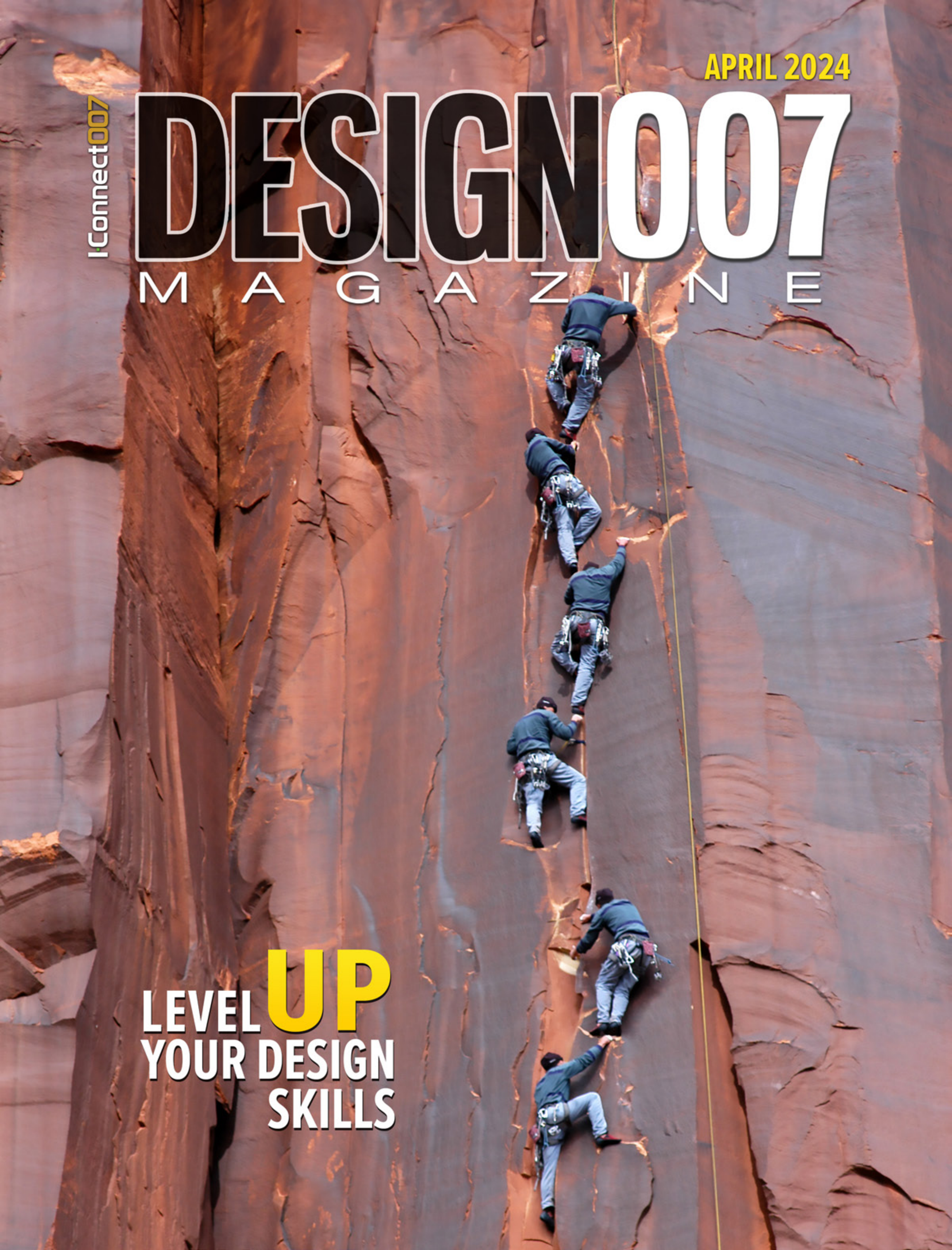
I-Connect007

APRIL 2024

# DESIGN007 MAGAZINE



## LEVEL UP YOUR DESIGN SKILLS

DESIGN007
MAGAZINE

# Level Up Your Design Skills

In this issue, our contributors discuss the PCB design classes available at IPC APEX EXPO. As they explain, these courses cover everything from the basics of design to avoiding over-constraining high-speed boards, designing for flex, test, design for reality, chiplet design, and DFM. If you don't come to the conference, you're missing out.

## FEATURE INTERVIEWS









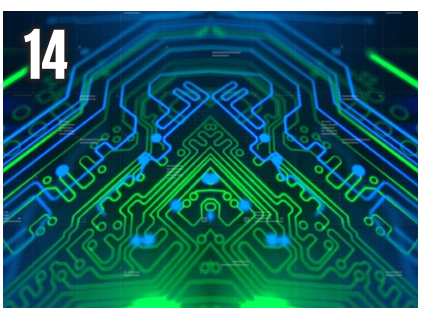
## FEATURE ARTICLES






## FEATURE COLUMN



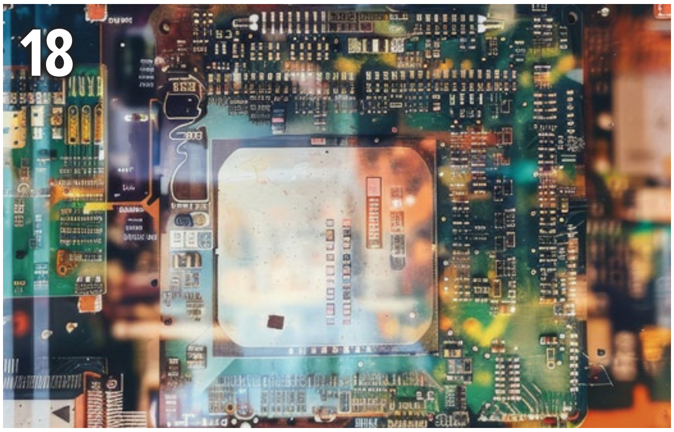



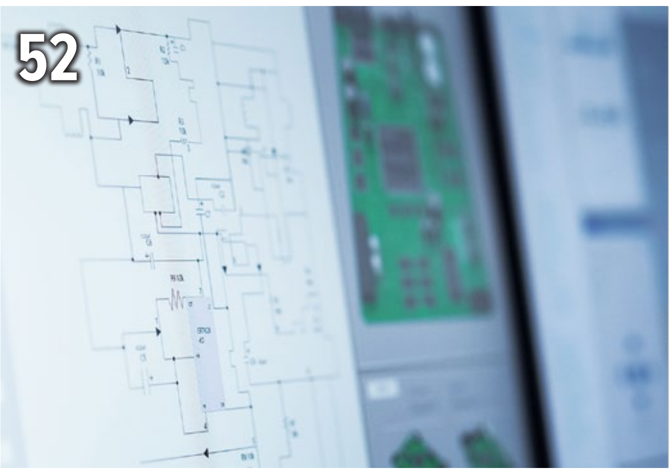




American Standard Circuits
Sunstone Circuits
ascsunstone.com
630-639-5444
Design Software
24 HOUR
Quickturn PCBs
Quickturn Assembly
High Tech
High Volume
High Reliability
Your Ultimate PCB Solution
Experience the Full Spectrum of Capabilities, Certifications, & Services
—from Design to Delivery—we have everything you need!
CERTIFICATIONS
ISO 9001:2015
Intertek
IATF 16949:2016
ITAR
MIL Certified
IEEE
IPC
UL
PCB CAD DESIGN
PCB123
ExpressPCB

# DESIGN007 MAGAZINE



# Design Takes Center Stage at IPC APEX EXPO

## The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

IPC APEX EXPO has been known for years as the largest PCB fabrication and assembly show in the U.S. In the past decade, the conference portion has been increasing its design content and this year's event in Anaheim, California, offers a cornucopia of PCB design curriculum.

The IPC Design Competition, for example, moves into its third year, with entrants from all over the world competing in Anaheim in the final heat. This show is shaping up as the place to be for PCB designers and design engineers.

If you're a designer and you've never attended IPC APEX EXPO, this is a great year to start. There's more design curriculum at the conference this year than ever before.; I counted 14 design-related Professional Development Courses and a half-dozen technical presentations related to design.

The classes don't just focus on design techniques. They cover everything from the basics of design to avoiding over-constraining high-speed boards, designing for flex, test, design for reality, chiplet design, and DFM. If you

don't come to the conference, you're missing out.

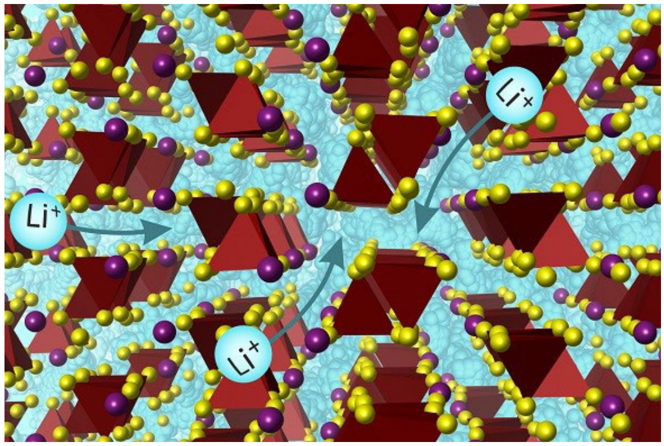
In this issue, we start with a conversation with IPC's Patrick Crawford and Kris Moyer, who discuss the IPC Design Competition, which takes place midweek during the show. Then, Cory Blaylock explains how IPC has lobbied the Labor Department to designate PCB design engineer as an official career, which will allow analysts to more accurately track data about this segment's demographics, salaries, etc. Kelly Allen outlines IPC's philosophy for PCB design training and education. Carlos Plaza explains what industry topics are hot in education, and why the hottest topic of all may be onboarding.

Filbert Arzola walks us through his two Professional Development Courses—one focusing on constraining your board correctly and the other on mixed-signal wire-bond techniques. As Fil says, today's designers need to think like electrical engineers when setting up constraints. Graham Blacksmith discusses what he learned in Kris Moyer's *PCB Design I and II* classes, and how he was able to utilize his takeaways immediately on the job. IPC's Robert Erickson updates us on IPC's courses for PCB designers and design engineers, at IPC APEX EXPO and year-round.

We also bring you columns from Vern Solberg, Martyn Gaudion, and Matt Stevenson, and part two of an article on heavy copper by Yash Sutariya.

We'll be bringing you full coverage of IPC APEX EXPO through our Real Time with... video interview program. I hope to see you in Anaheim. **DESIGN007**

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




# Li-ion Conductor Discovery Unlocks New Direction for Sustainable Batteries

In a paper published in the journal *Science,* researchers at the University of Liverpool have discovered a solid material that rapidly conducts lithium ions. Consisting of non-toxic earth-abundant elements, the new material has high enough Li-ion conductivity to replace the liquid electrolytes in current Li-ion battery technology, improving safety and energy capacity.

Using a transformative scientific approach to design the material, the interdisciplinary research team from the University synthesised the material in the laboratory, determined its structure (the arrangement of the atoms in space) and demonstrated it in a battery cell.

The new material is one of a very small number of solid materials that achieve Li-ion conductivity high enough to replace liquid electrolytes, and operates in a new way because of its structure.

Professor Matt Rosseinsky, from the University of Liverpool's Department of Chemistry, said, "This research demonstrates the design and discovery of a material that is both new and functional. The structure of this material changes previous understanding of what a high-performance solid-state electrolyte looks like. Our disruptive design approach offers a new route to discovery of these and other high-performance materials that rely on the fast motion of ions in solids."

(Source: University of Liverpool)

# IPC Design Competition: On Your Mark, Get Set, GO!

**Feature Interview by Andy Shaughnessy**
I-CONNECT007

I recently spoke with IPC's Patrick Crawford, manager of design standards and related industry programs, and Kris Moyer, certified IPC master instructor, about this year's IPC Design Competition. Now in its third year, the preliminary heat began in January, and the winners will compete in the final heat at IPC APEX EXPO in Anaheim.

I asked Patrick and Kris to discuss the Design Competition, advice for contestants, and why designers should enter next year's competition. As Kris points out, an open mind and knowledge of IPC standards will go a long way.

***Andy Shaughnessy: Patrick, give us an overview of the IPC Design Competition.***

*Patrick Crawford:* The competition was launched in 2022 from a desire to engage with designers in an extracurricular way, outside of standards, development, and work. This is our third year and it's been successful so far. We opened registration on Jan. 1, and 50 people signed up in the first week, which is really cool. We do two heats—a preliminary and a final heat. The preliminary heat is an at-home design. We provide contestants with a schematic, a bill of materials, and a scope of work. It's all on paper, and we say, "Go forth

Patrick Crawford

and read the design, schematic, specs, and requirements." We received several in Gerber and the associated manufacturing files, but in theory, they could have given us IPC-2581 as well.

IPC India has its own competition designed a little differently than ours, but we invite their two finalists. They're essentially assessing the same criteria in a different format. We invite the IPC India winner and runner-up, but we only invite three of the top-placing competitors from the prelims to the final heat. Last year, the final heat featured five competitors: two in the room at IPC APEX EXPO and three people calling in. This year, based on travel budgets and some competitors requiring a visa, it won't surprise me to have full virtual attendance. Last year, Sathish Vijayakumar from India won the IPC India competition and eventually the whole shebang. He couldn't be in India's competition this year, so he signed up for our competition. We'll see if he can take the top prize again.

This year, the finals competition will take place on Tuesday, April 9 during IPC APEX EXPO. It's free to attend; you can just walk in. We'll have screens set up for contestants from India, who will be dialing in. We do the judging onsite, and the winner is announced the next day. It's a four-and-a-half-hour competition. We provide them with an Altium Designer file package, and the board geometry is already finished. The nets are all defined, so they don't have to worry about the schematic or electrical engineering stuff. They are just responsible for component placement, topology, routing, etc. They're not starting from scratch.

***Kris, you are one of the judges, and you created the preliminary design and the final heat design. How do these designs compare to last year's?***

*Kris Moyer:* I won't give away anything about the final competition because the competitors don't find out until the day of the finals. We give you a schematic and say, "Here you go." For the prelim, we came up with a simple design: It's a driver circuit, like you have in your home theater speaker. You have two subwoofers, two mid-range speakers, and two tweeter speakers, and this driver circuit takes your audio signal and splits it across the three frequency ranges. Your high frequency goes up the tweeters, the mid-range goes to the mid-range speakers, the base goes down to the woofers, and so on, along with some amplification circuitry to actually drive the speakers. It's something a do-it-yourselfer might want to figure out—how a home theater speaker system works. This was a fun design.

**We give you a schematic and say, "Here you go."**

***Last year, none of the contestants finished the entire design. It must be hard to design a competition that's challenging without being too tough.***

*Crawford:* Yes. It's worth noting, too, that we originally thought most of the competitors would be "seasoned" industry folks with decades of experience. But experience taught us it's not the case. The actual median number of years of hands-on design experience for our competitors is three-and-a-half to four years, with a few outliers who have been in the industry for 20. So, in the prelim design, it's not like they're dealing with a 128-pin FPGA with a breakout route to a memory chip. It's only 111 pins. I'm kidding!

***Kris, what advice did you give for the contestants?***

*Moyer:* There are a couple of things. First, don't come with any preconceived notions. Second, remember that we're really only looking at IPC standards. Keep an open mind and look for those opportunities to be as efficient as possible with the design. For example, last year when one competitor was doing surface mount, he tried to put all the surface mount parts on one surface because he didn't like to have any parts on the second side. It made the parts spread out and made the routing nearly impossible. Whereas he could have looked at the design and said, "Hey, I've got the whole bottom surface of the board, and I can put those little resistors on the bottom. Now my parts all fit, and my routing is easier." Again, an open mind comes in handy. Follow, and be familiar and comfortable with the IPC standards.

*Crawford:* That's right. What's really interesting is this: We provide a library of IPC standards to the competitors for the prelims. Because we have IPC's DRM protection on them, I can actually go through, and see which contestant has opened them and how many times. For the past two years, there's been a 1:1 correlation between how many times the competitor opened their IPC standards vs. how well they do in the scoring of the prelims. That's been true for the past two years, and I anticipate it will be true for this year.

Kris Moyer

***So, what's next? Are you all working on the IPC Design Competition for 2025?***

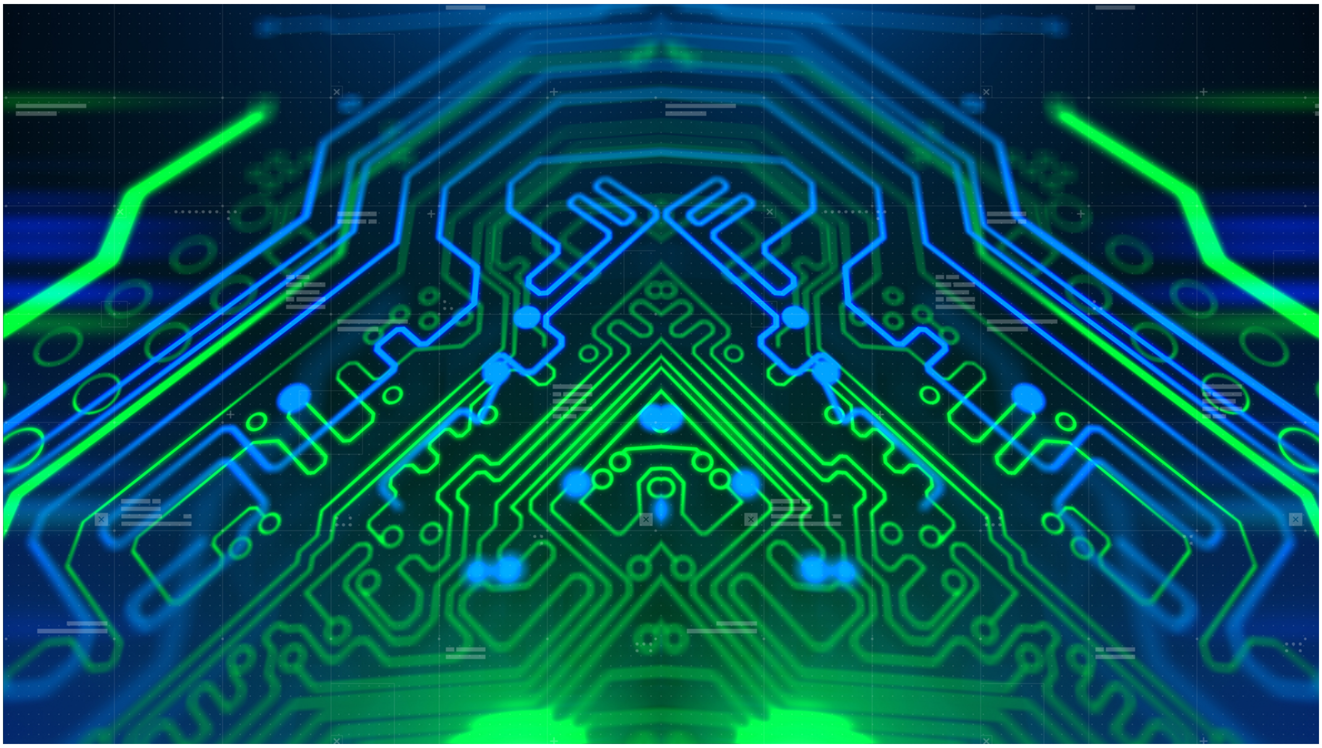
*Crawford:* As a matter of fact, we are. This is just a teaser, but next year's IPC Design Competition will be radically different. We've learned a lot over the past three years, and going forward, there will be much more emphasis on working in teams. There will be more emphasis on not just the board design but also product design. It will be completely different.

***That sounds like fun.***

*Crawford:* Yes, and challenging. We're still hammering out the details, but stay tuned.

***Thank you both. See you at the show.***

*Crawford:* Thank you, Andy. DESIGN007



# PCB Design Engineer Now 'Official Occupation'

**Feature Article by Cory Blaylock**
IPC

In a monumental stride for the electronics manufacturing industry, IPC has successfully championed the recognition of the PCB Design Engineer as an official occupation by the U.S. Department of Labor (DOL). This pivotal achievement not only underscores the critical role of PCB design engineers within the technology landscape, but also marks the beginning of a transformative journey toward nurturing a robust, skilled workforce ready to propel our industry into the future.

At the heart of this journey is IPC's unwavering commitment to developing National Program Standards that cater to the intricate needs of electronics manufacturing, emphasizing the importance of a highly trained workforce. The approval of the PCB design engineer occupation by the U.S. DOL is a testament to the collective vision and relentless efforts of countless individuals within the IPC community as well as the industry, which believes in the power of education, training, and standardization.

The PCB design engineer role is foundational to the electronics manufacturing value stream, serving as the linchpin that transforms innovative ideas into tangible, functioning technologies. These engineers are tasked with the intricate design of PCBs, which serve as the backbone for nearly all electronic devices. Their work supports not just the creation of new products but also the enhancement of existing technologies, ensuring they meet


a community of learners, educators, and industry leaders who share a common goal of advancing electronics manufacturing. It is about creating a sustainable ecosystem where knowledge is transferred, skills are honed, and innovation flourishes. This collaborative approach is essential for addressing the skills gap in the industry and for preparing a new generation of PCB design engineers ready to tackle the challenges of tomorrow.

the ever-evolving demands of consumers and industries alike.

Recognizing the PCB design engineer as a distinct occupation highlights the specialized skill set these professionals bring to the table. It acknowledges the complexity of their work, which requires a deep understanding of electrical engineering principles, proficiency in design software, and an acute awareness of manufacturing processes and constraints. More importantly, it shines a light on the critical need for tailored training and education programs that can prepare individuals for success in this challenging and rewarding field.

IPC's approved National Program Standards for the PCB design engineer occupation are designed with the future in mind. These standards define the competencies required for excellence in the field and lay the groundwork for comprehensive training programs that bridge the gap between theoretical knowledge and practical application. By aligning educational efforts with industry needs, IPC aims to ensure that the workforce is equipped with the skills necessary to drive innovation and maintain the competitiveness of the electronics manufacturing sector on a global scale.

The focus of IPC's standards goes beyond individual achievement; it is about fostering

As we celebrate this significant milestone, it is important to recognize that the journey does not end here. The approval of the electronics assembler, PCB fabricator, and PCB design engineer occupations is just the beginning. It opens new opportunities for collaboration between IPC, educational institutions, and industry partners to develop and implement training programs that meet the highest standards of excellence. It also paves the way for further recognition of other critical occupations within the electronics manufacturing value stream, reinforcing the importance of a skilled and knowledgeable workforce.

Let us continue to work hand in hand, forging pathways for aspiring PCB design engineers and ensuring that our industry remains at the forefront of technological advancement. Together, we are not just designing circuit boards; we are designing a brighter future for electronics manufacturing, powered by education, innovation, and collaboration. DESIGN007

**Cory Blaylock** is director of workforce partnerships for IPC.

# What Designers Need to Know About Manufacturing, Part 2

## Designers Notebook

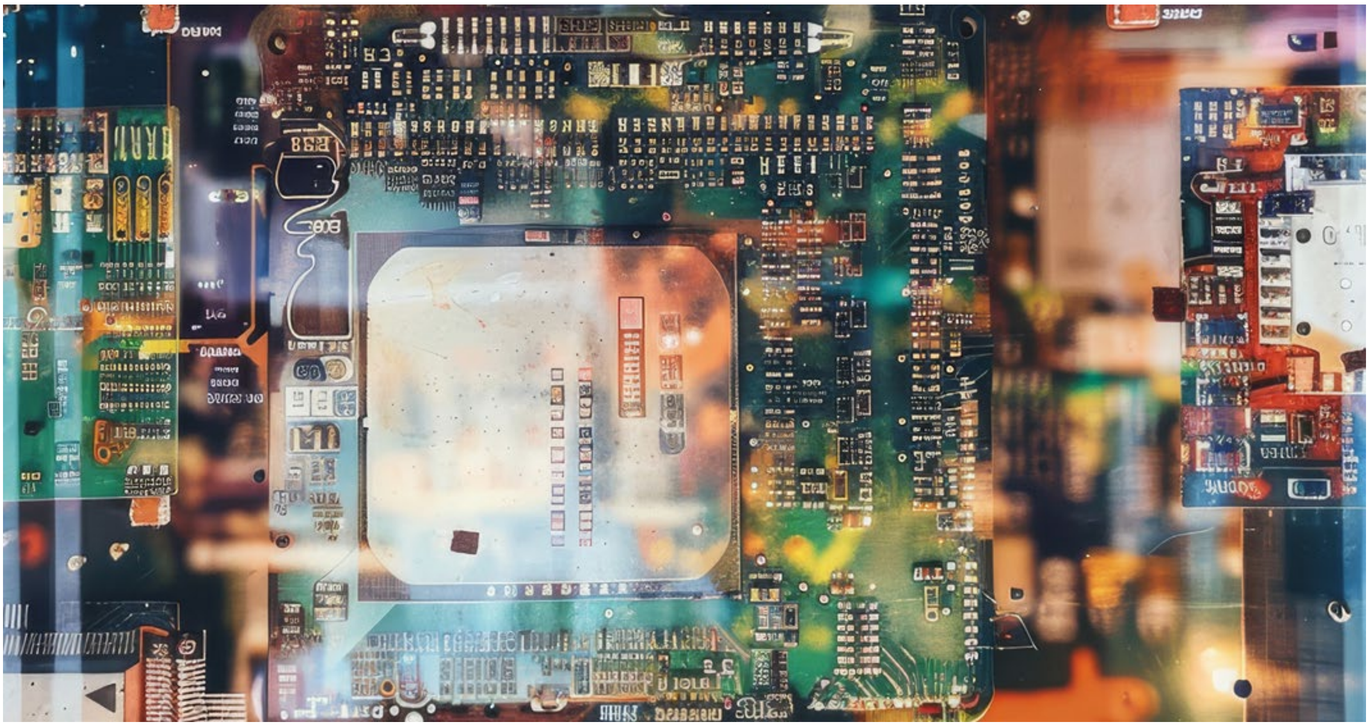
by Vern Solberg, CONSULTANT

The printed circuit board (PCB) is the primary base element for providing the interconnect platform for mounting and electrically joining electronic components. When assessing PCB design complexity, first consider the component area and board area ratio. If the surface area for the component interface is restricted, it may justify adopting multilayer or multilayer sequential buildup (SBU) PCB fabrication to enable a more efficient sub-surface circuit interconnect.

Developing the circuit board to accommodate surface mount passive and the various semiconductor package families, the designer and assembly process engineer must respect the disciplines, capabilities, and limitations within each other's realm. Additionally, each must be cognizant of the industry-developed standards and tolerances that control IC packaging and the requirements established by the industry for reliably attaching these devices to the printed circuit board.

The three primary PCB commodities being fabricated:

1. Single-sided circuits have a copper circuit pattern on only one side and can accommodate both SMT component mounting and through-hole components that are mounted onto the opposite surface, with their terminals protruding through mechanically drilled holes for termination. The single-sided circuit boards are gener-




Your survival guide for
the harshest environments.
Learn how encapsulation resins are used to ruggedize electronics.
THE PRINTED CIRCUIT ASSEMBLER'S GUIDE TO...™
Encapsulating Sustainability for Electronics
Beth Turner
MacDermid Alpha Electronics Solutions
I-007e Books
Look inside
I-007e Books

ally for less complex and/or cost-sensitive applications.

2. Double-sided circuits will rely on copper-plated holes for interconnecting conductor patterns between opposing circuit layers. For surface mount applications, component land patterns are incorporated into the design to enable solder attachment. Land patterns for surface-mounted components may be furnished on one or both sides.

3. Multilayer circuits will rely on copper-plated holes for interconnecting the outer layer conductor patterns and the inner layer conductor patterns. Typical of the less complex two-sided PCB, surface mount component land patterns may be incorporated into the design to enable solder attachment on one or both sides of the circuit board.

## Providing a Copper Balanced PCB Structure

The most desirable PCB construction is one where copper layers are built up in pairs and arranged symmetrically about the core of the board: two, four, six, eight, and so on. The various pre-etched circuit layers will be assembled and aligned with partially cured prepreg material between layers, as compared in Figure 1.

The sequence in which the circuit board layers are assembled (signal, power, ground, etc.) is a key factor that will affect signal transmission performance. In addition to performance concerns, controlling fabrication cost should be a priority as well. Layer count and the method selected for interconnecting between circuit layers, for example, will have significant influence in controlling process complexity. Implementing blind microvia technology for layer-to-layer interconnect will enable significantly greater circuit routing density, but the build-up process has substantial impact on the manufacturing complexity since it will affect the number of lamination cycles. Any layer on which a microvia begins or ends requires a sub-construction, and each sub-construction will require an extra lamination cycle. Some of the more complex sequential buildup (SBU) multilayer designs will require several lamination cycles, and with each lamination cycle, the core or base materials will also be subjected to repeated exposure to elevated temperatures and high pressure. The concern is that excessive lamination cycles can contribute to base material decomposition.

## Stacked and Staggered Microvias: Pros and Cons

Two variations of microvia stacking are commonly employed: offset via stacking and vertical via-on-via stacking (Figure 2). Multilayer SBU circuit boards can use the vertically stacked microvia format when routing channels are restricted, or when the subsurface circuit layers are less restricted, the staggered microvia that offsets the blind via hole's position from one layer to the nex, is recommended.

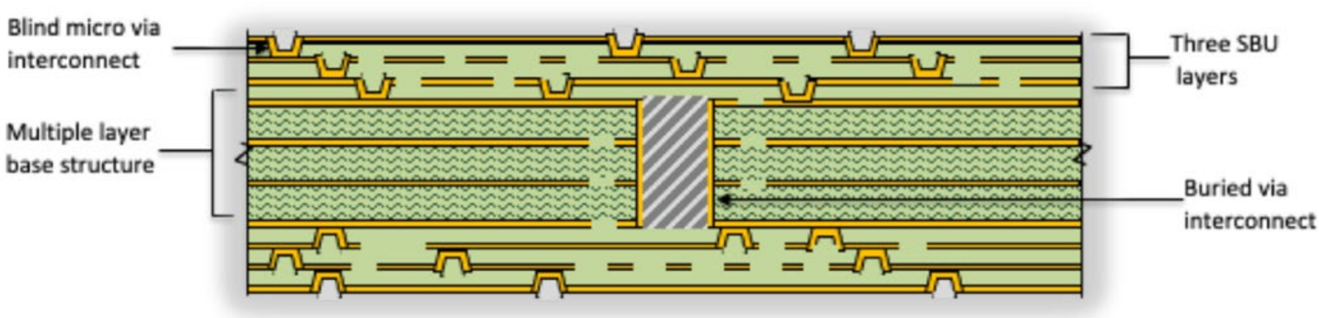



Figure 1: Layer balanced 3-4-3 circuit board, cross-section view.

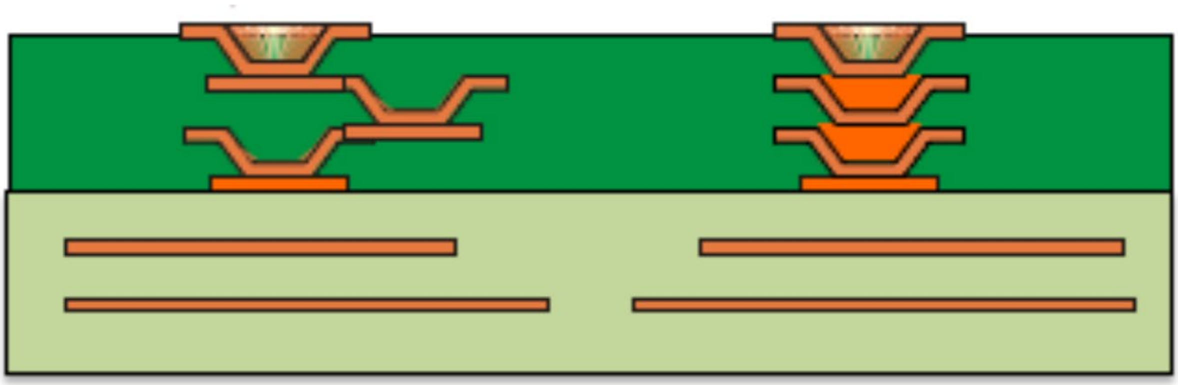


Figure 2: Comparing a staggered to a vertically stacked microvia interface.

Stacking microvia holes directly in line with one another will simplify circuit routing during the CAD development process, but additional plating process steps to fill and planarize the resulting via cavities will impact fabrication throughput. When implementing vertically aligned microvias, each stage of the lamination sequence will require that the depression in the center of the microvia be plated flush with the surface of the copper conductor's surface before lamination of the next circuit layer. Circuit fabricators note that providing a stable copper-to-copper interface between the filled microvia levels may be at risk when subjected to long-term physical or thermal stresses. When clustered together (fine-pitch array type semiconductor package interconnect, for example), thermal cycle testing of the in-line, vertically stacked microvia holes exhibited positive results.

Note: Studies conducted by Summit Interconnect found after extensive thermal test evaluation that the clustered, three-stack direct via-to-via interface format, typically required for fine-pitch array configured components, proved to be reliable.

Although the staggered three-level microvia will require slightly more surface area than the vertically stacked alternative, circuit board fabricators prefer the offset microvia option for circuit interconnect when vias are distributed randomly on the circuit board because the multilayer lamination sequence requires fewer process steps. With the microvias arranged in the staggered format, the copper fill operation is eliminated, saving both time and process complexity.

For both vertically stacked and staggered via-in-land component attachment sites on the circuit board's outer layer(s), an additional copper plating step will be necessary to fill microvias flush with the land pattern's surface to negate potential void formation in the solder interface during assembly processing.

## High Density Conductor Routing

Conductor routing protocols must be established in advance. Adapting blind and buried microvia holes and furnishing pre-defined routing channels will help the circuit board designer to facilitate efficient routing of these often very fine-pitch and array terminal configured semiconductor packages. To aid the designer in establishing copper conductor width and spacing for circuit routing, IPC-2226 has defined three HDI circuit board complexity levels (Table 1) for both external and internal locations.

The space separating via lands, microvia lands, and/or component attachment lands is referred to as channel width. The channel widths for routing array-configured semiconductors can easily be mathematically calculated using the terminal pitch (center-to-center distance), overall land pattern size, and the width and spacing established for conductors. This will provide the maximum number

Table 1: Establishing HDI complexity levels (Source: IPC-2226)

| Aspect Ratio | Level A | Level B | Level C |
|---|---|---|---|
| Internal conductor width | 127 µm (~.005”) | 75 µm (~.003”) | 50 µm (~.002”) |
| Internal conductor spacing | 127 µm (~.005”) | 100 µm (~.004”) | 50 µm (~.002”) |
| External conductor width | 127 µm (~.005”) | 75 µm (~.003”) | 25 µm (~.001”) |
| External conductor spacing | 127 µm (~.005”) | 100 µm (~.004”) | 25 µm (~.001”) |

of conductors that can be routed between each channel (conductors per channel). When the surface channels are not wide enough to facilitate the conductor routing, the designer will need to consider reducing the conductor width and spacing on the circuit board surface or resort to sub-surface circuit routing to achieve interconnect.

Note: Whenever possible, to reduce bow and twist and to increase dimensional stability, conductor routing density should be uniformly distributed throughout the printed circuit board's layer structure, and conductor density should be balanced within individual layers.

## Supplier Assessment and Process Refinement

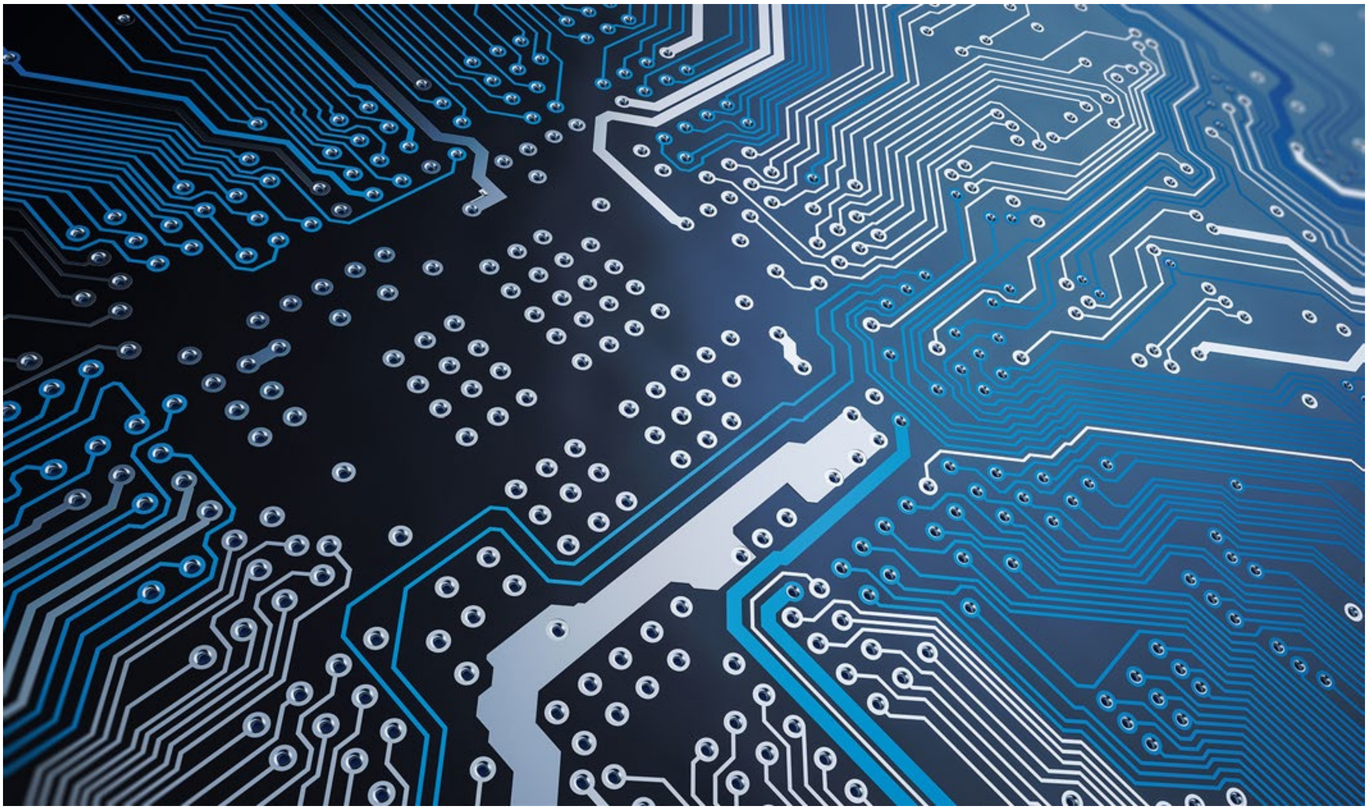
Know your PCB supplier and their capabilities; you don't want to end up designing a board that can't be manufactured. In addition to the design guidelines furnished in IPC-2221 and IPC-2222 for rigid circuit boards, suppliers will often publish their own design rules. To ensure a successful outcome for the circuit board design, it will be important that the designer recognize the manufacturing process complexities and become familiar with the circuit fabrication supplier's technical capability. But before committing to adopting HDI or UHDI technology, the designer should confirm that the PCB supplier(s) selected can furnish the preferred complexity level at the anticipated production quantity.

Many OEM companies have already established a business relationship with their primary circuit board suppliers and have qualified their level of expertise and capability. Review your PCB suppliers' material offerings:

- Discuss any unique base material or copper foil needs
- Clarify layer stacking alternatives with the supplier
- Define power and ground distribution objectives
- Specify controlled impedance requirements

When estimating the circuit board's manufacturing cost, the designer must consider materials, circuit density, the number of circuit layers, and fabrication process complexity. While mechanically drilled and plated via holes are a mature technology for multilayer circuit board fabrication, the buried via and laser-ablated blind microvia hole forming requires more sophisticated systems. DESIGN007

**Vern Solberg** is an independent technical consultant, specializing in SMT and microelectronics design and manufacturing technology. To read past columns, click here.



# What's New in Design Education at IPC APEX EXPO?

**Feature Q&A With Kelly Allen**
IPC TRAINING MANAGER

Kelly (Kel) Allen shares her thoughts on the educational offerings at IPC APEX EXPO and beyond. In this interview, she discusses some of the newest classes taking place during the conference in Anaheim, covering everything from design, fab, and assembly through mil/aero, test, and supply chain issues.

***Kel, why should PCB designers and design engineers attend the conference at IPC APEX EXPO?***

*Kelly Allen:* Adult learners appreciate practical knowledge. In our online classes, PCB designers and design engineers connect theoretical concepts to real-world situations, making our online classroom discussions dynamic and relevant. Many are driven by personal and professional goals so attending an APEX EXPO Professional Development Course allows them to actively participate, ask the speaker thought-provoking questions, and engage in meaningful dialogue. Our industry keeps our designers so busy that many do not have time to take one of our long-form courses. Yet they can invest a few days to take PD courses, attend committee meetings, network with colleagues, and see the latest in manufacturing and products on the show floor.

Kelly Allen

***Tell us about the new online design courses this year. What design classes are hot right now?***

The "I" in IPC also stands for innovation, and this year, we are offering several new online instructor-led courses covering a variety of topics for all experience levels:

***Technical Overview of the Semiconductor Chip Industry:*** This course provides a technical overview of the industry from design and manufacturing to assembly and testing. Instructor Cheah Soo Lan is an engineering manager with more than 30 years of experience in the electronics and semiconductor industry. This course is designed for all individuals, regardless of their experience or knowledge of the subject matter, and is presented in an easy-to-understand format.

***Counterfeit Electronic Parts Mitigation in Mission Critical & Life Saving Supply Chains:*** Led by Anthony J. Bryant, an expert with over 35 years of experience in aerospace, defense, and semiconductors, this course is essential for professionals who identify, report, and dispose of counterfeit material to prevent its use in the supply chain.

***Traditional DFM: Is it Dead or Alive?:*** What do you do when industry standards and DFM reviews do not address the material limitations, assembly tolerances, and other manufacturability factors posed by today's increasingly complex designs? Engineers, technicians, and others involved in creating manufacturable designs will want to join industry expert and presenter Dale Lee as he shares valuable insights and practical solutions from recent real-world challenges.

***Reliability of Electronics—Tin Whiskers—Everything you Need to Know:*** International Hall of Famer and presenter Dr. Jennie Hwang delivers the third course in IPC's Reliability of Electronics curriculum. Dr. Hwang will share insights from a career spent solving challenging production issues for high-reliability products for commercial and military applications. Participants will obtain a comprehensive understanding of tin whiskers, including what they are, how they occur, and how to prevent them.

The most popular classes in IPC's PCB design curriculum are those taught by IPC instructor and presenter Kris Moyer. These include *Introduction to PCB Design I & II* and *PCB Design for Military and Aerospace Applications.* We also launched a new course last year, *PCB Design for Advanced Design Concepts,* covering HDI (high-density interconnect), advanced packaging, embedded components, and wearable electronics.

***I understand that IPC is seeking new PCB design instructors. What classes are you hiring for?***

We conduct regular surveys of students and members to identify topics of interest, and many of our new courses are driven by the support of the Professional Development Courses at IPC APEX EXPO. This year, we are proud to present courses from renowned experts such as Tim Burke, Ph.D., on machine data analytics, assembly veterans Jim Hall and Phil Zarrow on troubleshooting and defect analysis, and Fil Arzola on engineering courses geared toward building better boards. These courses cover a diverse range of topics and highlight the wide variety of fields in our industry.

If you are an expert and comfortable with instructing in a virtual platform, have engaging classroom techniques, keep up to date with industry trends and technological advancements, can connect theory to on-the-job performance, and believe you have a compelling course topic, please click here to get in touch with me. We would be happy to hear from you and consider your proposal.

***What is the most challenging part of your job as a training manager?***

One of my primary responsibilities is to stay up to date with the ever-evolving landscape of our industry. I must ensure that the courses we offer provide value to our members and are relevant, meaningful, and not outdated or ineffective. It is essential to engage students during training sessions to promote active learning. To achieve this, the IPC Education team works collaboratively with our instructors to design engaging content that caters to different generations, learning preferences, and skill levels. This approach enables me to help employees learn and grow, fostering a skilled and engaged workforce that drives productivity and job satisfaction.

***What is the best part of your job?***

One of the best parts is that I get to work with some of the most incredibly brilliant and committed individuals in the electronics industry. Second, helping adult students achieve their learning and career goals is a profoundly rewarding experience. As an educator, witnessing the transformation of these dedicated learners is both inspiring and fulfilling. Adult learners bring a wealth of life experiences to the classroom. Their diverse backgrounds enrich our online discussions, providing unique industry perspectives that enhance everyone's learning journey. It's incredibly rewarding to witness the "aha" moment when a student grasps a concept and demonstrates their ability to apply it. Whether it's acing an exam, completing a project, or hearing about a promotion, celebrating their achievements and career successes only adds to the satisfaction.

***Is there anything else you would like to add?***

Many adult students juggle multiple responsibilities from jobs, families, and other commitments. Despite these challenges, they persist. Adult learners embody the spirit of lifelong learning; it is a continuous journey. Helping adult students reach their learning goals is more than disseminating knowledge; it is about empowering individuals to transform their lives. IPC offers fundamental, intermediate, and advanced courses to expand students' design and manufacturing capabilities. To review our course calendar, click here.

***We appreciate your time.***

Thank you. **DESIGN007**


# IPC Focuses on Education and Onboarding

**Feature Interview by Andy Shaughnessy**
I-CONNECT007

I recently spoke with Carlos Plaza, senior director of education for IPC, about expanding IPC's educational efforts in the PCB design, fabrication, and assembly segments. As Carlos explains, PCB design is a hot topic, but onboarding may be the hottest one of all.

***Andy Shaughnessy: Carlos, as IPC's senior director of education, I know you've been busy preparing for IPC APEX EXPO. Why don't you give me a brief overview of what that entails?***

*Carlos Plaza:* I work with industry experts and IPC learning specialists to help identify and meet the training and certification needs of our members. That's my job in a nutshell. Certification is an essential part of the workforce training equation. Customers should be confident that their boards are being fabricated and assembled by personnel who have demonstrated their ability to adhere to IPC standards, particularly for high precision and reliability applications.

However, a few years ago, we discovered that there were many new employees having a hard time getting certified because they didn't know the terminology, materials, or processes and tools used to build and assemble PCBs and wire harnesses. Of course, when you think about it, it doesn't make much sense to teach someone about the criteria that apply to things like turrets, conductors, and annular rings if they don't know what they are and how they're used.

That's why we created workforce training programs: to help people acquire the knowledge and skills they need to do the job of an operator, a technician, or an engineer. Once they've learned to build, assemble, and inspect Class 2 and 3 boards, they're ready for certification. That's the sequence. You don't take the bar exam to learn about law; you take it to validate what you learned in law school.

What does IPC workforce training encompass? We said we would create training for industry job roles. How do you perform the job of an operator, a technician, an inspector, an engineer, a program manager, or a PCB designer? Those are the major disciplines, and there are sublevels as well. The engineer could be a manufacturing engineer or a production engineer. One operator might focus on hands-on soldering, and another could operate a reflow oven on an SMT line. A third operator could be doing rework and repair. We dedicated ourselves to the saying, "Let's find the jobs that are the most critical and start filling in those gaps." In the process, we discovered that onboarding was the most critical point.

**Let's find the jobs that are the most critical and start filling in those gaps.**

*Why is onboarding so critical?*

The electronics manufacturing industry has grown quickly, and it competes with many other industries for the large number of workers needed to keep up with demand. Today, most new hires have zero experience on their first day. They are fresh out of school, or maybe they were working at Starbucks on Friday and now starting at a PCB fab or assembly facility on Monday. Trainers need to onboard all these novice employees in the most efficient and effective way possible for their companies to remain competitive. That's why we first built onboarding programs like Electronics Assembly for Operators and Wire Harness Assembly for Operators. We understood the need, and the stakes, so our training specialists worked with industry experts to make sure that IPC workforce training programs met two important criteria. First, the content covers only what an operator needs to know to do their job to industry standards. Second, we use learning strategies that significantly increase the likelihood that students will apply those skills on the job, the right way every time. This combination of industry expertise and learning science is the key. It actually allows trainers to do more in less time, so it reduces training costs, time to proficiency, and even rework down the line.

***Longtime employees are retiring every day, and we're losing our SMEs. What can we do to preserve all that know-how?***

When I talk to trainers, engineers, and administrators about training, there is a sense of urgency because many of their veteran workers are retiring and walking out the door with decades of tribal knowledge. So, we decided to meet some of those veterans at the door and ask them to work with us to share their hard-won, real-life expertise, and experience with the next generation of workers. Our online instructor-led courses, like PCB Design I and II and PCB Design for Manufacturability, are led by long-time experts in the field who really know how to talk about their areas of expertise because they love what they do. In our course surveys, students express their appreciation for these instructors and the valuable insights they provide. These nuggets of wisdom are not found in textbooks but are immensely helpful in improving job performance.

***I understand IPC offers different types of online courses.***

Yes, there's a mix. We have asynchronous courses, which are online and self-paced. You log into our learning management system (called EDGE), which is available 24/7, and you can go through the course at your own pace. We also have online instructor-led courses where you meet the instructor online at a certain time, twice a week, usually for about two hours. Those courses can run anywhere from a few days to a few weeks, depending on the course.

***You're responsible for identifying the next classes and disciplines that need to be addressed. How do you stay on top of it all?***

I talk to a lot of people. IPC is a volunteer-driven organization, and we work with and for the electronics manufacturing industry, so I routinely call trainers, engineers, and administrators to talk about their training programs and learn more about how IPC can help with their pain points. What's working and what's not? Where do trainees tend to struggle? What is the most critical problem you need to solve?

Sometimes people will walk up to me after a meeting to discuss an issue, and I'll do an on-the-spot discovery call, or I'll ask, "Can we meet next week and talk more about this?" Every year at IPC APEX EXPO, I schedule separate meetings with people I want to touch base with to learn more about what and how they're doing. Often, I'll do that in conjunction with Mike Hoyt, our workforce training advisor. We are both well-versed in the fields of education, and we've learned a great deal about electronics manufacturing over the past few years. Together, we do our best to assist our colleagues in the training room and on the factory floor. While many companies have adopted IPC workforce training to improve their training processes, Mike and I are committed to using our expertise to help our colleagues achieve the best possible results, regardless of the methods employed.

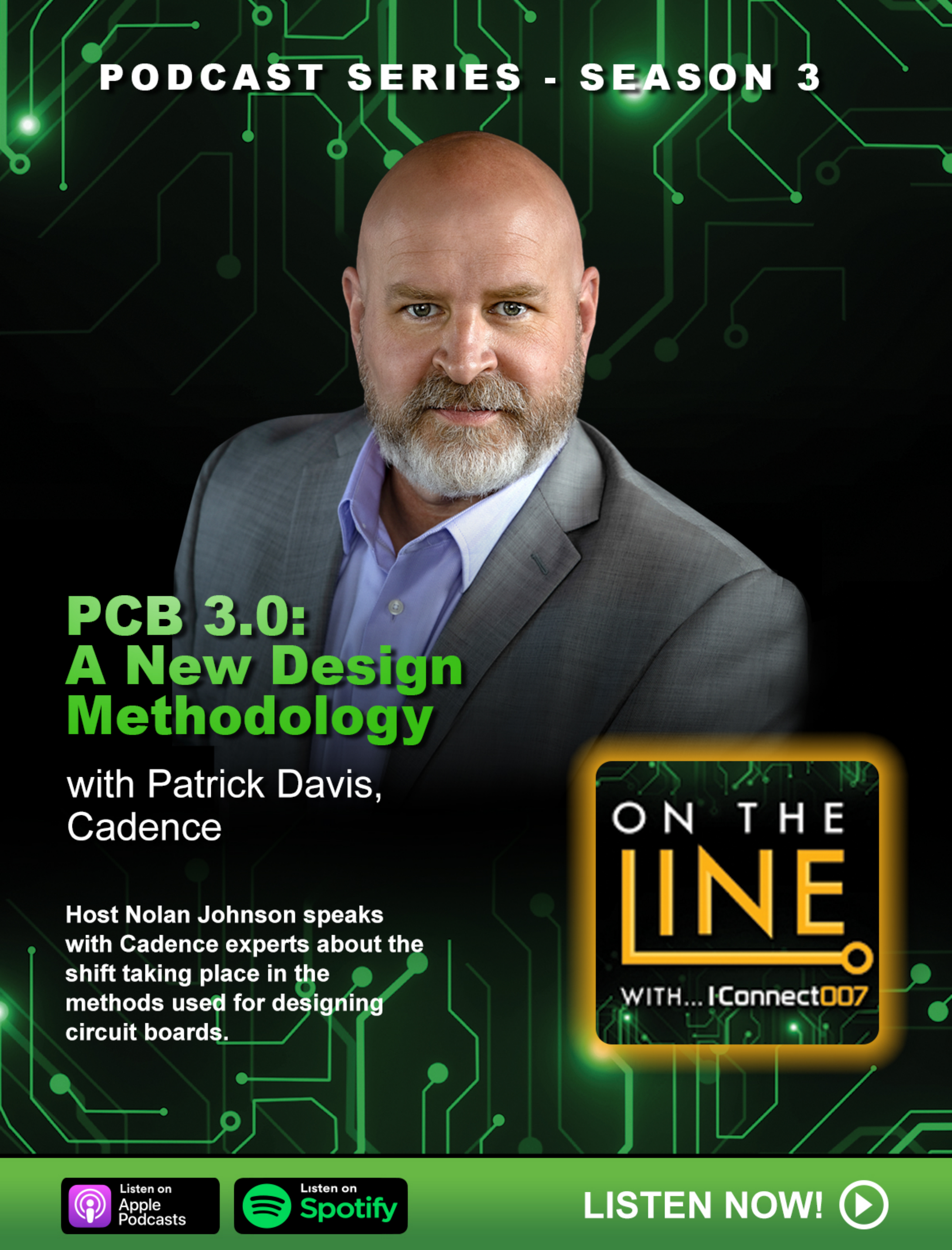
Carlos Plaza

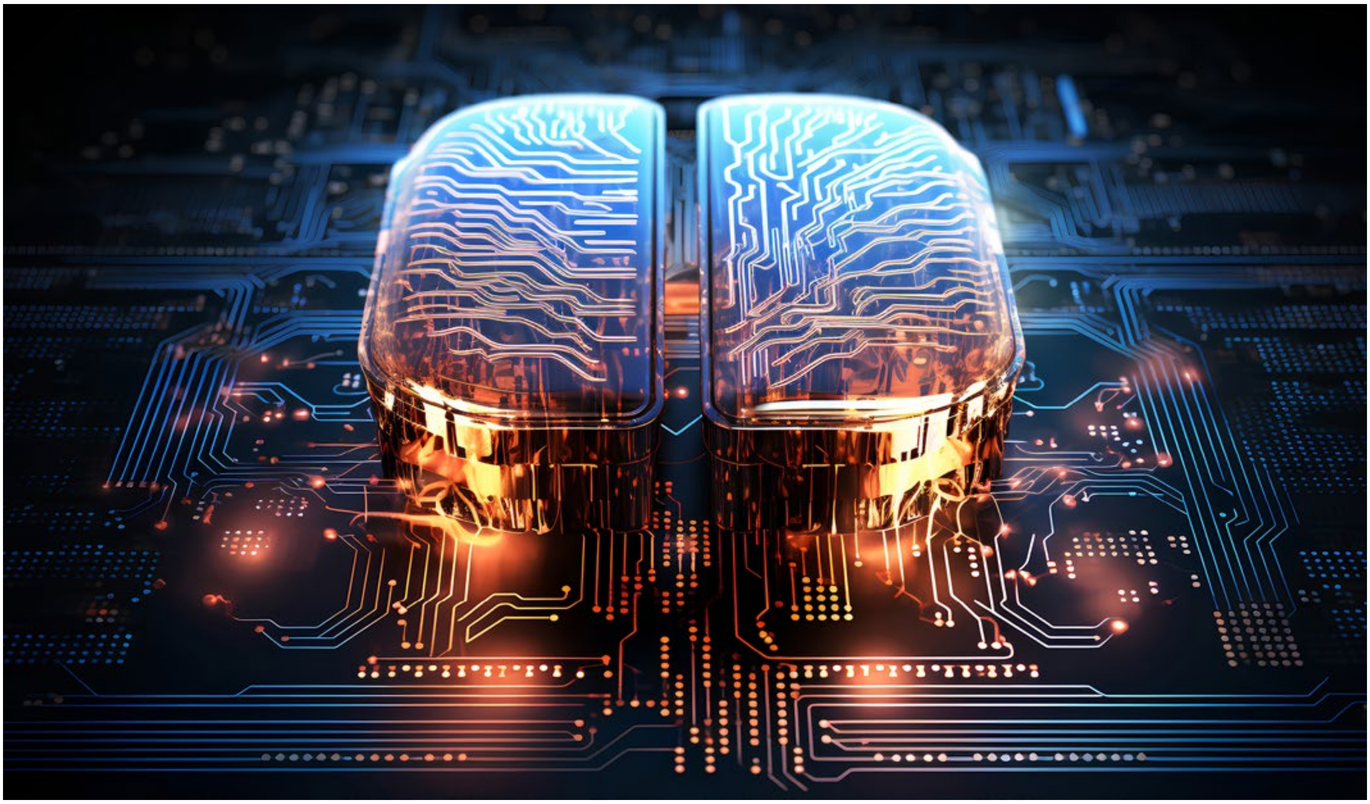
***Any final thoughts?***

We need to hear from you. Tell us what you're doing and how we can help you sharpen your competitive edge. Then help us execute. Volunteer your time to review course content or take a beta test on the latest course. You may also want to consider becoming an instructor. Students really enjoy IPC online instructor-led courses, but the instructors get a lot out of the experience as well. Kris Moyer, who teaches several of IPC's PCB design courses, says, "It's very rewarding to witness those a-ha moments, when you realize that you have just helped that student become a better, even happier, designer."

***Thanks for your time, Carlos.***

Thank you, Andy. DESIGN007


PODCAST SERIES - SEASON 3
PCB 3.0: A New Design Methodology
with Patrick Davis, Cadence
Host Nolan Johnson speaks with Cadence experts about the shift taking place in the methods used for designing circuit boards.
ON THE LINE
WITH... I-Connect007
Listen on Apple Podcasts
Listen on Spotify
LISTEN NOW!



# Time to Think Like an Engineer

**Feature Interview by Andy Shaughnessy**
I-CONNECT007

Filbert (Fil) Arzola, a senior principal engineer at Raytheon, is returning to IPC APEX EXPO to teach a pair of Professional Development Courses that go far beyond typical design courses. I checked out one of his classes last year, and it was packed.

Fil explains what attendees can expect, why designers need to think like engineers, and how he aims to make his mixed-signal wire-bond class "fun." How is that even possible, you ask? Read on.

***Andy Shaughnessy: Fil, you're scheduled to teach a couple of Professional Development Courses at IPC APEX EXPO on Sunday, April 7. Can you walk us through these classes?***

*Filbert Arzola:* Sure. Just like last year, I'm teaching two classes. One focuses on how to constrain your design. By that, I mean doing things to make your design work better in engineering.

The second class focuses on mixed-signal wire-bond designs. We'll make it a fun how-to class where everyone can participate if they want, or they can just listen. We'll take a sample wire bond design and make sure that everybody understands the proper steps from beginning to end.

***How will you make that wire-bond class fun? It's not exactly a Don Rickles sort of topic.***

You're right. Over the years, my colleagues have always said that wire-bond designs are a huge challenge. This is especially true for folks who have spent five to 10 years doing digital boards, power supplies, or other such things.

Filbert Arzola

They've never worked on any kind of complex RF boards or had one of their boards redone because they didn't have the correct part. It's really amazing when you're working with these young engineers, they see chip-on-board for the first time, and they say, "What on earth is that? I've never seen that." It's so much fun to show them what's going on here and how it works. They're amazed. I ask them, "Do you want to learn more? Yes? Well, let me show you how to do these designs."

For this course, I thought it would be cool to just have a wire-bond design that goes from beginning to end—a really easy one, nothing too difficult—but also throw in some control difficulty. I'm putting together a good sample design, and I'm sure by the end of the class, everybody will have a really good idea about how to place parts, set up your constraint management, and a whole bunch of different things related to wire-bond design. It will be a fun class.

***That's great because it's fun for the students too. What are some of the biggest challenges that the students face when constraining the board?***

A lot of folks just don't put in the rules when they start a design; those constraints help you route faster and better, and route so that you don't have to redo the design repeatedly. By learning how to set constraints, and by keeping the design rule editor on, you can route and not make any mistakes. Then, once you do the route, it's done. You don't have to clean it up later unless something really bad happens. By understanding better methods to set up your engineering constraints, it gives students a better idea of how they can be more productive and make their designs more engineered than ever before. That's what we're facing right now. We have to engineer our work a lot better than ever before.

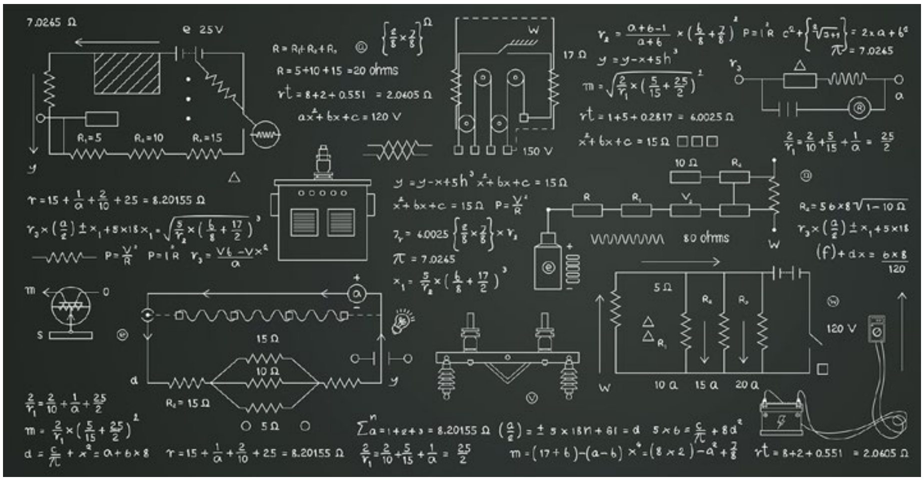
***We also hear from designers who are trying to avoid over-constraining. Some high-speed designers throw the baby out with the bathwater and go with the most expensive materials every time.***

It's a challenge to find that fine line and not waver. I've been in some webinars where certain software packages can really over-constrain things. That takes away the magic and the beauty of what we're doing. We're solving a puzzle and making it correct engineering-wise. But I'll lean toward over-constraint every now and then to engineer everything. Over-constraint can really hurt a design during assembly; we can really hurt ourselves by over-constraining. Then we're back to redesigning.

***One engineer said, "Don't show me how perfect you can make your design; show me the dirtiest signal I can have that still passes and makes the product work."***

Right. Like you mentioned, we can get over-constrained if we're trying to be too perfect. Sometimes EEs will try to constrain things too much. Every PCB design engineer should ask, "What speed are we at? Do we really need to match length to 1 mil? Will 10 mil work? Do we really need it?" A few friends say I ask too many questions, but I can't help it. I'm an engineer.

I'm doing these two courses at IPC APEX EXPO, but I'm teaching other classes for IPC. I did two last year and I'll do two again this year, improving on them and adding more material to each one. They're all through IPC, and I think the course dates have been released.



IPC has many courses on how to design a board and work with schematics. In working with IPC, I told them, "Instead of just showing everyone how to do design, why don't we show them how to engineer their design?" They asked me about the difference, and I said, "A lot of folks don't think that PCB designers are engineers. They just look at us as drafters."

These aren't your mom and dad's boards nowadays. Everybody has to worry about the material to stack up. You have questions like: How much metal is on each layer? Are we using a foil? Do we have to worry about the size of the vias or whether we're stacking them? There are so many more constraints now that designers need to focus on engineering as well as design. Anybody can design a board, but to take that design and engineer it, you need a nice skill set.

***I recently interviewed some 18-year-old freshmen, and they weren't sure what they wanted to do, but they knew they needed to know how to design a circuit board for whatever career they picked. They already knew how to code at 18. So, there's hope for the future.***

Those kids are the future for board designers. I think everything will change in the next 10 years.

***We are seeing change. For years, it was just not hip for young people to be PCB designers, but now it's coming back.***

So, maybe it's finally hip to be square?

***As Huey Lewis once said. What are the big takeaways from these classes? What do you hope the students leave with?***

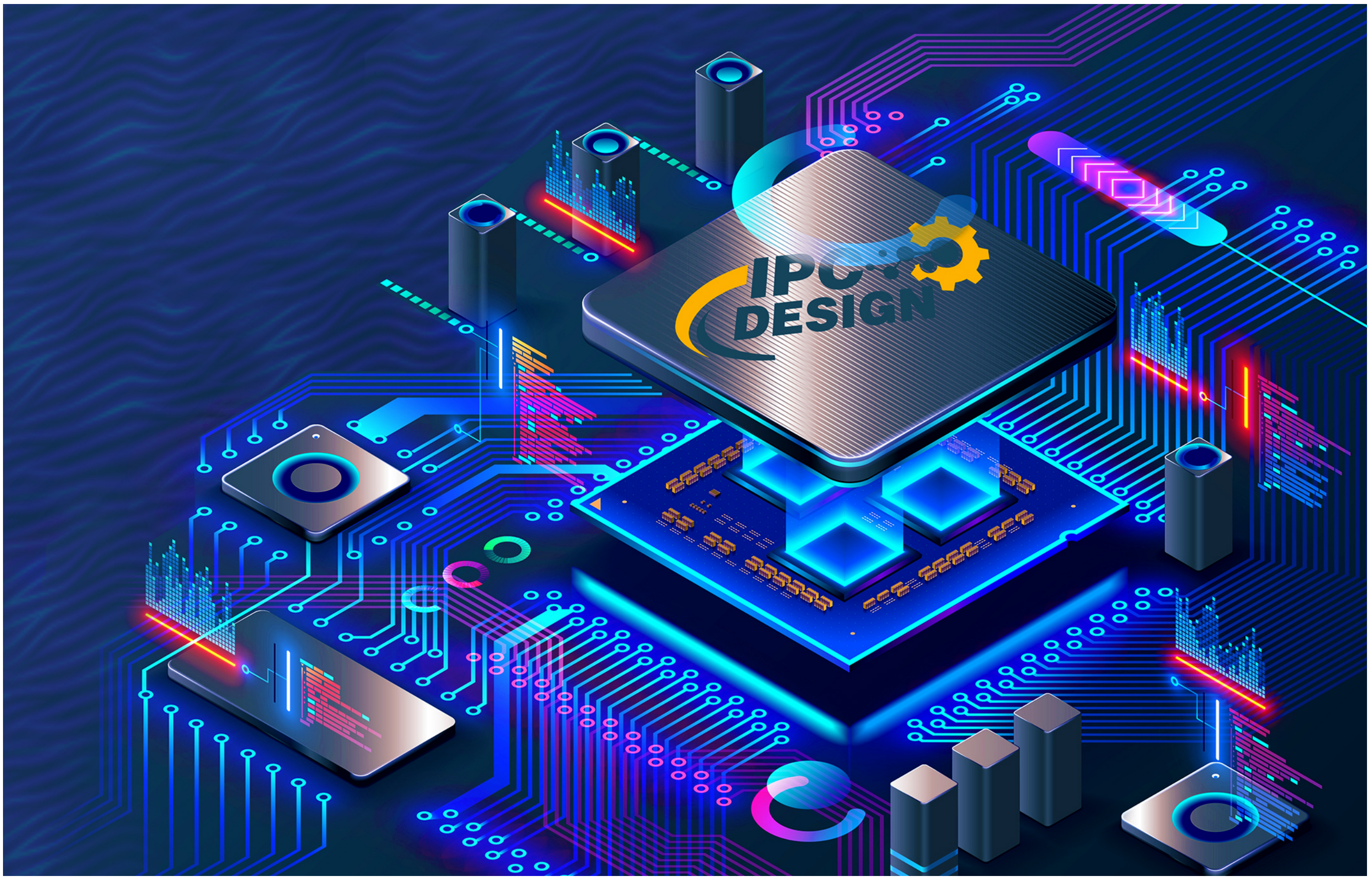
I want them to take away the knowledge, first and foremost, that we're doing engineering work. I want them to walk away knowing that this is a design community; we all work for different companies but we can still talk to each other and ask, "Hey, how do you manage this or that?" Try to keep in touch with folks. You should take what you learn at IPC back to your company and ask your manager, "Can I do a presentation, and instruct others on what I learned? I want to put my education to good use." If you have three or four people in your design center, after a while, everybody knows what you've learned. Everybody can be that much better. That's one of the biggest takeaways you can have.

***Final thoughts?***

I'm looking forward to seeing everyone at IPC APEX EXPO. If you happen to see me walking around, stop and talk to me. I'm available. You don't have to buy me a coffee, but that would help. We'll try to do some podcasts later in the year that will focus on engineering.

***Thanks for your time, Fil.***

Thank you, Andy. **DESIGN007**




# Design Classes Abound at IPC APEX EXPO

**Feature Interview by Andy Shaughnessy**
I-CONNECT007

There are quite a few design classes taking place at IPC APEX EXPO this year, such as Kris Moyer's Professional Development Course, "The Impact of Silicon Geometry Reduction on Signal Integrity and PCB Performance." I asked Kris's former student, Graham Blacksmith, to tell us about the class and Kris's teaching style.

***Andy Shaughnessy: Graham, you've taken some of Kris Moyer's PCB design classes. He's teaching a class at IPC APEX EXPO this year in Anaheim. Tell us about Kris's classes.***

*Graham Blacksmith:* Sure. I've actually taken his other classes, *"Introduction to PCB Design I and II."* Both were excellent. Kris taught us the IPC standards and the practical application of the information directly in a CAD environment. We were also given access to software that is not exactly cheap. Kris gave direct feedback about all the assignments and work, as well as answering everyone's questions down to the last detail. He was concise, very patient, and able to work around difficult schedules. Kris provided plenty of time for those who work more than the average. He has an immense wealth of experience in PCB design.

***What were some of your takeaways from his classes?***

The best takeaways I had were related to Altium use. Learning about PCB standards and prac-


Every Successful Design Starts With Your CAD Library
OrCAD CIP is here to ensure your library is up to the task.
Learn More
Centralized Component Data Management
Real-Time Parametric Data Search & Update
Automated Part History & Lifecycle Tracking
Predictive Supply Chain Intelligence
OrCAD™
CADENCE PCB SOLUTIONS
EMA Design Automation®

tices is one thing, but being able to learn and then use them day-to-day is where the value lies.

***Did you learn any design techniques that you could use on your job right away?***

I am an electrical test engineer in the solar tracking industry. What I learned and was able to use right away was the ability to create and output manufacturing files, drawings, and data for projects on my team. We recently lost our layout engineer, and we needed to fill the gap. With the practice and information I learned from Kris's courses, we were able to pick back up without much loss. We would not have been able to do so if I hadn't taken those courses.

***That's a pretty good example. So, why should designers and design engineers attend the conference at IPC APEX EXPO this year?***

Designers should attend this conference if they want to learn about the most important IPC standards without having to pore over every page of every standard. Of course, reading the standards is still necessary, but with these classes, you get a sense of which standards apply to which elements of design, and you learn exactly where to look when you need something. Combined with the help of someone who can answer almost any question (and I mean it), it's hard not to come up with the answer you are looking for.

Graham Blacksmith

***Thanks for speaking with me, Graham.***

Thank you, Andy. **DESIGN007**

## Breakthrough in Efficiency for Narrow-Bandgap Perovskite Solar Cells

A research team led by Professor Sung-Yeon Jang in the School of Energy and Chemical Engineering at UNIST has achieved a significant advancement in solar cell technology. Through a collaborative effort with Professor Sang Kyu Kwak and his team at Korea University, the researchers have developed a groundbreaking technology that greatly enhances the efficiency of solar cell devices by integrating tin-lead halide perovskites (TLHPs) photoactive layers with quantum dot layers.

One of the challenges with tin-lead halide complexes is their limited energy gap between bands, despite their excellent light absorption capabilities in the near-infrared region. The presence of internal defects and short charge movement distances has hindered stable charge extraction in the past.

Moreover, the alignment of energy levels and efficient collection of electric charges, enabled by the unique properties of the materials, now allow for enhanced electron extraction and increased extraction of electric charges generated by sunlight.

Professor Jang emphasized the significance of the integration of quantum dot and perovskite solar cell technologies, highlighting the potential for high-efficiency solar cells in the future. This research breakthrough opens up new possibilities for the development of next-generation solar cells with enhanced performance and efficiency.

(Source: UNIST)


# IPC's Vision for Empowering PCB Design Engineers

**Feature Article by Robert Erickson**
IPC

As architects of innovation, printed circuit board designers are tasked with translating increasingly complex concepts into tangible designs that power our modern world. IPC provides the necessary community, standards framework, and education to prepare these pioneers as they explore the boundaries of what's possible, equipping engineers with the knowledge, skills, and resources required to thrive in an increasingly dynamic field.

## Setting the Stage for Success

All except the youngest of us remember when telephones were only for talking, and a refrigerator didn't do much at all. Today, traditionally mundane devices like refrigerators, doorbells, and thermostats include advanced electronic components that offer connectivity, smart features, and the ability to communicate. These and the thousands of other devices that make our modern way of life possible rely on printed circuit boards and the skilled individuals who design them.

These designers need to keep up with the latest developments in PCB design as it evolves in tandem with the increasingly complex components required to deliver the functionality provided by this wide variety of products. IPC fulfills this need by providing comprehensive training that helps designers gain a

deep understanding of PCB design principles, industry best practices, and emerging technologies. From foundational courses like *PCB Design I and II* to advanced offerings such as *PCB Design for Manufacturability* and *PCB Design for Advanced Packaging,* IPC's curriculum covers the wide range of skills required to achieve success in PCB design.

Additionally, IPC training is built with adult learners in mind, providing industry-sourced content effectively and efficiently, as follows:

- **Foundational concepts:** The essentials, including PCB design principles, terminology, IPC standards, and industry requirements
- **Software proficiency:** Key tools such as Altium Designer, Cadence Allegro, Siemens EDA, and PADS
- **Design techniques:** Advanced design considerations, including signal integrity, power integrity, high-speed design, and thermal management
- **Design for manufacturability:** Best practices that ensure delivery of production-ready designs for assembly and testing
- **IPC standards:** Compliance with IPC standards for PCB design, fabrication, and assembly ensures quality, reliability, and customer satisfaction
- **Immediate applicability:** Hands-on projects and practical exercises facilitate real-world application of newly acquired skills

## Fostering Innovation and Collaboration

IPC is committed to advancing the field of PCB design through education. However, the organization's commitment goes beyond imparting current knowledge. IPC conferences, working groups, and online forums provide an environment of innovation and collaboration where researchers and practitioners share insights, explore new ideas, and write the future of PCB design.

## Leading the Way in Standards Development

IPC's touchstone standards, such as IPC-2221, IPC-2222, and IPC-6012, are globally recognized benchmarks for excellence in PCB design and manufacturing. IPC collaborates with industry experts and stakeholders to ensure that these standards remain relevant, rigorous, and reflective of the latest advancements in technology and industry best practices. By adhering to current standards, engineers can design with confidence, knowing that their products meet the highest quality and reliability standards.

## Embracing Emerging Technologies

Our forward-looking commitment to the PCB design community is evidenced in regular updates to PCB design standards and curriculum, including the latest advances in technologies such as flexible electronics, additive manufacturing, and Internet of Things (IoT) connectivity. By equipping designers with the knowledge and skills required to harness cutting-edge technologies, IPC empowers them to drive innovation and shape the future of electronics manufacturing.

## Looking Ahead

As we look ahead, PCB designers will play an increasingly important role in the creation of ever more complex and connected devices. These architects of the digital age can rest assured that IPC will be by their side, seeking their input and providing the tools, training, and support they need to succeed in an ever-changing landscape. DESIGN007

**Robert Erickson** is director of product management and NPI for IPC.

# MilAero007 Highlights

## DARPA Explores Additive Manufacturing's Potential for Futuristic Microsystems ▸

DARPA played a seminal role in establishing materials science as a discipline. One of the latest disruptive efforts in new materials and applications, the Additive Manufacturing of Microelectronic systEms (AMME) program, seeks to launch microsystems manufacturing far beyond today's state of the art.

## The Government Circuit: Driving Resiliency and Economic Security on Both Sides of the Atlantic ▸

Welcome to the latest edition of The Government Circuit, where I share updates on the government policy developments shaping the electronics manufacturing industry. Since the year began, we've already witnessed several significant milestones and strategic engagements that may ultimately affect the way you do business.

## Major Investors Expect First Commercial eVTOL Passenger Routes by 2026 ▸

New Horizon Aircraft Ltd. indicates global investors are anticipating the first commercial passenger routes of electric vertical takeoff and landing (eVTOL) aircraft to be operational in the next few years as the future air mobility market continues to advance rapidly.

## BAE Systems Launches MethaneSAT Satellite for Global Greenhouse Gas Emissions Data ▸

BAE Systems is celebrating alongside its customers at the Environmental Defense Fund (EDF) following the successful launch of the MethaneSAT satellite from Vandenberg Space Force Base in California. The satellite will provide the public with reliable scientific data about the sources and scale of methane emissions globally, with the ultimate goal of driving reductions in the near future.

## Lockheed Martin Awarded $219M To Produce PrSM Units for U.S. Army ▸

The U.S. Army has awarded Lockheed Martin a $219 million contract to produce more Early Operational Capability (EOC) Precision Strike Missiles (PrSM). The award is the fourth production contract to date for the long-range surface-to-surface missile, which will allow for a significant increase in production capacity to meet Army demand.

## Sikorsky Looks to Future Family of VTOL Systems ▸

Sikorsky, a Lockheed Martin company unveiled its plan to build, test and fly a hybrid-electric vertical takeoff and landing demonstrator (HEX / VTOL) with a tilt-wing configuration. The design is the first in a series of large, next generation VTOL aircraft—ranging from more traditional helicopters to winged configurations—which will feature varying degrees of electrification, and an advanced autonomy system for optionally piloted flight.

## DARPA's REMA Program to Add Mission Autonomy to Commercial Drones ▸

DARPA's Rapid Experimental Missionized Autonomy (REMA) program aims to enable a drone to autonomously continue its predefined mission when connection to the operator is lost.

# DESIGN TIPS #124:

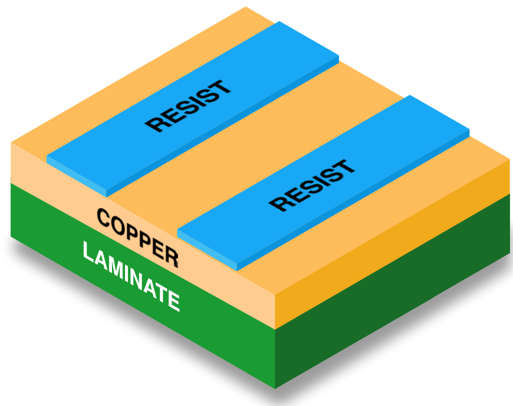
## ETCH COMPENSATION

What is minimum space and trace? The answer depends on the starting copper weight.

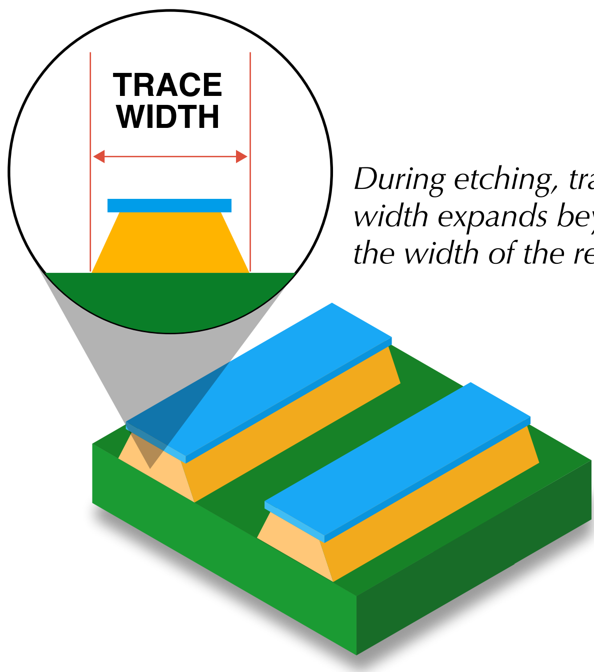
This is because we must do an etch comp on the traces in CAM to compensate for known etch loss. The space between traces after compensation will play a role in whether a board can be manufactured.

The lower the spacing width, the higher the cost. Designers don't always account for the proper starting copper weight after edge compensation.

### Before etching




### After etching




*During etching, trace width expands beyond the width of the resist.*

### Design tips:

- For accurate starting copper weight, **add a half mil (.0005″) to all copper features**.
- **Start with 3/8 or 1/4 oz. foil**, reducing etch comp and less likely to cause a spacing issue.
- **Boards that call for full body electrolytic gold are not comped** to avoid gold slivers occurring during the etching process.

**Prototron Circuits**
*America's* **Board Source**
prototron.com (520) 745-8515

# Drilling Down on Documentation

## The Pulse

by Martyn Gaudion, POLAR INSTRUMENTS

How did a product aimed at signal integrity end up being more about documentation? For a little backstory, the Polar team has an unspoken "no business speak" rule at certain times. So, why is this column titled "Drilling Down?" I find it fascinating when a company sets off in one direction, but customers steer it in another. That's what has happened here as customers took a product down a fork in the road we couldn't predict. Your destination isn't always where you initially set off to go, and that's how we got to our subject of drills and drill documentation.

### Laser-focused, But Rotating Drills Still Play a Major Role

Conventional drilling has come a long way, and from a signal integrity perspective, the advent of precision-controlled depth drilling—especially when combined with contact detection—has transformed the high-speed performance of thicker boards, eliminating troublesome via stubs which behave to the signal as though they are twice as deep electrically as they are physically. Historically, you had to calculate the stub length, which would cap the maximum frequency at which a channel could operate.

Backdrilling the via can release the performance of the channel. The increased availability of PCB drilling machines with precision depth drilling has revolutionized the signal integrity performance of high-layer count PCBs. I have often mentioned that good modeling tools can't fix an inherent design flaw or limitation. In the case of via stubs, the mod-

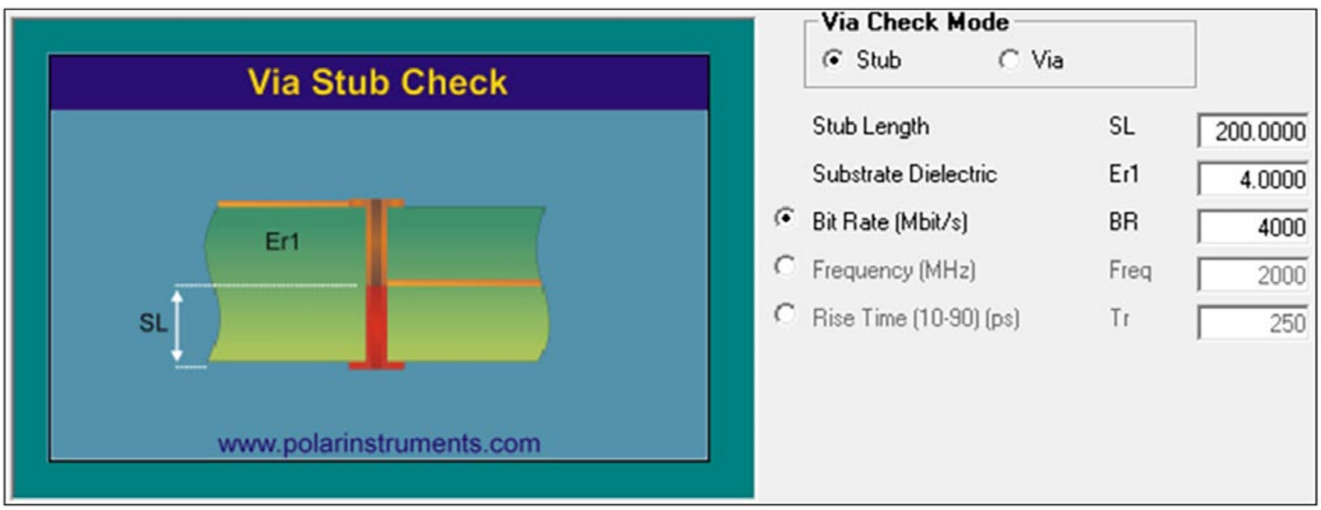



Figure 1: Via stub length of 200 mils requires attention at 4 Gbit/s data rate. Find the math behind this in *Signal and Power Integrity Simplified,* by Dr. Eric Bogatin.

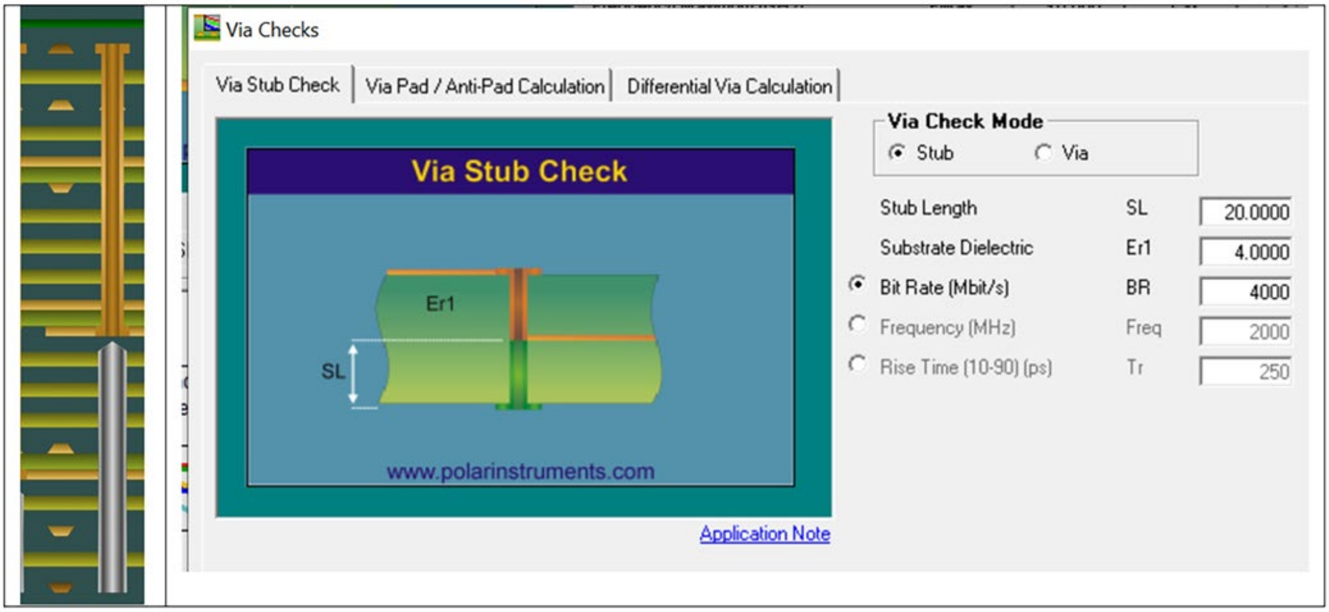



Figure 2: Via stub length backdrilled to 20 mils gives a green light at 4 Gbit/s data rate. Find the math behind this in *Signal and Power Integrity Simplified,* by Dr. Eric Bogatin.

eling tool can flag where stub length becomes an issue and direct the PCB technologist or designer to deploy back drilling in the appropriate cases.

## Via Documentation Fills a Variety of Applications

Aside from backdrilling, where the unused stub of the via is removed, vias can be treated in myriad ways to suit the specific application or functional area of the PCB. In high-power applications, the via may be copper or conductive epoxy-filled. Both methods aid in reducing the resistance of the via; aiding high-power applications have the added benefit of increasing the thermal conductivity and helping heat exit the PCB in a managed way.

Vias may also be filled non-conductively simply to prevent the entrapment of contaminants and for increased reliability.

The variety of options and implications of getting the via fills and styles incorrect mean that clear documentation is vital for supply chain communication.

## Conclusion

I started my column talking about Polar's informal ban on business speak—no reaching out. We email or call each other, no "going forward," no "dog and pony shows," "deep dives" (unless compressed gasses and seawater are involved), and certainly no circling back or "Let's 'parking lot' this." So, why does the title focus on drilling down? It must be on the list of "verboten" words and phrases. In this case, I used the words literally as this article is all about

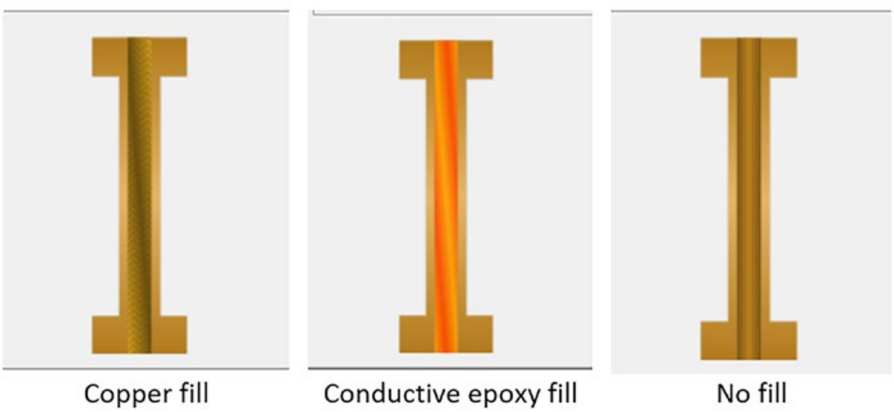



Figure 3: To fill or not to fill?

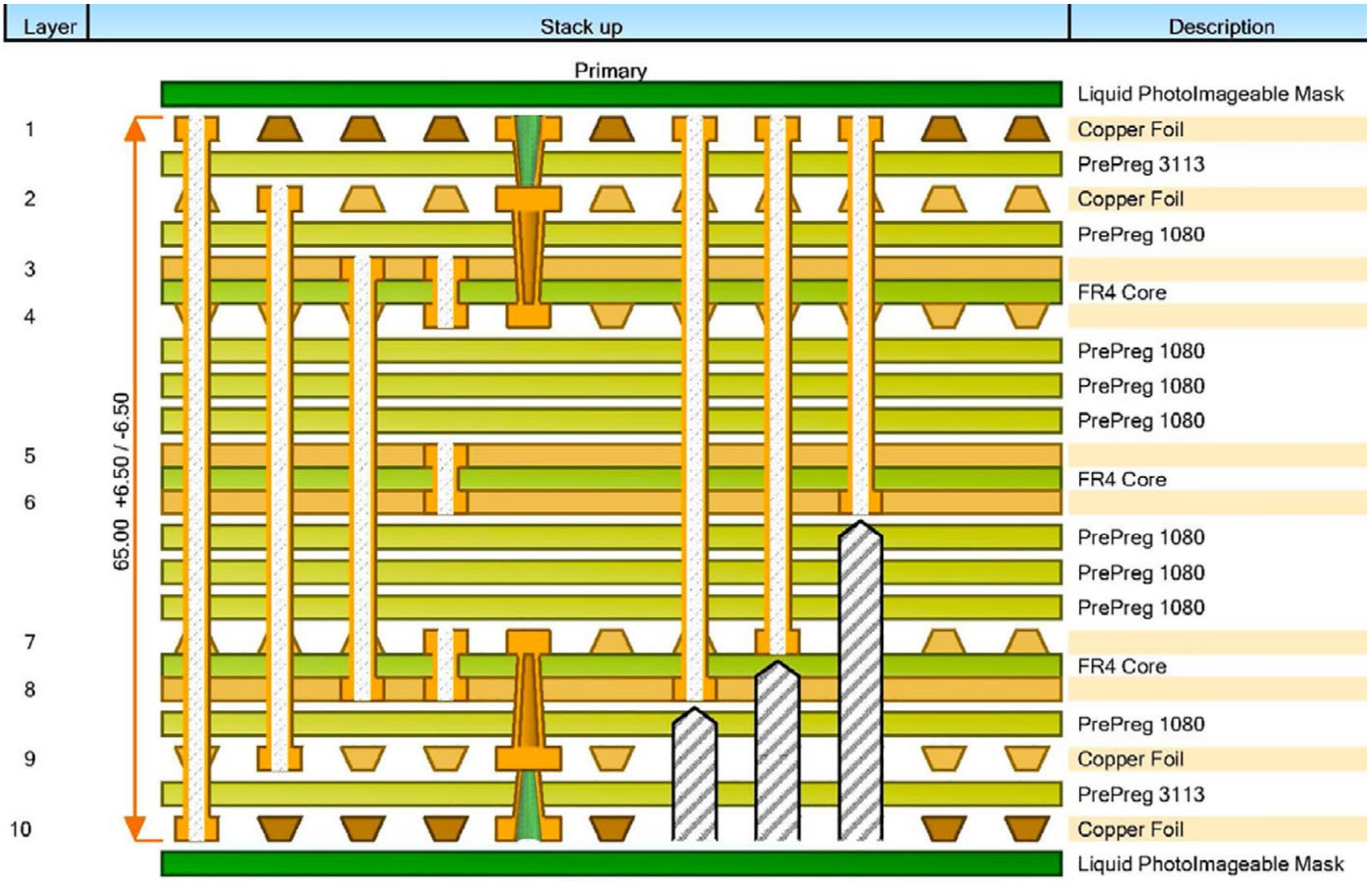



Figure 4: A 10-layer HDI board illustrating just a few of the via fills and drills available to the PCB designer.

the increasing need for drill documentation in PCB stackups.

I also mentioned that sometimes a product doesn't end up where you thought it would. In the early 2000s, Polar was nudged in the direction of providing a simple stackup documentation tool—SB200, the forerunner of today's Speedstack. With the aim for a no-frills stackup tool, we added impedance control to the tool as that was what we were known for then. Over time, we added more impedance capability and extended the tool to cover insertion loss. At the same time, there was a drip, drip, drip of requests for more drill-related documentation. Back then, there was some HDI, but nothing like the widespread adoption of complex drilling we see now. Now, the prime drivers of Speedstack are customer requests on the documentation side, especially drill-related ones, plus the sharing of that documentation information with other tools in use by the industry. It will be interesting to see where our customer base takes us in the next 20 years.

Drilling capabilities have changed radically and the variety of drills and fills available to fabricator and designers mean that a variety of challenges in high power and high speed can be addressed in ways that would have seemed impossible or just an R&D dream a mere 10 or so years ago. **DESIGN007**

**Martyn Gaudion** is managing director of Polar Instruments Ltd. To read past columns, click here. Martyn is the author of *The Printed Circuit Designer's Guide to... Secrets of High-Speed PCBs, Parts 1* and *2*.

# Designing for Reality: The Pre-Manufacturing Process

## Connect the Dots

by Matt Stevenson, SUNSTONE CIRCUITS

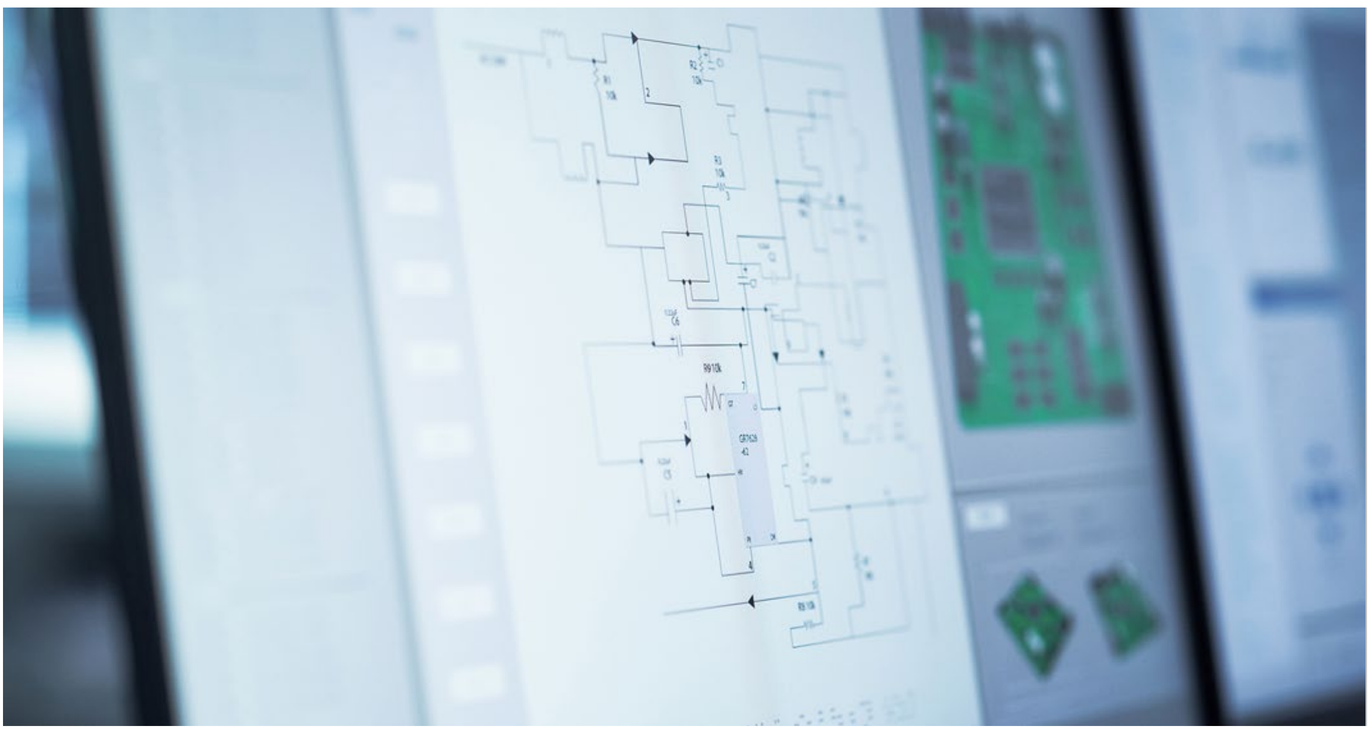
I have been working with Nolan Johnson on a podcast series about designing PCBs for the reality of manufacturing. By sharing lessons learned over a long career in the PCB industry, we hope to shorten learning curves and help designers produce better boards with less hassle and rework. Episode 2 deals with the electronic pre-manufacturing process. (Access the podcast series here.)

Moving from CAD (computer-aided design) to CAM (computer-aided manufacturing) is a key step in PCB manufacturing. CAM turns digital designs into instructions that machines can use to actually build the PCB. This involves exporting files in a specific format, such as Gerber files. Essentially, the pre-manufacturing process shifts from designing a PCB in a computer to the practical work of manufacturing it. This ensures that what was designed can be precisely made into a physical product with actual, working boards.

### Designing for Manufacturability (DFM)

Realistic designs should prioritize manufacturability and reliability of the PCB as well as meet the other design requirements. To do so, one must account for the production variables associated with individual manufacturing partners.

DFM is a great approach for vetting your design and making sure that it will be manufacturable before moving to the quote stage. DFM



is all about designing your product in a way that makes it easy and cost-effective to manufacture without compromising quality. This process spots potential issues early and makes the manufacturing process smoother, ensuring the final product is of high quality.

## Consider the Manufacturer's Design Requirements

One important factor to consider in the DFM stage is that design requirements are specific to the manufacturer. Be aware of this when choosing which tool to use for DFM. Your preferred manufacturer may offer a tool or suggest specific tools. In addition, some CAD tools have DFM-type features, but those need to be configured to match your preferred manufacturer's tolerances and requirements. Note that DFM tools operate on either a completed design file or on individual modules.

Understanding both your CAD tool's features and the requirements of your preferred manufacturer is key. To get a better understanding of DFM and this part of the process, please refer to my book, Designing for Reality.

## Key Takeaway: Important Design Considerations

Designers should use wider traces and gaps (when possible) in their PCB layouts for the best reliability/cost, though sometimes narrower ones might be necessary. It is important to avoid treating every trace the same way. Utilizing larger holes for vias is another tip for increasing manufacturability and lowering cost. Old concerns about acid traps—caused by sharp angles in traces—may not be much of a manufacturing issue anymore, but best practice for design and signal routing would have you minimize these sharp angle routes.

Maintaining enough distance between component pads with protruding "fingers" is critical to minimize potential solder shorting between these features. A minimum of 4 mils (0.004") solder mask webbing between SMT pads is advised to maintain a solder mask dam between the pads. These practices increase board reliability and durability with today's manufacturing techniques.

## Preparing Your Design to Send to the Manufacturer

In the pre-manufacturing phase, different file formats become important. We start with native file formats, which are specific to the CAD software used. However, these files can't always be directly used by manufacturing equipment, so the designer may need to supply Gerber files or have Gerbers created from the native file by the manufacturer. Gerber files have long been the go-to format for sharing PCB design data with manufacturers because they provide the essential information needed to make the boards. This is a venerable file format with a history going back several decades to the early days of PCB manufacturing, though it has been improved over time.

In addition to Gerber files, some manufacturers might require smart data files, like ODB++ and IPC-2581. These carry not just layout information but details about components, assembly, and tests. Sometimes, certain CAD tools or manufacturers might instead use proprietary file types.

When it is time to send your design to your preferred manufacturer, you will need to export your design into the proper file format, such as the Gerber files I've mentioned. Once you have exported the file, I highly recommend viewing it in an external file viewer to make sure everything looks right. Gerber files, in particular, can benefit from this additional step.

## How the Manufacturer Triages Inbound Designs

At the manufacturer, your PCB design will undergo a triage process. First, CAM software checks the design against the specifications provided in the order, including dimensions,

materials, and layer counts, to confirm everything aligns with your expectations and their manufacturing capabilities.

Following this is a manufacturability review, which identifies any elements in the design that might pose challenges during production, such as intricate features that exceed machinery capabilities or materials that are difficult to source. Component availability and lead times are increasingly important, given recent supply chain fluctuations. Ensuring that specified components are readily available helps avoid delays once the production process begins.

It is important to identify and rectify any design issues before the manufacturing phase commences. Although it might seem like an extra step, this saves time, reduces costs, and guarantees that the final products meet both quality standards and client requirements. Essentially, it bridges the gap between design and the practicalities of manufacturing, setting the stage for a smooth and efficient production process.

## Key Takeaway: When CAM Changes PCB Designs

Noticed some unexpected tweaks in your design? It's likely in the solder mask layers; more than 50% of all CAM edits occur in these two layers. While it might seem like setting up the solder mask layers should be pretty straightforward, this is not always the case. Your design may flood the entire board with a solder mask, and then you can carefully carve out spaces around all things that need soldering—like SMT pads, through-hole pads, and test points.

It's a delicate art of ensuring these spaces are big enough for soldering to be effective but not so large that they leave nearby copper unnecessarily exposed, which could lead to dreaded solder bridges. Indeed, achieving the ideal balance in solder mask adjustments is an intricate dance the CAM software performs to keep everything in harmony.

## The Feedback Loop Between the Designer and the Manufacturer

Once a manufacturer approves a design, the work doesn't stop there. It can kick off a feedback loop between the manufacturer and the designer, which, though it might seem like extra work at first, can be valuable in the long run. This feedback loop allows the manufacturer to highlight any problems they encounter while trying to make the design a reality, giving the designer a chance to tweak and improve the design. This not only improves the current project but also smooths the path for future designs. This back-and-forth helps spot and fix issues early, avoiding delays and ensuring a better final product.

The feedback loop teaches designers valuable lessons about manufacturing needs and limits, helping them create smarter, more feasible designs right from the start. Over time, as designers become better versed in the practical requirements of their preferred manufacturer, they start to design with manufacturing in mind, leading to reliable, finished PCBs sooner. Therefore, while it may require more effort and patience up front, this collaborative process between designers and manufacturers is important for creating better products and constantly improving design practices.

I hope this has given you greater insight into the electronic pre-manufacturing process of PCB design. In our next episode of On the Line with…, we will be discussing the inner layer process, including materials, imaging, etching, and inspection. **DESIGN007**

**Matt Stevenson** is vice president and general manager of ASC Sunstone Circuits. To read past columns, click here. Download *The Printed Circuit Designer's Guide to... Designing for Reality* by Matt Stevenson.

# Heavy Copper PCBs:
## Bridging the Gap Between Design and Fabrication, Part 2

**Article by Yash Sutariya**
SATURN ELECTRONICS CORPORATION

Editor's note: To read Part 1 of this article, which appeared in the March 2024 issue of *Design007 Magazine,* click here.

Another factor designers must consider is minimum spacing. Often, we run across designs that would work well for 0.5- or 1-ounce base copper but are actually intended for 3- or 4-ounce copper. Failing to account for etch compensation is one of the most frequent mistakes we see in designs. The most common method for etching is using an etchant chemistry called ammonium hydroxide. As you etch down the unwanted surrounding copper, you also etch against the edges of the desired copper, resulting in smaller features.

**Design issue:** Feature-to-feature spacing too small

**Manufacturing pitfall:** Etch factor/compensation

For this reason, our front-end CAM operators increase all features proportionally to the finished desired copper weight. Etch factor (etch compensation) predicts the mechanics of etching so it is important that designers assign adequate spacing between features. Figure 7 is a brief pictorial description of the outer layer etching process. A similar mechanism applies to the inner layer etching process.

### *Design Solution*

To create a universally buildable design, you should account for 0.001"/25 microns of inward

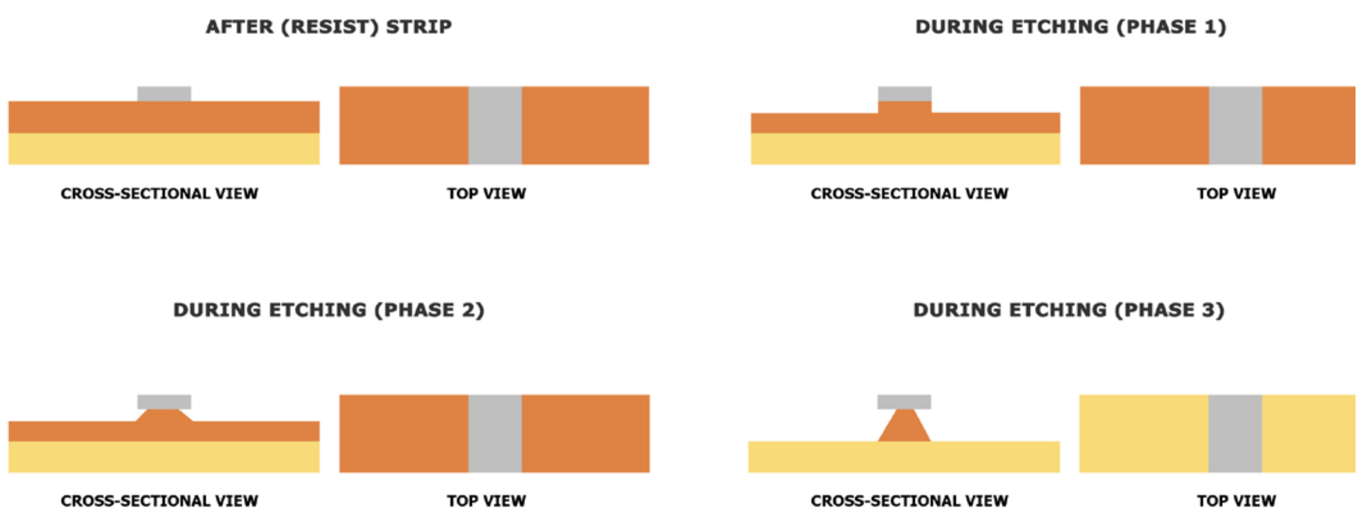


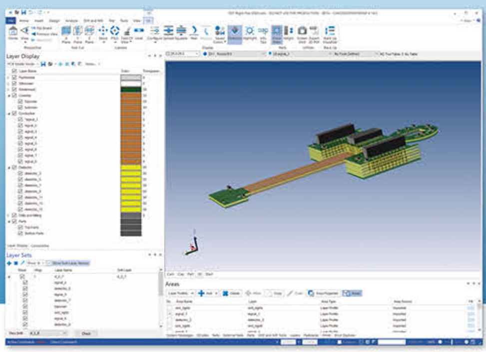

Figure 7: A description of the outer layer etching process.

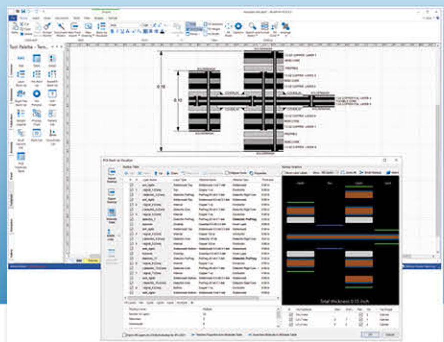
# Support For Flex, Rigid Flex and Embedded Component Designs Now Available.

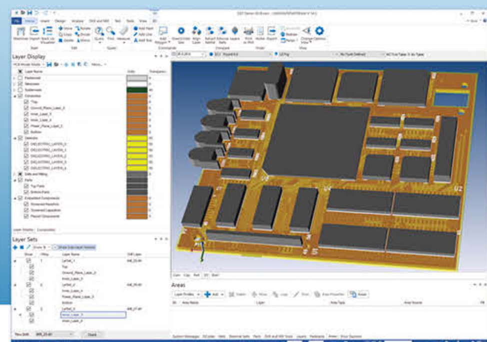
**BluePrint·PCB®**

**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







**DownStream Technologies**

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

Table 6.

| 3 oz inner layer Cu design |
|---|
| **Bare Minimum**<br>3 (ounce) x 0.001" x 2 (adjacent features) + 0.004" (minimum final spacing) = 0.010" minimum designed spacing. |
| **Robust Design**<br>3 (ounce) x 0.001" x 2 (adjacent features) + 0.006" (optimal final spacing) = 0.012" minimum designed spacing. |

etching per 1 ounce of base copper. Base copper for outer layers is 1 ounce less than the desired finished copper while the base copper for inner layers is the same as the desired finished copper weight (excluding a termination layer for a plated blind or buried vias). Final spacing should be at least 0.004"/100 microns for good yield results.

### *Fabricator Solution for Inner Layers*

The inner layers require more scrutiny as they tend to feature thicker amounts of copper. Do not worry if your design does not allow for this amount of spacing (typically in windings, via pad diameters, etc.) as larger fabricators employ two distinct methods for etching the inner and outer layers.

The ideal method for inner layer etching features the etchant chemistry cupric chloride. As it etches down, it leaves a coating against the now-exposed sidewall of active features to inhibit further etching. A good rule of thumb states that for every ounce of copper etched down, the etch into each feature's sidewall is only one-quarter to one-half of 1 mil (0.00025"–0.0005"/6.25 to 12.5 microns). For good yield results, final spacing should be at least 0.004"/100 microns. With that in mind, we will now recalculate the 3-ounce inner layer copper design using cupric chloride as the etchant (Table 7).

Table 7.

**Heavy Cu PCB Design & Fabrication Solutions**

| Design Issue | MFG Pitfall | Solutions |
|---|---|---|
| **Stackup dielectric thicknesses too thin** | Delamination / Resin Voiding; Inner Layer Misalignment | Get Help! Consult vendor for stackup generation; Greater allowances for layer-to-layer registration; auto-align X-ray drill |
| **Feature-to-feature spacing too small** | Over-Etching (anti-Puddling); Etch Factor / Compensation | Make some Space! Account for 25 microns of lateral etching per one ounce of copper; distinguish between inner and outer layer etch methods. |
| **Via diameters too small** | Drill Breakage / Reduced Throughput | Go Big! Be mindful of the copper density when selecting via diameters. |

Table 8: Comparison of etching methods

| Etching Method | Bare Minimum Spacing | Robust Design Spacing |
|---|---|---|
| Ammonium Hyrdroxide | 0.010” | 0.012” |
| Cupric Chloride | 0.0055” | 0.0075” |


Figure 8: We have developed a spacing calculator to aid designers.

**Design issue:** Too small vias

**Manufacturing pitfall:** Drill breakage/reduced throughput

Remember in “Armageddon” when Harry Stamper (Bruce Willis) and his team were struggling to drill through the dense metal alloy to reach the asteroid’s core? We face a similar problem when drilling through heavy copper circuit boards.

Typical via diameters in standard technology PCBs bottom out at 0.008"/200 microns (for reference, a typical human hair is 0.002"/50 micron). Given that, we may drill through one-half and 1-ounce copper with a total PCB thickness of 0.062"/1.6 mm. The copper density in a 10-layer design half-ounce copper might be 11–22%. Meanwhile, a 10-layer design with 3-ounce copper can easily hit 0.093"/2.36 mm.

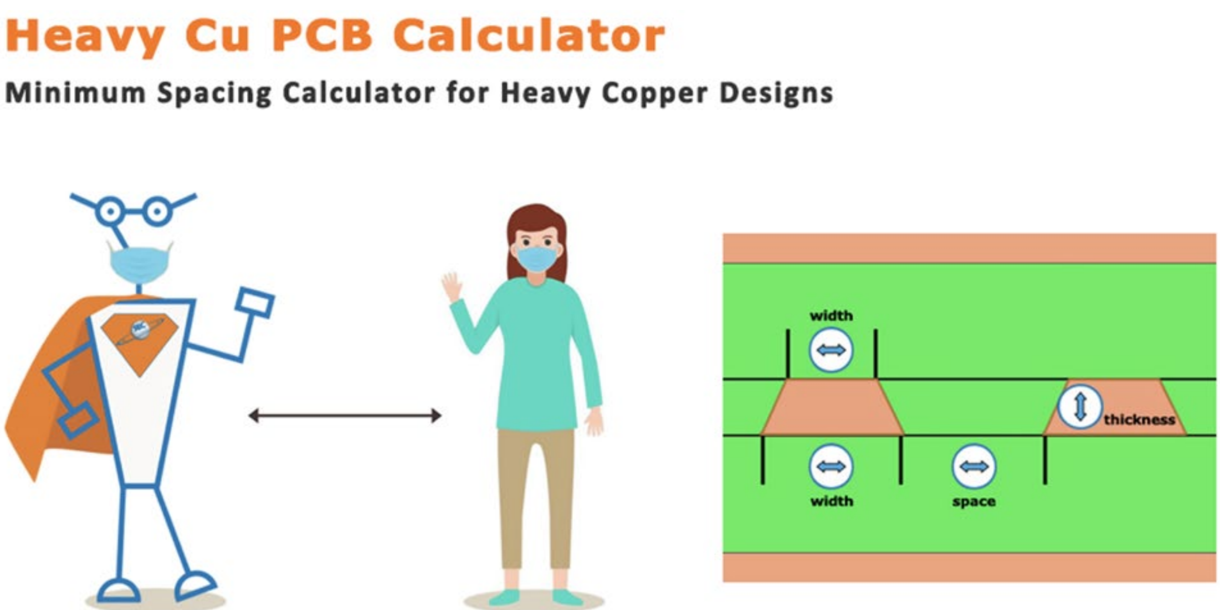



Figure 9.

Table 9.

**Heavy Cu PCB Design & Fabrication Solutions**

| Design Issue | MFG Pitfall | Solutions |
|---|---|---|
| **Stackup dielectric thicknesses too thin** | Delamination / Resin Voiding; Inner Layer Misalignment | Get Help! Consult vendor for stackup generation; Greater allowances for layer-to-layer registration; auto-align X-ray drill |
| **Feature-to-feature spacing too small** | Over-Etching (anti-Puddling); Etch Factor / Compensation | Make some Space! Account for 25 microns of lateral etching per one ounce of copper; distinguish between inner and outer layer etch methods. |
| **Via diameters too small** | Drill Breakage / Reduced Throughput | Go Big! Be mindful of the copper density when selecting via diameters. |

The copper density in this design would be closer to 45%. Even Bruce would have a hard time drilling through this with a 0.012" drill bit.

While peck-drilling and flip-drilling could help combat this, the design solution is the best method to resolve the issue; therefore, increase via size to at least 0.016" to 0.018" on 3+ ounce copper multilayer designs.

## Design/Manufacturing Methods

Three of the most common design and manufacturing methods are etch-down, hybrid (buried heavy Cu), and plate-up.

### *Etch-Down*

With its lower cost and simplified process flow, etch-down is part of the standard process used for fabricating your run-of-the-shop board. This is fine for most boards—but heavy copper PCBs are not most boards. In most boards, that trapezoid shape resulting from the uneven sidewalls will not affect current carrying capacity; however, this is a major issue for heavy copper boards where mechanical strength and subsequent field life depend on thick, powerful traces to conduct power and transfer heat.

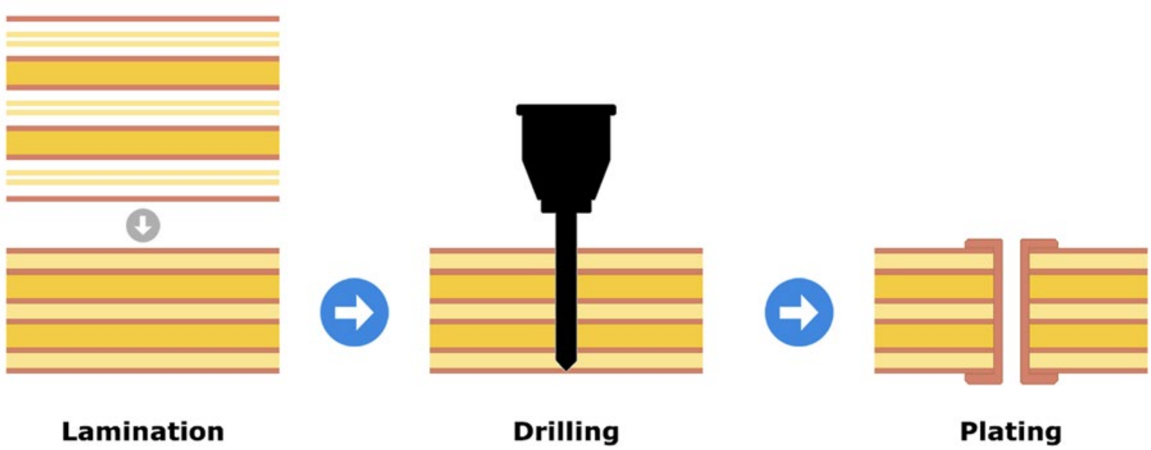



Figure 10: The best way to reduce drill breakage is to increase drill size.

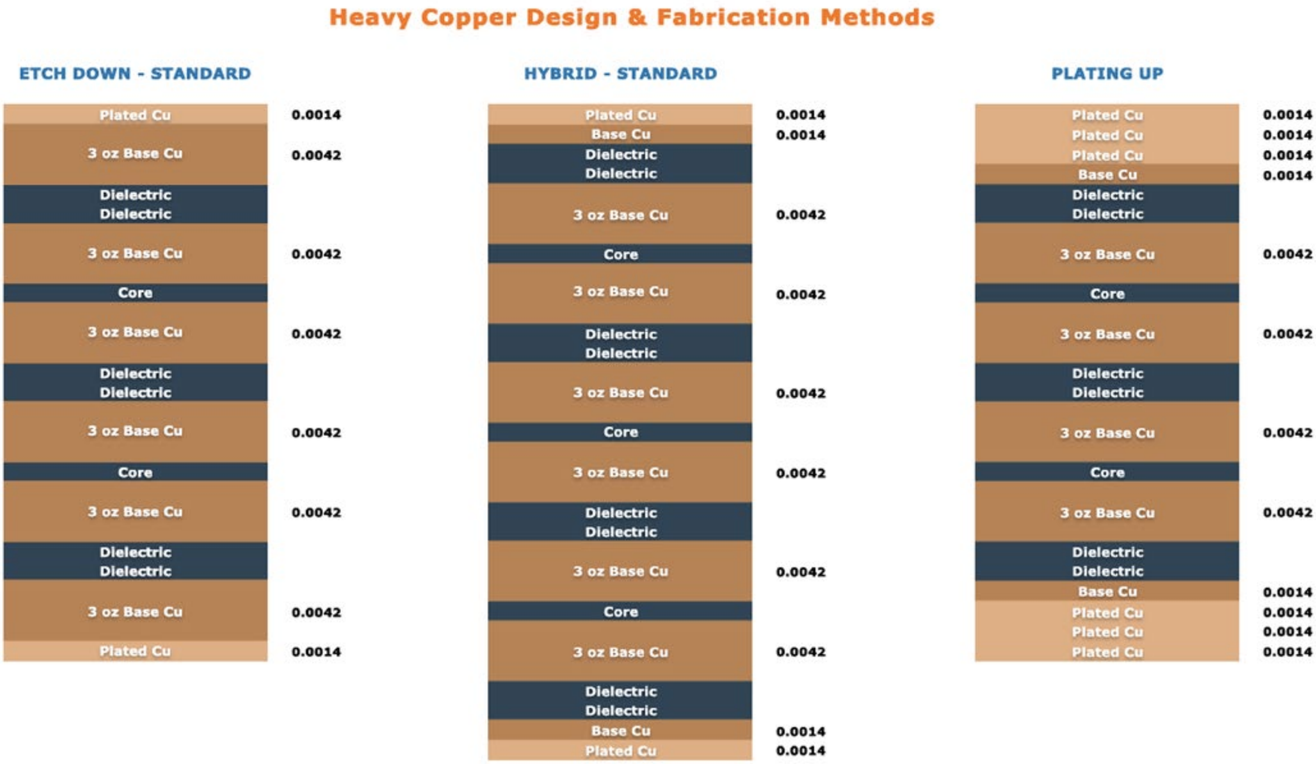



Figure 11.

There are considerable minimum line and spacing requirements and the prodigious copper weight impedes most finer pitch SMT components. There are also a limited number of capable suppliers as UL-certified materials have a 6-ounce ceiling.

### *Hybrid*

By bringing digital features to the surface and relegating heavy copper to the inner layers, the hybrid method allows for finer lines and spaces to accommodate finer pitch components. Hybrid uses standard manufacturing methods to connect the heavy copper inner layers with vias and component holes. Nevertheless, the increased materials and processing make it more expensive than a standard PCB—though still more economical than the plate-up method.

The fabricator might advise increasing the overall thickness to accommodate the increased prepreg plies required to fill internal copper layers (utilizing layer ink provides a solution here as we will discuss in the next section). While some shops lack the necessary equipment for the hybrid method, most domestic fabricators should be able to provide it.

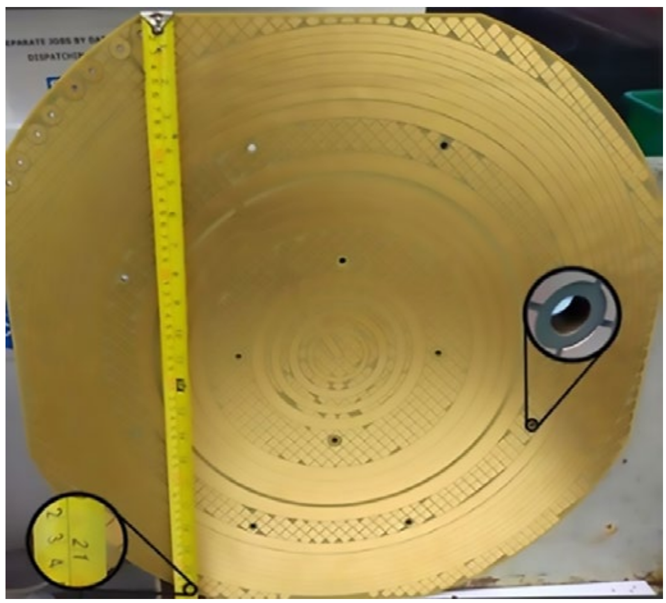


Figure 12: Example of the hybrid method.

### *Plate-up*

Plating-up reaches the desired finished copper weight by starting at a lower copper weight

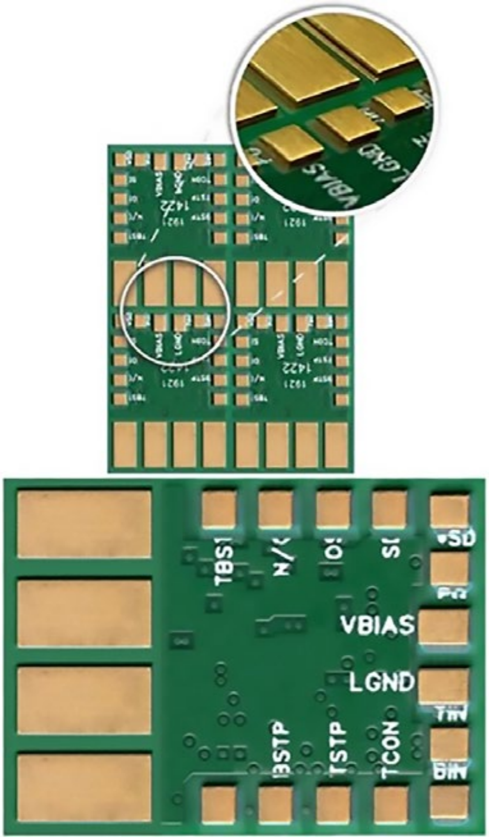



Figure 13: Plate up.

and then processing multiple plating cycles to hit the target. A typical plating cycle presumably adds one ounce of copper to the PCB surface (in order to plate a minimum of 1 mil in through-holes). At a lower amperage setting, this will take two hours in the copper plating bath alone (resulting in an overall plating cycle of over three hours). Consequently, this is one of the longest cycle times of all the PCB fabrication processes. When all is said and done, plating-up to 3-ounce finished copper from a half-ounce base adds multiple days to the lead time.

Since it uses standard DFM rules, plating-up allows for smaller minimum line and spacing requirements. Most fabricators have the capacity for plating up, even if they are not aware of it. Having said that, there is a much higher cost resulting from the repeated back-and-forth from imaging-to-plating. There are also an increased number of failure modes, considerable fluctuation in the final copper weight, and longer lead times. Additionally, the sidewalls will not be flush/straight. They may look more like a Jenga wall.

This method is a last resort to avoid higher costs and longer lead times.

## Layer Fill Ink

Sometimes, we encounter a predicament where the desired inner layer copper thickness does not jibe with the desired maximum total thickness. The primary reason is that in order to have enough prepreg plies to fill the etched gaps requires a set minimum thickness allowance between layers. By the time we add all of them up, we are well beyond the maximum desired thickness. So, what do you do when overall thickness cannot be increased *and* inner layer copper weight cannot be decreased? The answer is layer fill ink (Figure 14).

Figure 14: Layer fill ink.

Instead of relying on the prepreg to fill the etched gaps between layers, we pre-fill those gaps using an epoxy ink that is compatible with the PCB dielectric materials. This method involves screening the ink onto one layer at a time, baking, and then planarizing the ink to insure flatness. You then have to flip the panel over and repeat on the opposing side layer. Not only is this time-consuming but it also requires the use of expensive layer fill ink. That being said, it has allowed us to create some pretty intense heavy copper multilayer products in the past. Our record was 20 layers of 20-ounce copper.

## Design Considerations

Knowing the methods used to create heavy copper PCBs, design considerations should now make more sense. The primary consideration will be the overall thickness. If the constraint is too tight it could result in a much more expensive production method. This the easiest place to start to seek resolution since fabricators should be able to provide stackup consultation with just a proposed layer count, a description of each layer (e.g., power, ground, signal), minimum desired dielectric thickness, and maximum overall desired thickness.

An aid to successful stackup design and multilayer lamination will be copper retention—especially on signal layers. A good rule of thumb states that the more copper you retain in the design, the less prepreg you will need to ensure the filling of the etched areas. As such, we recommend filling in all "blank" areas on inner layers with dummy copper (or include a fabrication note allowing the fabricator to do so with a minimum clearance to active areas) and retaining "dummy" pads in inner layer clearances.

### *Silkscreen/Nomenclature*

Another frequent design flaw is the use of silkscreen wherein features traverse across

Table 10: Extreme heavy Cu design and fabrication comparison chart

| | Etch-Down | Plate-Up | Hybrid |
|---|---|---|---|
| Cost | Medium | High | Standard |
| Process Flow | Standard | Complex | Standard |
| Min Line / Space | Large | Medium | Standard |
| Materials | max out at 6oz; non UL up to 20 oz | max recommended 3 oz. | max out at 6oz; non UL up to 20 oz |
| Supplier Availability | Limited | Limited | Standard |
| Solder Mask | Trace fill ink; spray method | Standard | Standard |
| DFM Rules | Standard | Variable | Standard |
| Failure Modes | Standard | High | Standard |
| Cu Weight Variation | Standard | High | Standard |
| Lead Times | Standard | Long | Standard |
| Level Sidewalls | No | Yes | Yes |
| SMT Compatible / Compliant | Dependent on Cu Weight | Yes | Yes |

copper and non-copper areas. The difference in depth often deems the silkscreen illegible. Designers should avoid silkscreen altogether. When that is not an option, be sure to relegate silkscreen to the top of copper features. While this does not affect board functionality it may be an issue for customers meticulous about aesthetics, which we have seen rise exponentially since the release of Steve Jobs' biography.

### *Final Finish*

Since heavy copper PCBs are already more costly than their skinnier counterparts, a common cost-saving habit is to specify SnPb or Pb-free HASL final finish. We encourage customers to use either OSP or an immersion finish as these are applied chemically to the PCB surface. Conversely, HASL finishes involve coating the PCBs with flux and then dipping them into a pot of molten solder. On the way out, we clear the holes of solder and create a flat surface by blowing at the panel with high pressure air knives. While this works well for standard designs, heavy copper PCBs often heat sink the solderpot resulting in an uneven surface finish. HASL operators habitually re-dip these panels and create yet another thermal excursion. Multiple thermal shocks like this are harmful to thicker copper designs and can result in delamination or, at a minimum, a weakening of the overall package.

## Conclusion

Heavy copper designs are a neat segment of the PCB market. A close working relationship between the designer and fabricator is an essential formula for success. Conversely, designing in a vacuum and ignoring the manufacturing pitfalls can lead to a path of failure and frustration.

There are numerous other technologies that could be considered heavy copper, particularly in the thermal management PCB realm. These include E-coins, U-coins, via farms, and filled plated-shut vias that could serve as alternate solutions for high power needs. Regrettably, this article is not a fully comprehensive compilation of every possible design consideration. This is merely an opening salvo intended to launch you down the right path to producing these high-calorie PCBs. DESIGN007

**Yash Sutariya** is president of Saturn Electronics Corporation.

# Tapping Into the 300 GHz Band with an Innovative CMOS Transmitter

New phased-array transmitter design overcomes common problems of CMOS technology in the 300 GHz band, as reported by scientists from Tokyo Tech. Today, most frequencies above the 250 GHz mark remain unallocated. However, high-frequency electromagnetic waves become weaker at a fast pace when travelling through free space.

Now, a research team led by Professor Kenichi Okada from Tokyo Institute of Technology (Tokyo Tech) and NTT Corporation have recently developed a 300 GHz-band transmitter that solves these issues through several key innovations. Their work will be presented in the 2024 IEEE International Solid-State Circuits Conference.

The researchers tested their design through both simulations and experiments, obtaining very promising results. Remarkably, the proposed transmitter achieved a data rate of 108 Gb/s in on-PCB probe measurements, which is substantially higher than other state-of-the-art 300 GHz-band transmitters.

Moreover, the transmitter also displayed remarkable area efficiency compared to other CMOS-based designs alongside low power consumption, highlighting its potential for miniaturized and power-constrained applications. Some notable use cases are sixth-generation (6G) wireless communications, high-resolution terahertz sensors, and human body and cell monitoring. (Source: Tokyo Institute of Technology)


PODCAST SERIES - SEASON 2
Designing for Reality
with Matt Stevenson, ASC Sunstone
Season 2 of this popular podcast systematically details the interrelationships between design, fabrication, yields, and cost optimization.
ON THE LINE
WITH... I-Connect007
Listen on Spotify
LISTEN NOW!

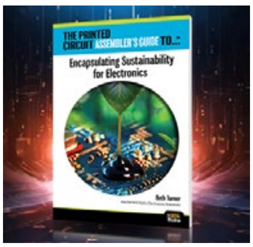
# TOP TEN

EDITOR'S PICKS


## Being the Best Design Engineer

During DesignCon 2024, I met with Bill Hargin, founder and CEO (Chief Everything Officer) of the stackup design software company Z-zero. In this interview, Bill discusses his stackup curriculum, what it takes to become a great design engineer, and why the best designers keep reading and challenging themselves constantly so they can stay on top of their game.

## MacDermid Alpha Electronics Solutions' New Book Now Available for Download




I-Connect007 is pleased to announce the launch of *The Printed Circuit Assembler's Guide to... Encapsulating Sustainability for Electronics* by Beth Turner at MacDermid Alpha Electronics Solutions. The latest in a series of educational books published by I-Connect007, this book is an introductory guide to understanding encapsulation resins and a tool for application troubleshooting.

## On the Line With... Talks With Cadence Experts on Changing How PCBs Are Designed and the Role of AI


In this second episode, Cadence Distinguished Engineer Taylor Hogan and Product Management Director Patrick Davis discuss the role artificial intelligence and machine learning play in intelligent system design.

## The Challenges, Opportunities, and Future Specialties of PCB Design

What were once specialties have become more generalized over time—PCB designers must learn about design automation, signal integrity (SI), electromagnetic compatibility (EMC), complex high-speed design, mechanical design, and manufacturability/producibility.

## Beyond Design: The Art of Presenting PCB Design Courses

Whether it's presenting software courses or demonstrating a product's value, effective communication is key. By imparting knowledge, you enable others to appreciate the technology. As a presenter, your role extends beyond information delivery; it's about igniting curiosity and competence.

## The Survey Said: What Does It Take to Become a Great Designer?

What does it take to become a great designer? After all, there isn't exactly a critical path to becoming a great designer. How does a designer qualify as "great" in the first place? We posed that question to our PCB designer readers in a recent survey, and, as usual, the readers did not disappoint.

## UHDI Fundamentals: Talking UHDI with John Johnson, Part 3

American Standard Circuits is an early adopter of Averatek's A-SAP process for its ultra high density interconnect (UHDI) products. In this final part of my interview with industry veteran John Johnson, vice president of business development at American Standard Circuits, we use photos, slides, and materials to discuss what he learned from his previous role at Averatek.

## OrCADX: High Performance That's Easy to Use

Chris Banton

During DesignCon, I spoke with Chris Banton of EMA about the newest developments around OrCADX, which provides PCB designers with more design automation capability across the whole design process in an easy-to-use GUI. As Chris says, this tool is designed for agile design teams, and features an updated PCB UX.

## Connect the Dots: Are You Ready for 2024?

I see 2023 as one of the most exciting and significant years at Sunstone Circuits since I began my career with the organization. We joined forces with American Standard Circuits (ASC), an organization that excels in producing ultra HDI, metal-backed/core, RF/microwave, flex, and rigid-flex PCBs for diverse industries.

## ICAPE Webinar: Want to Know How to Order PCBs in 5 Minutes?

ICAPE Group is excited to announce an exclusive webinar that will unveil the secrets to streamlined PCB procurement in under 20 minutes. Titled "Unlock the secrets of rapid online PCB ordering," this webinar promises to revolutionize the way you approach prototype and small series PCB ordering.

For the latest news and information, visit **PCBDesign007.com**

# Career Opportunities

## Account Manager

### Eastern United States Territory

EMX US Inc. is currently in search of an account manager for the Eastern United States territory. We are looking for a motivated individual to manage accounts, develop sales activity within the territory, and provide excellent customer support.

**Key Responsibilities**

- Support existing sales logistically and commercially
- Provide field assistance to distribution accounts
- Liaise with suppliers
- Grow sales through increased market share and the conversion of new business opportunities into sales
- Develop and maintain account plans to achieve sales targets and objectives
- Forecast sales projections and provide regular reports on market activity
- Regular travel within the territory is required

**Experience**

Ideal candidate has technical sales experience in the PCB industry.

**About EMX US Inc.**

EMX US Inc. is a distributor of consumable materials to the US printed circuit board industry. We strive to provide unparalleled service and support to our customers. EMX US Inc. is located in Manchester, NH.

Please submit inquiries to
careers@emxus.com

apply now

## Field Service Engineer (or) Field Service Technologist

SCHMID Group is currently in search of a Field Service Engineer or Field Service Technologist for its USA subsidiary SCHMID Systems, Inc. (SSI). This position acts as an advocate for the company providing worldwide customer service on-site or remotely.

General scope of duties includes machine installation, commissioning, maintenance, and repair of PLC and PC-controlled systems primarily in the company's proprietary industrial machines within the wet chemical processing industry as well as automation technology.

This is a full-time exempt position with limited supervision. SSI provides full-time employees different options for benefits including medical, dental, vision, flex, 401K, and more.

Contact Bob Ferguson:
Ferguson.ro@schmid-group.com

apply now

# Career Opportunities

## Senior Printed Circuit Board (PCB) Designer

Garmin is seeking a full-time Senior Printed Circuit Board (PCB) Designer in our Olathe, KS, Cary, NC or Salem, OR location. Relocation allowance provided.

### Essential Functions

- Partners with Product Design Engineers and Mechanical Engineers to produce loosely defined complex PCB Designs that are timely, robust, and economical
- Key technical contributor and ECAD/MCAD interface expert in the design of new or challenging PCB designs or projects bearing directly on organizational objectives
- Leverages thorough knowledge of Garmin processes and procedures through leadership of major research or product development projects consisting of multiple modules or sub-projects that align the team with Garmin and departmental mission and vision
- Leads the advancement of team capabilities through identification and testing of new PCB design technologies for ECAD software
- Connects engineering teams, communicating effectively with all project stakeholders (ex. Electrical, Process and Mechanical Engineering)
- Serves as an expert in PCB Design and Engineering processes including mentoring one or more PCB Designers

### Basic Qualifications

- Associate's Degree in Electronics Technology or related field AND a minimum of 10 years relevant experience performing similar consumer electronics industry duties OR an equivalent combination of education and experience
- Demonstrates expert proficiency using Garmin's ECAD tools (Cadence Allegro)

apply now

## Lead eCAD Librarian

Garmin is seeking a full-time Lead eCAD Librarian in our Olathe, KS or Cary, NC location. Relocation allowance provided.

### Essential Functions

- Ability to define library solutions with a cross functional understanding of the overarching library impact
- Manages/delivers a global library database solution within established Garmin standards
- Develops reliable solutions for exceedingly complex eCad Library parts which require the regular use of individual thought and creativity
- Verifies/validates schematic symbols and physical footprints for parts created by other librarians for accuracy
- Leads advancement of team capabilities through identification and definition of eCAD Library technical strategy
- Expert in evaluation of new eCAD features and capabilities as they relate to the eCAD Library
- Ability to define eCAD Library process for new technologies and capabilities
- Ability to mentor one or more eCAD Librarians

### Basic Qualifications

- Possess a minimum of 15 years experience in an eCAD librarian position OR an equivalent combination of education and relevant experience
- Demonstrates expert proficiency of eCAD Library best practices and design standards for all PCB technologies used in current Garmin designs
- Demonstrates a working knowledge of all types of electronic components
- Demonstrates proficiency to interpret Manufacturer Data Sheets
- Demonstrates proficiency of PCB manufacturing processes

apply now

# Career Opportunities

pluritec®

MACHINES FOR PRINTED CIRCUIT BOARDS

## Sales Manager, Remote

**Location:** North America

**Experience:** Minimum of 4 years in the PCB industry

**Job Description:** We are looking for a highly motivated and experienced sales manager to join our team. The ideal candidate will have a minimum of 4 years of experience in the PCB industry and a proven track record of success in sales. The successful candidate will be responsible for developing new business and sales network, maintaining existing accounts, and achieving sales targets. The candidate must be able to work independently, have excellent communication and interpersonal skills, and be willing to travel.

**Qualifications:**
- Minimum of 4 years of experience in the PCB industry
- Proven track record of success in sales
- Excellent communication and interpersonal skills
- Strong technical process background
- Ability to work independently.
- Willingness to travel

**Education:** Technical or related field preferred

**Compensation:** Competitive salary and benefits package

Pluritec develops high end equipment for the printed circuit board (PCB & PCBA) manufacturing industry. We offer a wide range of equipment including drilling and routing, wet processing, spray coating and more. We are a global supplier with more than 3,000 systems installed worldwide.

**Contact Nicola Doria nicola.doria@pluritec.org to apply.**

apply now

TTC The Test Connection, Inc.

## Rewarding Careers

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

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

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

## Associate Electronics Technician/ Engineer (ATE-MD)

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

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

apply now

# Career Opportunities

TAIYO AMERICA

## 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 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 customer's 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

BLACKFOX Premier Training & Certification

## 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. Candidate 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

Prototron Circuits

## 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

EMA Design Automation®

## 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

ARLON ELECTRONIC MATERIALS

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

INSULECTRO

## Are You Our Next Superstar?!

Insulectro, the largest national distributor of printed circuit board materials, is looking to add superstars to our dynamic technical and sales teams. We are always looking for good talent to enhance our service level to our customers and drive our purpose to enable our customers 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

# Career Opportunities

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

eptac
TRAIN. WORK SMARTER. SUCCEED.

## Become a Certified IPC Master Instructor

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

### Qualifications and skills

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

### Benefits

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

apply now

# Career Opportunities

American Standard Circuits

Creative Innovations In Flex, Digital & Microwave Circuits

## CAD/CAM Engineer

### Summary of Functions

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

### Essential Duties and Responsibilities

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

### Organizational Relationship

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

### Qualifications

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

### Physical Demands

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

apply now

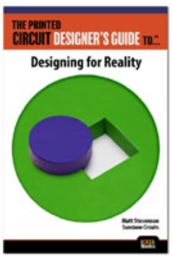
## Educational Resources


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

### *Manufacturing Driven Design*

**by Max Clark, Siemens**

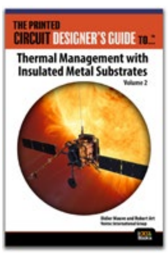
This book introduces a new process workflow for optimizing your design called Manufacturing Driven Design (MDD) and is a distinct evolution from DFM. Manufacturing certainly plays a critical role in this process change, and manufacturers do certainly benefit from the improved process, but it is design teams that ultimately own their overall product workflow; they are the ones who need to drive this shift. Get empowered now!



### *Designing for Reality*

**by Matt Stevenson, Sunstone Circuits**

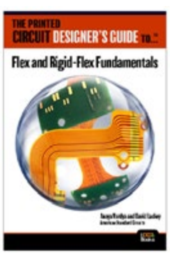
Based on the wisdom of 50 years of PCB manufacturing at Sunstone Circuits, this book is a must-have reference for designers seeking to understand the PCB manufacturing process as it relates to their design. Designing for manufacturability requires understanding the production process fundamentals and factors within the process. 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.



### Flex and Rigid-Flex Fundamentals

**by Anaya Vardya and David Lackey, American Standard Circuits**

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

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

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

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

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

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

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

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

BUSINESS DEVELOPMENT 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 © GOL**

DESIGN007 MAGAZINE

DESIGN007 MAGAZINE®
is published by IPC Publishing Group, Inc.
3000 Lakeside Dr., Suite 105N, Bannockburn, IL 60015



April 2024, Volume 13, Number 4
DESIGN007 MAGAZINE is published monthly
by IPC Publishing Group, Inc., dba I-Connect007

# ADVERTISER INDEX

Problems solved!
PCB007 MAGAZINE
JANUARY 2024
Getting to Know Your Designer
PCB007 MAGAZINE
FEBRUARY 2024
SNEAK PEEK: IPC APEX EXPO PRE-SHOW ISSUE
WELCOME
DESIGN007 MAGAZINE
FEBRUARY 2024
DESIGN007 MAGAZINE
MARCH 2024
OPPORTUNITIES
DESIGN007 MAGAZINE
APRIL 2024
LEVEL UP YOUR DESIGN SKILLS
SMT007 MAGAZINE
MARCH 2024
SNEAK PEEK IPC APEX EXPO PRE-SHOW ISSUE
SMT007 MAGAZINE
APRIL 2024
SUBSCRIBE NOW