
TRAIN. WORK SMARTER. SUCCEED.

Become a Certified IPC Master Instructor

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

Qualifications and skills

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

Benefits

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

[apply now](#)

Career Opportunities



IPC Instructor Longmont, CO; Phoenix, AZ; U.S.-based remote

*Independent contractor,
possible full-time employment*

Job Description

This position is responsible for delivering effective electronics manufacturing training, including IPC Certification, to students from the electronics manufacturing industry. IPC instructors primarily train and certify operators, inspectors, engineers, and other trainers to one of six IPC Certification Programs: IPC-A-600, IPC-A-610, IPC/WHMA-A-620, IPC J-STD-001, IPC 7711/7721, and IPC-6012.

IPC instructors will conduct training at one of our public training centers or will travel directly to the customer's facility. A candidate's close proximity to Longmont, CO, or Phoenix, AZ, is a plus. Several IPC Certification Courses can be taught remotely and require no travel.

Qualifications

Candidates must have a minimum of five years of electronics manufacturing experience. This experience can include printed circuit board fabrication, circuit board assembly, and/or wire and cable harness assembly. Soldering experience of through-hole and/or surface-mount components is highly preferred.

Candidate must have IPC training experience, either currently or in the past. A current and valid certified IPC trainer certificate holder is highly preferred.

Applicants must have the ability to work with little to no supervision and make appropriate and professional decisions.

Send resumes to Sharon Montana-Beard at
sharonm@blackfox.com.

[apply now](#)



APCT, Printed Circuit Board Solutions: Opportunities Await

APCT, a leading manufacturer of printed circuit boards, has experienced rapid growth over the past year and has multiple opportunities for highly skilled individuals looking to join a progressive and growing company. APCT is always eager to speak with professionals who understand the value of hard work, quality craftsmanship, and being part of a culture that not only serves the customer but one another.

APCT currently has opportunities in Santa Clara, CA; Orange County, CA; Anaheim, CA; Wallingford, CT; and Austin, TX. Positions available range from manufacturing to quality control, sales, and finance.

We invite you to read about APCT at APCT.com and encourage you to understand our core values of passion, commitment, and trust. If you can embrace these principles and what they entail, then you may be a great match to join our team! Peruse the opportunities by clicking the link below.

Thank you, and we look forward to
hearing from you soon.

[apply now](#)

Career Opportunities



U.S. CIRCUIT

Sales Representatives (Specific Territories)

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

- Florida
- Denver
- Washington
- Los Angeles

Experience:

- Candidates must have previous PCB sales experience.

Compensation:

- 7% commission

Contact Mike Fariba for
more information.

mfariba@uscircuit.com

[apply now](#)



JOHNS HOPKINS

CAM / Process Engineer

The JHU/APL PCB Fabrication team is seeking a Computer Aided Manufacturing Engineer to support front-end data processing of APL manufactured hardware. You will directly contribute to hardware fabrication in support of National Security, Military Readiness, Space Exploration, National Health, and Research related to fundamental scientific advancement. This position includes a variable mix of core CAM work scope with additional opportunities for hands-on support such as bare board electrical testing, laser drilling, and mechanical CNC drilling and routing.

Responsibilities:

1. Computer Aided Manufacturing for rigid PCB, rigid-flex, and flexible circuits
 - a) Perform design checks, panel layout, coupon generation, file generation, stackups
 - b) Support manufacturability reviews with internal APL engineers (customers)
 - c) Generate work travelers
 - d) Communicate status to supervisors and internal customers
2. Support transition of software tools (Genesis 2000 to InCAM Pro)
 - a) Edit design rules checks and generate automation scripts
 - b) Develop new ideas to further the technical progress of our product
 - c) Develop CAM area through continuous improvement initiatives
3. Interface and inform APL Engineers on PCB design for manufacturing guidelines
4. Operate bare board electrical tester
5. Backup operator for CNC drilling, routing, laser drilling (on-site training)

For more details and to apply:
www.jhuapl.edu/careers and search for CAM.

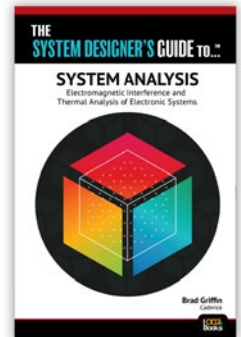
[apply now](#)

Introducing:

The System Designer's Guide to... System Analysis

Electromagnetic Interference and Thermal Analysis of Electronic Systems

In this latest title from I-007eBooks, readers will learn how system-level analysis of complex and high-speed electronic designs is critical to solve electromagnetic, electrothermal, and electromechanical simulation challenges and to ensure that the system works under wide-ranging operating conditions. **Get your copy now!**



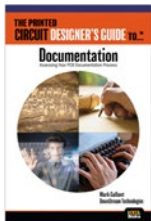
I-007eBooks The Printed Circuit Designer's Guide to...



Thermal Management: A Fabricator's Perspective

by Anaya Vardya, American Standard Circuits

Beat the heat in your designs through thermal management design processes. This book serves as a desk reference on the most current techniques and methods from a PCB fabricator's perspective.



Documentation

by Mark Gallant, Downstream Technologies

When the PCB layout is finished, the designer is still not quite done. The designer's intent must still be communicated to the fabricator through accurate PCB documentation.



Thermal Management with Insulated Metal Substrates

by Didier Mauve and Ian Mayoh, Ventec International Group

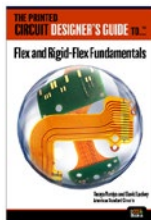
Considering thermal issues in the earliest stages of the design process is critical. This book highlights the need to dissipate heat from electronic devices.



Fundamentals of RF/Microwave PCBs

by John Bushie and Anaya Vardya, American Standard Circuits

Today's designers are challenged more than ever with the task of finding the optimal balance between cost and performance when designing radio frequency/microwave PCBs. This micro eBook provides information needed to understand the unique challenges of RF PCBs.



Flex and Rigid-Flex Fundamentals

by Anaya Vardya and David Lackey, American Standard Circuits

Flexible circuits are rapidly becoming a preferred interconnection technology for electronic products. By their intrinsic nature, FPCBs require a good deal more understanding and planning than their rigid PCB counterparts to be assured of first-pass success.

Our library is open 24/7/365. Visit us at: I-007eBooks.com

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

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

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

EDITOR | COLUMNIST COORDINATOR: **MICHELLE TE**
michelle@iconnect007.com

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

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

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

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

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

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

TECHNOLOGY SPECIALIST: **BRYSON MATTIES**

COVER: **SHELLY STEIN**

COVER IMAGE: **ADOBE STOCK © OLINCHUK**

DESIGN007

M A G A Z I N E

DESIGN007 MAGAZINE®

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

© 2021 BR Publishing, Inc. does not assume and hereby disclaims any liability to any person for loss or damage caused by errors or omissions in the material contained within this publication, regardless of whether such errors or omissions are caused accidentally, from negligence or any other cause.

July 2021, Volume 10, Number 7
DESIGN007 MAGAZINE is published monthly,
by BR Publishing, Inc.

ADVERTISER INDEX

Accurate Circuit Engineering.....	73
All Flex.....	95
Altium.....	65
American Standard Circuits.....	69
APCT.....	35
Burkle North America.....	9
Cadence.....	7
Candor Circuit Boards.....	5
DB Management.....	51
Downstream Technologies.....	17, 53
Eagle Electronics.....	49
EMA Design Automation.....	27
Flexible Circuit Technologies.....	99
I-Connect007.....	91
I-007 eBooks.....	2, 3
In-Circuit Design Pty Ltd.....	61
IPC.....	43, 79
MacDermid Alpha Electronic Solutions.....	93
NCAB.....	57
Siemens Digital Industries Software.....	39
Prototron Circuits.....	15
Pulsonix.....	83
Sheldahl.....	101
Summit Interconnect.....	81
Taiyo America.....	97
U.S. Circuit.....	23
Ventec International Group.....	31

Problems solved!



SUBSCRIBE NOW