

MARCH 2023

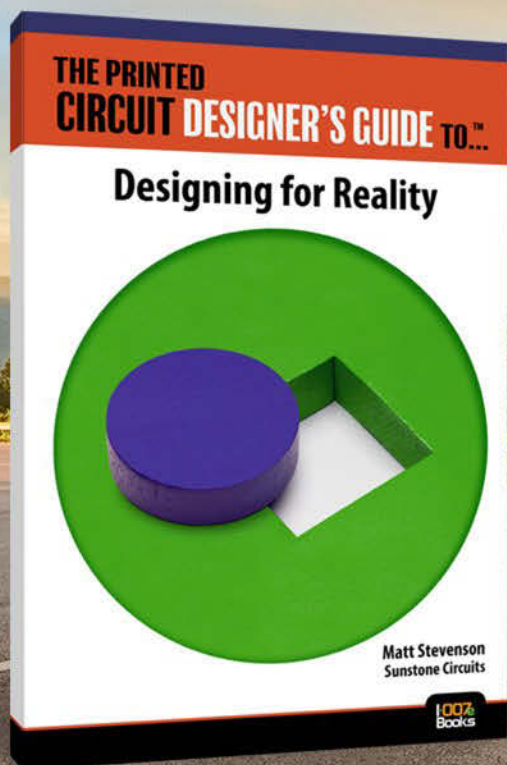
I-Connect007

DESIGN007

M A G A Z I N E



Tribal Knowledge
Friend or Foe?



Learn how to create
realistic designs and
advance your skills!



Download

1007e
Books

I-007eBooks

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

Tribal Knowledge: Friend or Foe?

Tribal knowledge is present in every organization, no matter what size. How do we distinguish this information from documented facts? In this issue, our expert contributors will provide readers with the tools and methodologies needed to identify tribal knowledge, as well as when to question such information, and how to document and transform this information into a process.

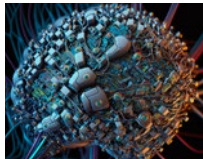
FEATURE INTERVIEWS

- 10** **With Tribal Knowledge, Trust but Verify**
with Tamara Jovanovic



- 22** **True Experts Can Cite Their Sources**
with Kris Moyer

- 28** **Embedding Know-how Into Automation**
with Michael Ford



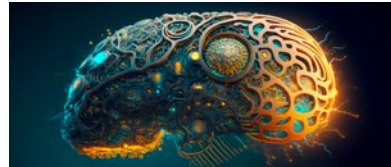
FEATURE ARTICLE

- 42** **Material Matters and Tribal Knowledge**
by Alun Morgan



FEATURE COLUMNS

- 8** **Tribal Knowledge: Friend or Foe?**
by Andy Shaughnessy



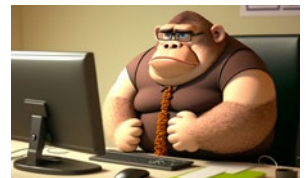
- 32** **Tribal Knowledge: Not the Villain You Thought**
by Tim Haag

- 44** **Instilling an Informal Information Culture**
by Martyn Gaudion



- 48** **If Not You, Then Who?**
by John Watson

- 68** **The Tale of Five CAD Monkeys**
by Kelly Dack



- 84** **Tribal Knowledge: A Personal Perspective**
by Joe Fjelstad



The fastest lead times in the US. Guaranteed.

We guarantee you the following:

- 1 Your single-sided or plated through printed circuit board can be PRODUCED WITHIN 5 WORKING DAYS from the receipt of your PCB data.
- 2 Your drilling data will be reproduced exactly both in position and diameter, improving assembly processes on your boards.
- 3 Your printed circuit boards are inspected at 7 separate stages to ensure a perfect final PCB.

[Request a Quote](#)

MILTRONIX

mil-tronix.com

DESIGN007

M A G A Z I N E



38

SHORTS

- 36** What Do Electrical Engineers Do?



- 52** A Week-long Industry Extravaganza

- 57** IPC Expands Instructor-led Courses on Technical Knowledge and Skills

- 71** Onboarding New Customers at FCT

- 87** Soft Touch Sensitivity

HIGHLIGHTS

- 26** MilAero007

- 82** Flex007

- 88** Top Ten Editor's Picks



DEPARTMENTS

- 91** Career Opportunities

- 102** Educational Resources

- 103** Advertiser Index & Masthead



COLUMNS

- 16** The Interaction of Electromagnetic Fields
by Barry Olney with Special Advisor Rick Hartley

- 38** The Power of Forward Thinking
by Matt Stevenson



- 72** PCB Design for Bare Board Testing
by Vern Solberg

ARTICLE

- 76** DFM 101: Final Finish Design Guidelines
by Anaya Vardya

SPECIAL COVERAGE



- 53** IPC APEX EXPO: A Focus on PCB Design with Andy Shaughnessy

- 54** IPC Design Competition Wrap-up with Kris Moyer

- 56** Q&A With the IPC Design Competition Winner with Sathish Vijayakumar

- 58** Runner-up Discusses IPC Design Competition with Adam Thorvaldson

- 62** Forcing Designers Out of Their Comfort Zone with Paul Brionez

- 64** Tips and Tricks in Today's Designs with Filbert Arzola

- 66** Solid Design Methodologies for Avoiding Obstacles with Christian Dominguez

BENDING THE POSSIBILITIES



BY TAIYO

AVAILABLE IN AMBER



AMBER

- ⌚ Excellent bendability
- ⌚ High Resolution by photolithography
- ⌚ High Reliability
- ⌚ Simplified process
- ⌚ UL94 VTM-0 certified
- ⌚ Low spring back force
- ⌚ Resolution Line/Space=50/50μm
- ⌚ Solder mask defined opening (30μm diameter)

OUR BIGGEST FLEX YET!

Contact your local sales representative today!

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



Follow us on social media



Tribal Knowledge: Friend or Foe?

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

The ongoing retirement of many of our colleagues has cast a spotlight on this month's topic: tribal knowledge. As designers and engineers with 30 or 40 years of experience start pricing condos in Boca Raton, the entire industry is wondering: How will we hand down the knowledge acquired by these "silverbacks" to the next generation of designers? How do we know we're not handing down tribal knowledge to the new crop of designers?

If we're going to discuss tribal knowledge, perhaps a definition is in order. iSixSigma has spent a lot of time studying tribal knowledge, and the company defines the term this way:

Tribal knowledge is any information pertaining to a product or service process that resides only in the minds of the employees. The information may reside with one or many employees, and

it may vary between employees, but it is undocumented in nature.

"Undocumented" means the method may or may not be the most efficient way of performing the work. It may not even be an effective or correct way to perform the work. It also means multiple employees are likely to perform the work in different ways, based on their own version of tribal knowledge.

I'll buy that; once it's documented, it ceases to be tribal knowledge. I especially like the part about employees all performing tasks in different ways based on their own tribal knowledge.

The term "tribal" conjures up images of island dwellers passing down myths and legends over millennia. Let's face it: Designers are basically a tribe, like the headhunters of Borneo. Your numbers are dwindling, you speak your own



language, and no one really understands what you do all day.

A CAD manager shared this illustrative tribal knowledge story with me: One of his senior designers started designing boards in the '80s, and his mentor had told him that every board needed at least 100 decoupling capacitors. The designer diligently sprinkled decaps like pixie dust on every board for 30 years. When the manager asked why he put so many decaps on every design, the designer said, "That's how I was taught." He kept decap distributors in business for years.

Tribal knowledge is present in every organization, no matter the size. Tribal knowledge isn't necessarily bad; all the processes that Bell Labs pioneered in the '60s and '70s started out as tribal knowledge. But there's a lot of bad tribal knowledge floating around out there. How do we distinguish tribal knowledge from documented facts?

In this issue, our expert contributors will provide readers with the tools and methodologies needed to identify tribal knowledge, as well as when to question such information, and how to document and transform tribal knowledge into a process.

We begin by interviewing Tamara Jovanovic, a designer who recently completed her master's degree in electrical engineering. She discusses how she identifies tribal knowledge, and when it's time to dig deeper when presented with suspect information. Next, IPC

instructor Kris Moyer explains the road signs that lead him to questionable data, and why you should ask experts to cite their sources. Alun Morgan lays out the need for better documentation for PCB materials.

Our columnists had quite a bit to say about tribal knowledge and their opinions varied. Michael Ford says it's almost always negative. Tim Haag believes tribal knowledge can be good, bad, accurate, or inaccurate. Martyn Gaudion discusses how to create an "informal information culture," and John Watson explains why every designer needs to be ready to step up and help counter bad information. Kelly Dack relates the tale of the "Five CAD Monkeys," and Joe Fjelstad shares a personal view of his experience with undocumented data.

On other topics, we have columns from Barry Olney, Matt Stevenson, and Vern Solberg. Anaya Vardya wraps up his series on final finishes.

Don't miss our IPC APEX EXPO special section, including interviews with IPC Design Competition contestants, as well as Kris Moyer, who explains the reasoning behind this year's more complex design and how he'll approach the competition next year.

See you next month. **DESIGN007**



Andy Shaughnessy is managing editor of *Design007 Magazine*. He has been covering PCB design for 23 years. To read past columns, [click here](#).

jobCONNECT007

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

jobconnect007.com



Feature Interview by the I-Connect007 Editorial Team

Tamara Jovanovic is a design engineer with Happiest Baby, a manufacturer of smart baby beds that alert parents if the infant needs attention and soothes the baby back to sleep. She also recently completed her master's degree in electrical engineering by studying around her work schedule.

Since Tamara has been absorbing new information from the halls of academia and her workplace, we asked for her thoughts on differentiating between tribal knowledge and documented fact. Is tribal knowledge a friend, foe, or a little of both?

Andy Shaughnessy: *Tamara, being a recent grad school student and working in the industry as a designer, you probably have some thoughts on tribal knowledge.*

As a person who started really young, in the beginning all you seek is help and validation

that you're doing a good job or confirmation that you're not making mistakes.

In those first few years, tribal knowledge is welcome. As a young person, you're fresh and eager to learn. You're trying to understand the ins and outs of your job and the industry. With time, you gain some experience, and you realize that suddenly, you have this knowledge that is your own.

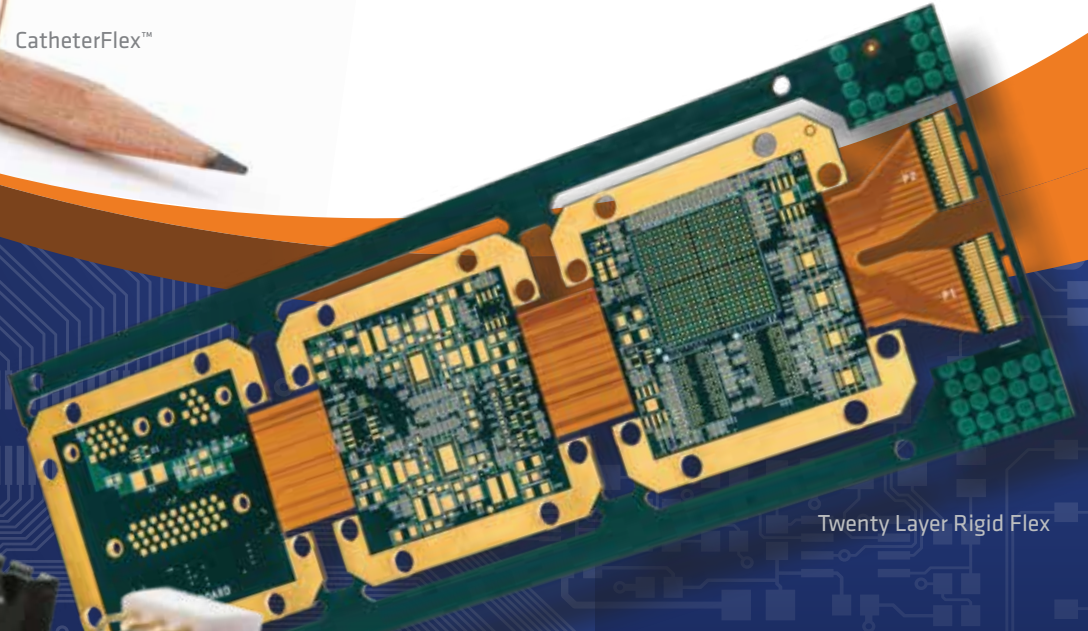
You slowly start to see certain things that may be outdated or that there are better ways to do things. However, I don't think tribal knowledge is a "fail," or necessarily bad. It's a great starting point, but the tech industry progresses so fast and there are new technologies and improvements coming out all the time. I need to do my own research.

Barry Matties: *Coming into the industry as a young person, and coworkers were sharing their knowledge, did you question it, or did you just accept it?*

ALL YOUR FLEXIBLE SOLUTIONS IN ONE PLACE



CatheterFlex™



Twenty Layer Rigid Flex



And, Assembly Too!

- ***Flexible circuit solutions***
- ***Rigid flex solutions***
- ***CatheterFlex™ solutions***
- ***Flexible heater solutions***
- ***And assembly solutions too!***

All Flex Solutions

1601 Cannon Lane
Northfield, MN 55057

(507) 663-7162
AllFlexInc.com



A little of both, really. At my company, it's not like they say, "Here's how we do things. You must follow these rules." It's more like, "You're young; here's what I know and how I do it." From there on, it's on me how I use that information. It's more of a positive way of sharing experiences and knowledge among each other.

We've always had a very young and small team, so we work very closely, and we are glad to get each other's input because you can look at something for hours and not realize that there is a small mistake right in front of you. A fresh set of eyes might spot such things immediately. We work together to evolve as a team and adapt better processes that would help save us time and money.

We have a very good culture of taking what we've already done with a grain of salt and seeing if we can improve it.

Matties: *What triggers the point where you say, "I need to look at this a little further?"*

There are just some processes where, if it seems too complicated or time-consuming, I'm thinking, "Could we maybe find something better than this?" I like to do my own research, regardless, so, I'm always looking for books, literature, or online content of what others are doing. Is there anything else out there that would make this easier or better? It's more like my curiosity is triggered by wanting to provide the best work for my teammates and my employers.

Matties: *There may be a lot of, perhaps, good and bad tribal knowledge on the design side, because there are so many ways to design a board.*

I think it's a matter of compiling what you already know and filtering that with what you learn from different sources. Obviously, some-

one who's been in the industry for 20 or 30 years has a lot more experience and is probably particular about the way that they like to design a board. It's something they would share. But as you said, there are many ways to do one task, and the more you know the better, as you can use the information at your disposal in your own way.

It's a matter of finding that middle ground. I love getting input, whether it's good or bad. Feedback makes us all better, and if somebody gives you advice, it's because they want you to do better. What we do with that feedback is completely up to us. Even if we don't use the advice, it can inspire us to find the best solution. In the end, you just have to find the best way to move forward.

Shaughnessy: *It's very subjective though. It's not like you can say, "Here's a hard set of points that I can apply to determine whether something's tribal knowledge." You just get a sense that it's not right.*

Matties: *To that point, how do you capture tribal knowledge? Is it a documented process that someone is sharing with you, or is it just words that are coming at you and it's up to you to document them?*

I would say it's a combination. In my company, we are, as I mentioned before, a small team, and it often ends up being my decision; there aren't a lot of documented examples of tribal knowledge. My team relies on me to do the right thing. People will give their input or present the tribal knowledge, but ultimately, it's okay. You may not be the expert, but you are the person in this field and the one responsible for a specific task. You should do your own research and make the best decision for the team. It is a bit subjective from that perspective.



Matties: *Perhaps information comes into a meeting, for example, and then someone must filter that information and make the choice to move forward on a particular path. Is that what I'm hearing?*

Yes, exactly. That's why we have experts and specific teams, whether it's mechanical, engineering, electrical, or firmware. We can't be micromanaging and checking what everyone's doing. There needs to be trust, respect, and the sense that your teammates will make the best decision for the project. There are discussions and constructive conversations where we share our ideas and opinions but, in the end, you're responsible for your own portion of the project.

Matties: *The key word is "trust." When someone's sharing tribal knowledge, the more you trust that person, the more likely you will accept that knowledge.*

That's true. After working with someone for many years, you see how they operate, and you can predict the decisions they are likely to make (or not make). It is easier to trust someone you know well and who you have experience working well with, as you know it worked in the past.

Matties: *Where do you begin?*

It starts with Google, but there's a lot of information online. There's definitely a filtering process to find out what's accurate and who to trust. I like to visit related blogs and read a little bit of everything. I like hearing real people talk about their issues and what they've struggled with. I also read forums



Tamara Jovanovic

from companies that sell specific chips or materials, depending on what I'm working on. That's my starting point, and then it's just a matter of where your research takes you.

Matties: *But Google can be an overwhelming resource that requires hours of filtering. I go back to trust. Often you don't know which "expert" you can trust. There are tons of unlimited streams about any topic, so it really becomes a challenge, doesn't it?*

Yes, the internet can be great, but it can also be a very dark place. If I'm working on a specific design with a specific chip, for example, TI has a variety of blogs, forums, notes, and good documentation, so it's good to start somewhere like that. Much of it is open source so you don't know who you can trust. You just have your own personal knowledge to go on and help you understand what is good and what's not; you need to learn to filter everything you see.

Shaughnessy: *As Ronald Reagan once said about dealing with the Soviets, "Trust but verify."*

Yeah, exactly.

Shaughnessy: *Congratulations on recently earning your master's degree while working full time as a designer. Did you learn anything in your graduate studies that contradicted what you saw in the industry or vice versa?*

Thank you. I wouldn't use the word "contradicted," but sometimes I might have felt that I was learning some outdated material, or going

through programs or information and realizing that it is no longer being used in the industry. The program I attended is very focused on the industry and what is being learned now so most of it has been very useful.

Matties: *You're dealing with coworkers from different generations. Do you find that some people are reluctant to change or to adopt something new?*

I don't personally run into that a lot in the workplace, but I have heard about it. Doing a certain process and then having to start over and try a new process or system can be frustrating at times, and it can be tough to adapt at first. But with a lot of practice and patience, everything comes together.

Matties: *How complex are the boards that you design and build?*

They are boards for consumer electronics. I hear about companies building 16-layer boards but the ones I work on are nothing like that. We have some complicated designs and some that are simpler. Usually, it's a multi-board assembly and there are processors and chips that are more complex.

Matties: *Do you bring in your supply base, EMS, fabricators, and material suppliers as part of your knowledge pool?*

Yes, on a regular basis. Since I am the one who is designing the boards I have a lot of contacts within the manufacturing industry, quick turn or otherwise. What's expected of me is to provide some sort of comparison and determine which manufacturer fits our needs the best, whether it's from the perspective of lead time, cost, or quality.


Matties: *How important is it for designers to understand the manufacturing process of a circuit board?*

It's very important. You have to know your manufacturer's capabilities, what they can and can't do. You have to design a board that is manufacturable and be sure that the processes used won't be too expensive because of poor design decisions. Those boards go in a consumer product, so cost is everything. Understanding the process helps you within that framework, and helps you take your design to completion without having to worry about radical redesign later.

Shaughnessy: *What are your thoughts on datasheets? Lee Ritchey famously said that you should assume they're wrong until you've proven otherwise. Again, trust but verify.*

Some datasheets will give you the exact details on how to use the parts in very different applications, and sometimes there are even additional application notes with further design suggestions and guidelines. There's such a variety of datasheets, though, and some are very detailed and useful, but in most cases it's the other way around. The basics are good for component selection but when it gets to the nitty gritty of hardware design, you have to figure out on your own how to implement it properly, because you can't just be creating designs that are full of noise or susceptible to crosstalk.

What if you must use a part that has really bad documentation? Then you're stuck and you must find other designers online who have used that part and had experience with it. In general, it feels as if there isn't enough information on specific designs available. With a BGA footprint, for instance, what is the most



With a lot of
practice and patience,
everything comes
together.

optimal layout? With chips like DC-to-DC converters, there is a specific way to lay them out properly, and that information may not always be in the datasheets.

Matties: What's your priority on training?

I'm an EE at a small company; it's very seldom that I have a lot of free time just for doing research, so I try to do it on my own time. Even before I start the schematic, I like to go in depth with all the information that I have available. I would say I spend at least a week doing research and getting familiar with the parts so I can understand how they're supposed to work within the system that I'm designing.

Matties: You've been in your career for a few years. What is the best piece of advice, either tribal knowledge or otherwise, that you've been given?

The best advice I've had is to take a step back and look at the problem from a different perspective. Here's an example: Recently, we were working on a design, and we had all the parts picked out, but halfway through we realized that we were fitting the design around the parts that we had chosen rather than making sure the design was right. We were so blinded by these new devices that we thought would be really good for our project, only to realize we were compensating on the design to use these specific parts; generally, that's not a great practice. So, it's good to take a step back and see if you're fulfilling the requirements for the project you're working on and not the other way around.

Matties: So, how did you choose this career? What was so appealing about it?

When I was younger, I liked science and electronics, and it seemed like a natural path forward. I studied electrical engineering as an undergrad. I realized that with engineering I could use it to help people all over the world by making life-changing devices. When I started out at Happiest Baby, PCB design was part of the job. Working here has allowed me to learn a lot more about PCB design and run with it and be creative. It's been a great opportunity and I'm really grateful for it. I still get to do the hands-on part of engineering: designing, testing, and troubleshooting, which I also very much enjoy. I get to focus on design problems, like the best way to lay out a board, or how to get the design to do exactly what we need.

Matties: Do you have any advice for new designers who may be sorting through tribal knowledge?

Don't do something just because someone told you that it's the right thing to do. Make sure that you understand why it's the right thing to do.

Don't do something just because someone told you that it's the right thing to do.

Matties: That's really good advice.

You'll still make mistakes; I'm not perfect by any means. I definitely have made my share of mistakes. But mistakes are okay. Learn from them and try not to make the same one again.

Shaughnessy: This has been great, Tamara. Thank you.

Thank you. I enjoyed it. **DESIGN007**

The Interaction of Electromagnetic Fields

Beyond Design

by Barry Olney, IN-CIRCUIT DESIGN PTY LTD / AUSTRALIA
with Special Advisor Rick Hartley, R HARTLEY ENTERPRISES

When two (or more) electromagnetic fields overlap or meet, they add vectorially at each point in space. Fields have direction and polarity. At any point in space, there can be only one field, so at some spatial points, they will cancel each other, and at others, they will re-enforce each other. James Clerk Maxwell described electromagnetic fields as being linear. Linearity implies superposition, meaning that the fields do not merge with each other but rather add vectorially—distorting the signal. This applies to both static (DC) and time-varying (AC) fields.

When an electromagnetic field propagates along a transmission line, there are electric field lines between the signal and return paths and rings of magnetic field lines around the signal and return path conductors. These fields are

not confined to the immediate space between the signal and return paths but rather, fringe fields spread out into the surrounding volume.

In a stripline (inner) configuration (Figure 1a), the electromagnetic field propagates between the planes, and the return displacement current flows in close proximity (mainly) to the nearest plane. As the frequency increases, the current is forced into the outer surface of the copper, due to the skin effect, dramatically increasing loss. Figure 1b shows the electromagnetic field in a microstrip (outer layer) configuration. Electric fields terminate when they come into contact with a solid plane, whereas magnetic fields tend to radiate into the dielectric or air above or below the PCB. However, stripline (embedded signals) are rather confined by the adjacent planes and only emit radiation on the

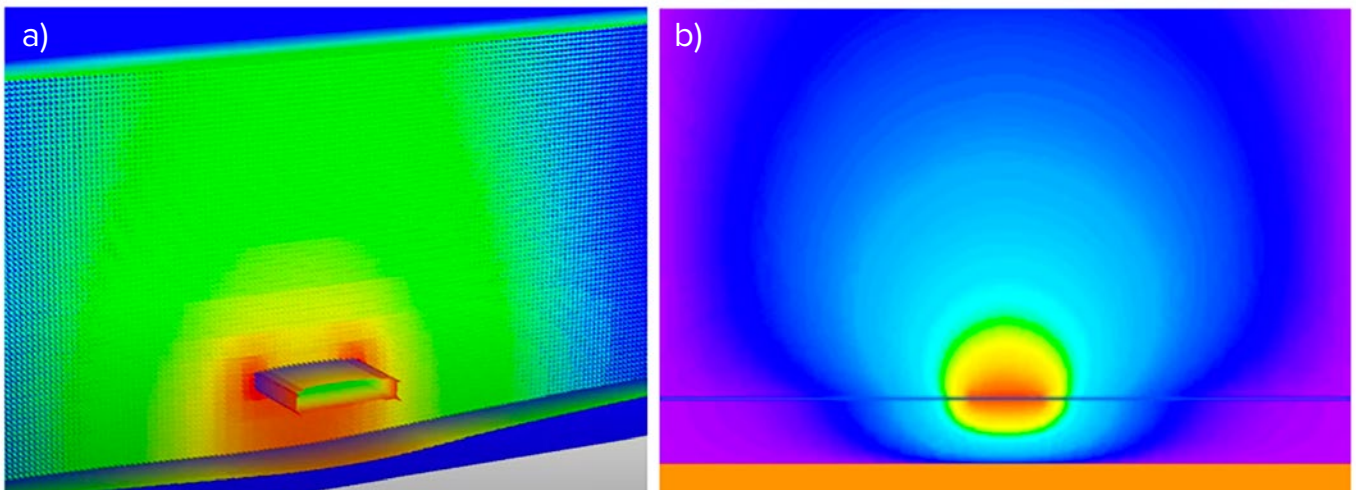
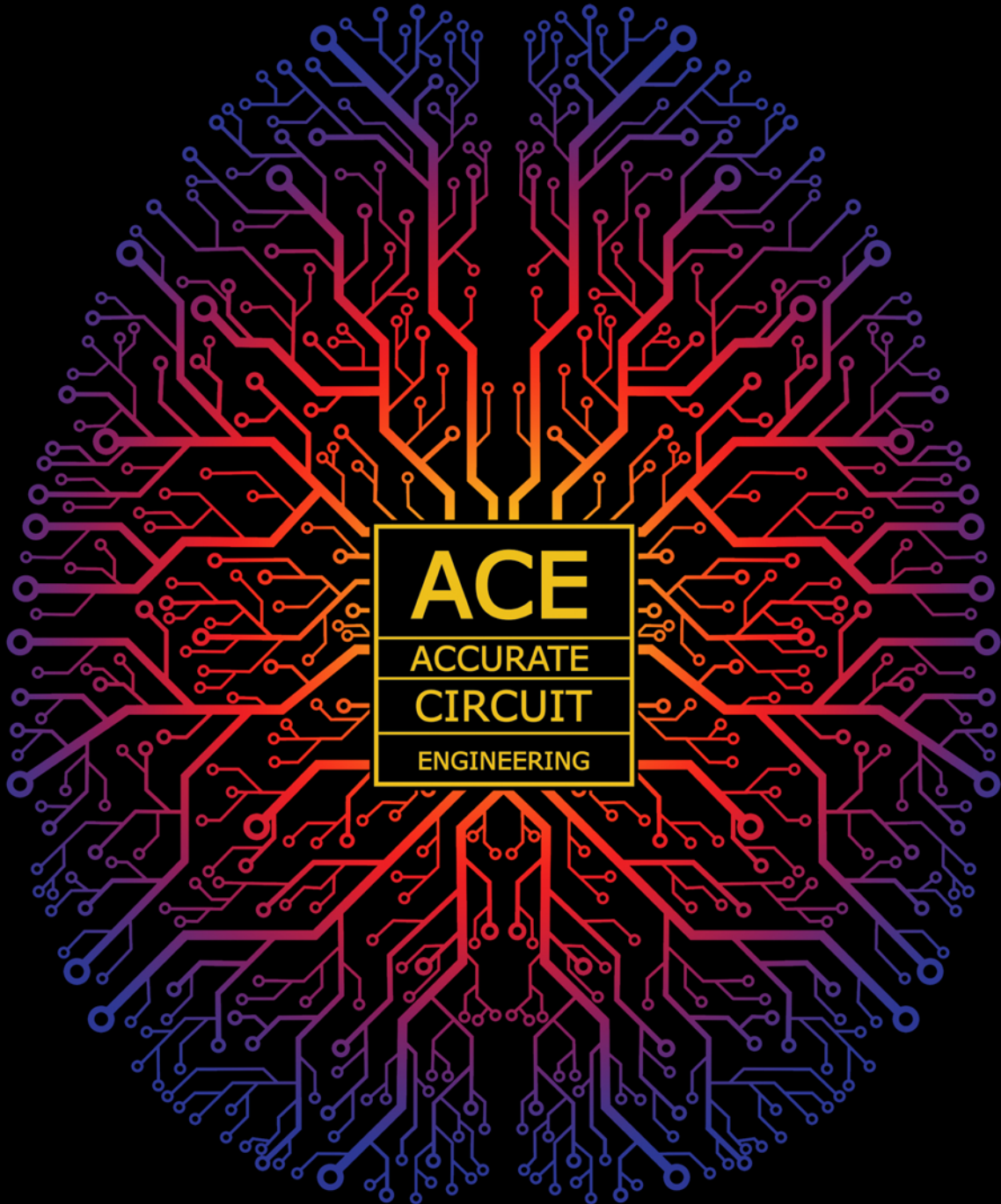


Figure 1: a) EM fields in stripline configuration.; b) Microstrip configuration.
(Source: Simboer & QuickField)

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

fringing fields of the PCB edge. This is a good reason to embed all critical signals.

Both electric and magnetic field coupling can occur on PCBs; however, crosstalk usually comes from electric field-based capacitive coupling. Trace segments running in parallel on the same layer or running vertically (broad-side) between two layers are extremely susceptible to crosstalk. Parallel traces act as capacitor plates separated by a dielectric. If there is a change in voltage, it will alter the electric field and generate a displacement current leading to crosstalk. Mutual capacitance between the two lines couples energy in both directions on the victim trace (Figure 2a).

Inductive coupling depends on the magnitude of the magnetic field surrounding the victim trace. The magnetic field change generated by the aggressor signal surrounding the victim trace results in a changing conduction current. Mutual inductance between the lines couples energy onto the victim trace segment in the backward direction (Figure 2b).

Though both mutual capacitance and mutual inductance play a role in crosstalk, there are two regimes to consider. When the return path is a wide, uniform plane, as is the case for most coupled transmission lines in a mul-

tilayer PCB, the capacitively coupled energy and inductively coupled energy are of the same order of magnitude. However, when the return path is a single lead in a package, or a single pin in a connector, the inductively coupled energy is much larger than the capacitively coupled energy. In this case, the noise behavior is dominated by the inductively coupled energy. The noise on the victim line is driven by a dI/dt in the aggressor net, which happens at the rising and falling edges of the signal when the driver switches.

Electromagnetic fields have direction and polarity. Forward (or far-end crosstalk, FEXT) and backward (or near-end crosstalk, NEXT) refer to the direction the crosstalk travels along a victim transmission line. Forward crosstalk propagates in the same direction as the aggressor signal, while backward crosstalk propagates in the opposite direction.

- FEXT = (Capacitive coupling – Inductive coupling)/2
- NEXT = (Capacitive coupling + Inductive coupling)/4

In an outer layer microstrip configuration, the mutual capacitive coupling between adjacent traces is generally weaker than the mutu-

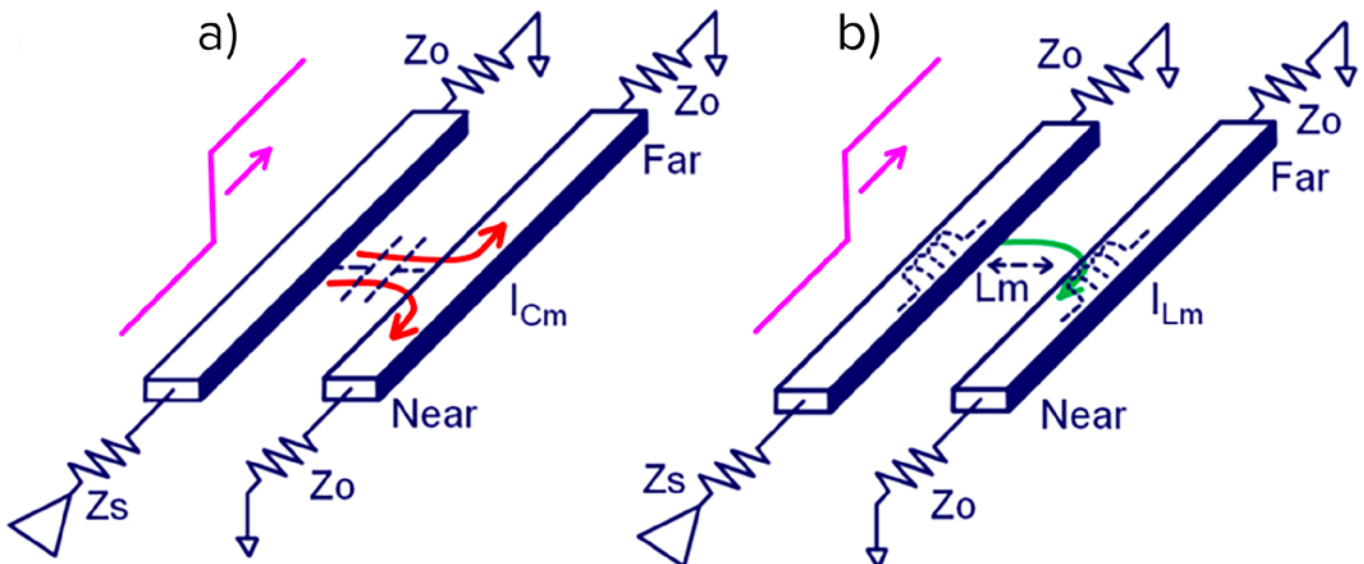


Figure 2: a) Capacitive coupling; b) Inductive coupling. (Source: Intel)

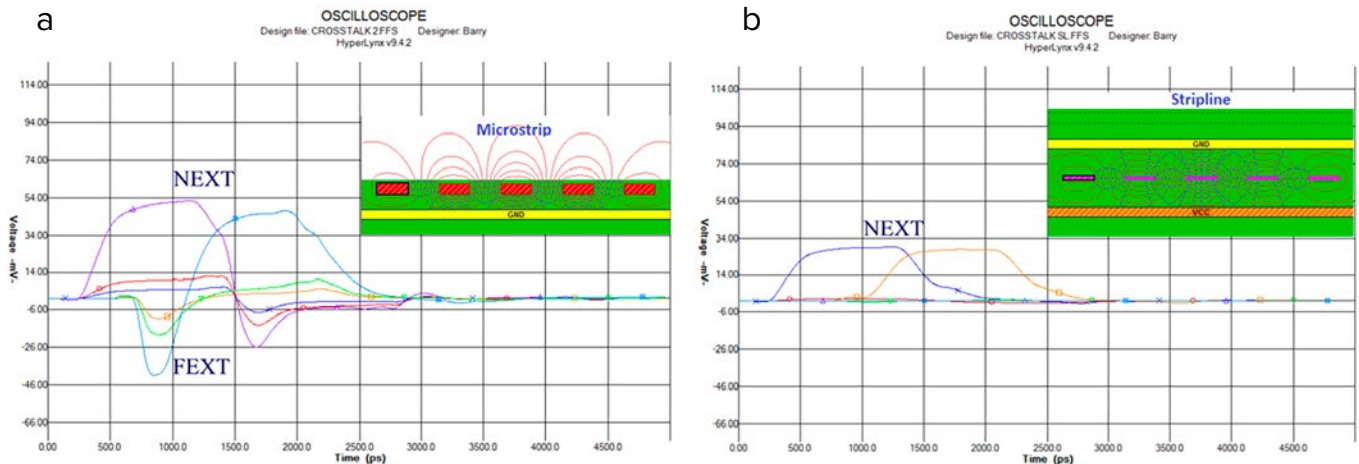


Figure 3: Forward FEXT; and backward NEXT crosstalk for a) microstrip and b) stripline configurations.

ally inductive coupling, driving the FEXT coefficient negative (Figure 3a). However, forward crosstalk does not exist in the stripline configuration. The fine balance between capacitive and inductive coupled crosstalk produces no observable forward crosstalk (Figure 3b). And, the NEXT component is halved. So, all other factors being equal, here is just another good reason why one should always route critical high-speed signals on the inner layers of a multilayer PCB.

Figure 4 illustrates the “backward crosstalk critical length.” This is a measure of how far an aggressor line and a victim line can route in parallel before the amplitude of backward crosstalk (NEXT) reaches its maximum level. Parallelism beyond this length will widen the coupled pulse in the victim line, but will not increase its amplitude. As rise time gets faster, “critical length” becomes shorter. In today’s systems, with extremely fast IC output switching times, almost any parallelism

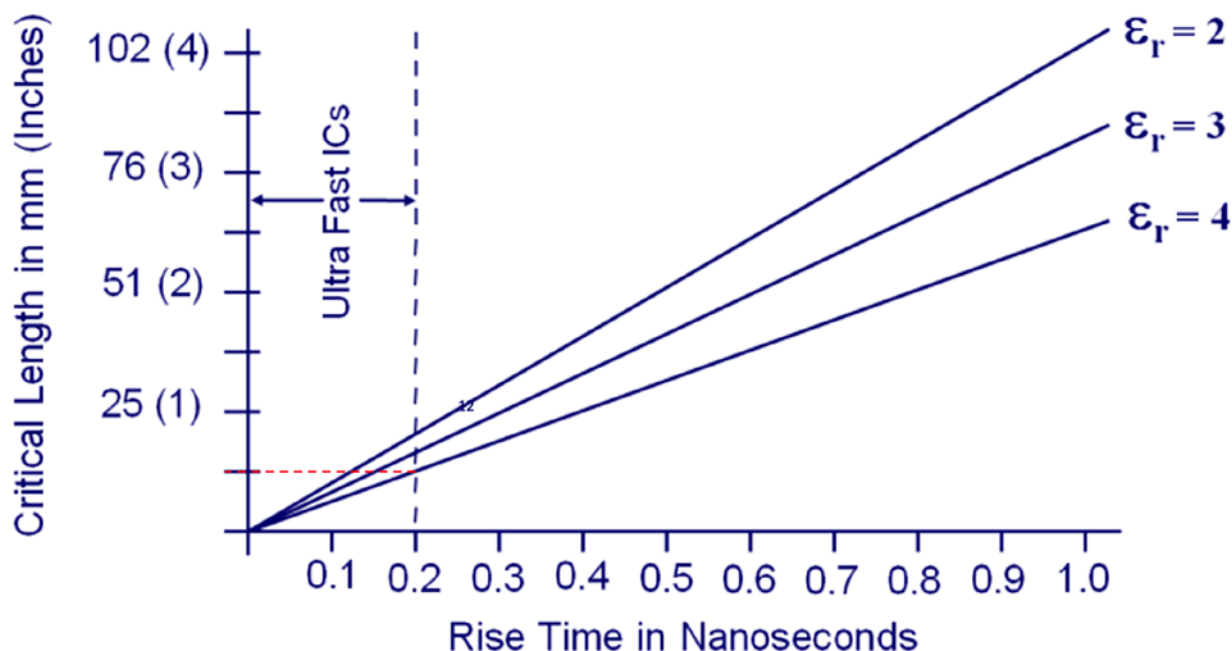


Figure 4: Backward crosstalk critical length.

between lines can be a problem. As seen in the graph of Figure 4, with a rise time of 200 ps, backward crosstalk reaches its maximum amplitude in 12.5 mm (1/2 in) of parallelism between lines. The absolute amplitude is determined by the distance between the aggressor and victim lines relative to their height above the reference plane (ground plane). For this reason, it is extremely important to route all traces on a layer that is one dielectric distance above or below a ground plane and to keep the dielectric thickness as thin as reasonably possible, within the constraints of good manufacturability rules.

Having a wide uniform plane as the return path is the configuration for the lowest radiation and crosstalk. Anything that changes the return path from a wide uniform plane will increase the amount of coupled noise between transmission lines. Usually this happens when the signal goes through a connector, and the return paths for more than one signal path are now shared by one of the pins rather than by a plane—the inductively coupled noise increases much more than the capacitively coupled noise. Simultaneous switching noise occurs mostly in connectors, IC packages, and vias, where the return path conductor is not a

wide, uniform plane. Ground bounce is really a special case of crosstalk caused when the fields from several lines overlap and couple into one another. As the fields all attempt to couple to a single reference line in the connector or IC package, or into a single ground via, the return currents overlap, creating very high mutual inductance.

Engineers often employ guard traces to help lower crosstalk. In a well-designed PCB with the ideal number of ground planes, guard traces offer little, if any, value unless properly attached to the ground plane(s) of the board. An ungrounded guard trace (Figure 5) or one attached to the ground plane(s) at one or both ends only (middle) can create a bandpass filter—an LC network—that radiates at one-quarter wavelength of its resonant frequency. Whereas, a guard trace that is grounded every 1/12th wavelength (bottom) prevents radiation and drastically lowers cross-coupling.

On one- and two-layer PCBs, with no ground planes, even guard traces that are connected to the 0V reference points at both ends only, are often better than no return reference. That said, employing one- and two-layer PCBs without planes is a very poor design practice. The traces have an impedance of 376 ohms

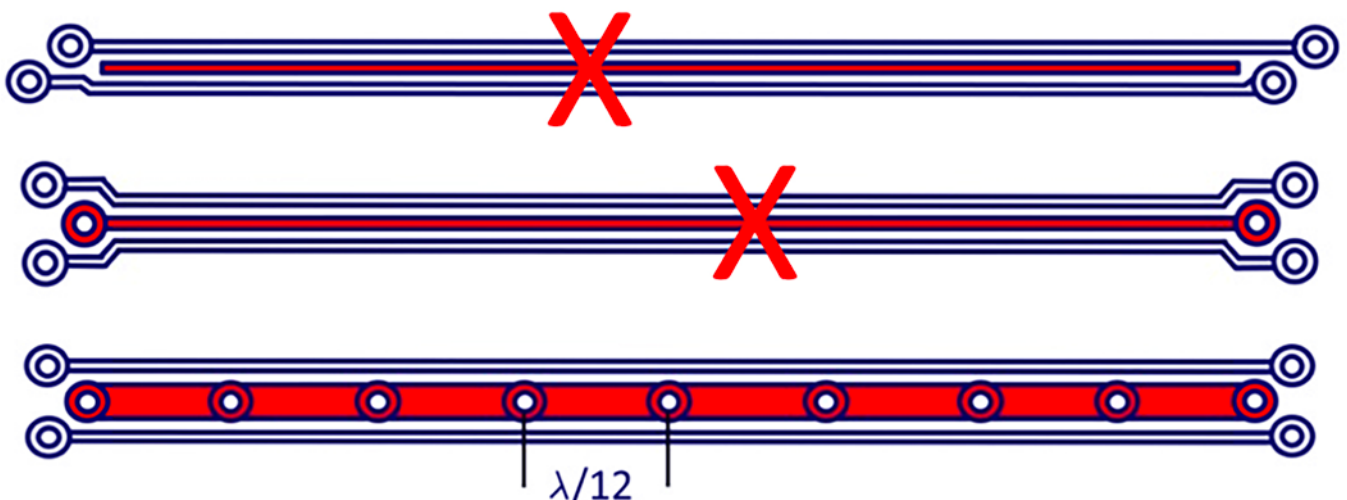


Figure 5: Ground guard trace routing.

(impedance of free space) and will often cause more problems than they solve.

To minimize the detrimental interaction of electromagnetic fields, it is good design practice to:

- Separate asynchronous signals, unrelated signals, and signals of different potentials.
- Avoid long parallel trace segments to minimize their coupling.
- Route all critical signals embedded between the planes to reduce crosstalk.
- Keep other signals away from I/O signals and connectors. Never allow anything to route into the I/O area of the board, except I/O lines.
- Use low dielectric constant material and closely couple the signal trace to the reference plane to decrease the radiation.

Key Points

- EM fields do not merge with each other but rather add vectorially—distorting the signal.
- Crosstalk usually comes from electric field-based capacitive coupling of trace segments running in parallel.
- Mutual capacitance between the two lines couples energy in both directions on the victim trace.
- A wide uniform plane as the return path is the configuration for the lowest radiation and crosstalk.
- Mutual inductance between the lines couples energy onto the victim trace segment in the backward direction.
- When multiple fields couple to a single reference line, in a connector or IC package, or into a single ground via, the return currents overlap, creating very high mutual inductance.
- Electromagnetic fields have direction and polarity.

- Forward crosstalk does not exist in the stripline configuration due to the fine balance between the capacitive and inductive coupling.
- For a rise time of 200 ps, backward crosstalk reaches its maximum amplitude in 12 mm of parallelism between lines.
- The amplitude of crosstalk is determined by the distance between the aggressor and victim lines relative to their height above the reference plane. It is extremely important to route all traces on a layer that is one dielectric distance above or below a ground plane.
- A guard trace that is grounded every 1/12th wavelength (bottom) prevents radiation and drastically lowers cross-coupling. **DESIGN007**

Resources

- Beyond Design columns by Barry Olney: “Return Path Discontinuities,” “The Dark Side—Return of the Signal,” “Stackup Configurations to Mitigate Crosstalk.”
- “Noise, EMI & SI and Single-ended Crosstalk,” by Rick Hartley.
- “Signal and Power Integrity Simplified,” by Eric Bogatin.



Rick Hartley is the principal engineer at R Hartley Enterprises and has been in the industry for over 50 years. He is one of the primary consultants for PCB manufacturing and design companies. Rick has

also conducted classes worldwide on EMI, signal integrity, and various other electrical topics for the last 30 years.



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 incor-

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



True Experts Can Cite Their Sources

Feature Interview by the I-Connect007 Editorial Team

We've heard a lot lately about the need to identify tribal knowledge within our organizations. How do you know whether an "expert" is sharing documented knowledge or it's just something they learned at their first job during the Carter administration?

We asked IPC design instructor Kris Moyer to explain his process for separating the wheat from the chaff, so to speak, in design knowledge. As he points out, a true expert will not be afraid to cite the sources and data sets behind their arguments. Ask questions; maybe there is a reason why they "always did it this way."

Barry Matties: *Kris, what are the signs when you think someone is presenting tribal knowledge?*

When someone says to me, "This is the way I was told to do it," I say "Why?" If their only answer is, "That's the way we've always done it," I say, "Do some research and understand what the history was."

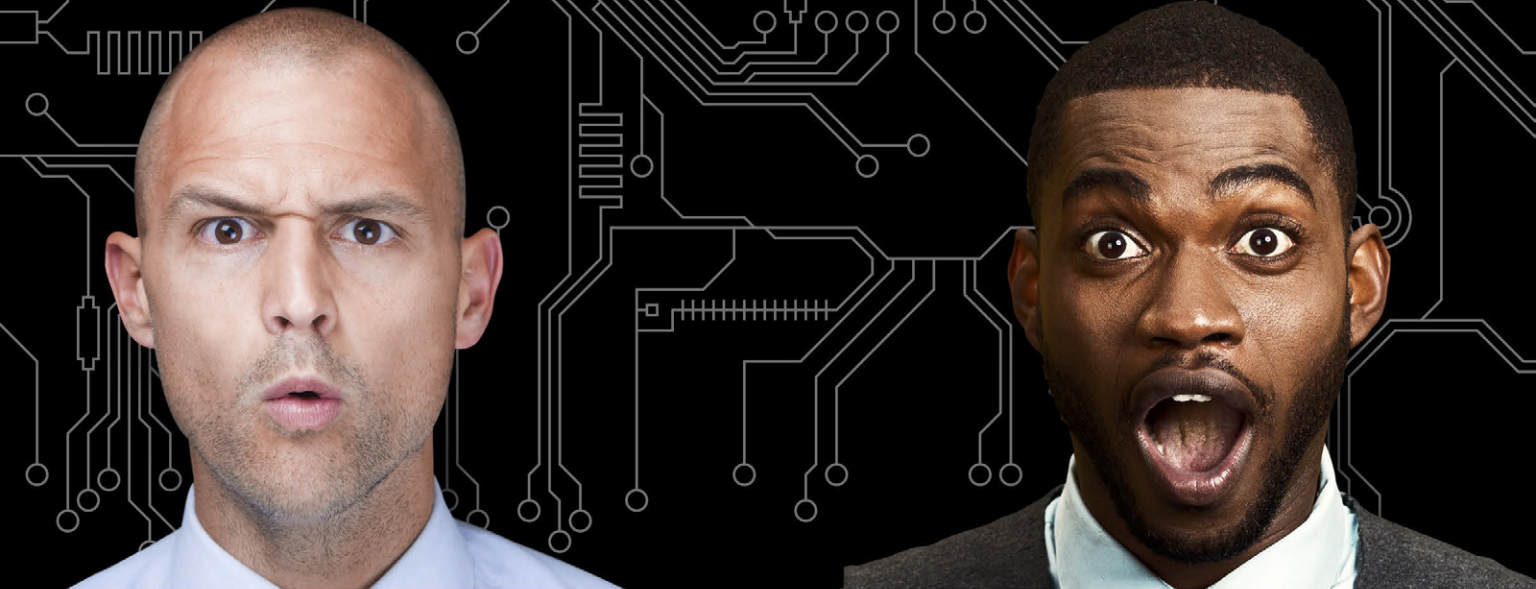
It is always important to have a history of the rationale behind decisions that were made on the design so that we don't lose that knowledge. We may have done things a certain way at one point in history, but we must back everything up with rationale and quantified engineering data. That's the biggest problem with tribal knowledge: "This is how we've always done it. The company way is fine." If you can answer why and have a technical rationale of why it's still valid, I'm all for it. Otherwise, I will push until you sit there and give me a technical rationale for why it's still necessary. Can you prove you're right?

Matties: *But when you're sitting in the classroom listening to a PCB design instructor, how do you validate whether that's just tribal knowledge? Do you accept that as the truth?*

First, is the instructor a well-respected PCB design instructor? They should be current on their data and able to cite and show their data

From Who? To Wow!

Stronger Together To Serve You



APCT Acquires Advanced Circuits, Inc.

APCT Then

2014 | \$10 Million | 1 Location

APCT Now

2023 | \$200 Million | 7 Locations

New Offerings Include:

- **Standard Technology**
 - 2-Layer Boards Built Domestically in 1 day
 - 4-Layer Boards Built Domestically in 1 day
 - Oversized Boards Up to 37" by 120"
- **Advanced Technology**
 - Rigid Through-Hole up to 40-Layers
 - HDI; Blind/Buried/Stacked Vias
 - Flex, Rigid Flex up to 22-Layers
 - RF Design Expertise
 - Cavity Boards
 - Heavy Copper up to 20 ounces

APCT Leading the Printed Circuit Board Industry



APCT
Passion | Commitment | Trust



**ADVANCED
CIRCUITS**
A Division of APCT

APCT.com | 4PCB.com

sources. The best teachers in the world love to be challenged. When a student says to me, “I came up with X, Y, and Z for that,” I’m always willing to look at their data, and see if it fits. If necessary, I’ll update my information.

Colleges and organizations love to be challenged on their data. You name it: IPC, IEEE, Johns Hopkins University Applied Physics Lab.

Any organization that does engineering, data engineering, or research engineering white papers is usually more than happy to answer your questions.

Matties: *Oftentimes, you ask your question, and the instructor says, “I have 40 years of experience. That’s the source.” But that’s also the danger.*

A great instructor will say, “Not only do I have 30 years of experience, but here’s the data; here’s the math that proves this. Here is the data set.” Then you can run your own analysis on that. Prove that it works.

Nolan Johnson:
It seems as if tribal knowledge could, and probably does, end up in our procedures and standards. I thought, “Doesn’t ISO 9000 help fix some of that?” Based on some of our other discussions, the answer there may be no. You may be documenting your tribal knowledge into your processes. Isn’t that right?

Sure. Let’s say the task is to tighten a Phillips head screw. If you were to describe this process, you’d say, “Insert the screw until it’s hand tight, and then hand-tighten it with a hand screwdriver.” But if you write the statement



Kris Moyer

in ISO 9000, it might say, “Install the screw using a Craftsman #2 Phillips head screwdriver.” You would probably get specific. If the next technician comes in and uses a Stanley screwdriver instead of a Craftsman, you just violated ISO 9000.

ISO 9000 is just a way to document your procedure and then file the procedure that you document.

So yes, you can get tribal knowledge into procedures and it’s up to the individual companies to determine what’s critical to have in the document.

Andy Shaughnessy: *What’s the best way to determine whether you’re dealing with tribal knowledge?*

Step one, from a board design point of view: Read and understand the various IPC standards and specifications for all aspects of PCB design. Understand the guides for design, fabrication, and assembly, so that you see the big picture. Then ask them to cite their sources. Go through that level of engineering analysis.

If you’re ever told, “I have 30 years of experience,” when you ask a question, ask them to cite their data source. If you disagree with it, look for an independent third-party expert in the field. It’s no different than getting a second opinion from a doctor. If you have any concerns with this data, repeat step one—understand the IPC standards; step two—ask them to cite their sources. You could even get a third opinion if you need to.

Matties: *Thank you, Kris. This has been great.*

Thank you all. DESIGN007

Any organization that does engineering is usually more than happy to answer your questions.

The most complete event coverage the industry has ever seen.



- Interviews
- Photo galleries
- Awards coverage
- Commentary

[Download Now](#)



MilAero007 Highlights



Time to ‘Finish the Job’ on PCB Funding ►

In his State of the Union address in early February, President Biden departed from his usual topics about education and the economy to mention the new CHIPS Act, as well as supply chain and infrastructure issues. For perspective on those remarks, Nolan Johnson chats with David Schild, executive director of the Printed Circuit Board Association of America (PCBAA).

Real Time with... IPC APEX EXPO 2023: Taking Our Message to Washington ►

Nolan Johnson talks with Chris Mitchell, vice president of Global Government Relations at IPC, about key issues IPC is working on today and how you can be involved in advocating for our industry in 2023. Their conversation also covers the CHIPS Act, advanced packaging, and much more.

A Conversation with ‘The Space Gal’ ►

In this interview with the I-Connect007 Editorial Team, Emily Calandrelli talks about her unconventional entry into science, what’s ahead for space commerce, advice for industry leaders, and what she really thinks about going into space.

Printed Circuit Boards Have Champions on Capitol Hill ►

IPC’s vice president of global government relations, Chris Mitchell, shared this letter, sent to DoD on Wednesday by Reps. Anna Eshoo (D-CA) and Blake Moore (R-UT), which insists DoD must “leverage all available resources, including the use of Title III of the Defense

Production Act (DPA), to increase domestic production of PCBs and IC substrates.”

Real Time with... IPC APEX EXPO 2023: PCBAA Legislative Update ►

PCBAA Chairman Travis Kelly and Editor Nolan Johnson discuss the association’s activities, including legislation in the range of \$3 billion to \$5 billion that follows the CHIPS Act, and why Congress needs to support the PCB industry as they did the semiconductor segment.

EIPC Winter Conference 2023: Day 1 Review ►

The EIPC Winter Conference returned to the Metropolis of Lyon in eastern France this month. In 2018, the venue was Villeurbanne in the Auvergne-Rhône-Alpes region. Five years later the setting was the Groupama Stadium in Décines-Charpieu, and leaders of the European printed circuit community gathered in expectation of a spectacular programme of 16 presentations, a visit to a nuclear power station, and an invaluable networking opportunity. They weren’t disappointed.

Terran Orbital Wins \$2.4 Billion Contract to Build 300 Satellites for Rivada Space Networks ►

Terran Orbital Corporation’s subsidiary Tyvak Nano-Satellite Systems has been awarded a \$2.4 billion contract to design, build, and deploy 288 low-earth orbit satellites for Rivada Space Networks. As part of the contract, Terran Orbital will also develop 12 “spare” satellites to produce a total of 300 spacecraft.


The RF Specialists


- ISO 9001:2015 Certified
- AS 9100 Rev D Certified
- MIL-PRF-31032-C Qualified
- MIL-PRF-55110 Qualified
- ITAR Registered

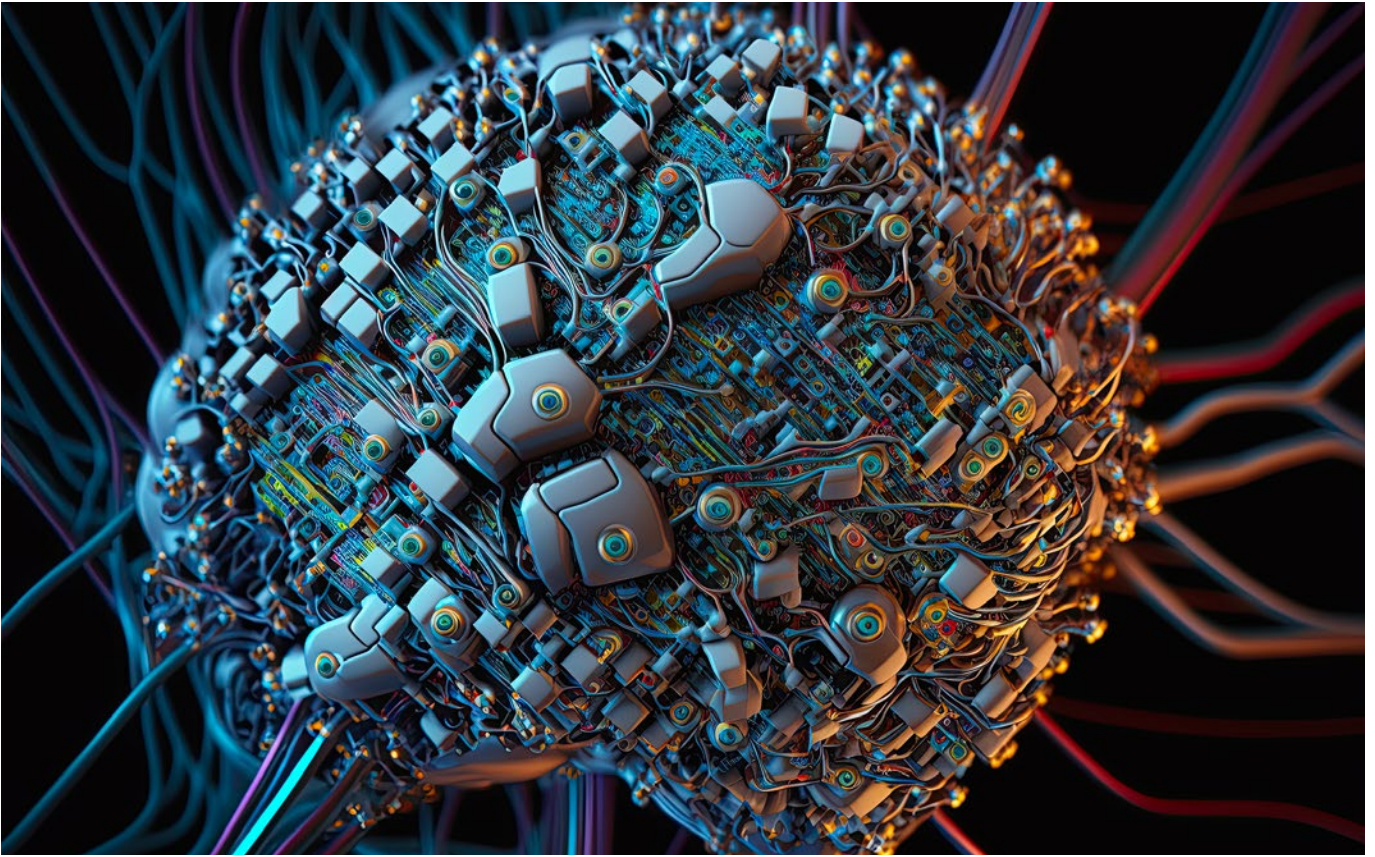


Prototron Circuits
America's Board Source

Serving the industry for over 35 years

 prototron.com

 (520) 745-8515



Embedding **Know-how** Into Automation

Feature Interview by Barry Matties

I-CONNECT007

Michael Ford, senior director of emerging industry strategy for Aegis Software, weighs in on the topic of tribal knowledge, its pros and cons, and its role in our industry. While it may be useful if your team never changes, the inevitability of a person's career progression means they take their knowledge with them. Michael brings his nuanced perspective to the conversation.

Michael, is tribal knowledge a friend or foe? What's your initial reaction?

My initial reaction is that tribal knowledge is definitely a foe.

Why do you say that?

Manufacturing has long been dependent on a significant amount of legacy skills. These have accumulated over many years as we've gained experience and know-how across various technical areas of operations. But as our experts grow older, the younger people coming in behind them don't have the opportunity to gain that same experience first-hand. We are, therefore, destined to lose the experience and know-how unless we can capture it somehow. Manufacturing cannot be sustainable if it continues to be dependent on that tribal knowledge.

Short of downloading somebody's experience into a computer, and transferring it to another person, the more practical approach is to represent that know-how within software



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

associated with machines and software solutions. Automation technology is developed using that know-how, so that key manufacturing dependencies on specific people are avoided. The value from that experience is easily realized in their choices of machines and software tools.

What I'm hearing is that tribal knowledge is valuable, but it will fade away unless we capture it.

There is a simple, but unfortunate pattern that emerges when we try to transfer knowledge from one person to another. Initially, you train someone to perform a certain role, and over time, they become absolute experts in what they do. The time comes, however, for them to move on. Everyone likes the opportunity for career progression. Their replacement must be trained. Will 100% of their knowledge and experience be transferred? No.

Some of that knowledge may not even be valid today, and that's where it gets tricky.

The priority is always to transfer the part of the knowledge that is in use, relevant to the current operation, which represents a fraction of their total knowledge. The same cycle then repeats, reducing the amount of knowledge and know-how overall. This is a natural process, as there is limited time in which to do those transitions. The worst part is that the operation appears to be operating as required but is actually becoming totally inflexible. Those performing their tasks know less and less about why they're doing it. They are only doing what they've been told and are not confident to deviate from that operation in any way because they don't know the reasons behind the tasks. Inflexibility breeds inefficiency, because where there needs to be change and evolution in the way that things are done, the ability to



Michael Ford

adapt and optimize are simply no longer there.

Who's responsible for deciding which knowledge is transferred?

It follows the law of diminishing returns. If the manager wants to make sure that the training is more comprehensive, whether for people moving internally or being replaced from the outside, resistance to

do so increases as the relevance to the immediate operation reduces, making it a lower priority over time vs. daily operational challenges. Clearly, the responsibility for drawing the line falls on managers. It's difficult to balance these needs for training against the needs of continuous operation. You can't blame them for that; it is, I'm afraid, a natural process.

Obviously, there is value if the tribal knowledge is captured correctly and validated within software tools. You still have to validate things one way or the other. I'm thinking this applies even down to the layout of a board, because a PCB can be designed an unlimited number of ways.

Exactly. I remember my relatively few days of designing printed circuit boards years ago. It was hard work having to engineer every aspect of the layout yourself, with only simplistic word-of-mouth design rules. Over the years, we've seen the know-how of design layout becoming increasingly automated within software, as know-how is incorporated into the software little by little. Over time, it became easier and quicker to perform the PCB layout. It soon got to the point where there was no longer any need to do all that initial low-level layout work. Today, you simply rely on the software tool's algorithms to put the components where they ought to be, and then route the layout, basically, replacing all the boring,

repetitive, and tedious work. Is there fine-tuning still to do? Sure, but hopefully, only a few tweaks. Software algorithmic and data content advances will one day resolve more outstanding issues toward that last little bit of automation. That is how we improve. The design engineer now completes many more layouts, even with massive complexities, in a way that is a lot less stressful, with few opportunities for mistakes. This example is relevant to many applications of software-based decision-making.

Does that mean there's ultimately no future to accumulate know-how in manufacturing?

Absolutely not, and in fact, as you go through this process, the role of the human is elevated within the manufacturing operation. Instead of having a really deep, specific technical knowledge that narrows their area of expertise and opportunity for contribution, you have people who are capable of identifying and understanding challenges holistically, deciding what needs to be done and how best to do it; they are aided by information, and based on accumulated know-how, it's built into the software. The focus of knowledge, experience, and expertise for the person becomes how to set up and configure such software tools, how to utilize them effectively, and to know how and when to use which tools to get the best possible result. Rather than being restricted to a narrow area of expertise, being both a dependency and vulnerability, manufacturers are now contributing value in a way that will significantly optimize operations.

Part of what takes away some of that tribal knowledge value is an emerging technology like additive. We don't have much tribal knowledge around new technology.

You're right. Companies are creating and evolving new technologies all the time, which creates opportunities for those who like to dive deep into innovative technical areas, working out the real details. I see it as a very positive progression, an opportunity for those who do have those inquiring minds, and that are able to invent the technologies of the future.

What advice about tribal knowledge would you offer a young engineer who's moving into PCB design?

There are two things. First, beware if you focus on a narrow specialty. Be sure to set it into context, to understand what your work is dependent on, and what is depending on your work. This makes your experience, performance, and contribution more flexible and valuable.

Second, based on this strong foundation, get yourself in a position where you can widen your scope, based on your core experience, together with the context of the business, and

the way in which technology is applied and used. It's essential to create additional value from the use of

available tools—to have the how, where, and when to apply them. If you're taking a deep dive into an area that's new, there aren't many experts to help. It is a very interesting and creative environment, just don't think that you will be doing that same thing forever. Be prepared.

How many years have you had in your career?

Too many (laughs).

Michael, thank you so much for this conversation. I appreciate it.

Thank you, Barry. Always wonderful chatting with you. DESIGN007



the way in which technology is applied and used. It's essential to create additional value from the use of available tools—to have the how, where, and when to apply them. If you're taking a deep dive into an area that's new, there aren't many experts to help. It is a very interesting and creative environment, just don't think that you will be doing that same thing forever. Be prepared.

Tribal Knowledge: Not the Villain You Thought

Tim's Takeaways

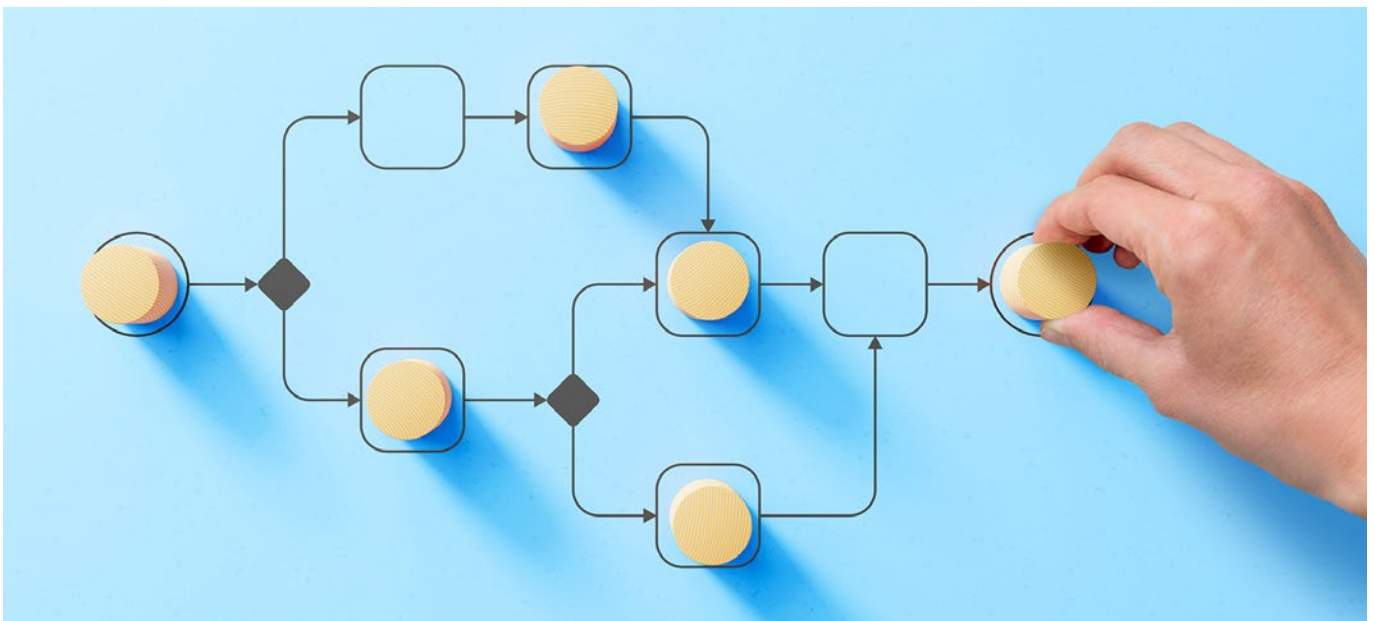
Feature Column by Tim Haag, FIRST PAGE SAGE


There's a lot to be said about the pros and cons of tribal knowledge in the workplace and there really is more than one definition. To put it simply, tribal knowledge is information or skills known by an individual or group that is not known outside that group. One of the trademarks of this definition is that it's commonly used to describe functional—but undocumented—knowledge essential to the operation of an organization. Of course, this is a broad generalization and just one definition of tribal knowledge, but it will do for our purposes.

There are good reasons these undocumented processes are usually looked on as a liability. For example, the smooth operation of a PCB design group requires specific functional skills and knowledge. If this information isn't acces-

sible through clear and concise documentation, it can only be passed from one employee to another through mentoring or by example. Without proper documentation, critical operational knowledge and procedures can be lost. This can be due to employee turnover, or like the children's game Telephone, it can become confused and misapplied. After passing from person to person, the final state of a secret message in the game is often hilariously altered from how it originated. In the workplace, however, wasted time and effort aren't nearly as amusing, especially when it results in lost revenue because those processes and procedures were misunderstood.

It can reasonably be expected that an experienced PCB designer will understand how to lay out different technologies of circuit



A person in a yellow shirt is sitting on a suspension bridge that spans a deep valley. Below the bridge is a calm lake reflecting the surrounding snow-capped mountains. The sky is a mix of blue and orange, suggesting dawn or dusk. The bridge is made of wooden planks and metal cables. The mountains are rugged and covered in patches of snow and ice.

Hmm, what is the recommended **minimum solder mask** width to be able to get a solder mask bridge **between two copper pads**?

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?

Every day we get questions like those. And we love it. We have more than 500 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/pcb-design-mistakes/

Reliable answers. **Reliable** PCBs.



boards: digital, analog, power, RF, etc. However, a designer won't necessarily understand the design flow, critical layout techniques, and manufacturing data requirements of a specific company, especially if they are new to the organization. Even something as simple as, "Our standard trace width in this application is X," can end up as an error if it isn't properly documented. Although it is true that the newer output file formats, such as IPC-2581, can help mitigate complex procedural problems, such as learning file generation processes, there are still many hurdles caused by tribal knowledge that must be overcome. These include understanding specific manufacturing processes, design documentation details, and specific legal or certification requirements, to name a few.

Another problem with tribal knowledge is how those processes can become stale with time due to the inability to review, refresh, and update undocumented procedures. It makes you wonder just how many errors have been introduced into designs because the "tried and true" methods embedded in tribal knowledge are hopelessly antiquated, which allows mistakes to happen. Along those same lines, how much unnecessary effort has been expended and time lost due to outdated processes still in place? For instance, many newer design systems will automatically track and propagate part numbers and descriptions. Yet some people will still manually record this information because "that's the way we've always done it."

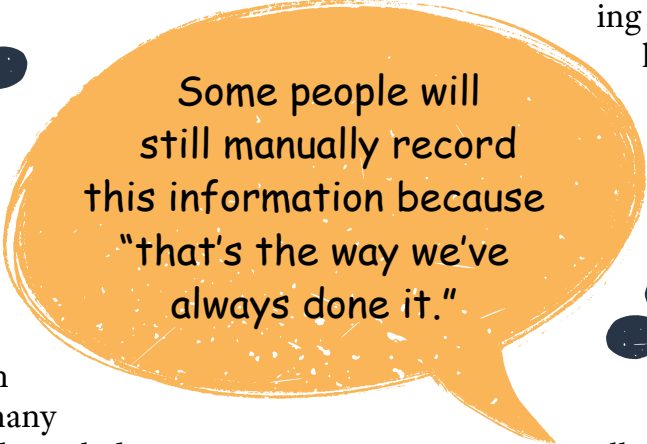
So, tribal knowledge is bad, right? Well, maybe not in all circumstances. I would agree that relying on undocumented processes to convey critical procedural information can, in some circumstances, lead to negative or even catastrophic results. But

there are some positive outcomes when relying on tribal knowledge as well. Consider the following story of how I once benefited from someone's tribal knowledge in a completely unexpected way.

Years ago, we moved across town from one house to another. I hired a moving company to do the heavy lifting for us and went with them to supervise the first delivery. However, I witnessed a terrible sight when we returned for the next load. A landscaper hired by the new owner already had ripped out all the existing plants and bushes, as well as unloaded

all the new shrubbery on the driveway. Since this is Oregon, it was raining cats and dogs, and the dirt from the new shrubs was creating a sea of mud over the entire driveway. I was in total shock and had no idea how to resolve this situation. We still had a large amount of furniture and possessions to move, and my driveway was completely inaccessible.

At this point, the moving crew supervisor noticed my "deer-in-the-headlights" expression and realized that I was completely incapacitated. Springing into action, he said quite firmly, "Tim, you stay here, and I will take care of this problem." Hopping out of the truck, he demanded to know who was in charge of the landscape team and then laid down the law very succinctly to their supervisor: "We are moving the previous homeowners out of this house today and will be using this driveway. If the new homeowners don't want us to drag mud through their brand-new house, you will have this driveway cleaned off in five minutes so that we can proceed." With the landscapers properly chastised, in very short order we had a clean driveway and proceeded with the move.



Some people will
still manually record
this information because
"that's the way we've
always done it."

I hope the new owner loved our former house as much as we did and ended up with a beautiful yard of new plants and bushes. My point in relating this story, however, is in how the moving crew supervisor used his experience to save the day. I was having a crisis moment, and needed someone who knew what to do and wasn't afraid to use their knowledge to get the job done. Yet, if you were to look up the processes and procedures for this moving company, I will bet that there wasn't a single piece of documentation that detailed how to handle a mud-caked driveway filled with shrubbery. Instead, to successfully solve the problem, this supervisor relied on his past experiences in dealing with various scenarios and his gut instincts.

Obviously, documented procedures are essential. But it's important to react to unique situations with only our experiences and knowledge to guide us; this is what some would label as reliance on tribal knowledge. There is a fine line here, and, realistically, the need for both is important. Considering this, how can we manage tribal knowledge within our organizations to ensure that we get the best results no matter which course of action is required? Here are some ideas.

Facilitate Open Communication

Isolation is a negative aspect of tribal knowledge that should be avoided. Without the ability to freely share information between co-workers, critical procedures will inevitably get siloed by groups or individuals, bringing about the bad results we've discussed. Instead, ensure that communication with your co-workers is unencumbered by either technology or company culture. Open communication gives everyone a stake in the game and helps

bring to light important process steps that are all too often known only to a few.

Keep Documentation Up to Date

If you don't want the critical processes in your workflow relegated to tribal knowledge, it's essential that you keep your documentation fresh. This requires regularly scheduled reviews of your procedures and putting a system in place for workers to report outdated or incorrect processes.

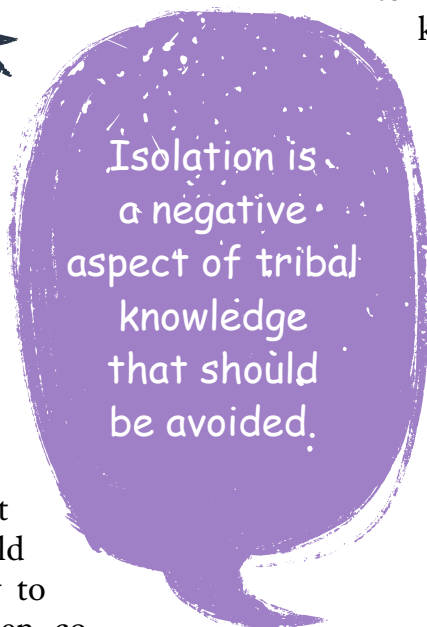
Make Documentation Easily Accessible

Another important but often overlooked element of good documentation is its accessibility. How many times have you needed a procedure, and yet you couldn't quickly find what you were looking for? As corporations grow, so do their file systems and documentation processes. Workers may not know where to look for their procedures or that they even exist in the first place. One of the quickest ways to foster the creation of unwanted tribal knowledge is to bury important process documentation so deep that no one can find it.

Encourage Initiative

Even with all the best documentation in the world, there will be those times when someone must deal with a muddy driveway. To ensure that your corporation doesn't stall out when critical non-standard decisions have to be made, encourage your team to feel they have the initiative and freedom to do so. You may

lose control in some circumstances, and there may even be a few errors, but if you want to be ready to handle the unexpected, it requires some risk. You will develop greater operational flexibility in your company, foster a new sense of ownership, and encourage a desire to excel.



In my company, we have reorganized our department into smaller vertical teams that focus on different areas of the business. Not only has this helped our overall department efficiency, but the smaller team structures have opened communication between co-workers, which fosters the development of new processes and procedures. One example is how the teams have initiated cross-functional training to better support each other in overload situations. Instead of allowing work to pile up when the one person trained to do that job is unavailable, the work can now be spread around; this prevents a bottleneck. By freeing up our teams to take the initiative, they identified areas that have improved the workflow and increased our productivity.

So, is tribal knowledge good or bad? I would say it's a little of both. The best approach is to find the parts of tribal knowledge that are most useful, then harvest and manage them for the benefit of the entire corporation. What do you think?

Until next time, everyone, keep on designing. **DESIGN007**



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, [click here](#).

What Do Electrical Engineers Do?



Electrical engineers work in various industries and the skills required also vary. These skills can range from basic circuit theory to those required to be a project manager. The tools and equipment that an electrical engineer may need are also variable, and can range from a straightforward voltmeter to a top end analyzer to advanced design and manufacturing software.

Electrical engineering specifically deals with electricity, electro-magnetism and electronics. It also covers power, control systems, telecommunications, and signal processing.

An electrical engineer applies the physics and mathematics of electricity, electromagnetism, and

electronics to design and develop new electrical equipment and systems, to solve problems, and to test equipment.

Electrical engineers can work on various projects, from designing household appliances to designing large-scale electrical telecommunication systems, electrical power stations, and satellite communications systems.

More and more, electrical engineers are relying on computer-aided design (CAD) systems for the creation of schematics and to lay out circuits, and they use computers to simulate how electrical devices and systems will function.

Interested in joining the electronics industry? We have over 50 career paths for you to explore! Whether you are interested in design, soldering, robots, chemistry, machinery, or computers, we have a path for you!

Learn more at: [Careers in Electronics](#)



careers in
ELECTRONICS



Hmm, what is recommended
**minimum distance for
copper to board edge?**

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?

Every day we get questions like those. And we love it. We have more than 500 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/pcb-design-mistakes/

Reliable answers. **Reliable** PCBs.



The Power of Forward Thinking

Connect the Dots

by Matt Stevenson, SUNSTONE CIRCUITS

Innovation is everywhere we look. From Apple's air tag luggage tracker to robot vacuums to gadgets that will feed our pets, innovative electronic devices are changing how we live. Though these devices serve a wide range of needs, they all have two things in common: They need innovators to imagine them and printed circuit boards to make them operate.

My experience and expertise keep me focused on the PCB manufacturing component of innovation, but it is important sometimes to step back and look at the bigger picture.

Getting Started

In any industry, the organizations leading the way are the ones that recognize a need that is not being served, determine how a technology can be improved, or discover something revolutionary. Because successful innovation

requires the ability to conceptualize, design, prototype, and build something that will be embraced by the marketplace, innovators need to be skilled in more disciplines than is humanly possible.

While enterprise-level product development teams sometimes possess the resources necessary to take a project from beginning to end, most organizations need to fill gaps in areas such as:

- Product design
- Mechanical and electrical engineering
- PCB design and layout
- PCB assembly
- Troubleshooting
- Marketing and sales

Our industry is a key partner for organizations turning innovative ideas and concepts into technology that [enables green living](#),



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.



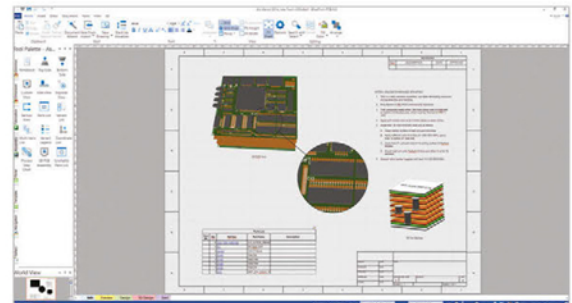
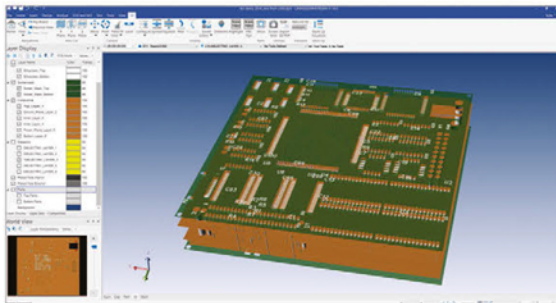
Panelize

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**

improves education for students with special needs, and enhances transportation safety. To support their innovations, we have to innovate as well. New devices often demand more from designers because of complex functionality, unique environmental tolerances, or extreme constraints on board size.

The Road Ahead

We must be forward-thinking about how we can serve technology innovators. If PCB manufacturers don't keep an eye on where today's trends will lead, we may not have the knowledge and tools needed to support an innovator's vision. Was limited board functionality the difference between the iPod and the Zune? Probably not, but the margin between a multi-billion-dollar success and infamy can be razor thin.

Over the years, we've seen products become great successes, unmitigated failures, and everything in between. Projects will struggle to get out of the drawing phase without solid market research, product validation, patent searches, prototyping, product feedback, and iterative improvement. Innovating is hard work and developing a product that will be successful in the market is challenging—but not impossible.

Sometimes the first prototype evolves into a marketable product, and when innovators really endeavor to think outside the rectangle, they create something special. Countless others simply don't have the patience for innovation, and their ideas never make it out of the concept phase.

To increase the chances of an idea becoming a profitable product, you must do the legwork: Figure out and really understand what people want or need, try to ascertain how many of them there are, and determine if there are other products or patents that already satisfy this need.

The tech community is a big, open place. People like to share ideas, so talk to them. Today's stranger might be tomorrow's colleague. Don't

be afraid to pull the plug if the research is not showing that there is a viable market for the product. I have seen many examples of a product going to market and failing because the inventor was convinced that it was a home run and ignored all the signs pointing to a bust.

Taking the Leap

Once you have done your research and planning and have determined that you have a good idea, next comes the fun (and the cash investment): Building the prototype.

Preparation and research are also vital in this phase. Even if you are building a prototype in your shop using pieces from existing products, it is wise to know beforehand if and where you can get what you need for future production runs. Do your homework on materials suppliers, PCB manufacturers, and assembly partners.

Designing a product is one thing, but nothing is more rewarding than seeing it transform into something tangible for people to touch, feel, and interact with. The feedback from those interactions with members of the target market can inform modifications to the product before undertaking larger production runs. The more your potential customers can see, touch, and experience your prototype, the better the feedback you will get, and the better the end product will be. This may take time and several iterations, but it will ultimately be worth the effort.

So, go be an innovator. Make cool things, invent the next biggest gadget, solve a problem, design a PCB—but don't forget to think ahead and do your research to avoid common pitfalls. **DESIGN007**

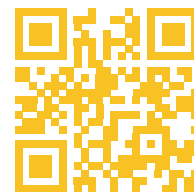


Matt Stevenson is vice president at Sunstone Circuits. To read past columns, [click here](#).

Download *The Printed Circuit Designer's Guide to... Designing for Reality* by Matt Stevenson.



edu.ipc.org/pcb-design



PCB DESIGN COURSES

DRIVE QUALITY AND INNOVATION

Introduction to PCB Design II

Two upcoming classes:

3/14 to 5/4

Design for Radio Frequency

Upcoming classes:

3/13 to 4/19

Advanced Design Concepts

Upcoming classes:

3/13 to 5/3

PCB Troubleshooting & Defect Analysis

Upcoming class:

4/3 to 4/26

Rigid-Flex Boards

Upcoming class:

6/5 to 7/12

Design for Mil-Aero

Upcoming class:

6/5 to 7/12

New PCB Assembly Process & Quality Courses

- BTC Package & Assembly: Materials, Process and Reliability
Jennie Hwang | 4/4 to 4/20
- Lead-free Reliability for Harsh Environment Electronics
Jennie Hwang | 5/16 to 5/18
- Top Lead-free Production Defects and Issues, Causes, Remedies and Prevention
Jennie Hwang | 7/11 to 7/20

Learn more about our [PCB Design Courses](http://edu.ipc.org/pcb-design).

Material Matters and Tribal Knowledge

Feature Article by Alun Morgan

VENTEC INTERNATIONAL GROUP

At a recent IPC Laminate/Prepreg Materials Subcommittee meeting, I made two observations: It was very well-chaired and managed, and despite a full attendance of very knowledgeable materials experts, none could remember the rationale for several of the specification sheet descriptions—slash sheets—that comprise IPC-4101E, “Specification for Base Materials for Rigid and Multilayer Printed Boards.”

The thought naturally followed that if a comprehensive cross-section of materials experts struggled with this, what hope do PCB designers have of making sense of it?

It was a revelation to learn at a previous materials subcommittee that the headings for each specification sheet do not form part of the specification. These headings include reference definitions for the material, including any reinforcements, resin systems, flame retardants, and fillers that may be used, as well as its other known identifications and the glass transition temperature. However, the requirements for the material are

solely defined by the ensuing specific line items within the specification sheets. Bearing in mind that the specification sheets are organized according to the reinforcement type, resin system, and/or construction—although these do not form part of the specification require-

ments—one could be forgiven for wondering what value this classification system has.

Hypothetically, it would be possible for a materials manufacturer supplying a glass-reinforced polyimide laminate material, which would ordinarily be classified as IPC-4101/40, as an IPC-4101/126 material which would ordinarily be classified as an epoxy woven-

glass material. There would be nothing to stop this from happening if the material satisfies the specification sheet requirements. Clearly absurd, but possible under the classification system.

So how do PCB designers specify materials? Perhaps unsurprisingly, many specify whatever material worked last time on a similar



design. Or they ask whoever is next in the supply chain, usually the board shop, what they would recommend. That's where the tribal knowledge comes in.

There is nothing wrong with this approach, thankfully, as in my view this is common practice. However, there can be disadvantages. It can mitigate against the use of new and untried materials, which may offer significant performance gains compared to established products. There is also a real risk of over-specifying materials by erring "on the safe side." Over-specifying could potentially make a project uncompetitive, causing management to pull the plug before it reaches commercialization.

This leads one to wonder if there is a better way that doesn't rely on tribal knowledge or the, "It worked okay last time," approach. The clue is that IPC-4101 acknowledges that the specification requirements are defined not by the description, but by the specification values—in other words, the performance of the material. This is hardly a Eureka moment; what else would a designer care about? The question is why we don't offer the designer a practical guide to materials selection based on their real requirements rather than an arcane system based on old and largely inadequate descriptions of chemistry which don't even form part of the specification. After all, when most of us buy new tech, we don't describe our needs by the underlying chipset (well, most of us anyway), but focus rather on the function and performance based on our needs.

Ventec has for some time been curating products into groupings based on performance requirements in specific application areas. One example is the "autolam" portfolio, a base material set specifically curated for the diverse and often extremely demanding requirements of automotive applications. Imagine being a designer in that space; consider how much more easily you could select the right materials for any given project if you could start with

a subset containing those that are conceived for the application area, and then quickly find a handful of candidate material types matching your specific requirement.

The hierarchy of such a material set may look something like this: automotive/infotainment, automotive/thermal management, and automotive/radar. If interested in thermal management, you may have an interior or exterior lighting application, or a power converter, inverter,

or charger, for which several candidate materials may be positioned. Your final choice may then depend on specific requirements such as power rating, reliability, etc., that may direct you to a particular grade of insulated metal substrate, or various ceramic-filled materials, coming with the necessary automotive qualifications and documentation.

Similarly, Ventec has created the "aerolam" portfolio, which adopts the same approach to make life easier for designers in the aerospace and defense sectors.

It was interesting to observe at IPC APEX EXPO in San Diego that a number of materials suppliers have now followed suit in grouping their product offerings into application-specific areas. The famous quote from Oscar Wilde, "Imitation is the sincerest form of flattery," comes to mind.

The road to converting the old classification system to one based on performance in application-specific areas will likely not be smooth and will inevitably require a concerted effort by all stakeholders. However, the reward will be to put the real power of informed material selection in the hands of the designer—surely a worthy aim and one we should embrace as an industry. **DESIGN007**



Alun Morgan is technology ambassador for Ventec International Group.

Instilling an **Informal** Information Culture

The Pulse

Feature Column by Martyn Gaudion, POLAR INSTRUMENTS

Informal information exists within most companies and distilling this knowledge into tools is not an easy task—maybe even an impossible one. But what's most important in maximizing this informal information is to have a good communication network, those “go to” people for a particular purpose. It is important that the company promotes a culture of openness and sharing, or knowledge has a risk of being ring-fenced and locked away.

More Than Just the Company

Writing this on my return from an informative and inspiring visit to the EIPC Winter Conference—this time in the City of Lyon with a visit to a nearby nuclear power facility—it dawns on me that just as important as having that company knowledge is having the net-

work of knowledge that exists in all the myriad suppliers of base material, chemistry, drill machines, plating equipment, and so on. The extended network provided through organisations like EIPC acts as a knowledge amplifier—and more importantly, those networks allow you to channel customer questions to the appropriate place. This human interaction is difficult (fortunately) to substitute with tools.

Modelling Tools

When modelling tools are used for prediction on ideal materials, very good predictions can be created; however, when working with composite materials—or materials that have to compromise properties for a variety of reasons (cost, regulation, reliability, signal integrity)—it is extremely impractical to cover all those



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

FLEXIBLE CIRCUITS | EMS/ASSEMBLY | CONTRACT MANUFACTURING

Industry leading Flex Design Support

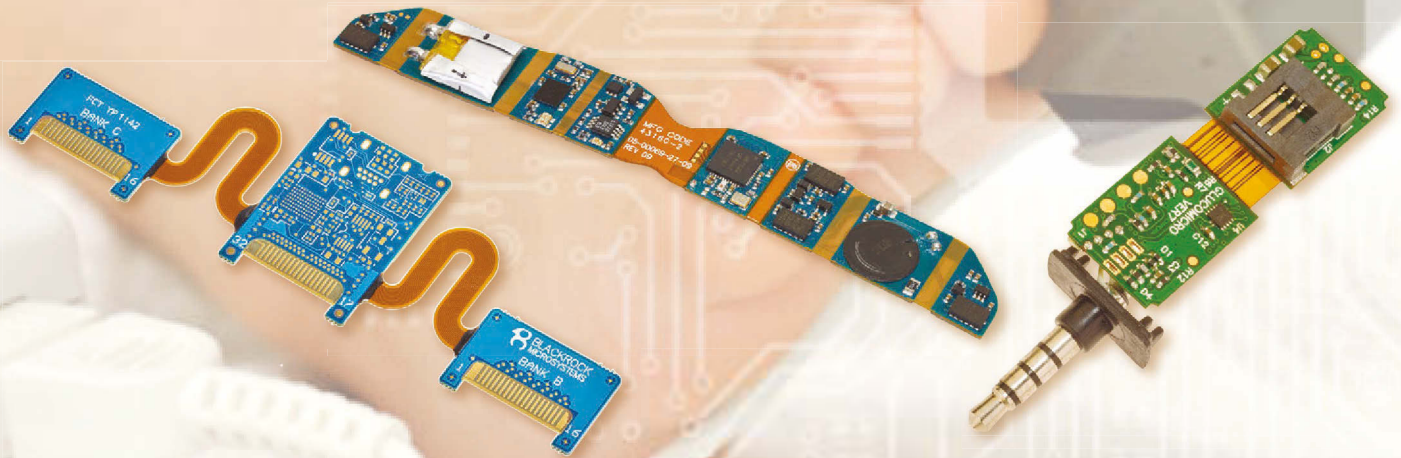
**Flexible Circuits, Rigid Flex, Flexible Heaters
Membrane Switches, Plastic Moldings**

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

bases. One approach from a tool provider is to alert the user that they are headed into uncertain territory and direct them toward the appropriate source of knowledge for their question.

Interpreting the Internet

“Tribal knowledge” is not a term I’m very familiar with, maybe because it’s not used so much in the UK. One internet search yielded a potentially racist and insensitive term.

Other results say it’s fine to use as it does not refer to any tribe in particular; it’s just a concept. More than that, in a company, tribal knowledge is the lifeblood of the organization. That said, it is far from easy to see whether the term has really fallen from favour or the result is simply a very vocal opinion. Much the same applies with PCB fabrication knowledge; each specialist application needs to draw on the skills of a variety of people deeply knowledgeable in their own discipline. This knowledge is most valuable when each party knows where there is a need to draw in the skills of other disciplines so the best engineering judgment, the best price, and yield for the specific application can be met.

Communication

The thread that runs through all this is open communication, both within a company and within a supplier network. Because change can happen fast, tools can’t always instantly respond to every move in the market—not the least because it is hard to predict which innovation will succeed and which will lead into a blind alley—so avoid cramming tools with functionality for innovations that for whatever reason don’t ramp up into the mainstream.

Not on Paper

Some knowledge is not easy to impart on paper, especially while understanding is being built on a new process or technology so that during ramp-up phases of new processes or technology the shared knowledge of the development team and the beta customers may morph and guidance may change as the new technology stabilises. Only once it’s stabilised is it possible to document. Such is the process of development—going through iterations before a final stable process can be documented.

Conclusion

Informal information is like a neural network that holds a company together, the optimum for keeping the company working at its best

is to promote sharing of that information so that if a member moves on to new pastures, the network “heals” and recovers in the fastest possible time. Remember also to extend the network to those companies you work in partnership with in order to deliver value to your closest asset: your customer. **DESIGN007**

The thread that runs through all this is open communication, both within a company and within a supplier network.



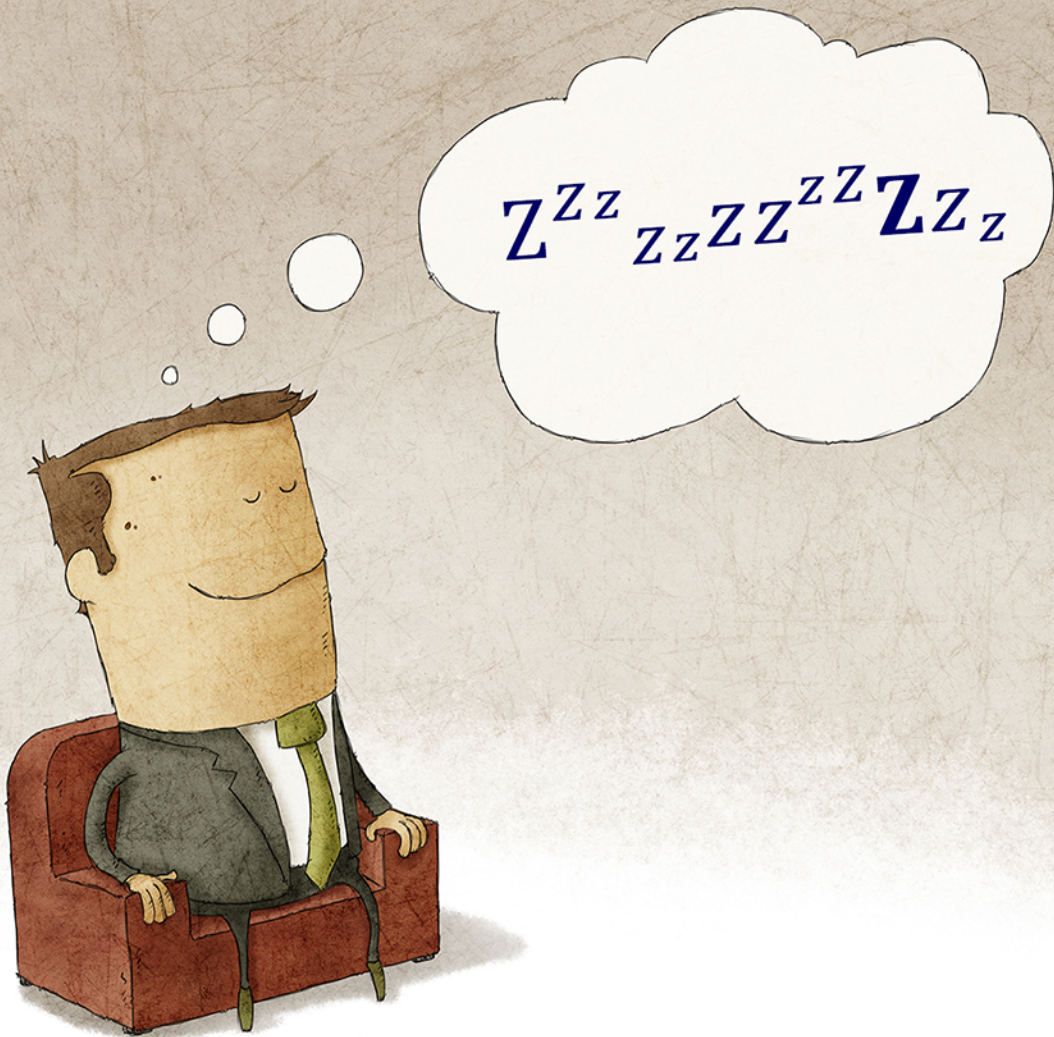
Martyn Gaudion is managing director of Polar Instruments Ltd. To read past columns or contact Gaudion, [click here](#).

Additional content from Polar:

- *The Printed Circuit Designer’s Guide to... Secrets of High-Speed PCBs, Part 1* by Martyn Gaudion
- *The Printed Circuit Designer’s Guide to... Secrets of High-Speed PCBs, Part 2* by Martyn Gaudion

You can also view other titles in our full **I-007eBooks library**.

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.



If Not You, Then Who?

Elementary, Mr. Watson

Feature Column by John Watson, CID, ALTium

I have done a considerable amount of traveling over the years. In my many visits to the Philippines, I noticed a distinct lack of nursing homes, and learned that it is common practice for children to take care of their parents at home. Their designated term for this practice is *ma alaga*, roughly translated as “take care of.” This practice leads to some fantastic outcomes in the culture, resulting in each age group taking care of its own, with a beautiful overlapping of different generations. What a great concept.

If you haven’t noticed recently, the PCB design industry is struggling. It is an understatement to say that we are facing a talent shortage. I regularly get phone calls regarding open positions and the need for more designers. At one time, there were designated PCB designers. But with the first economic

downturn in 2008, talent shortages hit many companies.

Next in line to fill the designer vacancies were electrical engineers (EE). But that came with inherent problems. First, very talented engineers got pulled away from what they do best: engineering. Second, we quickly found out that many EEs never received any formal PCB design training. They spent years in higher learning, but very little time in PCB design.

To make matters worse, the trend seems to be getting worse as Baby Boomers head for retirement. Some surveys estimate that a staggering 63% of PCB designers and design engineers are planning to retire within a year. This explains the massive sucking sound we all hear. It is the departure of talented and experienced



We are dedicated
to excellence through

innovation, technology and most importantly, service.



COMMERCIAL • MILITARY • MEDICAL • BAREBOARD



U.S.CIRCUIT

www.uscircuit.com



PCB designers leaving the industry in droves, taking with them their years of experience and knowledge.

Furthermore, the experience and know-how we are losing is at a completely different level: it is real-world knowledge. This “tribal knowledge” could be defined as “any unwritten knowledge within a company that is not widely known elsewhere.” That real-world practical knowledge is priceless to a company, but it’s walking out the front door, and it’s likely to get worse before it gets better.

Now the situation has escalated to a critical new level. With many companies badly needing new PCB designers, they are simply looking for anyone who has a commitment to learning. These companies are willing to invest time, energy, and money into those individuals and train them for the position. I teach PCB design at Palomar College and just recently a company reached out and asked to partner with us to secure a supply chain of new PCB designers to fill their positions. They were willing to continue their mentoring and education.

It is said that the definition of insanity is doing the same thing over and over again and expecting different results. If we are to break into a new paradigm in our industry, we need to change what we’re doing. More companies are realizing we need to find a new way to find and retain PCB design talent.

Time for Mentoring?

To accomplish this, we need to prioritize mentoring for the next generation of designers. We are quickly losing the needed talent to support an ever-growing electronics industry with new technologies and innovations. Maybe it’s time for a change; let’s conduct business to have our *ma alaga* moment as an industry. We

must care for our own and prioritize nurturing the next generation of younger designers.

The main goal of mentoring is to help mentees tap into the knowledge of those with more experience than themselves and learn faster than they would on their own. Also, it’s an opportunity to grow their network and connect with leaders rather than only their peers.

Create a Plan of Attack

To succeed in any mentoring program, there must be a plan of attack. In the words of Zig Ziglar, “If you aim for nothing, you will hit it every time.”

With no clear objectives or goals, you will never reach them. It’s only a matter of time before your shiny, new mentoring program, which everyone put their hope into for your company’s future, becomes an utter failure only

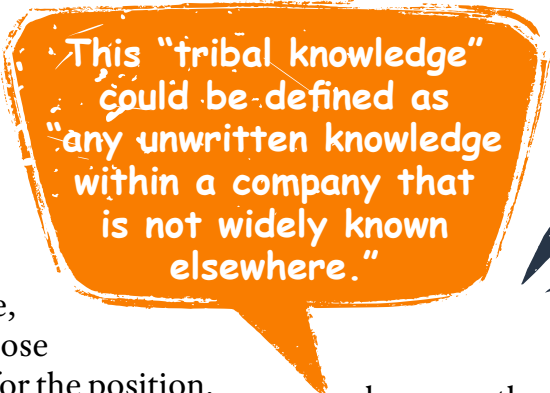
because there were no defined, detailed objectives. Connecting knowledgeable team members with those who want to learn is the first step; I would even say the easy step.

Have a Lesson Plan

One of the great joys in my life is the opportunity to serve as a college professor. I know each week and in each class session what I will teach. It would be irresponsible not to prepare the material and be fully prepared to present it to the best of my ability. But many mentoring programs I have seen fail simply because there was no plan with a detailed structure.

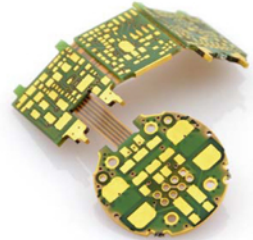
Create Metrics

Along that line, have measurable and specific metrics to determine when the goal is reached. Evaluate how the information is received. By not having these primary objectives and measurable metrics, it’s the engineering equivalent of sailing a boat with no rudder.



This “tribal knowledge” could be defined as “any unwritten knowledge within a company that is not widely known elsewhere.”

Focused on Flex

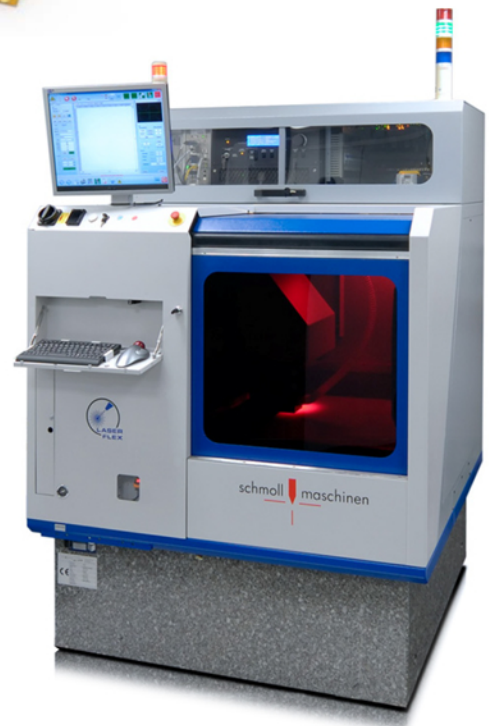


Laser μ Flex series

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
- Microvia and through-hole drilling



BURKLE

NORTH AMERICA, INC.

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

burkleamerica.com

Practice, Practice, Practice

I recently spoke to an engineering professor in the Midwest, and he mentioned that there are plenty of training resources, particularly with video. Still, at some point, you need to turn off the video and practice. Practical learning is a must with PCB design. You must understand the steps from concept to finished design. It's something you only understand in the abstract until you do it.

Make It Relatable

Whatever mentoring program you develop, make it practical and relatable to the industry. This is something we practice in the PCB class at Palomar College, just as the students are finishing their designs, we throw the proverbial monkey wrench into the mix. It might look like this: The sales department issues an ECO notice that we must cut \$100 out of the final cost of our design. What do we do?

That is not just book knowledge, but critical thinking and problem-solving. As we know,

that is precisely how it happens in a company environment.

We can sum this entire discussion up with a single word: obligation. No matter what level you're at, you have something to give back to the industry. Some feel it is someone else's responsibility, but I would argue that it falls on all PCB designers to step up and fill the gap; give back to the industry that has given so much to us. It's time for our *ma alaga* moment, and we must care for our own. If not you, then who? **DESIGN007**



John Watson, CID, is a customer success manager at Altium.

Download [The Printed Circuit Designer's Guide to... Design for Manufacturing](#) by David Marrakchi. You can also view other titles in [our full I-007eBooks library](#).

A Week-long Industry Extravaganza

by Happy Holden

We were back in San Diego, maybe for a "long vacation," as IPC APEX EXPO is in Anaheim next year. Before we get to Anaheim, though, let's talk about the show: It was a great week.

I spent Sunday and Monday visiting some excellent Professional Development courses. As you know, this is a fast-changing industry, and we have to keep learning to stay ahead. On Monday, I attended the Microvia Weak Interface/Reliability subcommittee, and on Tuesday, the Ultra-HDI Technology Committee. These turned out to be popular, as over 90 people showed up to the Microvia Reliability meeting and around 85 at the UHDI meeting. Those committees may have many new members soon.

On Wednesday, I attended the "DoD Executive Agent for PCB and Interconnect Technology" (PrCB

EA) Programs Update. This was very informative, with speakers updating everyone on the various activities of PrCB EA, the Defense Production Act Title III, the Defense Microelectronics Cross-Functional Team, and Marc Carter's EMTE education outreach. IPC's Chris Mitchell and Chris Peters of USPAE provided updates on how both organizations are facilitating government/industry cooperation and projects.

The IPC standard A-610 is just the start, and EVs have different specs and components than cars with internal combustion engines. The technical challenges for EVs are temperature variation, heavy copper PCBs, density requirements, thermal dissipation, electromagnetics, and vibration/shaking requirements.

This article originally appeared in [Real Time with... IPC APEX EXPO 2023 Show & Tell Magazine](#).



IPC APEX EXPO: A Focus on PCB Design

by Andy Shaughnessy



IPC APEX EXPO 2023 featured more PCB design content than I can recall in prior years, and I've been going to this show for a long time. PCB design and design-related issues were the focus of a variety of committee meetings and conference classes—not just design classes, but sessions that cover manufacturing challenges and process optimization.

I didn't have to walk far to find a designer or design engineer. Even the classes that focused on process optimization covered some aspect of PCB design.

I spoke with process engineers who now spend much of their day working with their design teams to head off potential DFM and DFA problems early in the design cycle before they become budget-busting missteps. Likewise, designers are learning how to best optimize their processes for DFM and DFA. Hearing designers and manufacturers talking to each other was refreshing; is this the start of a new trend?

One of the most exciting events was the second annual IPC Design Competition, with five finalists from around the globe battling it out for four hours. Three contestants worked remotely and two competed live at the San Diego show. Out of the original 49 contestants who began the preliminary heat last fall, these five finalists truly represented the cream of the crop.

This year's design was a tough one; no one was able to finish the design completely, so judges graded them on what they did complete, and what kind of decisions they made during the competition.

In the end, Sathishkumar Vijayakumar took home first-place bragging rights. Vijayakumar, a senior design engineer with Tessolve India, narrowly edged out Adam Thorvaldson, CID, lead PCB designer with Innovex Design Service.

Let's all congratulate Sathishkumar Vijayakumar and the rest of the design competitors for sticking it out over this months-long competition.

In this special section, you'll read interviews from the top two winners, as well as an interview with Kris Moyer, who designed the competition. We also talked to a Professional Development course instructor and one of his students.



Sathishkumar Vijayakumar:
IPC Design Competition
Winner



IPC Design Competition Wrap-up

Interview by the I-Connect007 Editorial Team

The IPC Design Competition took place during IPC APEX EXPO 2023, with five finalists competing—a far cry from the original 49 entries last fall. After the competition was over, we spoke with Kris Moyer, the IPC instructor who created the design used in the competition and served as a design advisor and judge for the event.

We asked Kris to give us a quick wrap-up of the competition, as well as his thoughts on this year's design, which none of the finalists completed.

Andy Shaughnessy: *Kris, would you give us a brief rundown on the design competition?*

To provide a better challenge for our competitors, we added some additional levels of complexity over last year's design. We had five finalists, three here at IPC APEX EXPO, plus the two finalists from our India branch. We ran them through the design yesterday, and we saw some very interesting designs. We came down to one clear winner, although I will say it was very close between the first and second place.

As far as complexity, this year we ended up with a multilayer rigid-flex board incorporating several more advanced complexities: sequential lamination, HDI, advanced packaging, controlled impedance, high-speed digital design, and power distribution design. Our competitors didn't finish, but they showed a lot of excellent technique and capability.

Shaughnessy: *Some said that they never worked with rigid-flex before.*



Kris Moyer

That's right. Unlike last year, all the competitors this year were given the opportunity for a one- to two-hour individual session with me on Monday evening before the competition. I also provided a two-hour cram session where we reviewed not only the tool, but all the different features and technologies that we have to use. So, although they may never have routed rigid-flex before, I introduced them to it and some of the issues they would be involved with.

Shaughnessy: *Who was the winner?*

The winner was Sathishkumar Vijayakumar of Tessolve, one of our finalists from India. He did quite well in the work that he was able to complete. He showed good technique, com-

plexity, and the broadest range of features and methodologies. Adam Thorvaldson of Innovex came in second. Adam also had a good amount of design complexity and features, so it was very close.

Kelly Dack: *Kris, were these designs graded on manufacturability?*

Yes, that was one of the criteria, and we gave them some rules up front. The bulk of the design rules were provided to them already in the design because of the time limitations, but we said, “Here’s the range of via sizes you can make. You can make microvias for the BGA escape routing all the way up to larger, higher-current vias for the power distribution, etc.”

We said that this had to meet IPC Class 3. The finalists were provided access to the appropriate IPC standards through Heat 1 and Heat 2. It was up to them to know how to use the standards to ensure that the features they created met those Class 3 performance, classification, and manufacturability concerns.

Dack: *Were there any notes or documentation criteria that they needed to fill in?*

Not for this. This one was straight routing design, again because of the time limitations. But in Heat 1 last fall, they had a whole month, and we gave them an entire schematic. But Heat 2 was purely board routing. There wasn’t even silkscreen clean-up. It wasn’t any of that for the finalists; it was purely routing, the key aspect of board design. You can make silkscreen as pretty as you want, but if the copper’s not done right, it doesn’t work.

Dack: *Last year’s design was very simple, and this year’s design was more complex. For next year, will we have the*

Goldilocks syndrome, where the design is just right?

This year, I don’t think it was so much the complexity of the design, but the allotment of time for completion. What we’re really looking at is the time allowance.

Dack: *What advice would you give future contestants to better prepare for this design competition?*

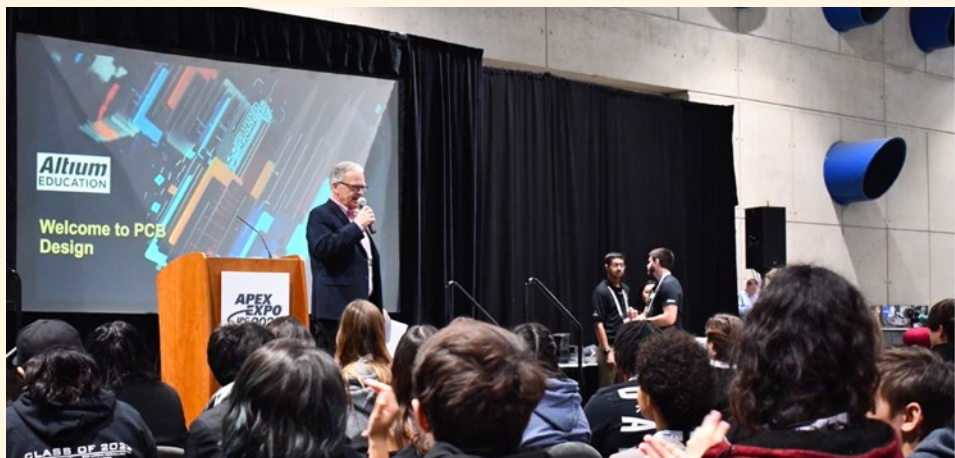
Be familiar with the IPC design standards, especially the IPC-2220 series. Know what we mean when we specify Class 3, Level 8 producibility. Become proficient with the various tools. Part of that’s on us to make sure that we have a standardized platform.

Shaughnessy: *That’s good. Would you do anything differently for next year?*

It’s still early, but maybe. For next year, I will be talking to Patrick Crawford (IPC member of design programs) about several different concepts we might do for next year. We’ll have more understanding after Patrick and I flesh it out, but we’re looking at some ideas for next year.

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

No problem. Thank you. DESIGN007



Q&A With the IPC Design Competition Winner

Interview by Andy Shaughnessy

I-CONNECT007

Design engineer Sathishkumar Vijayakumar with Tessolve Semiconductor, India, took home top honors in this year's IPC Design Competition, besting the other four finalists in a rigid-flex design showdown during IPC APEX EXPO.

Unlike last year, no one finished the design completely, so judges graded competitors on what they did finish, as well as criteria such as design decisions they made, and whether they followed electrical and DFM rules. The first heat of the contest began last fall with an original field of 49 competitors, so it's safe to say that these five finalists represent the cream of the crop.

I asked Sathish to tell us a little about himself and the competition, and what winning this contest means to him as a PCB designer.

Sathish, congratulations on winning the IPC Design Competition for 2023! Why don't you give us some background about yourself, and how you got into PCB design?

Thanks for your wishes, Andy. I am working as a senior PCB design engineer at Tessolve Semiconductor, India. I completed my bachelor's degree in Electronics and Communication Engineering at Anna University, Chennai. I was happy to start a career in 2014 as a PCB design engineer as my background was in electronics and more specifically, I consider PCB design as an art where I can show my unique skills. At Tessolve, we cater to some of the top semiconductor companies in the world, and I get to work on cutting-edge technologies up front, which makes me feel proud as a PCB designer.



Sathish Vijayakumar

What did you think of the design used in the competition? None of the competitors finished it, whereas last year's design was considered too easy by a few of those contestants.

Yes, the design was quite complicated compared to last year's design. The design was a rigid-flex PCB with a shape that would fit inside a robotic arm. You can imagine the traces need to be routed with odd angles for such board shapes, which generally takes more time compared to 45-degree angle traces. It's a real challenge to complete the routing within the allocated time. I feel it was one of the reasons why none of us were able to complete the design. I started working on items by priority and completed whatever I could following the guidelines. I believe this process helped me.

Remember: “The process is more important than the results. And if you take care of the process, you will get the results.”

What does winning this competition mean to you as a designer?

This is one of the greatest milestones in my career and I am very proud for participating in this world competition. Winning this competition gives me more confidence in designing complex PCBs.

I believe that I received the best training and exposure to best practices from my organization while working on different types of PCBs over the years. It certainly helped me a lot with winning this competition. I would like to thank Tessolve, IPC India, and IPC International for helping me participate in the com-

petition. I also thank my family members for being a part of my success.

What advice would you give to other designers considering entering a design contest like this?

Nothing is more expensive than losing an opportunity. I would tell all my fellow designers to volunteer themselves and participate in design contests like this. Preparing for such contests itself will make designers learn more about PCB design and it's certainly going to help their careers. I also think IPC contests create a positive vibe among PCB designers to come forward and participate in upcoming events. I believe that participating in the contests is more important than winning or losing. **DESIGN007**

IPC Expands Instructor-led Courses on Technical Knowledge and Skills

by Kelly Allen, IPC TRAINING MANAGER

If you're looking to enhance your learning in electronics manufacturing, IPC can provide a personalized approach where you feel engaged and ready to strengthen your on-the-job performance.

2022 was a great year for our instructor-led courses. We have reached a point where our industry has embraced the flexibility of online learning. Online lectures are a great option.

Why do they work so well? Because students in these classes most often have full-time jobs or other commitments that prevent them from attending in-person instruction.

Interestingly, students can still build important and unique professional relationships among their instructors and fellow students because they have opportunities to schedule virtual meetings outside of class time. IPC encourages

students to reach out and discuss assignments, and IPC course instructors are available via email or Zoom to review material for better clarity.

Industry experts lead these online courses, which are always recorded so students can revisit and review the lectures on their own time.

Here are two courses you can see this year:

PCB Design I and II: These course offerings are driven by the high demand, where courses have been near capacity all year.

PCB Design for Military and Aerospace Applications: PCB design students have demonstrated great enthusiasm for the coursework and new knowledge of this course.

This article originally appeared in the Winter 2023 issue of IPC Community.



Runner-up Discusses IPC Design Competition

Interview by the I-Connect007 Editorial Team

PCB designer Adam Thorvaldson of Innovex was a finalist in this year's IPC Design Competition at IPC APEX EXPO. He came in second place in this final heat, which is quite a feat, considering that the contest started last fall with 49 contestants from around the globe. We asked Adam to share his thoughts on the competition, what it means to be one of the winners, and any ideas about improving the contest for 2024 in Anaheim.

Kelly Dack: *Adam, you've finished up the design competition here at IPC APEX EXPO. What did you think of the experience?*

This experience has been very eye-opening. The event is great; the competition has been very interesting, and the town is beautiful.

Dack: *How did you hear about the competition? What were the previous heats like?*

I learned about the competition on LinkedIn. I like finding ways to better myself and develop new foundations for future growth. I applied, signed up, got the project files, and competed in the first round. My first round took about 49 hours of work, and later I found out I was a finalist.

Andy Shaughnessy: *That was when we had 49 finalists?*

Yes, and that was all wrapped up in November.

Dack: *What about the EDA tools? For the final heat, everyone used Altium Designer.*

But in the first round we were allowed to use any software package. I was able to use the tool



Adam Thorvaldson

that I was most familiar with, that I've been on for 25+ years, so it made it really easy for me. I'm not sure what everybody else was using in the earlier heats.

Dack: *Designers often have to switch software in their careers. To participate in a competition like this, some of you had to learn the ropes of a new program fairly quickly.*

Sure. For the competition, the finalists received a 30-day trial license to learn Altium beforehand. Unfortunately, I got my license only 10 days before the show, so it's been a crash course. The tool I'm most familiar with is Allegro (and OrCAD) PCB Designer from Cadence. They're a little different. I learned Eagle years ago. I've known about Altium for a long time, but I finally got my hands on it about 10 days ago and it's very bewildering. There

are some features that I like but I haven't had enough time on the tool to become familiar with its behavior. For the competition today, I didn't feel comfortable, but I didn't give up.

Dack: *You're a true designer. First, because you didn't give up, and you saw the importance of the task at hand. Given the tools that you had, you made the best of it. Congratulations for that.*

Thank you.

Shaughnessy: *Let's look at this design. It was a little tougher than last year's, which was a simple conference name badge with an LED light. This year's design was a rigid-flex. Have you done much rigid-flex before?*

It's been about 10 years since I touched rigid-flex, and I had a lot of help back then. These past few months, when it's been a little bit

slow, I've been digging into various materials for PCB design and one of them was a rigid-flex. I wanted to know more about it. I studied the stackups, all the different gaps, transition zones, and everything that goes along with it, like cross-hatching and when you apply it. Is it dynamic flex or not? I did everything I could to prepare for whatever I was about to walk into.

Dack: *Designers often call design tools "a bag of hammers." They all hit the nail on the head, and the nail goes in, no matter what tool you use. But designers who are familiar with Altium may have had an edge here. Do you agree?*

That's certainly an assumption I had. After spending time on any tool, you know where the buttons are, how it behaves, how to anticipate things. I was told that Altium is a memory hog, so when I put a large plane in early on, that slowed me down. I didn't realize that



beforehand. Once I removed that plane, things sped up.

Just having foreknowledge of how the tool works in that regard would be very beneficial to knowing what steps should be done first—for efficiency, to get to the end quickly without having to redo stuff. I had trouble with adjusting the vias (stacked microvias) while moving a circuit; maybe I wasn't grabbing the vias right, but the behavior was not what I expected so I had to improvise and accomplish what I could.

Dack: *What would you do differently or like to see?*

Shaughnessy: *Could it have been different if you were on your familiar tool?*

I have just hundreds of scripts, hotkeys, and aliases that shortcut all sorts of commands; and I've got a library of elements and 26 years of developing my own environment. If I could have used a Cadence tool, I could have gotten at least twice as far. Again, I've only been on Altium for 10 days, maybe eight hours total. But I was really impressed with the install of Altium. That was just amazing.

I don't really know what could be better with Altium, because there are so many ways to do the same thing that I don't yet know what's efficient and what's not. I like some of the menus and how they correlate. I haven't gotten into the query aspects, which is something I would really like to manipulate. You know, they have all these predetermined filters, sets, and queries so you don't have to hit all the buttons.

Shaughnessy: *Tell us about your job at Innovex.*

I own a service bureau and I've been doing that for six years.

Shaughnessy: *That might be why you're used to picking up new tools—it's not your first time*

learning on the fly. What design tools have you used in your career?

It's been Allegro for my whole career, and about 10 projects in Eagle in 2016. I touched PADS briefly years ago, when I was working with another service bureau, but I never finished any projects on PADS. Now I'm learning Altium.

Dack: *Given the experience in this competition, I say, "Everybody wins." You get exposure, networking, learning a new tool, and laying foundations for future growth. Would you recommend it for any new designers?*

Yes, the score sheet that I got from the first round was very informative. It highlighted the areas where I still need to grow.

Dack: *Did they do a design for manufacturability review or a design performance type of review?*

Yes, they checked it all because certainly that was in the criteria of project. We needed to be aware of the IPC reference material and all the classes, levels, and types.

Dack: *What's your message to the design and engineering community? What do you recommend to emerging designers who will be picking up new tools?*

Never stop learning or challenging yourself. In every failure, there's a lesson, if not multiple lessons, so don't be scared of new tools even if you don't like the idea. Just learn as much as you can, and find a mentor that will open doors for you and teach you things.

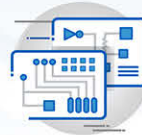
Dack: *Thank you very much. It's been great.*

Thank you both. I enjoyed it. DESIGN007

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

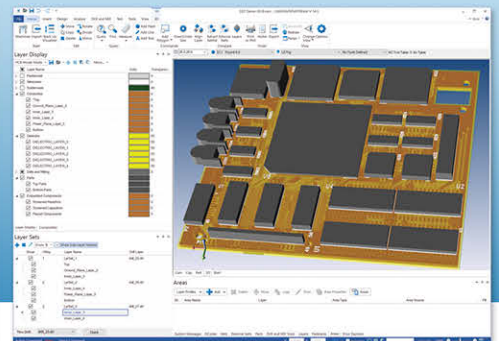
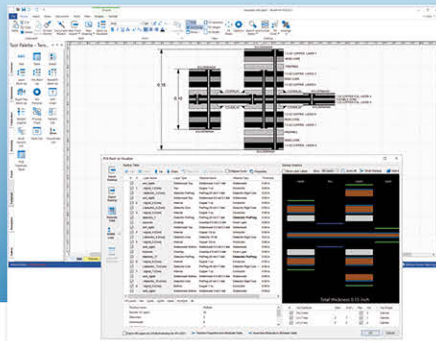
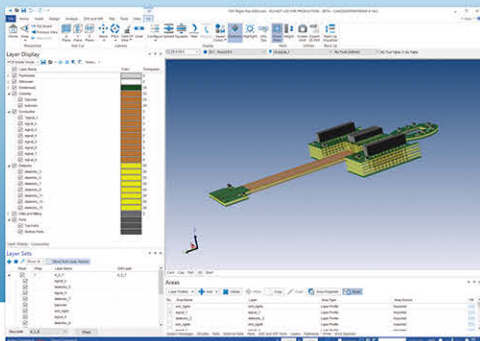


BluePrint-PCB®



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

Forcing Designers Out of Their Comfort Zone

Interview by Andy Shaughnessy

I-CONNECT007

After the IPC Design Competition at IPC APEX EXPO, I spoke with contestant Paul Brionez. He's a senior PCB layout and design engineer with Wisk Aerospace in Mountain View, California. I asked Paul to share his thoughts on this contest, and the rigid-flex technology that was the focus of this year's event. As Paul explains, he was happy to be pushed out of his "comfort zone" in this design competition.



Paul Brionez

Paul, tell me a little about the competition. It looked very interesting, and potentially problematic—a hand-shaped virtual reality glove made of rigid-flex. Last year's design was just a name badge with a light.

It's quite a unique design. I don't really know what the end-product was. It looked like a wearable, but we don't really know. It was just an oddly shaped board. Much of what I've done over the years has been more rigid, so it's a unique opportunity to work on something like that.

You were on your own.

I've seen some rigid-flex, but nothing quite as complex as that.

It forced you out of your comfort zone, but that's timely, because many of our readers are being forced into rigid-flex now.

Sure, and sometimes you learn by being out of your comfort zone. It has a lot to do with pushing you to your limits in that type of scenario.

We were watching your screens live during the contest, and we could see where you'd realize, "Oh, this is RF right here."

Definitely. I saw a lot of things that I haven't seen much of before. I've spent much of my time in aerospace, and we really never venture into something like this because everything we look at has to be more rigid and stable. So, that's where I came from.

Are you familiar with Altium?

I've been using it since August when I moved to Wisk, so my only real "adventure" in Altium was doing this competition, although I've attended a couple of classes at Christmas. Yes, it's a new tool for me, but it comes from a tool that I used many years ago, Protel and P-CAD. I started my career in layout 30+ years ago with

those tools. First it was Protel DOS, and then Protel 99SE.

That was a little while ago.

I have a funny story from those days. At the beginning of my career, in the early 1990s, we were using DOS-based Protel and OrCAD. One of the designs we were working was a strobe light for a forklift, and the challenge was to draw round circuit boards. Protel DOS had two options, but when we were doing the designs in OrCAD, the only option we had was to do the board layout, then take a coffee cup, and draw a circle around the artwork. After that, we would send it to the board house. But if we grabbed the wrong coffee cup that day and sent that design to the board house, we would get the wrong

board size. So, we had to make sure we never lost that coffee cup.

Don't lose the known good cup.

That was a challenge.

What would you say to PCB designers who are considering entering this contest next year?

The design competition adds a variety of challenges for everyone, designers, and the like, and it gives you a chance to challenge yourself. Technology is always advancing, so if you don't challenge yourself, you don't learn.

That's great. Thanks, Paul.

Thank you, Andy. DESIGN007



Tips and Tricks in Today's Designs

Interview by Kelly Dack

Filbert Arzola of Raytheon Intelligence & Space taught a Professional Development course at IPC APEX EXPO on general design practices. During a break, Filbert spoke with Kelly Dack about some of today's design strategies and how designers are reacting to them. What matters most?

Filbert, your class is more about tips and tricks. I've listened to a couple of topics on ground bonding and strategies, and length matching and strategies. What can you tell me about it?

It's a two-part class and it's going very well. We've had a lot of questions, as well as positive comments on the class topics. We're trying to cover everything as basic as possible, but also get into some of the more intense parts of what we want to accomplish as board designers. The second part of the class will include RF design and some other topics, so I'm looking forward to what everyone thinks. The point of the class is to help folks outside printed wiring boards understand why our job isn't as easy as they think. Depending on how you look at it, they may also see it's not as hard as they thought it was. I'm very happy with how it's going.

How about any material or stackup strategies? Do you get in-depth with any of them?

I get into the stackup strategy in that it is the spine of the design producibility, and that's the only way that you can start a design. You must have a stackup so you can plan. I talk about the design being producible and stable.



Filbert Arzola

I don't really get into the materials because I could talk for hours about that and we don't have the time.

How about cost strategies? You're in the aerospace industry and there are a lot of industries represented here. Are you concerned about cost for the aerospace industry?

I'm very concerned about cost because we make things airborne. We do things in space, so cost always depends on where you put things. Cost can increase exponentially at times. We have a concept called LC swap, and LC stands for "low cost." It also includes sizing, power, weight, and we talk about how we combine all that to drive it together to meet your requirement. So, if low cost is really your high point,

all the others have to make it low cost and they all work together in that same way.

It's a work together thing and there's the trade-off of costs; hand in hand with that is volume aerospace to products, and that can be high-, mid-, or low-volume. How does that play into design?

If I make something that weighs less or uses less power and I'm making a lot of them, it will make everything better. If something weighed

five pounds and I make 10,000 of them, that's a lot of weight. But if I get it down to one pound and I make the same quantity, your volume now changes, and you're at less cost. It all works together.

Sounds great, and you have a well-attended class with a room full of attendees. Thank you for presenting.

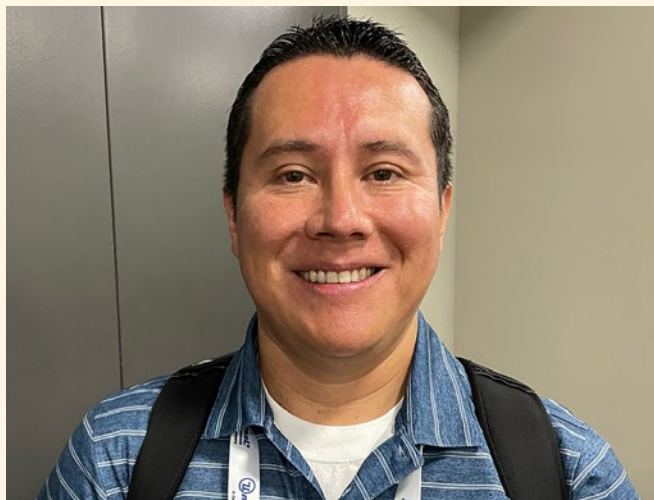
I love the critique, good and bad. That's what drives us. **DESIGN007**



Solid Design Methodologies for Avoiding Obstacles

Interview by the I-Connect007 Editorial Team

Christian Dominguez, an engineering manager with an aerospace company, attended Filbert Arzola's Professional Development class at IPC APEX EXPO. We asked Christian to share his thoughts on the class, his job, and hiring and retaining talented employees in this tight job market.



Christian Dominguez

Kelly Dack: *I've heard Filbert Arzola talk about some strategies on ground bonding and length-matching strategies, and it's often in an aerospace context. What was your takeaway from the class on PCB design strategies and methodologies? What have you learned?*

The course material is very insightful. The instructor is giving us a pretty good overview, and a lot of tips on designing. He's also giving us the tools to look ahead and prepare for any obstacles in terms of design, so that's been very important and a big takeaway.

Dack: *In your job, do you use a lot of exotic materials which require exotic design strategies?*

Yes, we use very innovative strategies for design that match the complexity of the tasks.

Andy Shaughnessy: *Tell us a little about your job.*

I manage a team of layout engineers, and they work on everything from power supplies to RF; it's all over the spectrum.

Dack: *Since you're a design engineering manager, what is the workforce experience like today? Are you able to find and retain*

new design talent? Do you see that as waxing or waning in the industry?

Yes, and yes. We can find them, and we can retain them.

Shaughnessy: *A lot of our friends in the industry have had multiple positions open for six or nine months now. What's your secret? Do you have an outreach team? Do you work with colleges?*

Yes, we work with colleges, and we have an outreach team.

Dack: *Are you hiring fresh designers out of school who want to get into PCB design or are they typically engineering students who can be on an electronic engineer track, pick up the design, and begin doing layout?*

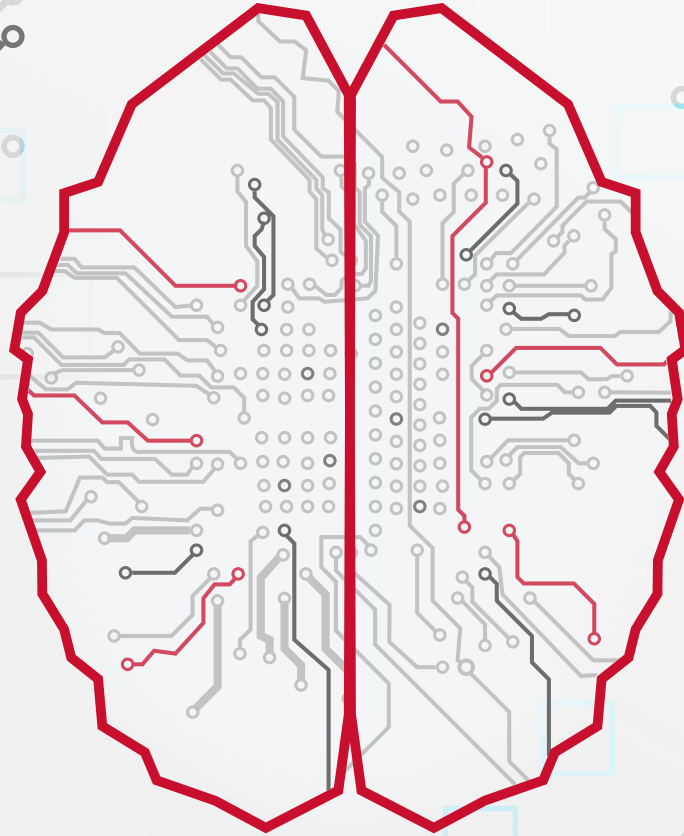
We do hire younger engineers. It's important to train the future workforce.

Shaughnessy: *All right. Very good. Thank you.*

Thanks. DESIGN007

EMA|Academy

Our most comprehensive learning tool, built just for you.



Take Your PCB Design Skills to the Next Level

Learn at your own pace with our self-guided, online training classes complete with videos, labs, and comprehensive training materials.

Start Learning >>



The Tale of Five CAD Monkeys

Target Condition

Feature Column by Kelly Dack, CIT, CID+

“But that’s the way we’ve always done it.”

Many electronics engineering groups have a hierarchical structure, with senior-, mid-, and junior-level engineering personnel possessing various depths of subject matter expertise. The head of an engineering group is usually one who has risen through the ranks of the organization and has fought and won many project battles; they’ve survived and can pass on knowledge, wisdom, and guidance.

Every so often, though, an engineering group leader must call together their team, and tell an ancient story passed down from the “elders” to steer the group in the right direction.

I recall a gathering like this: Several PCB project stakeholders could not agree upon a way to move forward with a challenging PCB

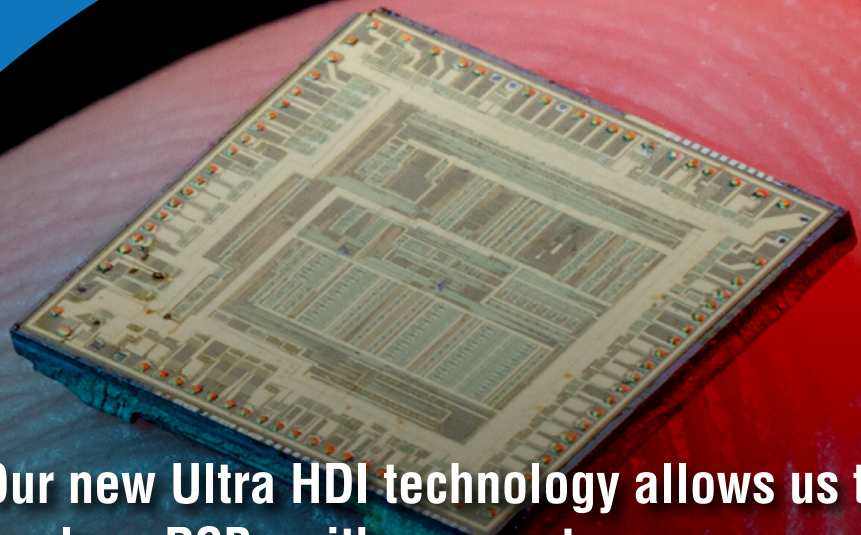
assembly which had suffered failures on the SMT line. Small, newly-specified chip capacitors were “tombstoning” and “skewing” on the PCB, and the root cause could have been attributed to many different variables.

The conference meeting room was packed, and the VP of engineering wisely started the meeting by giving each one an opportunity to share their experience regarding the assembly defects.

One by one, they methodically went around the room. The first one spoke up, “We’re using all our standard equipment, materials, and processes the same way our people always have; our manufacturing constraints have not changed for many moons.” Then he looked to the customer’s PCB design stakeholder, who



WE ARE READY TO BRING YOU THE FUTURE



Our new Ultra HDI technology allows us to produce PCBs with parameters never seen before in our industry. With lines down to 1 mil with a line aspect ratio of 1:1 at production volumes, the future is here!

Check out our UHDI capabilities



American Standard Circuits

Creative Innovations In Flex, Digital & Microwave Circuits

pronounced: “Our DRC rules have remained set to the defaults of our people since the beginning of time.” This continued around the table as they pointed to charts and data to show how their workflows were never altered in ways that would cause component defects such as these.

After the stakeholder feedback was given—and just before they had a chance to begin pointing fingers at each other—the VP of engineering began to talk about a profound parable.

“Sit down and I’ll tell you about the ‘Five Monkeys Experiment,’” he said, and began weaving a version of the story specially adapted for his group. It went something like this:

Long, long ago (in the ’90s), five “CAD monkeys” were placed in a cage as part of an experiment. In the cage were a bunch of bananas hanging by a string from the ceiling, as well as a spring-loaded test bed and a ladder, which needed to be precisely repositioned under the bananas to reach them.

The CAD monkeys had no use for the test bed at first. They knew intuitively that the ladder was the key to reaching the bananas. But each time a CAD monkey reset the ladder to climb toward the bananas, a crazy PCB cleaning process engineer sprayed all the CAD monkeys with icy de-ionized water. The CAD monkeys quickly formed some tribal knowledge about being sprayed down by the icy de-ionized water whenever attempting to reset the ladder. From then on, they agreed to inflict severe beatings on any who would dare go near the ladder.

Following their new tribal agreement, not a single CAD monkey went near the ladder. Instead, they spent all their time trying to figure out other creative ways to reach the bananas. Spontaneously, all five CAD monkeys

tried reaching the bananas by bouncing higher and higher on the test bed. But you know what happens when five CAD monkeys jumped on a bed? Yep, one fell off and broke his head—a program manager then came along and substituted the injured CAD monkey in the cage with a new CAD monkey.

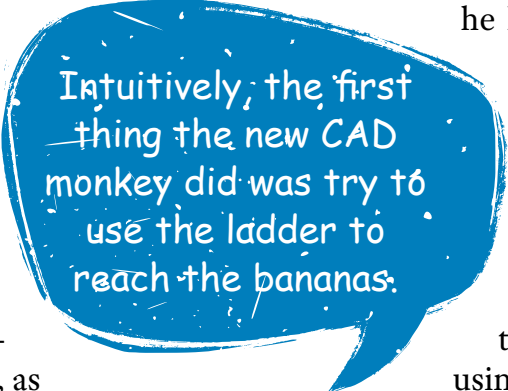
Intuitively, the first thing the new CAD monkey did was try to use the ladder to reach the bananas. After a training session involving several tail lashings from the already schooled CAD monkeys, the new CAD monkey learned the social norm. He was never taught “why” the other CAD monkeys wouldn’t let him use the ladder to reach the bananas because

he had never been sprayed with icy de-ionized water. Nevertheless, he quickly learned that this behavior would not be tolerated by the other CAD monkeys.

They continued trying to reach the banana without using the ladder. They even tried making a CAD monkey pyramid, but again, another monkey was injured.

One by one, each CAD monkey in the cage was injured trying to find a workaround for fear of using the ladder. Each time, the injured CAD monkey would be replaced by a new CAD monkey, who intuitively tried to climb the ladder, only to get beaten up. This happened a few more times until none of the original group remained.

By the end of the experiment, all five CAD monkeys in the cage had learned to never touch the ladder, though none knew the source of this strange reasoning. None of the remaining CAD monkeys knew about the icy de-ionized water abuse—the true root cause of their behavior modification (or tribal knowledge) in the first place. If they had, would they have called HR and recommended that the abusive test engineer be terminated? Once HR created a less toxic, experimental workplace, would



Intuitively, the first thing the new CAD monkey did was try to use the ladder to reach the bananas.

the CAD monkeys have freely moved the ladder and reached the bananas?

If we could have asked the CAD monkeys for their rationale behind not letting their cage mates reset the ladder to reach the bananas, their answer might have been, “I was just following orders.” In other words, “That’s the way we’ve always done it.”

The Moral of the Story

Whether you consider yourself a PCB engineering stakeholder, PCB designer, or even a CAD monkey, let us avoid assuming or creating unfounded conspiracy theories for workflow items in your company which may seem backward. Let’s get to the root cause of stakeholder’s problems and challenges. Let’s communicate and find ways to climb the ladder with the understanding to reach the goals. If icy water is about to be dumped on you,

refuse to be subjected to it. Bust out of the cage and head for HR.

Our engineering managers must see that holding on to the old ways can be a two-edged sword. Values and character traits—both good and bad—exist in the past. A group can be easily wiped out by the competition unless it cultivates a culture which fosters continual exploration and experimentation with new processes, materials, and machinery. **DESIGN007**



Kelly Dack, CIT, CID+, provides DfX centered PCB design and manufacturing liaison expertise for a dynamic EMS provider in the Pacific Northwest while also serving as an IPC design certification instructor (CID) for EPTAC.

To read past columns, [click here](#).

Onboarding New Customers at FCT

Interview by Andy Shaughnessy

After you’ve signed that new customer, then the fun—and hard work—really begins. How do you create a solid customer-vendor relationship from the very first meeting?

At DesignCon 2023, I met with Terrill Schmidt, director of technology for Flexible Circuit Technologies. I asked him to describe FCT’s new customer process, and what it takes to turn a new customer into a long-term partner.

Terrill, what’s your process when you first meet with a new customer? What are some of the challenges they come to you with, and how do you address that?

With new customers, it’s really an education process. We’re learning and understanding what they need, then we go back to them with the right questions and the right knowledge to educate them on what flex can and can’t do. We educate them on the capabilities of flex and help them through the design, everything from basic layout to material stackups.



Terrill Schmidt

We want to be a knowledge resource for them and help them get the product they need, designed for cost-effectiveness and quality. We’re a resource for customers in a variety of ways—not only the actual manufacturing, but also on the design, as well as reliability and quality. We work with all our customers, making sure that these things are there for them. We offer them education, a good product, and good service under one roof.

Educating the customer is a big part of what you offer your customers?

Yes, the more we know and can help them with the design, the faster we can get through prototyping to C stage, B stage, samples, and into production. You know, a lot of us here have multiple decades of experience. We’re able to leverage that information and knowledge, and then work with the customer’s engineering and design group to find an effective way to solve their problem.

Sounds good. Thanks, Terrill.

Thank you, Andy.

PCB Design for Bare Board Testing

Designers Notebook

by Vern Solberg, CONSULTANT

There are several testing methods used for certifying that the circuit board meets its defined quality level and intended functional criteria. Multilayer circuit boards have become increasingly complex: finer conductor line definition, blind and buried microvia interface, and smaller, finer-pitch SMT land pattern geometries. Visual assessment using automated optical inspection (AOI) is adopted to check all circuit layers prior to lamination and as a final examination of the end product.

Flying probe testing, in particular, is a method that uses multiple contactors designed to make rapid electrical contact on exposed conductive terminal sites of the PCB surface. Users have found that the flying probe electri-

cal test is very accurate; it is capable of quickly measuring the electrical characteristics of each board with a high accuracy, ensuring that the board meets the specifications of the design. Probe contactors rapidly move across the circuit board's surface, allowing the test to be completed in a short amount of time. This is beneficial when time is of the essence, such as when a product needs to be released to market quickly. The probes travel across the surface of the board, making contact at pre-designated locations to verify interconnect integrity.

Compared to fixed-probe testing, the flying probe test is not dedicated to a single board design and can test a variety of circuit board designs using the same testing system. By uti-



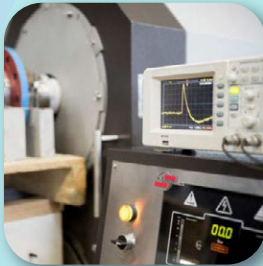


Candor

Circuit Boards

Innovative solutions for complex circuit board manufacturing

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



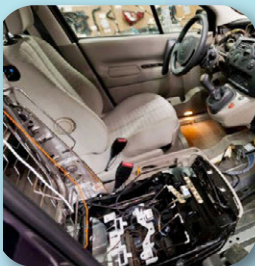
RF/Microwave
PCBs



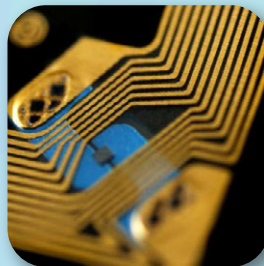
Power
Electronics



High-Speed
Data



Automotive
Electronics



Lamination



Startups

Our technology produces what our customers dream!

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

lizing the CAD data, the probes can be preset to comply with an unlimited number of circuit patterns, enabling the uncompromised testing of individual or panel format circuit boards. Finally, the flying probe test is safe. To prevent the probe contactors from coming into contact with any other components or people, the probes are protected by a guard, minimizing the risk of damage or injury caused by faulty circuit boards.

Flying probe testing is one of the most efficient and cost-effective methods of testing PCBs. It is accurate, fast, flexible, and safe, making it the ideal solution for many types of testing requirements. A leading circuit board fabricator in the Silicon Valley recommends:

- Designated probe contact surfaces must be free of dielectric coatings, resist material, or solder mask
- The preferred surface area provided for probe contact is 0.50 mm (~0.020")

- Minimum probe contact area should not be less than 0.15 mm (~0.006")
- When probing via holes, the preferred land diameter is 0.25 mm (~0.010") but, depending on the test system employed, they may range between 0.20 mm and 0.50 mm (~0.008" and 0.020") diameter

The system shown in Figure 1 is equipped with eight probe heads (4X on the topside and 4X on the bottom) and can simultaneously probe both surfaces of the bare circuit board or panel, enabling micro-pad probe targets as small as 0.30 μ m.

The test system can provide probing speeds up to 180 touches per second and accuracy in the range of ± 10 μ m, and repeatability of ± 5 μ m.

Circuit board manufacturers note that the manufacturing process of printed circuit boards is complex, and the process is cumbersome. Any defects not detected during the post etching phase of the manufacturing process, if not corrected, will seriously affect the quality of the final product. In order to effectively control the progression of quality defects in the production process, and prevent the imperfections from occurring on future runs, prompt corrective action is warranted.

The following identifies defects that may occur during the circuit board fabrication process:

- **Electroplating short circuit:** When the copper conductors are closely spaced and equal in both height and width, copper bridging between circuit paths can occur. The cause of bridging is likely due to dry film resist breakdown during the plating process.
- **Copper bridge short circuit:** A very thin wire-like copper connection between two conductors forming a short circuit. The cause may be attributed to foreign particles that remain trapped between the resist film and copper surface.



Figure 1: Two-sided, high-speed flying probe test system. (Source: SPEA Automated Test Equipment)



Figure 2: Isolating circuit defect locations using X-ray imaging. (Source: OlinaPCB)

- **Insufficient etching:** Where random copper blotches remain in the open, copper-free area, and where closely spaced conductors run in parallel for long distances. This may be due to etch chemical imbalance or when the duration for the etching process is too short.
- **Scratch short circuit:** When a short circuit remains after electroplating and etching. Typically attributed to scratches in the dry film resist by mechanical, human contact, or hair and other fine fibers falling on the board surface. These particles, if not removed before imaging and electroplating, will impede light access, resulting in copper bridging.

Overall, the flying probe test is an efficient and cost-effective method for testing circuit boards after fabrication is completed. It is accurate, fast, flexible, and safe, making it the ideal solution for many types of testing requirements. However, when the circuit board fails during electrical testing, there are other testing methods that can be implemented to identify defects.

The 3D X-ray inspection systems can provide 3D images of the circuit board by creating

a series of 2D cross-sections to expose defects embedded in the board after lamination. While removing the defect on the inner layers of the board may not be practical, identifying and discarding a specific circuit board within a multiple unit panel format will ensure that it does not reach assembly. The images shown in Figure 2 represent the isolated X-ray view of defects within the inner-layer circuit conductor routing path.

In addition to visual and X-ray imaging, thermal imaging, EMI test, and solderability testing is often implemented to further certify that all boards are manufactured correctly and will be assembly process ready, and that any design flaws that would compromise product reliability are identified and corrected. **DESIGN007**



Vern Solberg is an independent technical consultant, specializing in SMT and microelectronics design and manufacturing technology. To read past columns, [click here](#).

Final Finish Design Guidelines

Article by Anaya Vardya

AMERICAN STANDARD CIRCUITS

One of the biggest challenges facing PCB designers is not understanding the cost drivers in the PCB manufacturing process. There are many design considerations that need to be factored into any decision of which final finish to select for a particular application. We will cover these in this final installment of final finish DFM.

Final Finish Design Guidelines

Shelf Life

The shelf lives shown in Table 1 assume that the materials have been stored in the original unopened package under conditions of 68° to 77°F (20-25°C) and < 50% RH.

Table 1: Final finishes and corresponding shelf lives

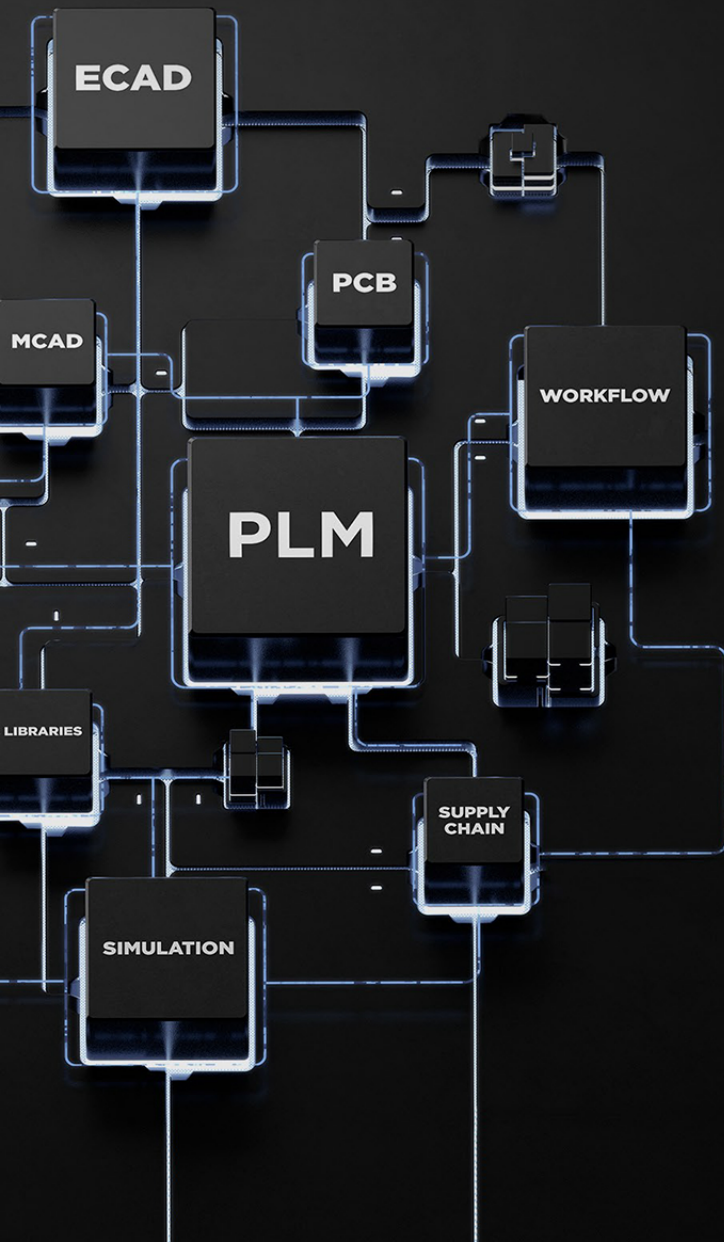
Finish	Shelf Life
Sn/Pb and LF HASL	12 months
Electrolytic Nickel/Gold	12 months
ENIG ¹	6+ months
ENIPIG, ENEPIG ²	6+ months
ImmAg with anti-tarnish ³	6-12 months
ImmSn ⁴	12 months
OSP	6 months

¹Per IPC-4552, ²Per IPC-4556, ³Per IPC-4553A, ⁴Per IPC-4554

Table 2: PCB surface finish material characteristics

Finish	Brand Name	Handling Concerns	Hold Time Between Reflow Cycles	Multi-thermal Cycles Assembly Applications	Shelf Life	Solderability
ENIG	AT5600EN & AT6000 IAU Technic	Organic and fingerprints	No concerns	Good	Good	Excellent
Electrolytic Ni/Au	Orosene 80RC Technic	Organic and fingerprints	No concerns	Good	Good	Fair
Soft Wire-bondable Au	Orosene 434 Technic	Organic and fingerprints	No concerns	Good	Good	Good
ImmAg	Sterling/MacDermid	Sulfates and Chlorides	No concerns	Good	Good	Excellent
ImmSn	TS28 White Tin Florida Cirtech	Organic and fingerprints & excessive heat cycles	24 hours	Fair	Fair	Excellent
OSP	Entek Plus HT Enthone	Organic and fingerprints	24 hours	Fair	Good	Good

A comprehensive, digitally integrated electronic design solution for enterprises of all sizes.



- Digitally-connected ECAD to the rest of the enterprise
- Cross-domain and cross-discipline collaboration
- Digitally-managed workflows, processes and resources

As the most lightweight, easy-to-implement electronics design solution for enterprises, Altium NEXUS makes the last mile of digital transformation not only possible - but easy.

Table 3: Availability of RoHS

SPECIFICATION	AVAILABILITY	
	STANDARD	AVAILABLE
LF HASL	X	
Immersion Silver	X	
Electroless Nickel/Immersion Gold (ENIG)	X	
OSP (Entek Plus CU-106A)	X	
Electroless Nickel/Electroless Pd/Immersion Gold (ENEPIG)		X
Electrolytic Hard Gold	X	
Electrolytic Soft Gold	X	
Electroplated	X	
Immersion Tin	X	

Table 4: Key attributes summary

Lead Free Hot Air Solder Leveling (LF HASL)

REQUIREMENT	SPECIFICATION	
	MINIMUM	MAXIMUM
	2.54µm	25.4µm

Immersion Silver

REQUIREMENT	SPECIFICATION	
	MINIMUM	MAXIMUM
Silver Thickness	6.0 µin.	25.0 µin.

ENIG

REQUIREMENT	SPECIFICATION	
	MINIMUM	MAXIMUM
Nickel Thickness	100 µin.	
Gold Thickness	2 µin.	8 µin.

Organic Solderability Preservative (OSP)

REQUIREMENT	SPECIFICATION	
	MINIMUM	MAXIMUM
Benzotriazole (ENTEK 106A) Thickness	~50 Angstroms	
Benzotriazole (ENTEK 106A) Solder Cycles		1
PCB Thickness	0.020"	0.250"

- The IPC-16012A requirements for OSP surface finishes are that they be solderable.
- No thickness is specified, as it is very difficult to measure.

Table 5: Comparison of key attributes for various finishes

ENEPIG

REQUIREMENT	SPECIFICATION	
	MINIMUM	MAXIMUM
Nickel Thickness	200 µin.	
Palladium Thickness	12 µin.	
Gold Thickness	2 µin.	8 µin.

Electrolytic Hard Gold

REQUIREMENT	SPECIFICATION	
	MINIMUM	MAXIMUM
Nickel Thickness	150 µin.	
Gold Thickness	40 µin.	
Flash Gold Thickness	3 µin.	

Electrolytic Soft Gold, Wire bondable

REQUIREMENT	SPECIFICATION	
	MINIMUM	MAXIMUM
Nickel Thickness	150 µin.	
Gold Thickness	40 µin.	

Electroplated

REQUIREMENT	SPECIFICATION	
	MINIMUM	MAXIMUM
Nickel Thickness	100 µin.	
Gold Thickness	20 µin.	
Finger Length		0.750"
Connector Span		23.25"
Distance from PCB Edge		5.5"

Immersion Tin

REQUIREMENT	SPECIFICATION	
	MINIMUM	MAXIMUM
Tin Thickness	25.6 µin.	

Availability of of Non-RoHS

REQUIREMENT	AVAILABILITY	
	STANDARD	AVAILABLE
Hot Air Solder Leveling	X	
Fused Tin Lead	X	

Table 6: Key attributes summary

Hot Air Solder Leveling (HASL)

Requirement	Specification	
	Minimum	Maximum
Standard SnPb Thickness	50 μin.	1500 μin.
Special SnPb Thickness	100 μin.	1000 μin.
Solder Thickness at the PTH Knee	10 μin.	25 μin.

- Thickness measurements are taken at the pad’s centroid
- As pad size decreases, solder thickness per unit area increases
- The solder thickness distribution of the PCB is determined by the solder thickness requirements of the smallest features

Table 7: Fused tin lead

Immersion Tin

Requirement	Specification	
	Minimum	Maximum
SnPb Thickness	Covered & Solderable	

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 educational titles. He also co-authored “Fundamentals of Printed Circuit Board Technologies” and provides a discussion of flex and rigid flex PCBs at *RealTime with... American Standard Circuits*.

1007e

Get smarter here.

NEW!



IPC Community Magazine is an exciting, new, quarterly publication with a strong editorial focus on members' success.



Download



Flex007 Highlights



Mapping the Future of Flexible and Printed Electronics ►

The “OE-A Roadmap for Flexible, Organic and Printed Electronics” white paper provides a close look into the advancing maturity of the industry. In addition to updates on key technologies and capabilities, more than 100 industry experts have contributed detailed short-, medium-, and long-term forecasts for markets including automotive, consumer electronics, healthcare, printing and packaging, smart building, and internet of things.

VTT: Environmental Impact of Flexible Electronics Can be Reduced by Almost 90% ►

VTT Technical Research Centre of Finland announces the results of a study investigating how changing the manufacturing of electronics can improve their environmental impact. For example, one of the project’s most significant findings showed that environmental impact could be reduced by 86% when additive printing methods are used to create flexible electronic components, as reported in detail by LUT University.

Real Time with... IPC APEX EXPO 2023: Flex Innovation ►

Casey Krueger from MKS Instruments speaks with Andy Shaughnessy about increasing yields and via quality in high-volume production manufacturing. They also discuss innovations in flex and improving the material work surface to increase via quality, and their integration with MKS’ Atotech, a company MKS acquired in 2021.

Real Time with... IPC APEX EXPO 2023: Flat Flex Cable a Potentially Disruptive Idea ►

Managing Editor Andy Shaughnessy and Jason Michaud, VP of sales and marketing for Miraco, discuss the company’s flat flex cable, which could replace the flexible circuit in certain situations. Jason explains why the company is happy to offer customers less expensive options, even if it costs them money in the short run.

Flexible Thinking: A Once and Future Idea ►

Electronic assemblies are typically comprised of a mix of discrete resistors, capacitors, inductors, and the like, along with numerous integrated circuit chips, each chip having a certain function or range of different functions. In such assemblies, there are also several different connectors and/or sockets that allow for the assembly to be connected to other assemblies.

Rogers Reports Q4, Full Year 2022 Results ►

Net sales of \$223.7 million decreased 9.5% versus the prior quarter resulting from the impact of China COVID-related restrictions, weaker demand in certain markets and unfavorable currency exchange rate fluctuations. AES net sales decreased by 4.1% related to unfavorable currency exchange rates and lower sales in the aerospace and defense and other markets. EMS net sales decreased by 15.6% primarily resulting from lower portable electronics and general industrial market demand.

Choose wisely.

With today's material performance requirements, it's more important than ever to understand and choose your materials wisely.



Download Now

I.007e
Books

Tribal Knowledge: A Personal Perspective

Flexible Thinking

Feature Column by Joe Fjelstad, VERDANT ELECTRONICS

With the benefit of more than half a century of experience in the printed circuit and electronics interconnection industry, I now enjoy a perspective that is not available to those just entering. I harken to a comment made by legendary Swedish film director, Ingmar Bergman, “Old age (growing old) is like climbing a mountain. You climb from ledge to ledge. The higher you get, the more tired and breathless you become, but your views become more extensive.” That perspective is truly a gift, regardless of what one does in life. It certainly has resonated with me (even before I arrived here.)

The young often balk at the knowledge of the “old folks” as foolish and old fashioned. But a

couple of timeless Mark Twain quotes come to mind at such times:

“When I was 17, my father was so stupid, I didn’t want to be seen with him in public. When I was 24, I was amazed at how much the old man had learned in just seven years.” Equally witty and more to the point when it comes to manufacturing: “It ain’t what you don’t know that gets you into trouble, it’s what you know for sure that just ain’t so.”

I know these things too well because I was young once. A final timeless aphorism: “A little knowledge is a dangerous thing.”

Enough of my referencing backwoods wisdom. I trust you get the idea.

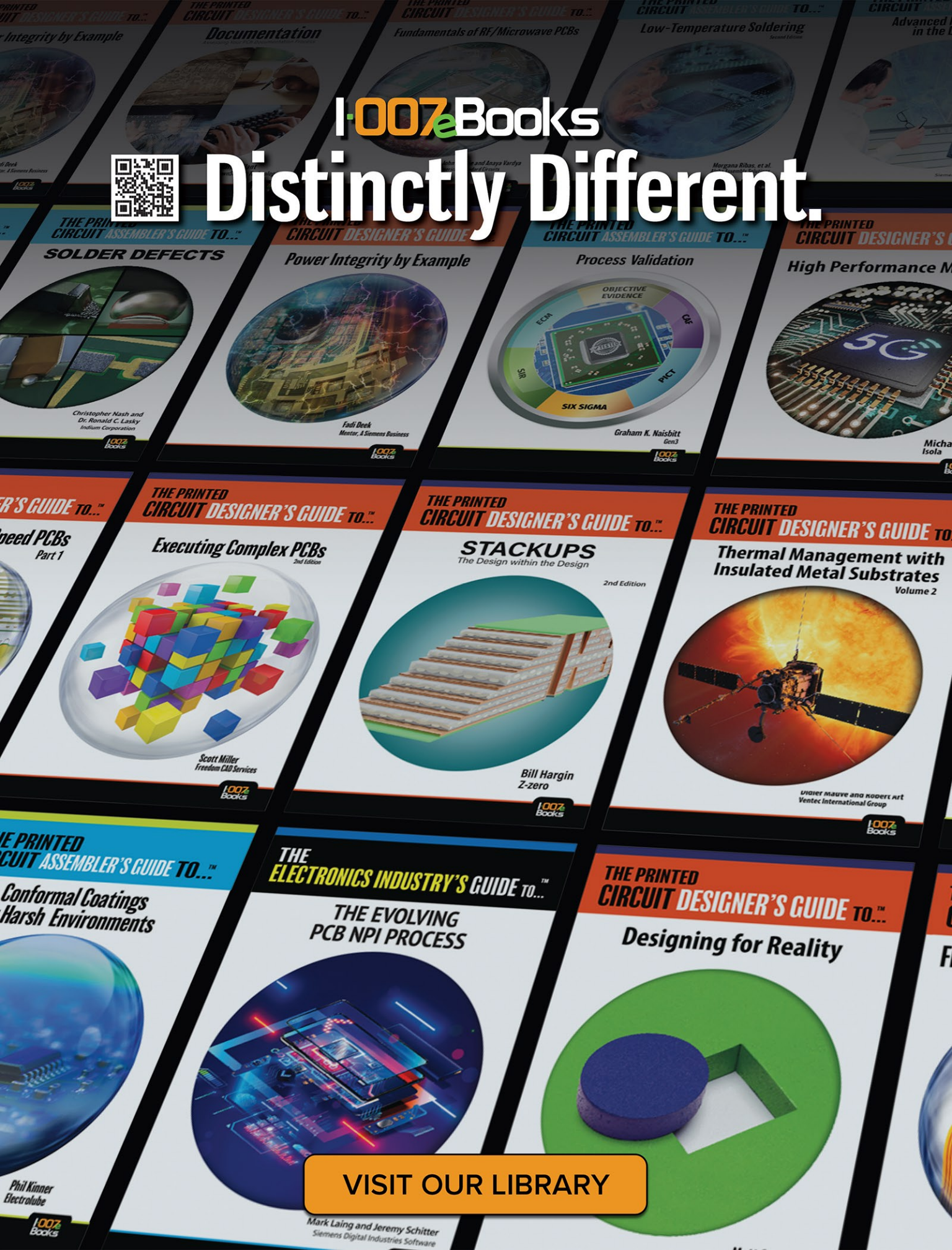
In my experience, using tribal knowledge





1007Books

Distinctly Different.



VISIT OUR LIBRARY

was how we managed to pass on our hard-won manufacturing experiences. Our personal scar tissues from trying and failing were how we informed those new to the industry with whom we were working what methods worked and what didn't. There was little generally available in the way of detailed codified knowledge in the days before my entry into the PCB industry.

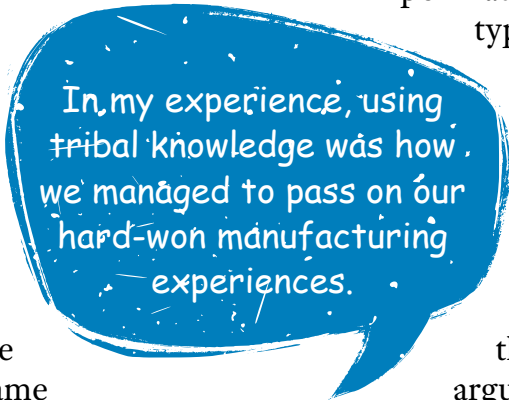
The Institute of Printed Circuits (the legacy name for IPC) created its first industry document, "How to Design and Specify Printed Circuits," in 1957, but what was lacking was detailed information on how to build them. An early attempt was made by a couple of unsung heroes of the industry, Cleo Brunetti and Roger W. Curtis, working for the National Bureau of Standards in the late 1940s. They surveyed the industry after WWII to collect and share what novel and useful printed circuit-related developments had been "born in the forage of war." The end product came from the U.S. Government Publishing Office a decade earlier than IPC's first publication in a short book titled "Printed Circuit Techniques,"¹ followed by another booklet a year later titled "New Advances in Printed Circuits."² These two technical showcases became teaching tools for commercial industries wanting to build products for a nation looking to enjoy the fruits of victory, especially electronic products such as radios and early televisions.

The PCB industry was somewhat spread out, but was largely located on the East Coast and some places in the Midwest. Silicon Valley was just being born and Hewlett-Packard was one of its most important early founding companies and the company needed PCBs to make their products. In the late 1960s, Clyde Coombs, with the approval of his managers at Hewlett-Packard, engaged the most knowl-

edgeable engineers in the early PCB manufacturing industry of the nation to write chapters covering their special experts from design to manufacture and Clyde served as editor for what is today still the bible of the PCB industry: *The Printed Circuit Handbook*, now in its seventh edition and co-edited with PCB industry icon (and revered I-Connect007 technical editor) Happy Holden.

These valuable books were published and evidently available but not always easy to find. Much knowledge transfer in the industry was accomplished through the diligence and efforts of field engineers of materials and manufacturing equipment and process chemistry product developers who served as the all-important "pollinators" of the PCB industry. They typically visited PCB shops both regionally and nationally, teaching and helping customers succeed with their products, while learning from customer shop process engineers what they found worked well. It is arguable that the PCB industry would have developed much slower without them.

I would be remiss at this point to not mention the importance of competition and the laissez-faire approach to employment at the time where engineers and technicians, in the days before non-compete contracts and NDAs, freely jumped from employer to employer, bringing with them knowledge gained at their last employer, and often being given a healthy pay raise. That said, there were also the behemoths of industry, such as IBM and AT&T, who were vertically integrated and insulated from the general industry, solving problems internally with their highly educated engineering staffs. In such facilities, it appeared that tribal knowledge was relied upon to develop and improve processes, but it was the discipline of codifying the knowledge into process



In my experience, using tribal knowledge was how we managed to pass on our hard-won manufacturing experiences.

specifications that allowed them to more easily transfer knowledge from one generation of workers to another.

The U.S. has lost a great deal of its tribal knowledge by the transfer of manufacturing to China and elsewhere over the last few decades. There is a great deal of concern in the U.S. today relative to the future of the industry as the older generation of engineers and technicians is “graying out” of the industry. This has not gone unnoticed, fortunately, and IPC, SMTA, and the DoD, along with some higher education providers, are actively supporting efforts to shore up, prime, and refill the pipeline of talent. We hope we are not too late. PCB manufacturing has been a great career for me. The myriad processes and technologies required to make a printed circuit have not greatly changed in intention and purpose, but they are endlessly fascinating and challenging to try and perfect.

In closing, I highly recommend that you take a moment to skim through the two earlier books I’ve cited. I think you will be amazed at how prescient the pioneers of the PCB industry were. There you will find a “genetic link” to nearly all of the processes we use today. We are blessed to be able to follow in their footsteps. **DESIGN007**

References

1. “Printed Circuit Techniques,” National Bureau of Standards, U.S. Department of Commerce, nvlpubs.nist.gov.
2. “New Advances in Printed Circuits,” U.S. Department of Commerce, nvlpubs.nist.gov.



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 copy of Fjelstad’s book *Flexible Circuit Technology, 4th Edition*, and watch his in-depth workshop series “Flexible Circuit Technology.”

Soft Touch Sensitivity



Vincent Tung

Compliant and conductive carbon nanomaterial could be the perfect fit for on-skin electronics.

A soft and flexible electronic “e-skin,” so sensitive it can detect the minute temperature difference between an inhaled and an exhaled breath, could form the basis of a new form of on-skin biosensor. The ultrathin material is also sensitive to touch and body motion, suggesting a wide array of potential applications.

“The skin plays a vital role in our interactions with the world,” says Vincent Tung from KAUST, who led the work. “Recreating its properties in an e-skin could have profound implications for wearable electronics, as well as for sensory prosthetics, soft robotics and human-machine interfaces,” he says.

A carbon nanomaterial called hydrogen-substituted graphdiyne (HsGDY) could be ideal for the task, Tung and his collaborators have shown. “The implementation of HsGDY into e-skin has long been touted by theorists, but had yet to be demonstrated experimentally,” Tung says.

The team were able to show what the theory had predicted: the resulting material was highly twistable, stretchable and mechanically durable. “At around 18 nanometers thick, our e-skin is a fraction of the thickness of human skin, enabling conformal contact and long-term adhesion to the body with maximum flexibility and comfort,” Tung says. (Source: KAUST)

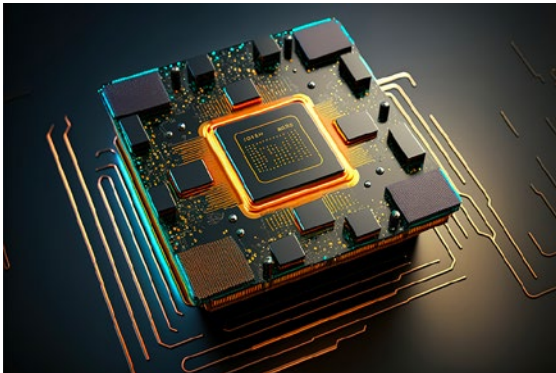
Six Money-saving Material Properties When Designing PCB Stackups



Stackup decisions are critical to every PCB and electronic product, but they don't always get the attention they deserve. With better stackup planning focusing on six key parameters, designers can select the best materials early in the design process and minimize the risk of under- and over-designing their PCB.

Shrinking Silicon, EMI, and SI

As IC features continue to shrink, the PCB designer's job gets more interesting—signal speeds and rise times are increasing, they're encountering EMI and signal integrity issues once only seen in the RF world.



Hexagon, Altium Partner to Improve the Sustainability of the Electronics Industry

Hexagon's Manufacturing Intelligence division and Altium have entered a strategic partnership which will help the design and manufacturing of electronics be more environmentally sustainable. The partnership combines Hexagon's expertise in utilizing data from design and engineering, manufacturing, and metrology with Altium's strengths in PCB design and electronics supply chain intelligence to help companies understand the impact of product development decisions on the sustainability of electronics and smart products.



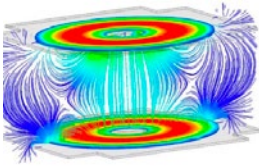
IPC Design Competition Champion Crowned at IPC APEX EXPO 2023

At IPC APEX EXPO in San Diego, five competitors squared off to determine who was the best of the best at PCB design. At the end of the four-hour round, each competitor delivered their project file to the judges who convened to review designs, with a winning PCB design going to Sathishkumar Vijayakumar, senior PCB design engineer at Tessolve Semiconductor Pvt Ltd. in Bengaluru, India.



Sathishkumar Vijayakumar

Beyond Design: Displacement Current—The Key to Electromagnetic Energy Propagation



The propagation of electromagnetic energy can be controlled in several ways depending on the medium the energy is traveling in.

However, electromagnetic waves do not require a medium to propagate. This means that electromagnetic waves can travel not only through liquids, solids, and air, but also through the vacuum of space.

Shrinking Geometries: Back to Fundamentals to Fight EMI

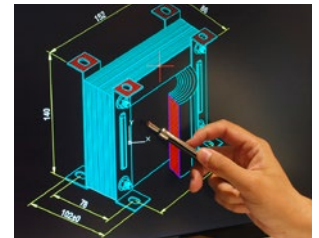
When silicon shrinks, a variety of things can happen—some positive, some negative. But for PCB designers, the fight against EMI becomes more complex as signal channels shrink and rise times increase. In this interview, Dan Beeker explains what happens when silicon shrinks, how feature size controls signal speed, and why this marks the perfect time to return to the fundamentals of physics and field theory.

Real Time with... IPC APEX EXPO 2023: Polar Driving Software Development for Customers

Andy Shaughnessy sits down with product specialist Erik Bateham, a new hire at Polar Instruments. Erik explains his circuitous route into circuit analysis tools. He also discusses the challenges that Polar's customers are facing, and how user input drives Polar's software development.

PCB Designers Are Really Product Designers

In the ECAD/EDA world, we still focus mostly on simulation and fit to enclosure, but there is so much more that goes into full-scale product development, and the ECAD software world should start to move much faster toward the goals I outline below.



Connect the Dots: Medical Technology—How PCBs Help Save Lives

During prototype design, manufacturers need effective communication and coordinated effort to ensure they get the quality boards their projects need to succeed. Close collaboration with a domestic PCB manufacturer offers more transparency during the manufacturing process, accelerates issue resolution, increases yield, and—most importantly—improves PCB quality.



Keysight Expands EDA Software Portfolio with the Acquisition of Clifsoft

Keysight Technologies has acquired Clifsoft and will be adding the company's line of hardware design data and intellectual property (IP) management software tools to its portfolio of electronic design automation (EDA) solutions.

For the latest news and information, visit PCBDesign007.com

Career Opportunities



Find industry-experienced candidates at jobConnect007.

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, suppliers and the academic community.

In addition, your ad will:

- be featured in at least one of our newsletters
- appear on our jobConnect007.com board, which is promoted in every newsletter
- appear in our monthly [Careers Guide](#), emailed to 26,000 potential candidates

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.

No contract required. Just send over your copy and company logo and we'll do the rest!

Contact barb@iconnect007.com to get your ad posted today!

+1 916.365.1727

I-Connect007
GOOD FOR THE INDUSTRY



Career Opportunities



Business Development Representative (Remote)

IPC Publishing Group (I-Connect007), a subsidiary of IPC International, Inc., is looking to hire a Business Development Representative. This position is a combination of sales and account management and will be responsible for building relationships with prospects to identify, qualify and close new business opportunities. The Sales Representative is responsible for selling the full suite of I-Connect007 products and services and working with the customer service team to ensure complete customer satisfaction. Must be well organized and an effective communicator.

RESPONSIBILITIES

- Build and develop new business through positive relationships with prospects
- Achieve monthly, quarterly, and annual revenue quotas and expected activity metrics
- Effectively manage internal and external relationships to maximize customer growth and total customer satisfaction
- Use CRM to document customer/prospect communication activities, opportunities, and customer service issues
- Conduct effective annual planning with quarterly updates
- Track and report sales projections, opportunities, monthly revenue, progress toward goals

REQUIREMENTS

Not all applicants will have skills that match a job description exactly. IPC values diverse experiences in other industries, and we encourage everyone who meets the required qualifications to apply. While having "desired" qualifications makes for a strong candidate, we encourage applicants with alternative experiences to also apply. If your career is just starting or hasn't followed a traditional path, don't let that stop you from considering IPC. We are always looking for people who will bring something new to the table!

- BA/BS preferred. Minimum 2 years successful experience in a high-energy selling/account management environment
- Demonstrated ability to think strategically to successfully engage and develop relationships
- Exceptional verbal and written communication skills with a strong business acumen
- Must be an excellent listener and relationship builder
- Domestic and international travel (up to 50%)
- Must work independently and as part of a team
- Ability to work in a fast-paced environment, on a broad range of projects, and maintain a sense of urgency

IPC International, Inc. is an Equal Opportunity Employer that supports diversity, equity & inclusion wherever we do business. Competitive salary and excellent employee benefits

MRA—The Management Association is conducting this search on behalf of IPC International, Inc. Please send resume and cover letter to e-mail: Lyssa.bernstein@mranet.org. Please include resume and cover letter. Subject line should say: BD

Since 1957, IPC has been guiding the electronics industry through dramatic changes. IPC is the only trade association that brings together all the players in this industry: designers, board manufacturers, assembly companies, suppliers, and original equipment manufacturers. More than 6,000 companies around the world depend on IPC programs and services to further their competitive excellence and financial success. As a member-driven organization and leading source for industry standards, training, market research and public policy advocacy, IPC supports programs to meet the needs of an estimated \$2 trillion global electronics industry.

NOTE: This job opening has been filled.

Career Opportunities



THERMAL SYSTEMS

Sales Engineer SMT North Mexico

Rehm Thermal Systems, a leading German manufacturer of reflow soldering systems with convection or condensation and drying and coating systems, has produced energy-efficient manufacturing equipment for the electronics and photovoltaics industry since 1990. We also offer tailor-made applications related to the soldering, coating and hardening of modules.

Responsibilities:

- This position is responsible for expanding our customer network and maintaining existing customer relationships in the Northeast Mexico region. The Sales Engineer would work closely with the German headquarters and the General Manager Rehm Mexico to implement the sales strategy.
- A candidate's proximity to Monterrey, Mexico, is a plus.

Qualifications:

- An Engineering degree or comparable qualification with a strong technical background is required.
- Sales-oriented attitude, good communication skills and willingness to travel frequently within Mexico is essential.

We offer innovative products, a great dynamic work environment and exciting training opportunities in our German headquarters.

To learn more about Rehm Group, please visit our website at www.rehm-group.com.

Please send resumes to: Mr. Luis Garcia at luis.garcia@rehm-group.com.

[apply now](#)



Sr. Test Engineer (STE-MD)

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.

- 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.

[apply now](#)

Career Opportunities



Europe Technical Sales Engineer

Taiyo is the world leader in solder mask products and inkjet technology, offering specialty dielectric inks and via filling inks for use with microvia and build-up technologies, as well as thermal-cure and UV-cure solder masks and inkjet and packaging inks.

PRIMARY FUNCTION:

1. To promote, demonstrate, sell, and service Taiyo's products
2. Assist colleagues with quotes for new customers from a technical perspective
3. Serve as primary technical point of contact to customers providing both pre- and post-sales advice
4. Interact regularly with other Taiyo team members, such as: Product design, development, production, purchasing, quality, and senior company managers from Taiyo's group of companies

ESSENTIAL DUTIES:

1. Maintain existing business and pursue new business to meet the sales goals
2. Build strong relationships with existing and new customers
3. Troubleshoot customer problems
4. Provide consultative sales solutions to customers technical issues
5. Write monthly reports
6. Conduct technical audits
7. Conduct product evaluations

QUALIFICATIONS / SKILLS:

1. College degree preferred, with solid knowledge of chemistry
2. Five years' technical sales experience, preferably in the PCB industry
3. Computer knowledge
4. Sales skills
5. Good interpersonal relationship skills
6. Bilingual (German/English) preferred

To apply, email: BobW@Taiyo-america.com
with a subject line of "Application for
Technical Sales Engineer."

[apply now](#)



IPC Instructor Longmont, CO

This position is responsible for delivering effective electronics manufacturing training, including IPC certification, to adult 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 primarily conduct training at our public training center in Longmont, Colo., or will travel directly to the customer's facility. It is highly preferred that the candidate be willing to travel 25–50% of the time. Several IPC certification courses can be taught remotely and require no travel or in-person training.

Required: A minimum of 5 years' experience in electronics manufacturing and familiarity with IPC standards. Candidates with current IPC CIS or CIT Trainer Specialist certifications are highly preferred.

Salary: Starting at \$30 per hour depending on experience

Benefits:

- 401k and 401k matching
- Dental and Vision Insurance
- Employee Assistance Program
- Flexible Spending Account
- Health Insurance
- Health Savings Account
- Life Insurance
- Paid Time Off

Schedule: Monday thru Friday, 8–5

Experience: Electronics Manufacturing:
5+ years (Required)

License/Certification: IPC Certification—
Preferred, Not Required

Willingness to travel: 25% (Required)

[apply now](#)

Career Opportunities



Technical Sales Manager

Objectives

Provide sales leadership and management for a regional sales territory. Responsible for retaining current customers as well as developing and attracting new customers and markets. Responsible for selling current and new products, keeping abreast of new technologies, market trends, and customer product needs.

Essential Functions and Responsibilities

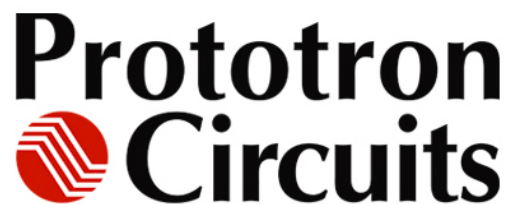
- Develop and service assigned geographic region
- Actively and consistently seek new customers
- Visit customers and potential customers to develop relationships, deliver sales presentations, follow up on leads, and close sales
- Provide technical support and product recommendations in person, by email, and phone
- Manage major accounts; establish long-term, ongoing relationships with key individuals
- Provide feedback to Chemcut as well as sales peers regarding competition, pricing, and marketing opportunities

Qualifications

- Bachelor's degree in mechanical, electrical, chemical engineering or related fields
- 3-5 years of field sales experience with technology-driven industrial products
- Well-developed sales and customer relations skills
- Ability to make decisions and evaluations to determine customer needs
- Ability to travel up to 50% of the time
- Excellent oral and written communication skills
- Knowledge of target market industries

To apply, please submit a cover letter and resume to hr@chemcut.net.

[apply now](#)



Sales Representatives

Prototron Circuits, a market-leading, quick-turn PCB manufacturer located in Tucson, AZ, is looking for sales representatives for the Utah/Colorado, and Northern California territories. With 35+ years of experience, our PCB manufacturing capabilities reach far beyond that of your typical fabricator.

Reasons you should work with Prototron:

- Solid reputation for on-time delivery (98+% on-time)
- Capacity for growth
- Excellent quality
- Production quality quick-turn services in as little as 24 hours
- 5-day standard lead time
- RF/microwave and special materials
- AS9100D
- MIL-PRF- 31032
- ITAR
- Global sourcing option (Taiwan)
- Engineering consultation, impedance modeling
- Completely customer-focused team

Interested? Please contact Russ Adams at (206) 351-0281 or russa@prototron.com.

[apply now](#)

Career Opportunities



Test Engineer, Electronics Engineer

Keytronic is a dynamic, team-based contract manufacturer with facilities worldwide. Innovation defines us. Come join us in Spokane, Washington! We invite you to bring your engineering expertise and passion for excellence. In turn, we provide meaningful opportunities for you to implement these attributes to their fullest while working together to bring our customer's high-tech automotive, aerospace, medical and commercial products to full production.

We encourage you to apply to one of our open positions below if you enjoy being challenged, working in a dynamic work setting and being a part of a team creating products to improve our world.

- **Test Engineer**—You will assist in conducting electrical test engineering support involving automation, assembly, maintenance, and data collection.
- **Electronics Engineer**—You will work on a team creating electronic circuitry, writing firmware for microprocessors and interfacing with customer development teams producing a wide array of products.

[apply now](#)



Regional Manager Midwest Region

General Summary: Manages sales of the company's products and services, Electronics and Industrial, within the Carolinas and Mid-Atlantic Region. Reports directly to Americas Manager. Collaborates with the Americas Manager to ensure consistent, profitable growth in sales revenues through positive planning, deployment and management of sales reps. Identifies objectives, strategies and action plans to improve short- and long-term sales and earnings for all product lines.

DETAILS OF FUNCTION:

- Develops and maintains strategic partner relationships
- Manages and develops sales reps:
 - Reviews progress of sales performance
 - Provides quarterly results assessments of sales reps' performance
 - Works with sales reps to identify and contact decision-makers
 - Setting growth targets for sales reps
 - Educates sales reps by conducting programs/seminars in the needed areas of knowledge
- Collects customer feedback and market research (products and competitors)
- Coordinates with other company departments to provide superior customer service

QUALIFICATIONS:

- 5-7+ years of related experience in the manufacturing sector or equivalent combination of formal education and experience
- Excellent oral and written communication skills
- Business-to-business sales experience a plus
- Good working knowledge of Microsoft Office Suite and common smart phone apps
- Valid driver's license
- 75-80% regional travel required

To apply, please submit a COVER LETTER and RESUME to: Fernando Rueda, Americas Manager

fernando_rueda@kyzen.com

[apply now](#)

Career Opportunities



Application Engineer

Flexible Circuit Technologies (FCT) is a global supplier providing design, prototyping and production of flexible circuits, rigid flex circuits, flexible heaters and full assembly services.

Responsibilities

- Gain understanding for customer/specific project requirements
- Review customer files, analyze - application, design, stack up, materials, mechanical requirements; develop cost-effective design to meet requirements
- Quote and follow-up to secure business
- Work with CAD: finalize files, attain customer approval prior to build
- Track timeline/provide customers with updates
- Follow up on prototype, assist with design changes (if needed), and push forward to production
- Work as the lead technician/program manager or as part of FCT team working with an assigned application engineer
- Help customer understand FCT's assembly, testing, and box build services
- Understand manufacturing and build process for flexible and rigid-flex circuits

Qualifications

- Demonstrated experience: flex circuit/rigid-flex design, including design rules, IPC; flex heater design +
- Ability to work in fast-paced environment, broad range of projects, maintain sense of urgency
- Ability to work as a team player
- Excellent written and verbal communication skills
- Willing to travel for sales support and customer support activities if needed

Competitive salary, bonus program, and benefits package. Preferred location Minneapolis, MN area.

[apply now](#)



Technical Marketing Engineer

EMA Design Automation, a leader in product development solutions, is in search of a detail-oriented individual who can apply their knowledge of electrical design and CAD software to assist marketing in the creation of videos, training materials, blog posts, and more. This Technical Marketing Engineer role is ideal for analytical problem-solvers who enjoy educating and teaching others.

Requirements:

- Bachelor's degree in electrical engineering or related field with a basic understanding of engineering theories and terminology required
- Basic knowledge of schematic design, PCB design, and simulation with experience in OrCAD or Allegro preferred
- Candidates must possess excellent writing skills with an understanding of sentence structure and grammar
- Basic knowledge of video editing and experience using Camtasia or Adobe Premiere Pro is preferred but not required
- Must be able to collaborate well with others and have excellent written and verbal communication skills for this remote position

EMA Design Automation is a small, family-owned company that fosters a flexible, collaborative environment and promotes professional growth.

Send Resumes to: resumes@ema-eda.com

[apply now](#)

Career Opportunities



MACHINES FOR PRINTED CIRCUIT BOARDS

Field Service Engineer

Location: West Coast, Midwest

Pluritec North America, Ltd., an innovative leader in drilling, routing, and automated inspection in the printed circuit board industry, is seeking a full-time field service engineer.

This individual will support service for North America in printed circuit board drill/routing and X-ray inspection equipment.

Duties included: Installation, training, maintenance, and repair. Must be able to troubleshoot electrical and mechanical issues in the field as well as calibrate products, perform modifications and retrofits. Diagnose effectively with customer via telephone support. Assist in optimization of machine operations.

A technical degree is preferred, along with strong verbal and written communication skills. Read and interpret schematics, collect data, write technical reports.

Valid driver's license is required, as well as a passport, and major credit card for travel.

Must be able to travel extensively.

[apply now](#)



ventec

INTERNATIONAL GROUP

騰輝電子

European Product Manager Taiyo Inks, Germany

We are looking for a European product manager to serve as the primary point of contact for product technical sales activities specifically for Taiyo Inks in Europe.

Duties include:

- Business development & sales growth in Europe
- Subject matter expert for Taiyo ink solutions
- Frequent travel to targeted strategic customers/OEMs in Europe
- Technical support to customers to solve application issues
- Liaising with operational and supply chain teams to support customer service

Skills and abilities required:

- Extensive sales, product management, product application experience
- European citizenship (or authorization to work in Europe/Germany)
- Fluency in English language (spoken & written)
- Good written & verbal communications skills
- Printed circuit board industry experience an advantage
- Ability to work well both independently and as part of a team
- Good user knowledge of common Microsoft Office programs
- Full driving license essential

What's on offer:

- Salary & sales commission--competitive and commensurate with experience
- Pension and health insurance following satisfactory probation
- Company car or car allowance

This is a fantastic opportunity to become part of a successful brand and leading team with excellent benefits. Please forward your resume to jobs@ventec-europe.com.

[apply now](#)

Career Opportunities



Technical Service & Applications Engineer Full-Time — Midwest (WI, IL, MI)

Koh Young Technology, founded in 2002 in Seoul, South Korea, is the world leader in 3D measurement-based inspection technology for electronics manufacturing. Located in Duluth, GA, Koh Young America has been serving its partners since 2010 and is expanding the team with an Applications Engineer to provide helpdesk support by delivering guidance on operation, maintenance, and programming remotely or on-site.

Responsibilities

- Provide support, preventive and corrective maintenance, process audits, and related services
- Train users on proper operation, maintenance, programming, and best practices
- Recommend and oversee operational, process, or other performance improvements
- Effectively troubleshoot and resolve machine, system, and process issues

Skills and Qualifications

- Bachelor's in a technical discipline, relevant Associate's, or equivalent vocational or military training
- Knowledge of electronics manufacturing, robotics, PCB assembly, and/or AI; 2-4 years of experience
- SPI/AOI programming, operation, and maintenance experience preferred
- 75% domestic and international travel (valid U.S. or Canadian passport, required)
- Able to work effectively and independently with minimal supervision
- Able to readily understand and interpret detailed documents, drawings, and specifications

Benefits

- Health/Dental/Vision/Life Insurance with no employee premium (including dependent coverage)
- 401K retirement plan
- Generous PTO and paid holidays

[apply now](#)



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](#)

Career Opportunities

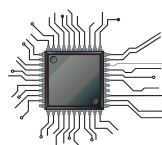


Are You Our Next Superstar?!

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 to 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.

[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



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



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, panelization 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



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](#)



APCT
Passion | Commitment | Trust

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](#)

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

Designing for Reality by Matt Stevenson, Sunstone Circuits

Based on the wisdom of 50 years of PCB manufacturing at Sunstone Circuits, this book is a must-have reference for designers seeking to understand the PCB manufacturing process as it relates to their design. Designing for manufacturability requires understanding the production process fundamentals and factors within the process that often lead to variations in manufacturability, reliability, and cost of the board. Speaking of making better decisions, [read it now!](#)



Thermal Management with Insulated Metal Substrates, Vol. 2

by Didier Mauve and Robert Art, Ventec International Group

This book covers the latest developments in the field of thermal management, particularly in insulated metal substrates, using state-of-the-art products as examples and focusing on specific solutions and enhanced properties of IMS. [Add this essential book to your library.](#)



High Performance Materials

by Michael Gay, Isola

This book provides the reader with a clearer picture of what to know when selecting which material is most desirable for their upcoming products and a solid base for making material selection decisions. [Get your copy now!](#)



Stackups: The Design within the Design

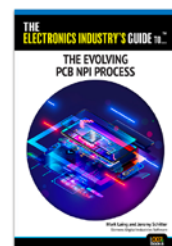
by Bill Hargin, Z-zero

Finally, a book about stackups! From material selection and understanding laminate data-sheets, to impedance planning, glass weave skew and rigid-flex materials, topic expert Bill Hargin has written a unique book on PCB stackups. [Get yours now!](#)

THE ELECTRONICS INDUSTRY'S GUIDE TO... The Evolving PCB NPI Process

by Mark Laing and Jeremy Schitter, Siemens Digital Industries Software

The authors of this book take a look at how market changes in the past 15 years, coupled with the current slowdown of production and delivery of materials and components, has affected the process for new product introduction (NPI) in the global marketplace. As a result, companies may need to adapt and take a new direction to navigate and thrive in an uncertain and rapidly evolving future. Learn how to streamline the NPI process and better manage the supply chain. [Get it Now!](#)



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) 365-1727; barb@iconnect007.com

MARKETING SERVICES: TOBEY MARSICOVETERE

(916) 266-9160; tobey@iconnect007.com

ART DIRECTOR/PRODUCTION MGR. SHELLY STEIN

shelly@iconnect007.com

MAGAZINE LAYOUT: RON MEOGROSSI

AD DESIGN: SHELLY STEIN, MIKE RADOGNA,

TOBEY MARSICOVETERE

TECHNOLOGY SPECIALIST: BRYSON MATTIES

COVER DESIGN: SHELLY STEIN

COVER IMAGE: ADOBE STOCK © NATALIA

DESIGN007

M A G A Z I N E

DESIGN007 MAGAZINE®

is published by IPC Publishing Group, Inc.

3000 Lakeside Dr., Suite 105N, Bannockburn, IL 60015

© 2023 IPC Publishing Group, 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.

March 2023, Volume 12, Number 3

DESIGN007 MAGAZINE is published monthly
by IPC Publishing Group, Inc., dba I-Connect007

ADVERTISER INDEX

Accurate Circuit Engineering.....	17
All Flex Solutions.....	11
Altium.....	77
American Standard Circuits.....	69
APCT.....	23
Burkle North America.....	51
Candor Industries.....	73
Downstream Technologies.....	39, 61
EMA Design Automation.....	67
Flexible Circuit Technologies.....	45
I-Connect007.....	85
I-007eBooks.....	2, 3, 83
In-Circuit Design Pty Ltd.....	47
IPC.....	41
IPC Community Magazine.....	81
Miltronix.....	5
NCAB Group.....	33, 37
Prototron Circuits.....	27
Show & Tell Magazine.....	25
Taiyo America.....	7
U.S. Circuit.....	49
Ventec International Group.....	29

Problems solved!



SUBSCRIBE NOW