MARCH 2024

I-Connect007

DESIGN007 MAGAZINE

A Distinct Evolution from DFM

Look Inside

"In an industry racing against time, MDD is the key to staying ahead."

Ammar Abusham
DFM Integration Engineer (AR EE Team)
Meta Reality Labs

DESIGN007 MAGAZINE

Opportunities and Challenges

In this issue, our expert contributors discuss the many challenges and opportunities in the PCB design community, and what can be done to grow the numbers of PCB designers—and design instructors. Join us; we truly need your help. How do you think we can attract more young people into the PCB design segment?

AMITRON

POWERING AMERICA'S ELECTRONICS.

- 62 PIECES OF NEW EQUIPMENT INSTALLED IN 2023
- 80K SQ/FT OF MANUFACTURING FOOTPRINT
- LEADING EDGE TECH AND ADVANCED AUTOMATION
- DOUBLED PRODUCTION CAPACITY FOR 2023

PRINTED CIRCUIT BOARD GLOBAL SUPPLY CHAIN ENGINEERING SOLUTIONS

- QUICK TURN
- LOW TO HIGH VOLUME PROJECTS
- RESEARCH AND DEVELOPMENT
- DESIGN SUPPORT

If you buy PCBs, have supply chain issues, and aren't talking to the new AMITRON you need to ask yourself, why not?

Come visit or connect with our team to learn how Amitron can help to stabilize and reduce risk in your PCB supply chain.

ITAR Registerd	MIL-PRF 31032	ISO 9001:2015	ISO 13485	AS9100	Minority Business	International Automotive Task Force	IATF 16949

MADE IN USA

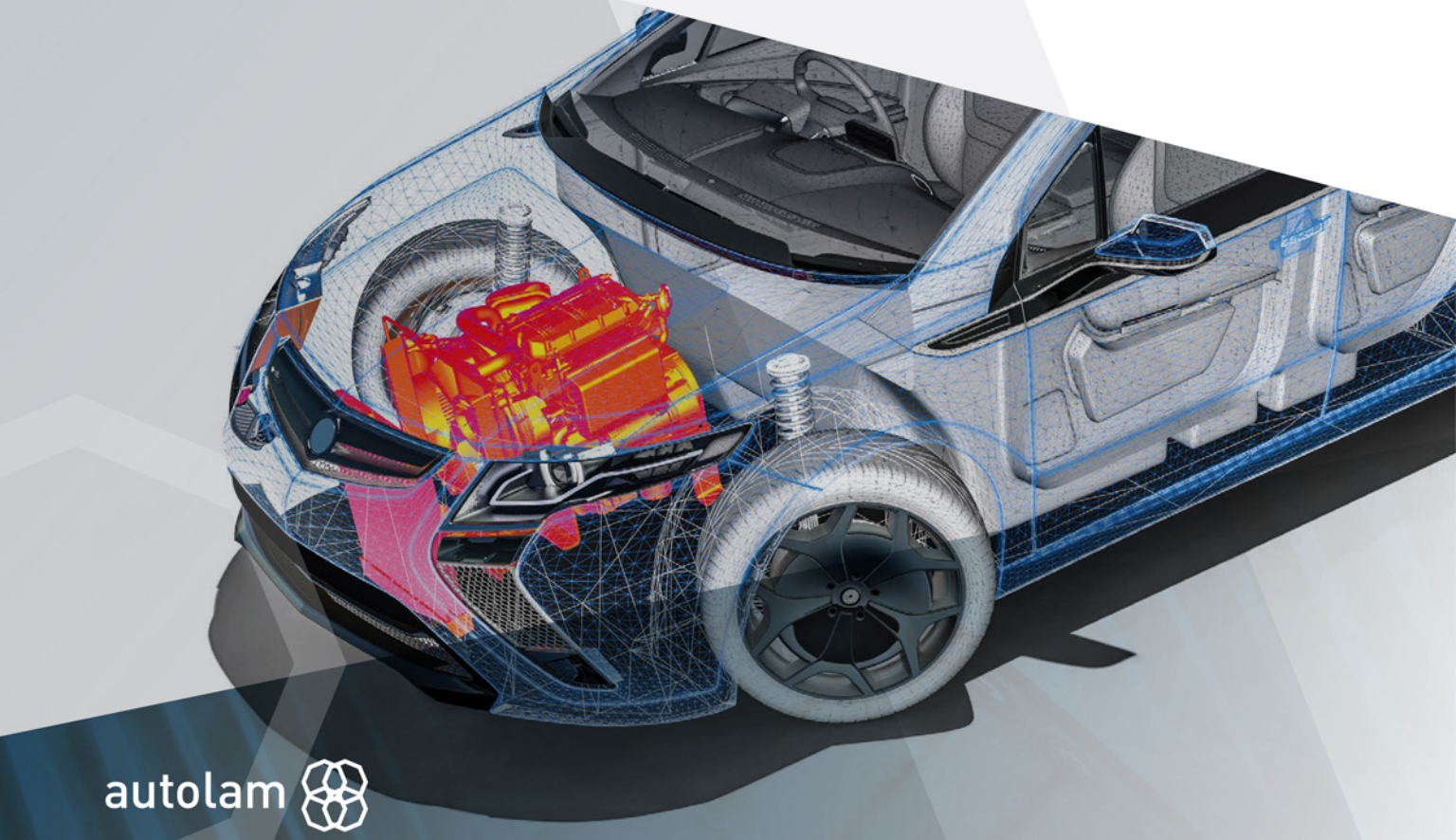
Visit us at www.amitron.com

support@amitron.com
+1 (888) 454-0679

DESIGN007 MAGAZINE

ventec
INTERNATIONAL GROUP
騰輝電子

autolam: Base-Material Solutions for Automotive Electronics



autolam

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

IATF 16949:2016 CERTIFICATION

venteclaminates.com

The Myriad Opportunities—and Challenges

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

If you look at the current state of PCB design, there are a lot of positive indicators. We have almost full employment in this great career that appeals to the techie side of the brain as well as the artistic side. You can make good money as a designer without ever seeing the inside of a college. This segment is in constant flux. It's challenging, and that's part of what makes it fun.

But there's one big problem: There aren't enough designers, and the problem is likely to get worse before it gets better.

Designers are retiring in droves, just when we need experienced folks more than ever to deal with high speeds, fast rise times, and UHDI. When they're gone, they'll be taking decades of knowledge with them.

On top of that, there aren't many young

PCB designers in the pipeline waiting to take their place. Most high school guidance counselors are still unaware that this career even exists, but we are seeing more PCB design curricula at colleges and universities. It's basically a supply-and-demand situation, but it's a serious one. What can we do to reverse this situation?

In this issue, we asked our expert contributors to discuss the many challenges and opportunities in the PCB design community, and they shared their thoughts on what can be done to grow the numbers of PCB designers and design instructors.

We kick things off with a conversation with John Watson, who looks into the causes and possible solutions to this "perfect storm" from his vantage point as a PCB design instructor. Next, Stephen Chavez breaks down the state of PCB design, and he predicts which specialties within design are likely to be in demand in the future. Bill Hargin explains what's needed to be the best design engineer you can be; much of it comes down to constantly reading industry books and white papers. Tim Haag, a former design instructor, discusses what it takes to be a great PCB design instructor. Joe Fjelstad explains what technical skills and soft skills are required to be a good PCB designer. Finally, Barry Olney sums it all up with his column, aptly titled "The Art of Presenting PCB Design Courses."

We're getting ready for IPC APEX EXPO in April. I hope to see you on the road. **DESIGN007**

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

All Flex Solutions Expands Rigid Flex Quick-turn Offering

All Flex's Rigid Flex Center of Excellence has recently expanded expedited services for their rigid flex customers. Recent capital improvements in All Flex Solutions' rigid flex manufacturing facility are helping to accelerate production lead times on rigid flex designs. All Flex Solutions recently installed and qualified an innerlayer factory allowing them to issue materials to the floor and complete innerlayers in hours instead of days. The reduced cycle time gets product into lamination faster to complete the remainder of the manufacturing cycle.

Matt Tannehill, President of the Rigid Flex Center of Excellence commented "we've always been capable of building very high quality, very high reliability rigid flex circuit boards. A couple of months ago we defined a set of criteria, that would allow us to support faster turn cycles for our partners. We know that if we help our customers prototype and de-bug their parts at the beginning, we'll likely have a partner when it comes to production volumes. This service will help us serve our customers better and collaborate with them on their initial designs."

The quick-turn service is based on capacity loading and material availability but offers designers of rigid flex printed circuits expedited services.

(Source: All Flex)

Great Opportunities and Challenges in PCB Design

Feature Interview by the I-Connect007 Editorial Team

You may have noticed that many designer friends are eyeing the exit door. Not only that, many of the best design instructors are well into their 70s—just when things are getting interesting.

Where will we find new designers and design instructors to replace the soon-to-be-retired? The I-Connect007 Editorial Team put that question to John Watson, CID, a Design007 columnist and professor at Palomar College. John teaches one of the most thorough PCB design courses, and likely the only one of its kind at a college or university.

In this interview, John shares his thoughts on what is needed to attract more people to the design field, how to become a great PCB designer, and which trends and challenges are driving PCB design right now.

Andy Shaughnessy: John, we've talked before about the stream of designers and design instructors retiring from the industry. We need experienced designers more than ever as silicon continues to shrink and signal speeds and rise times increase. It's almost a perfect storm. As a design instructor, what do you see happening?

John Watson: I'll give a plug for my design class at Palomar College in San Marcos, California. It's a 32-week program with basic and advanced PCB design. It's one of the longest courses available on PCB design in a college setting, so it's pretty exciting.

In my upcoming book, I talk about how we are quickly falling behind in providing good talent for PCB design. It's gone full circle. In 2008, we saw a definite downturn, a huge

SUMMIT INTERCONNECT

A Total PCB Solution from a Single Manufacturing Partner

Rigid. Flex. Rigid-Flex. HDI. RF. Assembly.

Summit Interconnect is the largest, privately held PCB manufacturer in North America offering a broad range of highly reliable, advanced technology PCBs. With eight high-tech facilities and thousands of employees, we work alongside customers to push the boundaries of what is possible, helping to create inventive and transformative electronic products.

Summit Interconnect proudly serves these high-growth markets:

- Aerospace & Defense
- Medical
- Industrial
- Semiconductor
- Automotive
- Space
- Consumer Electronics
- Computing & Datacom

summitinterconnect.com

The trusted technology partner for manufacturing mission-critical PCBs.

John Watson

paradigm shift in how people and companies operated. That was where it all started, and we haven't recovered. We're behind the eight ball on the available talent, and we have not caught up. We've all seen the surveys asking designers whether they plan to retire in the next 10 years, and 50% of respondents say yes.

Marcy LaRont: Many would agree with you. We've done a terrible job preparing our young people and making this an even remotely enticing career. It hasn't gotten better over time.

No, not much has changed. If anything, it's gotten worse because we've had huge advances in technology and electronic innovations, but we do not have the PCB talent to undergird all of us. This is very concerning, and a problem in our industry. People assume there's a set path or set of routes to become a printed circuit board designer.

Shaughnessy: But there's no critical path at all.

That's right. I have students coming from different segments of the industry, and even different careers. I have construction workers, and even a deli worker in this year's course. I have many students who are changing careers, different things like that. There's a belief that you must be an electrical engineer to be a PCB designer. No, you don't. An EE degree is not required.

LaRont: What trends do you notice among your students?

For one, there's been more interest in PCB design. They understand that it's a creative career, and it can pay well. As I mentioned, I have had construction workers and truck drivers decide to switch careers later in life to become PCB designers. At Palomar College, we concentrate on the internship side, where we work with companies waiting for graduates. It's a very attractive situation; they get the education they need and then step right into their new career. It's really gaining momentum. This class has been going on for nearly 25 years. I attended Palomar College myself as a brand-new designer, where I started my career.

In my book, I say that universities are primarily focused on the theoretical side, and they haven't gotten into the practical side, because you don't have time for the practical. Companies realize that even if you have EEs on staff, they may not have the practical experience to do the job. They know the theoretical, not the practical.

Shaughnessy: What is driving PCB design right now?

The industry is pushing smaller packaging, especially advanced packaging. Our footprints are so small that they can't possibly get any smaller. Now, they're building upward, stacking components on top of each other. It's a huge trend, and we don't have enough designers who are familiar with this technology.

Our industry has shifted because of the autonomous vehicle industry, and that entire segment has just taken off. As a result, we're now shifting to high power. I'm having many

conversations with companies that aren't looking for someone who can design high-speed boards. They want designers who do high-power boards—I'm talking about really high power. I'm working with a gentleman on a design that is 3,000 volts. That's huge. It's a paradigm shift in our industry, from high speed to high power. We have a need for designers who are cross-disciplined in high speed and high power. I also might want someone on the mechanical side. At Palomar College, we're expanding the focus of our classes so that we can cross-discipline some of these design students.

Shaughnessy: That's really interesting. I'm hearing more about ECAD designers learning to work with MCAD and IDF, DXF, and STEP files. Should ECAD designers be cross-trained in MCAD tools?

Sure. Designers should at least have a basic knowledge of, let's say, SolidWorks, or some 3D platform where they can work in that environment. I'm seeing more tools being developed that integrate ECAD and MCAD. Altium is integrating with SolidWorks. We're seeing more of this sort of collaboration, and the old ways of doing things are falling away.

Shaughnessy: Many EEs are now doing PCB design. Susy Webb's PCB design class can be almost all EEs. Are they the way forward?

That's actually a conversation I've had with Susy. I find that the most difficult people to teach are the EEs. They lay a trace down, and they'll see the connection. Their first question is, "Is this right?" Well, it's not whether it's right or wrong—it's a solution. They seem to have a problem thinking outside the box. I really enjoy teaching artists probably because I look at things as an artist. I'll say, "Give me some guidelines, and then I'll be free to do my own thing." If you look at the industry, many of the best designers are artists. Susy Webb was a landscape artist, and Bill Brooks, who taught at Palomar, is a world-renowned sculptor.

Shaughnessy: Artists would seem to be diametrically opposed to AI. Do you think AI will make its way into EDA tools? Maybe a designer could say, "Design me a board," and voilà, there's the data package.

AI is a big thing right now, but I don't know how much it will affect the designer's job. When I was on a panel at PCB West, I said, "We already have AI in EDA tools in autorouting. How many of you autoroute your boards? Almost no one raised their hand. Autorouting has not taken off, even with the advancements in the tools, because it pulls the designer out of that creative process. They want to be a part of creating something. Designers love the puzzle; they love getting in there and solving it. Ultimately, they want to say, "That's what I created right there."

Shaughnessy: So, how does a design instructor stay current?

I read a lot. I am constantly reading anything related to my job. I read Design007 Magazine, and everything by Dr. Eric Bogatin at the University of Colorado Boulder. As an instructor, you spend your time educating yourself on what's current and what's happening. You have to stay updated because things change constantly.

Shaughnessy: You say we need more designers; not just good designers, but great ones. What does it take to become a great designer?

I always distinguish between good designers and great designers. Good designers are a dime a dozen. You can go on LinkedIn and find a handful of good designers. But the great designers are the ones who go the extra mile. They go to conferences and constantly learn and relearn different things. They are growing as designers, and they become the great ones. They are both fab and design experts. The best designers talk to fabricators, and visit their fab houses so they can see what's going on. Visiting the board house helps you understand their job of taking your data and using it to build a successful product.

LaRont: Why isn't it compulsory for new PCB designers to visit a fab shop?

That's the million-dollar question. I would make it a requirement, just for the good of the products you're trying to build.

Shaughnessy: We appreciate your time, John. This has been great.

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

DRAM Industry Sees Nearly 30% Revenue Growth in 4Q23

TrendForce reports a 29.6% QoQ in DRAM industry revenue for 4Q23, reaching US$17.46 billion, propelled by revitalized stockpiling efforts and strategic production control by leading manufacturers. Looking ahead to 1Q24, the intent to further enhance profitability is evident, with a projected near 20% increase in DRAM contract prices—albeit with a slight decrease in shipment volumes to the traditional off-season.

Samsung led the pack with the highest revenue growth among the top manufacturers in Q4 as it jumped 50% QoQ to hit $7.95 billion, largely due to a surge in 1alpha nm DDR5 shipments, boosting server DRAM shipments by over 60%. SK hynix saw a modest 1–3% rise in shipment volumes but benefited from the pricing advantage of HBM and DDR5, especially from high-density server DRAM modules, leading to a 17–19% increase in ASP and a 20.2% rise in revenue to $5.56 billion. Micron witnessed growth in both volume and price, with a 4–6% increase in each, resulting in a more moderate revenue growth of 8.9%, totaling $3.35 billion for the quarter due to its comparatively lower share of DDR5 and HBM.

On the capacity planning front, Samsung's production bounced back in the first quarter of this year, reaching an 80% utilization rate after significantly cutting production back in 4Q23. Demand is expected to rise notably in 2H24, leading to a continuous increase in production capacity through 4Q24. SK hynix is actively expanding its HBM capacity and gradually increasing wafer starts, especially with the rollout of HBM3e's mass production. Micron is also warming up its wafer starts, aiming to boost its advanced 1beta nm process share for HBM, DDR5, and LPDDR5(X) products as it expects a convergence in capacity due to the increase in advanced process equipment.

(Source: TrendForce)

4Q23 Revenue Ranking of Branded DRAM Makers (Unit: Million USD)

Ranking	Company	Revenue			Market Share	
		4Q23	3Q23	QoQ	4Q23	3Q23
1	Samsung	7,950	5,250	51.4%	45.5%	38.9%
2	SK hynix	5,560	4,626	20.2%	31.8%	34.3%
3	Micron	3,350	3,075	8.9%	19.2%	22.8%
4	Nanya	274	244	12.1%	1.6%	1.8%
5	Winbond	133	112	19.5%	0.8%	0.8%
6	PSMC	39	19	110.0%	0.2%	0.1%
	Others	158	155	1.7%	0.9%	1.2%
	Total	17,464	13,480	29.6%	100.0%	100.0%

註：
1. 3Q23—USD:KRW = 1:1,313; USD:TWD = 1:31.7
2. 4Q23—USD:KRW = 1:1,322; USD:TWD = 1:31.8
3. PSMC's revenue calculation primarily pertains to its own production of consumer DRAM products and does not include DRAM foundry services

Source: TrendForce. Mar., 2024

My 9th birthday 1978 Montana

We love PCBs.
We always have.

We are the geeks. The nerds. The passionate. We all think, talk and dream about that little green masterpiece. We call it **Your Key Component.** Because that's what it is.

We are the NCAB Group. One of the world's leading producers of PCBs. We have been producing reliable PCBs, and only PCBs since 1993.

Reliable PCBs.
Because failure is not an option.

www.ncabgroup.com

NCAB GROUP
Integrated PCB Production

The Challenges, Opportunities, and Future Specialties of PCB Design

Feature Article by Stephen V. Chavez
SIEMENS EDA

We've witnessed a progression of the various roles in PCB design over the decades due to the ever-evolving field of electronics. These roles, which are key to PCB design, can be described as the all-encompassing domain of printed circuit engineering (PCE), which involves multiple disciplines and their respective domains, and plays a pivotal role in shaping the technology landscape. As technological advancements accelerate, the multi-disciplines within PCE face myriad challenges while simultaneously encountering exciting and promising opportunities.

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. Design engineers must learn about layout, simulation, and supply chains—and in their place new specialties have emerged, like multi-gigabit SerDes channel design, advanced manufacturing, IoT, and multi-physics system verification. Across the board (no pun intended), good engineering teams have learned to collaborate outside their siloed specialties, which increases their individual and collective value. Yet, it's not only designers grappling with this evolution but also numerous tech companies across several industries that must bolster their engineering resources to close the growing talent gap.

ME AND GERBER THE LIZARD 1986

We love PCBs.
We always have.

We are the geeks. The nerds. The passionate. We all think, talk and dream about that little green masterpiece. We call it **Your Key Component.** Because that's what it is.

We are the NCAB Group. One of the world's leading producers of PCBs. We have been producing reliable PCBs, and only PCBs since 1993.

Reliable PCBs.
Because failure is not an option.

www.ncabgroup.com

NCAB GROUP
Integrated PCB Production

The talent deficit within PCE persists, straining companies as they aim to uphold product quality, adhere to project timelines, manage budgets, and recruit new generations of designers and engineers. Securing senior-level talent is arduous, and recruiting entry-level professionals from educational institutions presents its own set of challenges. Let's delve into the current landscape of PCE to explore the challenges, identify growing opportunities, predict future in-demand specialties (disciplines), and contemplate strategies to attract young talent into this dynamic industry. It's important to note this article covers PCB design from PCB designer to a higher-level perspective of PCE.

A Closer View of the Challenges

The multitude of challenges facing PCE today include:

- **Miniaturization and component density.** With the increasing demand for smaller, more compact devices, printed circuit engineers must pack more functionality into limited space while ensuring optimal performance, thermal management, manufacturing, and supply chain.
- **High-speed design, SI, and EMC.** The proliferation of high-speed data transmission technologies like 5G and PCIe Gen 5 makes it more difficult to preserve signal integrity, minimize crosstalk, and manage impedance. These technologies require meticulous design and validation processes to ensure that electronic devices operate as intended in their target electromagnetic environment and don't interfere with other devices.
- **Complexity and integration.** As electronic devices become more feature-rich and multifunctional, printed circuit engineers must navigate intricate designs involving a multitude of components, interfaces, and functionalities, while ensuring seamless integration and interoperability.
- **Thermal management.** Efficient heat dissipation is crucial for maintaining the reliability and longevity of electronic devices. Printed circuit engineers must carefully plan the layout and placement of components to ensure proper thermal management. Thermal analysis is key to achieving success in addressing thermal management.
- **Cost and time-to-market pressure.** In a fiercely competitive market, companies are under constant pressure to reduce development costs and accelerate time-to-market, necessitating efficient design methodologies, rapid prototyping, and streamlined production processes.
- **Regulatory compliance.** Ensuring compliance with various industry standards and regulations, such as Restriction of Hazardous Substances (RoHS), adds complexity to PCB design projects.

It Also Comes With Opportunities

Along with these challenges come greater opportunities. Here are the biggest opportunities in PCE across diverse sectors of the industry:

- **Automotive industry:** The rising adoption of EVs, autonomous driving technologies, and in-vehicle connectivity necessitate more advanced PCB designs for power management, sensor integration, and communication systems.
- **5G and IoT:** The rollout of 5G networks and the proliferation of IoT devices create a greater need for printed circuit engineers

specializing in high-frequency and wireless communication technologies.

- **Aerospace and defense:** With the aerospace and defense industry witnessing rapid technological advancements that enable innovations in satellite communications, radar systems, and unmanned aerial vehicles, there is a growing demand for ruggedized, high-performance PCBs capable of withstanding extreme environments.
- **Medical devices:** The medical device industry relies heavily on PCBs for various applications, including diagnostic equipment, patient monitoring devices, and implantable medical devices. This sector continues to grow at a very high rate, calling for even more innovative and complex PCBs applied to a greater number of applications.
- **Consumer electronics:** With the unabated proliferation of smartphones, wearables, IoT devices, and smart appliances, the consumer electronics sector continues to drive innovation in PCE, including the creation of more compact, energy-efficient PCBs with enhanced connectivity and functionality.

Now a Look at Specialties

For more than a few years, several specialties in PCB design have been experiencing a shortage of or high demand for engineering talent due to various factors, such as technological advancements, industry growth, and shifts in manufacturing trends. Some of these specialties include:

- **High-speed digital design:** With the increasing demand for high-performance electronic devices, there's a growing need for printed circuit engineers skilled in designing circuits that can handle high-speed data transmissions without SI issues.
- **SI engineering:** With the proliferation of high-speed data transmission, SI engineers will play a critical role in ensuring SI, minimizing signal degradation, and mitigating EMI and EMC in PCB designs.
- **RF and microwave design:** The proliferation of wireless communication technologies, like 5G, IoT devices, and radar systems, has created a demand for printed circuit engineers proficient in RF and microwave design techniques to optimize signal performance and minimize interference.
- **Power electronics:** As the demand for energy-efficient electronic devices continues to rise, there's a need for printed circuit engineers with expertise in power electronics to design efficient power distribution systems, DC-DC converters, and voltage regulators.
- **Embedded systems and IoT:** The increasing integration of electronics into everyday objects and the growth of IoT require printed circuit engineers with expertise in microcontrollers, sensors, and wireless communication modules for the creation of compact, energy-efficient embedded systems.
- **Thermal management:** With electronic devices becoming more compact and powerful, managing heat dissipation on PCBs is crucial to ensuring reliability and longevity. Printed circuit engineers skilled in thermal management techniques are needed to design layouts that optimize heat dissipation and prevent overheating.
- **Flex and rigid-flex PCB design:** Flexible and rigid-flex PCBs are becoming increasingly popular in applications where space is limited or the PCB needs to conform to a specific shape. Printed circuit engineers with expertise in flex and rigid-flex PCB design are in high demand to create layouts that meet the mechanical and electrical requirements of these specialized boards.
- **Compliance:** The automotive and aerospace segments have stringent requirements for reliability, safety, and perfor-

mance. Printed circuit engineers with experience in these industries are in demand to design boards that can withstand harsh environmental conditions and meet industry standards and regulations.

Specialization in any of these will increase your job opportunities. It's worth noting that the demand for specific specialties can vary depending on factors such as geographic location, industry trends, and technological advancements. Therefore, it's essential for printed circuit engineers to stay updated on the latest developments in the field and continually develop their skills to remain competitive in the job market.

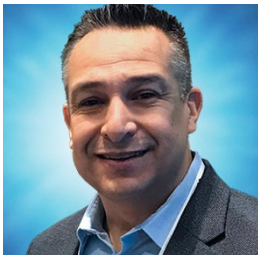
How to Attract Young Talent

With the talent gap continuing to grow, attracting young talent into the industry is no easy task. Here, we will focus on how to attract more young people into the multiple disciplines of PCE within the electronics industry. Several strategies can be employed in each:

- Collaborate with educational institutions to offer specialized courses and workshops on PCB design, providing students with hands-on experience and exposure to industry-relevant tools and technologies. Sadly, this is an area that is seriously lacking as most major colleges and universities in the U.S. don't teach PCB design nor offer PCB design as part of their engineering curriculum. EDA tool companies offer training, but this training tends to be specifically tailored to their respective tools or workflows. Yet, the fundamental concepts of PCB design are typically not taught, although there are signs this is changing.
- Establish internship programs and mentorship initiatives that provide aspiring printed circuit engineers with valuable real-world experience, guidance, and mentorship from seasoned professionals. Industry associations are a great place to investigate when seeking mentorship opportunities. PCEA is a great example of one.
- Highlight the innovative and creative aspects of PCE, showcasing how printed circuit engineers contribute to cutting-edge technologies and solve complex engineering challenges. Amplify the fact that a career in PCE has the potential to be long and fruitful with an increasing salary as engineers gain experience, develop professionally, and master the craft of PCE.
- Foster a diverse and inclusive work culture that welcomes individuals from different backgrounds and perspectives, creating opportunities for underrepresented groups to thrive and contribute to the industry's growth and innovation. A work environment and/or internal leadership along with company culture can make a positive or negative difference in bringing in and retaining talent.

Conclusion

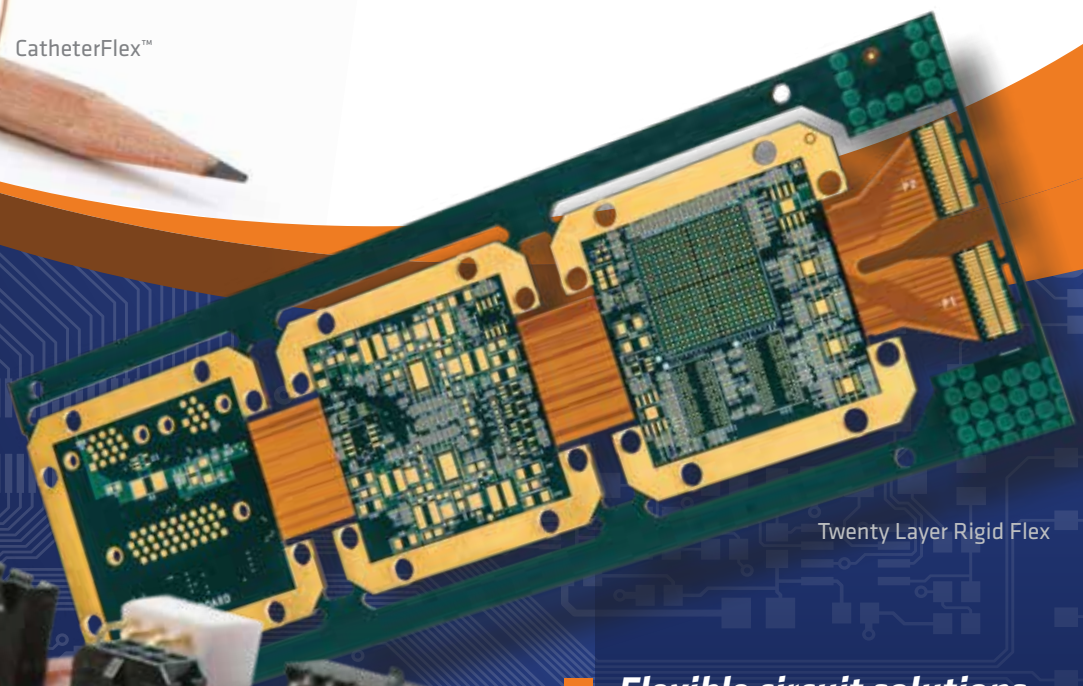
While printed circuit engineers encounter various challenges in navigating the complex and fast-paced landscape of electronics design, they also have the opportunity to shape the future of technology across diverse sectors, such as automotive, aerospace, and consumer electronics. By anticipating future in-demand specialties, fostering innovation, and attracting young talent into the industry, the PCE segment can continue to drive innovation and push the boundaries of technological advancement. **DESIGN007**



Stephen V. Chavez is principal technical product marketing manager for Siemens EDA and chairman of PCEA.

ALL YOUR FLEXIBLE SOLUTIONS IN ONE PLACE

CatheterFlex™



Twenty Layer Rigid Flex

And, Assembly Too!

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

All Flex Solutions
1601 Cannon Lane
Northfield, MN 55057

(507) 663-7162
AllFlexInc.com

all flex
SOLUTIONS

MilAero007 Highlights

After Three Years on Mars, NASA's Ingenuity Helicopter Mission Ends ▸

After its 72nd flight on Jan. 18, 2024, NASA's Ingenuity Mars Helicopter captured this color image showing the shadow of a rotor blade damaged during a rough landing. NASA's history-making Ingenuity Mars Helicopter has ended its mission at the Red Planet after surpassing expectations and making dozens more flights than planned. While the helicopter remains upright and in communication with ground controllers, imagery of its Jan. 18 flight sent to Earth this week indicates one or more of its rotor blades sustained damage during landing and it is no longer capable of flight.

NOAA's Newest Weather Satellite From Lockheed Martin Arrives in Florida to Begin Launch Preparations ▸

The next-generation Geostationary Operational Environmental Satellite (GOES)-U has successfully arrived at Kennedy Space Center, Florida, to begin preparing for its spring launch. It is the final of four satellites in the National Oceanic and Atmospheric Administration (NOAA)'s GOES-R weather satellite series.

NASA Puts Next-Gen Exoplanet-Imaging Technology to the Test ▸

The Coronagraph Instrument on NASA's Nancy Grace Roman Space Telescope will demonstrate new technologies that could vastly increase the number of planets outside our solar system (exoplanets) that scientists can directly observe. Designed and built at the agency's Jet Propulsion Laboratory in Southern California, it recently passed a series of critical tests ahead of launch.

U.S. Army Selects CACI for $382 Million Signals Intelligence and Electronic Warfare Systems Task Order ▸

CACI will provide advanced software and full life cycle support for the Trojan family of systems across the Army military intelligence enterprise at all echelons. CACI will deliver and enable cutting-edge intelligence collection, processing, exploitation, and dissemination (CPED) capabilities through dynamic data fusion and adaptable software solutions to ensure information advantage for multi-domain operations.

Airbus Boosts 'Make in India,' Awards Additional Manufacturing Contracts ▸

Demonstrating on its 'Make in India' commitment, Airbus has signed contracts with Tata Advanced Systems Limited (TASL) and Mahindra Aerospace Structures Private Limited (MASPL) to procure commercial aircraft components.

Honeywell to Invest $84 Million in Expansion of Kansas Aerospace Manufacturing Facility ▸

This expansion will create one of Honeywell's most technologically advanced aerospace manufacturing facilities. The 560,000-square-foot facility currently manufactures components for Honeywell's avionics, safety and flight control systems and complex radio frequency systems for traffic collision avoidance, radar altimeters, and weather radar.

DESIGN TIPS #124:

ETCH COMPENSATION

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

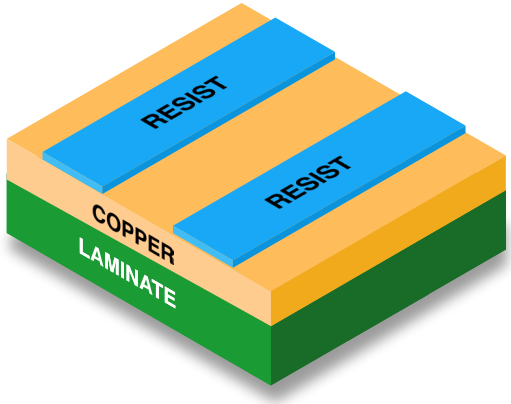
This is because we must do an etch comp on the traces in CAM to compensate for known etch loss. The space between traces after compensation will play a role in whether a board can be manufactured.

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

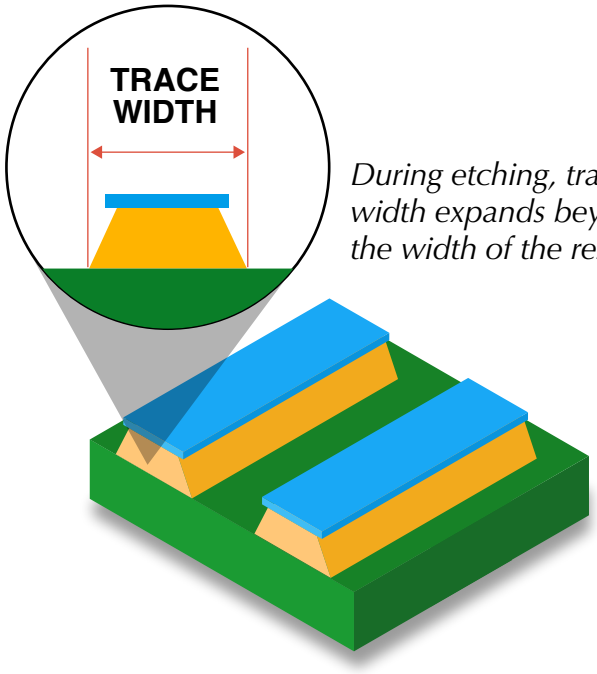
Design tips:

- 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.

Before etching



After etching



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

Prototron Circuits
America's Board Source

prototron.com (520) 745-8515

Being the Best Design Engineer

Feature Interview by Andy Shaughnessy
I-CONNECT007

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.

Andy Shaughnessy: Bill, you did a presentation here at DesignCon. Tell me about it.

Hargin: Yes, it was great—it's a popular topic, and the class was standing room only. The topic was the same as the title of the book I wrote, *The Printed Circuit Designer's Guide to… Stackups: The Design Within the Design.* It's kind of like a red pill vs. blue pill situation. You can have the blue pill and have your fabricators design your stackups, or the red pill, and you can learn to do the nitty gritty work yourself as a CAD designer or an engineer.

Generally, you want the freedom of the red pill, so that you know what's going on, I imagine.

Of course. There's a lot to learn, and there aren't college curriculums teaching stackup design. There isn't that much stackup training out there. So, I've really been leaning into researching it. If I wrote that book again today,

Candor
Circuit Boards

Innovative solutions for complex circuit board manufacturing

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

RF/Microwave PCBs

Power Electronics

High-Speed Data

Automotive Electronics

Lamination

Startups

Our technology produces what our customers dream!

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

it would be a totally different book. It's not that stackups have changed; I just keep learning. I'm researching at the same time that I'm developing software and teaching others how to do this stuff.

As Einstein said, "The more I learned about the universe, the more I realized how much I don't know." It sounds weird to say that about stackups, but it's a little like that. What I find is that the engineering hardware design part of the world is pretty interested in learning how to do a better job with the backbone of the PCB, the stackup.

As you say, the fabricator will do the stackup, but it benefits everybody if the person at the front end understands more about it. But stackups seem to give people such a hard time.

The problem is, let's say you have a master's degree in electrical engineering. You're a pretty smart guy. You've been trained on electromagnetics, and all the circuit theory that somebody could want. You might even be able to recite Maxwell's equations. But none of that helps you do a stackup, because the stackup is largely mechanical, with electrical properties. So, how do you learn how to do that? You need training, and having a good tool helps. I'm not trying to just sell my software like an infomercial. But the person who thought of this, the person who developed the class, and the person who developed the software, it's all the same guy. So, I can't separate those three things.

I remember talking to you when you started your company. You were personally invested in this business.

When I first thought of doing this, I was the HyperLynx marketing guy. The Isola director of marketing at that time approached me and said, "Hey, I'd like to get our library into your software." And I thought, yeah, that'd be a good idea. But we never really had time to do it. When I left Mentor and went to work in the laminate space, I took all the things I knew about signal integrity and added the fabrication and laminate side of the world to it. They all converged in the software and in the training that I do.

You were saying that the stackup is the manifestation or physical representation of what's going on in the electrical world: The real vs. the ideal.

That's right. Signal integrity is about the real vs. the ideal. That's in the book too. There's an ideal signal vs. a real signal, and the difference between the two is the physical world. I've created a spin on a Sun Tzu quote that I say at the beginning of all my training sessions. He said, "Keep your friends close, and keep your enemies closer." I say, "Keep your signals close and keep manufacturing closer." Your friends are your signals. The enemy of your signals is the mechanical world: fabrication. So, keep the mechanical world even closer. Learn about it and you'll be much better off as a design or hardware engineer. Your signals will be better off, and you will be better off.

Signal integrity is about the real vs. the ideal.

What does it take to become a great designer or design engineer?

First, ask yourself if you want to be great or just normal. Think about the normal distribution. I think 67% are within one sigma of the average. Roughly half are below average. "Great" might be defined as the top 10%. If you want to be great, you need to be inquisitive. Ask questions. Be willing to roll up your sleeves and do some research. Don't just do what you're told to do. Go beyond what you're required to do, and always be learning. I have a planned back-

log of things I'm reading because I want to know about stuff. I'm curious. People say that if you want to get something done, give it to a busy person because that's someone who has developed the muscle of doing. I don't have to be the biggest genius in the room, but I will win by sheer effort. Edison, in fact, said, "Genius is 1% inspiration and 99% perspiration."

Is every designer capable of being great?

Within reason. We all start with a certain hand we've been dealt. My point is more tied to what you might call "the pursuit of greatness." In addition to reading, I try to find people who are smarter than me about a subject. I track them down, and most people are nice enough to answer questions about something they have expertise in. I have a list in my mind of experts on certain topics. One really smart laminate guy told me, "You never ask easy questions." And I thought, "That's good. I like that I'm not asking easy questions."

There is an analogy I use in my class: There's a breed of caterpillar called the pine tree caterpillar. They're also called processional caterpillars. The ultimate goal, of course, is that they turn into butterflies. And if you put them in a row on top of a flowerpot, they will go around that circle following the one in front of them until they all starve. And, to me, that's an analogy for doing the bare minimum to "stay in line." Connecting it back to stackups and PCB design, they're just focused on the X-Y direction; if they focused on the Z direction, they could get off that flowerpot and go find a leaf to eat and survive. It's what I close my workshops with: "Don't be a pine tree caterpillar. Don't just follow what everyone else is doing. Be different. Be ambitious."

When I was a Boeing engineer, they hosted free classes after work, focused on software mainly, but also hardware. I took almost all of them. But I never once saw any of my co-workers in the classes. They got the hell out of dodge after work. There's always something to learn if you're curious. Right now, I want to know more about vias, power integrity, thermal effects, and rigid-flex design. So I read, research, and find people that are smarter than me on these subjects.

Bill Hargin

Being curious seems to be a key requirement to be a good designer.

Curiosity killed the cat, but it never killed an engineer.

I may steal that line. Do you have any final words of advice for new designers and design engineers?

Be inquisitive and curious. Find people with expertise and ask questions. Most of those guys are happy to teach you. We all want to leave a legacy behind. If you're an older guy and you can find younger, hungry, curious people, that's a good thing. If you're one of those young people, find those older people, and don't be afraid to ask them questions.

Thanks for speaking with me, Bill.

Thank you, Andy. **DESIGN007**

Designing for Reality: Prioritizing Manufacturability

Connect the Dots

by **Matt Stevenson,** SUNSTONE CIRCUITS

Realistic PCB 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.

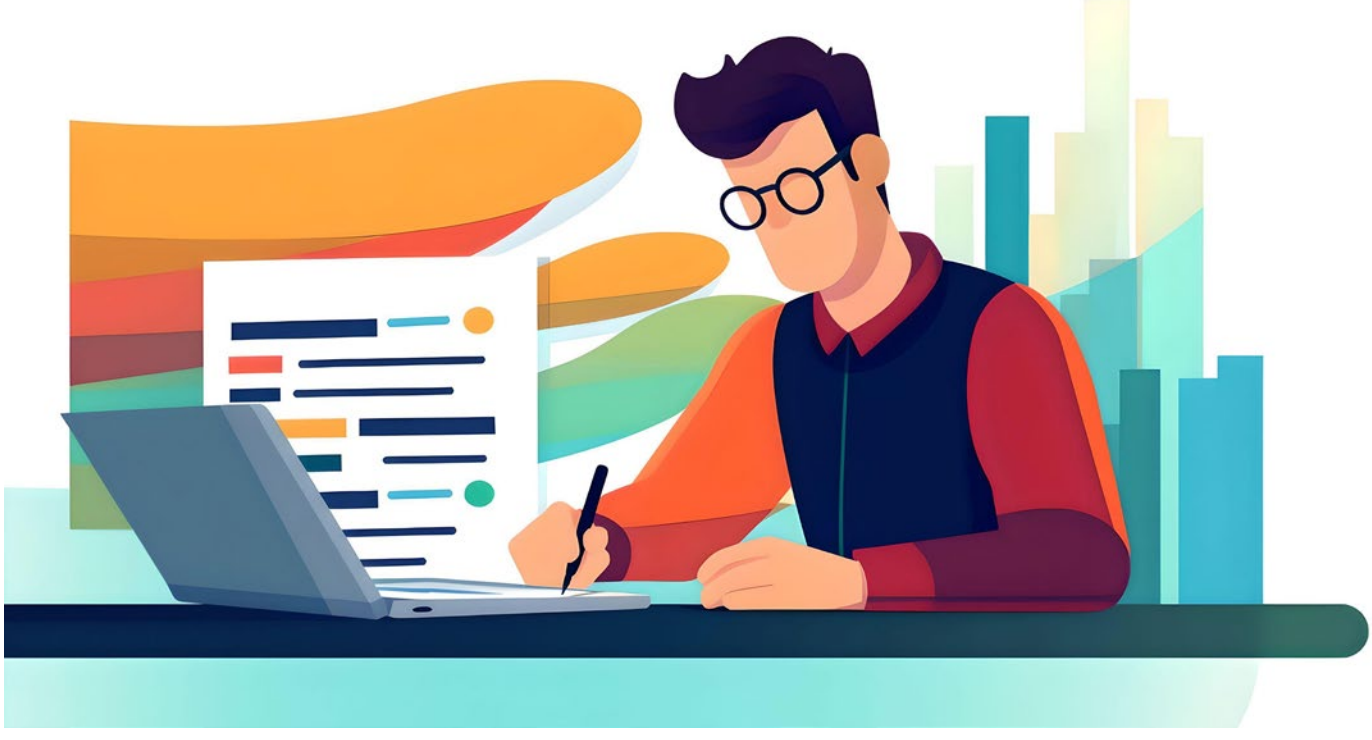
Understanding and creating robust PCB designs, especially in terms of board manufacturing, requires a lot of attention to detail. When more detail is included in the design, the manufacturing process goes more smoothly, and process yields are higher.

Product development professionals with limited PCB design experience get tasked with designs. Not immersed in the PCB design discipline, they don't know all the unwritten rules, best practices, techniques, and design requirements.

I recently sat down with Nolan Johnson for the first in a series of discussions 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.

PCB Manufacturing 101

PCB manufacturing is the physical manifestation of a digital design, and to design manufacturable boards, it's important to understand



BENDING THE POSSIBILITIES

FLEXFINER

BY TAIYO

AVAILABLE IN AMBER

AMBER

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

OUR BIGGEST FLEX YET!

Contact your local sales representative today!

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

TAIYO AMERICA

Follow us on social media

how they are produced. At a high level, these are the steps associated with PCB manufacturing:

- The manufacturer receives a design, and the computer-aided manufacturing (CAM) tooling department translates it into information usable during each step in the manufacturing process. This is where potential production issues are identified and corrected.
- Digital design files are transferred onto a physical copper-clad manufacturing panel, often using a high-intensity UV light source and a light-sensitive polymer (photoresist). The panel is the Rosetta Stone for production: defining traces, pads, and through-holes.
- Next, copper is added to through-holes and external circuitry, increasing the thickness and ensuring the electrical connection of layers and components.
- The photoresist and the exposed and unnecessary copper which lies underneath it are removed.
- The etching process then reveals the final printed board layout and makes it easier to visualize the finished board.
- Solder mask is applied to protect most of the board from oxidation, define solderable components, and prevent solder-bridging between them during assembly.
- The silkscreen, also called the legend or nomenclature, is applied. This is the process of printing specified labels, markers, and part reference designators onto the board that helps with component orientation, placement, and polarity during assembly.
- Surface finish is added to the remaining exposed copper, bringing the board much closer to its finished form.
- Finally, the PCBs are removed from the manufacturing panel, prepped for final inspection, and then prepared for shipment.

I encourage designers to tour the facilities where their boards are produced to get a better sense of how these processes are completed by manufacturers. For a deeper dive into the PCB manufacturing process, please refer to my book Designing for Reality.

Choosing the Right CAD Tools

Every profession has its tools, and the design phase begins by choosing one. Carpenters use hammers, doctors have scalpels, and tarot card readers draw from a deck of 78. For PCB designers, the tool of the trade is CAD software.

The market offers a wide range of PCB design software options and choosing which to use can become overwhelming. Your budget will likely influence your decision. CAD tools range from simple and free to complex and expensive to everything in between:

- There are many low-cost and free CAD tools available but be sure such tools are still being actively maintained and provide user support.
- Some low-cost and free CAD tools are funded and developed by PCB manufacturers. They produce tools that specifically work with their manufacturing process, which can be great if a designer has a specific manufacturer in mind for their design.
- There are also CAD tools available that charge a monthly subscription price. Most offer prices that range from about $75 all the way up to over $2,000. Pay attention to the limits placed on products with multiple tiers of pricing when choosing this option.

Regardless of budget requirements, designers should focus on software providers that are best able to support the transition from design to manufacture.

Look for design tools that will:

- Automate most or all of the prototype quote and ordering process

- Confirm the design integrity of your prototype and provide status reports
- Perform design rule checks (DRCs) as you design
- Offer unlimited customer support both via email and phone

With regard to features, stay focused on the basics. When evaluating design tools, you should first look for the following attributes:

- Be easy to learn
- Have an intuitive user interface (UI)
- Possess features key to your design needs
- Include access to an expansive parts library
- Be in widespread use
- Generates Gerber files usable by most manufacturers
- Integrates smoothly with your manufacturing process

Different tools will have different sweet spots with regard to the production process. Designers should learn and have more than one CAD tool in their toolbox to give them more flexibility from design to design. The selection process may involve some trial and error, but once you find tools that fit your needs, designing for reality will become easier.

Anticipating Variations in the Manufacturing Process

Though the basics of production are universal, variations in the manufacturing process are inevitable. The sources of variation during production are found in raw materials, equipment, human actions, environment, and method.

The raw materials required in a manufacturing process most often come in the form of a manufactured material or ore. There will inevitably be slight inconsistencies in the materials. This can have a cascading effect, creating variations during PCB production and potentially impacting board performance.

No two pieces of equipment operate identically in perpetuity, even if they are the same make and model. Over time, every piece of equipment will produce variation due to wear and tear, frequency of use, and quality of maintenance. Shop floors are filled with equipment from a wide array of eras and vendors, and every unique combination of machines will produce its own always-evolving output variations. Even using the same PCB manufacturer for every design will not eliminate variation. Given enough production runs, that manufacturer will inevitably produce slightly different results.

The human factor is arguably the biggest source of variability in any manufacturing process. Every person is different and reacts differently to stimuli. Each day presents manufacturing personnel with its own production variables. Many facilities run multiple shifts, so different shifts will have slightly different takes on process, and as a result every person is going to do the same task in slightly different ways that are most comfortable in the conditions of the moment.

The human factor is arguably the biggest source of variability in any manufacturing process.

Environmental factors can cause variation in a manufacturing process, impacting the process itself or the incoming raw materials. Heat and humidity are the most common environmental factors that can alter the PCB manufacturing process. Changes in air quality like volume of particulates (debris and vapors) can all produce variations in production output.

PCB manufacturers should always be looking for ways to improve, and when they do, the resulting changes to the steps or method of a manufacturing process can create variation.

Changing the order of production or the time allotted for a task creates variation. Changes to temperature in the facility or concentration levels in chemical processes also might affect the output.

Bringing it All Together for Ongoing Improvement

I encourage you to visit your preferred manufacturer to get a better feel for how they make boards and where variations are most likely to occur. With a good working knowledge of how the manufacturing process can drift and vary, potential issues can be anticipated and addressed in the design phase.

Understanding the fabrication process will help you align the PCB manufacturer's capabilities with the design requirements. Collaborating with them before creating the design can help ensure that elements like component placement are optimized for manufacturability. This can save time and effort and help realize a more manufacturable design.

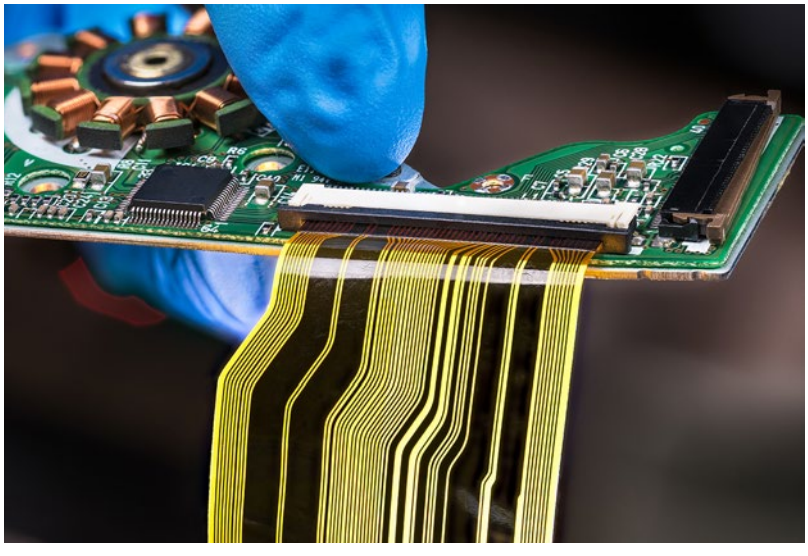
There is so much more to explore on the subject of designing for reality. I hope you will listen to our Episode 2 that dives into electronics pre-manufacturing processes. **DESIGN007**

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. You can view other titles in the I-007eBooks library.

Design Guidelines for Flexible Printed Circuits

An important but sometimes overlooked aspect of flex and rigid-flex fabrication and assembly is the flex circuit tail, which is attached to a rigid PCB with pressure-sensitive conductive adhesives. This subassembly is becoming very common. We often see this applied to glass displays and microelectronic applications.

But this method of attachment without connectors is not as straightforward as one may think. The wide varieties of circuitry to be bonded create many attachment challenges.



The best track configuration is not always possible, so one must be familiar with the optimum layouts as well as what is not recommended when considering these trade-offs. Less than optimum trace and pad layouts result in the need for more customization and smaller processing window variations for the bonding process.

Here are some common configurations that flex assemblers have developed, and the preferred approach for the method of bonding.

Bonding imbalance results from either bad co-planarity between the bonder head and stage or a thickness deviation of the flex and PCB materials. The interposer will absorb these discrepancies to some extent and help bring the bonding results in parallel.

If bonding is imbalanced, the developed color will also be uneven; adjust the bonder tool or stage until you achieve uniformity of color. Since Prescale will turn color with heat and pressure, confirm the color in short bonding time and low pressure. To confirm the condition of equipment, please do not use cushion materials.

Continue reading in the February 2024 issue of *Design007 Magazine*.

PODCAST SERIES - SEASON 2

Designing for Reality

with Matt Stevenson, ASC Sunstone

ON THE LINE WITH... I-Connect007

Season 2 of this popular podcast systematically details the interrelationships between design, fabrication, yields, and cost optimization.

Listen on Spotify

LISTEN NOW!

PCBflow Helps Designers Choose Best Manufacturer for the Job

Interview by Andy Shaughnessy
I-CONNECT007

IPC recently partnered with Siemens for the development of IPC DFM profiles in PCBflow, a cloud-based platform that organizes manufacturers' capabilities into DFM profiles, which helps designers ensure that they comply with their manufacturer's capabilities and pick the right manufacturer for each job.

I recently spoke with a few technologists who have first-hand experience with PCBflow: Susan Kayesar, technical product manager with Siemens; Evgeny Makhline, CTO of Nistec, a CEM based in Israel; and Peter Tranitz, senior director of technology solutions and leader of the IPC Design Initiative. They explain how PCBflow functions, from the designer's and manufacturer's viewpoint, and how this database helps break down the wall between these stakeholders.

Andy Shaughnessy: Susan, give us a quick background on PCBflow and the DFM profiles.

Susan Kayesar: Sure. PCBflow is an online platform and collaboration space for designers and manufacturers in the electronics industry. We enable transparent transfer of information between designers and manufacturers, which is something that is sorely missing in the industry. At a very functional level, we allow designers to upload their designs into a secure environment online and grab sets of pre-made DFM profiles. Basically, these are rules that represent either the constraints that they're looking to comply with or the actual manufacturing environment. To date, we have many suppliers of DFM profiles that are a representation of the real-life manufacturing environ-


We are dedicated
to excellence through
innovation,
technology
and most importantly,
service.
COMMERCIAL • MILITARY • MEDICAL • BAREBOARD
U.S.C.
U.S. CIRCUIT
www.uscircuit.com
ISO
9001 CERTIFIED
ITAR

ment. You can test your design for compliance with those rules, and you can communicate freely with the manufacturer that has provided these DFM profiles.

Our friends at IPC see this platform as critical to their objective of promoting the transparent transfer of necessary information. IPC came on board and provided nine DFM profiles, which enable our users to test for compliance with various aspects of IPC design standards. Evgeny Makhline is one of our "power users," and he can speak a little about how Nistec has been using those profiles.

Evgeny, walk us through how you use these DFM profiles.

Evgeny Makhline: At Nistec, we provide services for private companies looking for different levels of PCB design, including medicine, military, communication, and automotive. In one day we might check five or six different layouts, and PCBflow allows us to check every layout according to IPC standards: IPC class 2 or class 3, for example. We can also check according to the different manufacturers' specific profiles or according to our own profile, which I built and defined separately.

We are able to get results in five minutes, and that is very important to us. I can observe the results and be sure that we can send our files to manufacturing in exact compliance vs. having just a "good enough" estimate. We save a lot of my time, so we can move faster and end up with better quality. PCBflow also has a friendly GUI. It's very useful.

So, there's not much of a learning curve?

Makhline: No, not at all. The GUI is very simple and understandable, and everyone from our office, for example, can open the results reports and see reports for every issue found during checking. With the results report, we can see each issue or error, and then fix them.

The data doesn't have to be in any specific format, correct?

Kayesar: Yes, it's format-agnostic. When you come into the system, you could be creating your design in any EDA tool, with any data format—Gerber, ODB++, or IPC-2581. Even if you don't own an EDA tool, you can still investigate the results of the DFM analysis in the PCBflow online viewer, and it will actually point you to the exact XY location of the issue. So, if you have access to any CAD tool, you can directly jump to those results and change your design to comply.

Makhline: Right. This is an independent system; this means that I can skip some rules, then check to see if we're missing something during layout and send the ODB++ files for checking. Because it's an independent system, we can have different profiles and rules disconnected from the physical layout to get our own independent results. This is very important for us.

Designers say that some fabricators don't want to share their capabilities, but this lets manufacturers put their capabilities right in front of the designer.

Kayesar: Yes. Each manufacturer is a little bit unique regarding what's important to them, and what design challenges they see repeatedly, so they can put the emphasis where they want it. But the IPC profiles are enabling both the designer and manufacturers as well. Manufacturers can benefit from those IPC profiles.

It's not just the designer who needs to check their design for compliance; it could be an EMS company using this before they send a design off to fabrication. It's benefiting both sides of the equation in this case.

Do designers and manufacturers have to pay for this?

Kayesar: There is a 30-day free trial when starting. Beyond that, we are all about making information available. Manufacturers can upload and maintain their profiles free of charge. For DFM, this is a consumption-based, pay-per-use model. Designers, or anyone who wants to employ the DFM checks, can run the analysis on a pay-per-use basis. This opens up a lot of possibilities for small- and medium-sized businesses. This sort of thing was completely out of reach for them, but now they can consume the service on an as-needed basis, and it's a much more budget-friendly option.

That's good. How many manufacturers are involved in this?

Kayesar: There are hundreds of companies and dozens of manufacturers already involved directly in the platform, and you can certainly log in and look at the network to get a good idea of who's involved in it. Not everybody who joins wants to share their presence. We have a public network, and you have to submit your company to the network if you want to enable others to see you. So, we enable our customers to maintain their privacy and the privacy of their data, obviously, to any level that they want.

Does this include assembly providers and fabricators?

Kayesar: Yes, it's open to both. We support analysis for fabrication and assembly. The IPC profiles include information on both fabrication and assembly processes, so the standards that are being set may look at solderability, or pin-to-pad compliance, for example. The IPC-7351 standard is represented there. IPC did market research with their preferred suppliers and came up with trace width and spacing constraints that represent typical values in the industry and allow you to check for compliance with manufacturability standards. PCBflow allows you to organize all the accompanying files that you might need within a project, and these files can be passed between the designer and the manufacturer freely if they want to.

Peter, this sounds like a great partnership with Siemens.

Peter Tranitz: Yes, we're very happy with this collaboration. IPC and Siemens joined forces in mid-2023. We had meetings with experts from IPC like Kris Moyer, Patrick Crawford, Susan, Paul Carpine, and some other people from Siemens. Out of that joint initiative, we have set up these DFM profiles and put them into the PCBflow environment. We began marketing this campaign in January 2024. IPC already has approximately 30 to 40 companies connected in our network, with a very global footprint. We have a couple of very intensive users, and Nistec is one of them.

Will you all be discussing this at IPC APEX EXPO?

Tranitz: Yes. On Wednesday, April 10, we're hosting a Designer's Community Town Hall, and I will introduce IPC's design initiative, including PCBflow. We want to share this news because we believe it will help designers and manufacturers. For more information, check out our landing page.

I look forward to seeing you all at the show. Thanks for speaking with me, folks.

Kayesar: Thank you, Andy. **DESIGN007**

Heavy Copper PCBs: Bridging the Gap Between Design and Fabrication, Part 1

Article by Yash Sutariya
SATURN ELECTRONICS CORPORATION

They call me Sparky. This is due to my talent for getting shocked by a variety of voltages and because I cannot seem to keep my hands out of power control cabinets. While I do not have the time to throw the knife switch to the off position, that doesn't stop me from sticking screwdrivers into the fuse boxes. In all honesty, I'm lucky to be alive. Fortunately, I also have a talent for building high-voltage heavy copper circuit boards. Since this is where I spend most of my time, I can guide you through some potential design for manufacturability (DFM) hazards you may encounter with heavy copper design.

I have always perceived the term "heavy copper" as a printed circuit board with layers containing at least three ounces of finished copper. Recently, I've noticed a peculiar trend. We see monikers being used to describe higher copper weights (think 10-ounce, 20-ounce) that are seemingly pulled right out of a '90s-era Mountain Dew commercial: "extreme copper" or "super copper." I'm trying to coin the term "stupid crazy copper," but somehow, I don't think it will stick.

These are all fabricated descriptions to highlight achievements in building PCBs beyond our industry's standards. There is no standard definition for heavy copper from IPC, but that doesn't mean we can't have fun designing and building these beasts of the PCB world.

No matter what you call them or how you define them, we can all agree on why we use them. Heavy copper boards increase current carrying capacity and decrease circuit failures by reducing thermal resistance. To reduce layer count, some will contain more copper on the inner layers than the outer; moreover, some will even have different amounts of copper on the same layer. They can also function as a heat sink thanks to plated vias and their capacity to carry a higher current and heat through

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®

the board. Above all, they have eliminated the need for those complex wired buss configurations that C-3PO was forced to endure.

The purpose of this article is to answer the most frequently asked questions that designers have concerning heavy copper PCBs. By the same token, we seek to stoke the designer's imagination in creating heavy copper designs. Additionally, this article will cover the fabrication pitfalls as well as define the design solutions and manufacturing methods used in Heavy Cu builds.

Heavy Copper PCB Design Issues

Product designers are often flustered by certain DFM rules pertaining to heavy copper fabrication. I have listed the most common issues in Table 1.

Table 1

Heavy Cu PCB Design issues
1. Stackup dielectric thicknesses too thin
2. Feature-to-Feature spacing too small
3. Via diameters too small
4. Supplier UL Certifications only up to 2-3 oz Cu

Note: Failure to address these issues prior to completing your design can result in considerable rework.

Manufacturing Pitfalls

The best way to avoid design problems is to examine the ensuing manufacturing pitfalls of each feature (Figure 1).

Design issue: Stackup dielectric thicknesses too thin

Manufacturing pitfall: Delamination/resin voiding

Designers frequently call out core and prepreg placement. Ideally, the designer should stipulate minimum dielectric thicknesses and trust the fabricator to decide where a core or

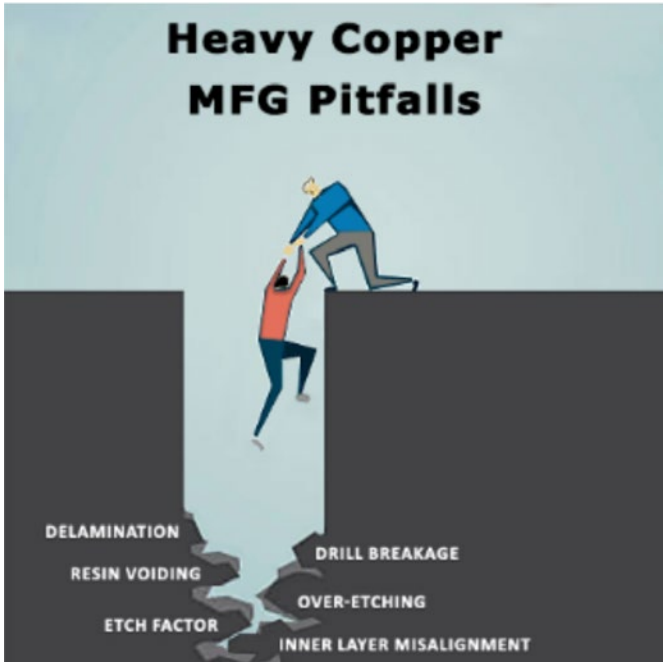


Figure 1: Heavy copper manufacturing pitfalls.

prepreg will be used. A common rule of thumb for electrical insulation is 750–1,000 volts DC per 1 mil (0.001"/25 micron) of FR-4 material. For example, if the designer requires 5,000 volts DC electrical insulation, then the manufacturer should use a minimum of 0.005"–0.007" dielectric thickness in their stackup design. Calling out your bare needs allows the fabricator to satisfy your design requirements, avoid manufacturing pitfalls, and ensure reliable builds.

Stackups are the most overlooked element of heavy copper design. Standard PCB designs are simple: Design your board with as many layers as necessary and inform the fabricator of the maximum allowable thickness. Generally, we can accommodate. However, in regard to heavy copper, we must consider the resin-filling capability of the prepreg.

A multilayer PCB is made up of two primary components: cores and prepregs. Core material is copper foil pre-bonded to each side of the fully-cured fiberglass epoxy sheets. These make up the inner layer circuit pairs. For example, the foil construction of the inner layer pairs would be Layer 2-Layer 3, Layer 4-Layer 5, etc. Yet, prepreg is just the fiberglass epoxy sheet used to make a core—merely a sheet of woven fiber-

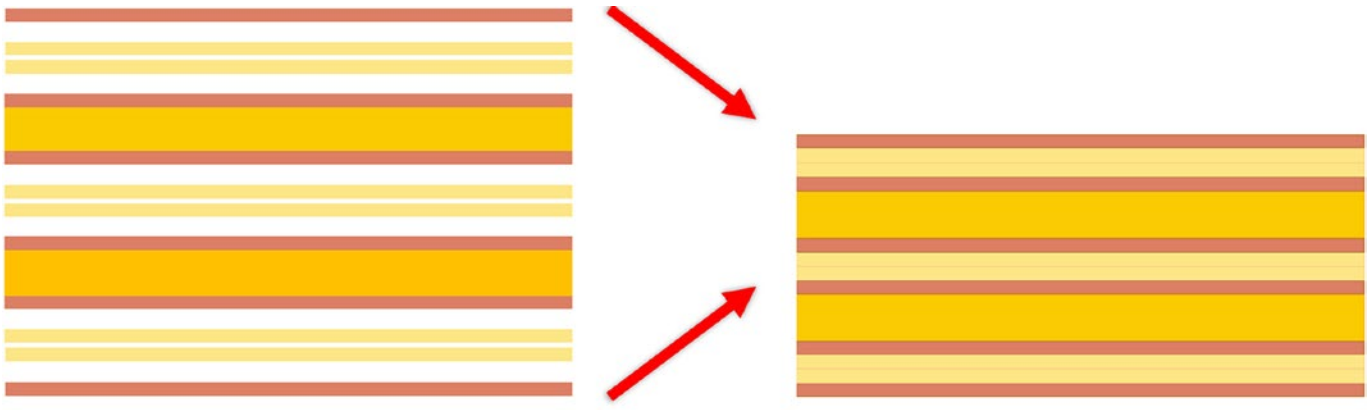


Figure 2: Material layup shown on the left is pressed into a single package with multilayer lamination.

glass yarn surrounded by epoxy. Be that as it may, a closer inspection reveals that the epoxy is not fully cured. We refer to this as "B-Stage" while the fully-cured core is referred to as "C-Stage."

Prepreg is available in a variety of glass thicknesses and epoxy resin content. Generally, the thicker the glass, the more you reduce your epoxy content in proportion to the overall thickness. Prepreg is the "glue" that holds the individual copper cores together. Hence, we align the inner layer cores and place the most applicable prepreg between them. Next, we insert these materials between heavy steel plates before placing the stack into a multilayer press. The press draws a vacuum before heating up to a controlled temperature, which liquefies the resin in the prepreg. This resin then flows in between the etched inner layer copper features on the opposing cores until achieving final cure.

Once cured, the epoxy becomes rigid, matching the "C-stage" state of the inner layer cores. Your chief concern is the amount of free epoxy required to displace the inner layer copper portions that were etched away. In conclusion, the heavier the copper, the greater the Z-axis fill requirement (i.e., thicker dielectric spacing in stackup).

The fabricator's prime directive is selecting the prepreg quantity and style accordingly to insure proper resin fill of opposing copper. For instance, a 6-ounce design will have etched recesses of 8.4 mils (in the extreme situation where an etched area opposes an etched area, there are 16.8 mils of void that require prepreg resin filling). Indeed, we have been known to push our fair share of boundaries.

Table 2: Standard readily available copper weights.

Copper oz.	Thickness (mils)
1/2	0.7
1	1.4
2	2.8
3	4.2
4	5.6
5	7.0
6	8.4

Table 3: Common FR-4 based resin system prepregs.

Preg style	Resin % content	Preg thickness	Glass thickness
106	75%	0.0023	0.0014
1080	65%	0.0030	0.0025
2113	57%	0.0039	0.0029
2116	56%	0.0051	0.0038
7628	42%	0.0069	0.0065

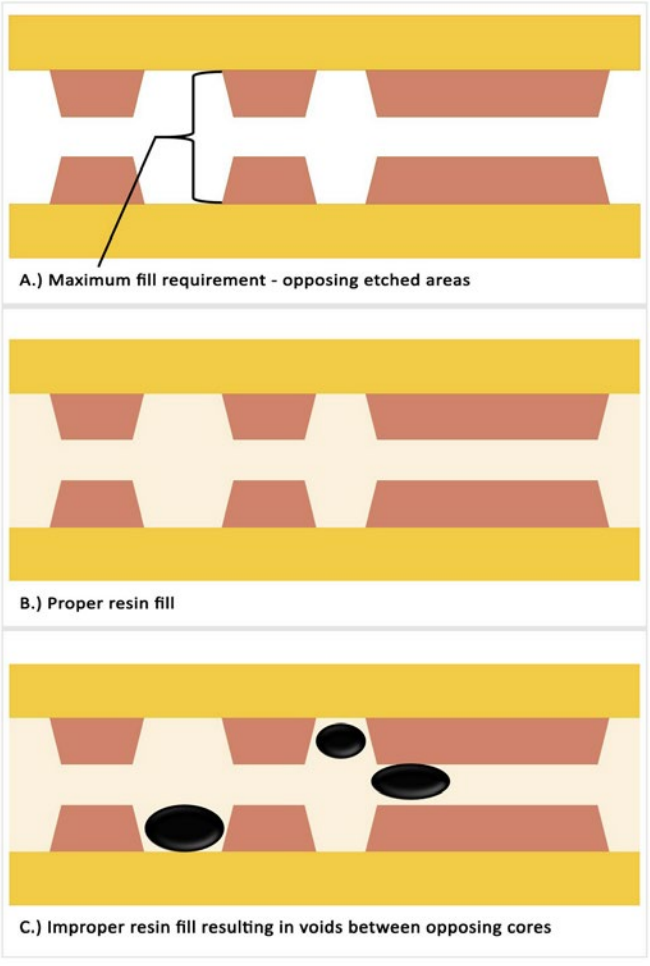


Figure 3: Maximum fill requirement.

Consequently, we have seen numerous jobs culminate in resin-voiding. It is for this reason that we have established stackup testing procedures as well as thickness calculators instrumental to realizing maximum thickness allowance and properly filling etched spaces with resin during pressing (lamination). Since each design is unique and there is no standard formula for creating heavy copper multilayer PCB stackups, collaboration between fabrication and design is imperative.

Design issue: Stackup dielectric thicknesses too thin

Manufacturing pitfall: Inner layer misalignment

Meeting the designer's maximum overall thickness requirements while still ensuring proper resin fill is another major contention between fabrication and design. In contrast, there is no dispute between copper weight and dielectric spacing as they are proportionally integral to one another (the heavier the copper, the more dielectric spacing required). For this reason, fabricators must assign sufficient prepreg plies to achieve the necessary resin volume, and the easiest way to compensate is by reducing the core material's thickness.

Figure 4: Resin voiding.

Material handling issues are primarily generated from older horizontal conveyorized lines not equipped for thinner materials. Materials are destroyed when dropped into the chemical sumps or wrapped around conveyors, so shops counter this by attaching them to thicker panels that tow them down the line. Obviously, this extra step affects lead times and cost.

Table 4

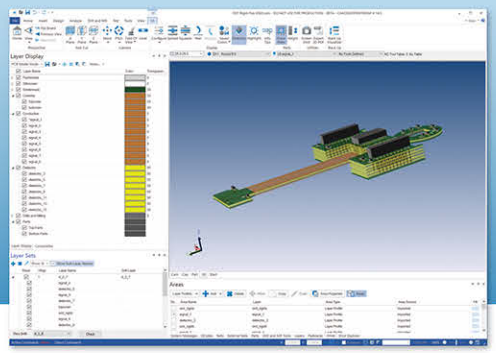
Fabrication concerns with thinner materials
1. Materials handling
2. Core-to-core alignment
3. Materials shrinkage

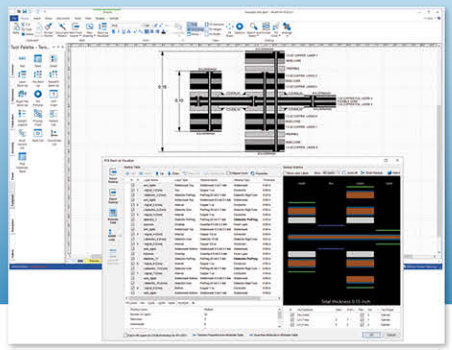
Support For Flex, Rigid Flex and Embedded Component Designs Now Available.

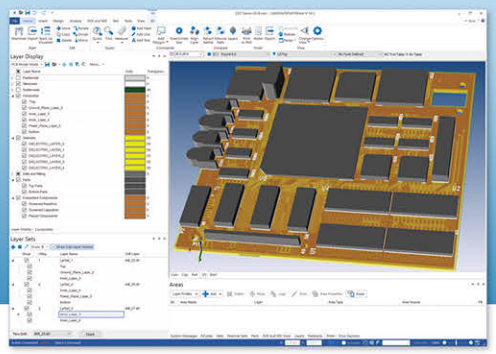
BluePrint-PCB®

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 5: Common design resolutions.

Common design resolutions
1. Increasing distance between inner layer copper features and edges of PCB (external perimeter, internal cutouts).
2. Enlarging internal capture pads for vias.
3. Modifying fabrication notes to remove any tight callouts for layer-to-layer alignment.

Core-to-core alignment can also be a challenge with thinner cores since heavy copper PCBs are more prone to registration issues that manifest in exposed copper in the winding cutouts as well as at the edge of the board. Most fabricators use fixed pins to align pre-punched cores to match these pin locations. Heavy copper on thin cores faces the dual threat of handling difficulty and a thin dielectric around the punch that often results in tearing, which, in turn, produces alignment issues.

Design Solution

The most cost-effective method is to create greater allowances for layer-to-layer registration.

Fabrication Solution

Hope remains if these design considerations cannot be met. Fabricators can enhance alignment by using either an indexed pin setup or auto-alignment lamination machines, which do not require the use of pins. Thin core materials require varying degrees of scale factors to account for material shrinkage during lamination. Both copper weight and circuit pattern affect shrinkage so experienced fabricators counter this through data collection. X-ray drills equipped with high-resolution cameras measure each layer and collect material shrinkage data to optimize future production runs. After collecting the data, the X-ray drills fresh tooling holes that optimize for layer-to-layer alignment, shift, and rotation to ensure drilled components and via holes are within capture pads.

Figure 5: Material shrinkage.

Design issue: Feature-to-feature spacing too small

Manufacturing pitfall: Over-etching (anti-puddling)

Small features and spaces (within acceptable limits) are a burden to be borne by the fabricator alone. The focus would need to be on the actual etching process whereby circuit features are created. In this process, etching chemistry is sprayed from above and below the panel during this horizontal process. While the chemistry sprayed from beneath the panel falls away because of gravity, the chemistry sprayed from above remains on the PCB in the form of a puddle, obstructing the surface and absorbing the force of the ensuing spray. Since etching is as much a function of physically hitting the copper as it is a chemical reaction, the etch rate of the top side is thereby reduced. As a result, the operator must reduce line speed to properly etch the top side which, in turn, over-etches the bottom. It goes without saying that etching heavy copper features will only exacerbate this predicament.

Figure 6

HDI requirements have triggered recent advancements in outer and inner layer etching lines—most notably with regard to vacuum etching. Vacuum etching uses suction units between the spray bars to extract the used etchant and allow the newly sprayed etchant direct access to the copper surface. In addition, these lines automatically control temperature and chemistry concentrations to ensure consistent production. While they may have been designed with HDI technologies in mind, there is no denying their positive impact on heavy copper.

Part 2 of this article will appear in the April 2024 issue of *Design007 Magazine.*

Yash Sutariya is president of Saturn Electronics Corporation

Heraeus Printed Electronics, SÜSS MicroTec Join Forces to Revolutionize High-Volume Semiconductor Manufacturing

Heraeus Printed Electronics GmbH and SÜSS MicroTec have announced the signing of a Joint Development Agreement (JDA) to pave the way for digital inkjet printing of metallic coatings for semiconductor manufacturing. The partnership combines the core competences of both companies to enable digital printing solutions for mass production in the electronics industry for the first time.

Heraeus Printed Electronics offers metallic inks, deep process knowledge and printing systems. It complements the broad product portfolio of Heraeus to the electronics industry. The Prexonics technology enables selective printing on different substrates for a wide range of applications such as shielding against electromagnetic interference (EMI), printing of conductive structures and metallization for heat dissipation.

"This collaboration represents the missing piece in the mosaic of scaling to mass production for us, having recently passed the important milestone of fulfilling the broad functional requirements of the semiconductor industry with our ground-breaking technology," said Franz Vollmann, head of Heraeus Printed Electronics.

SÜSS MicroTec, known for its advanced equipment and process solutions for semiconductor manufacturing, brings its leading-edge automation platform and industrial inkjet production capabilities to the table. Their high-volume production equipment, JETx, designed to integrate various printhead and substrate technologies, expands the application potential of Heraeus' digital printing technology.

The scope of the collaboration calls for the development of equipment that integrates Heraeus' digital printing technology, image processing software and advanced ink technology into SÜSS MicroTec's automation and JETx platform. The end goal is a state-of-the-art system with a competitive total cost of ownership (TCO) that not only sets new standards in the industry, but also aligns with Heraeus and SÜSS MicroTec's vision of cost and performance optimization. The product of this partnership looks to redefine the boundaries of what is possible and push forward into a future where technology serves as the cornerstone for innovation and growth.

(Source: Heraeus)

The Art of Technical Instruction

Tim's Takeaways

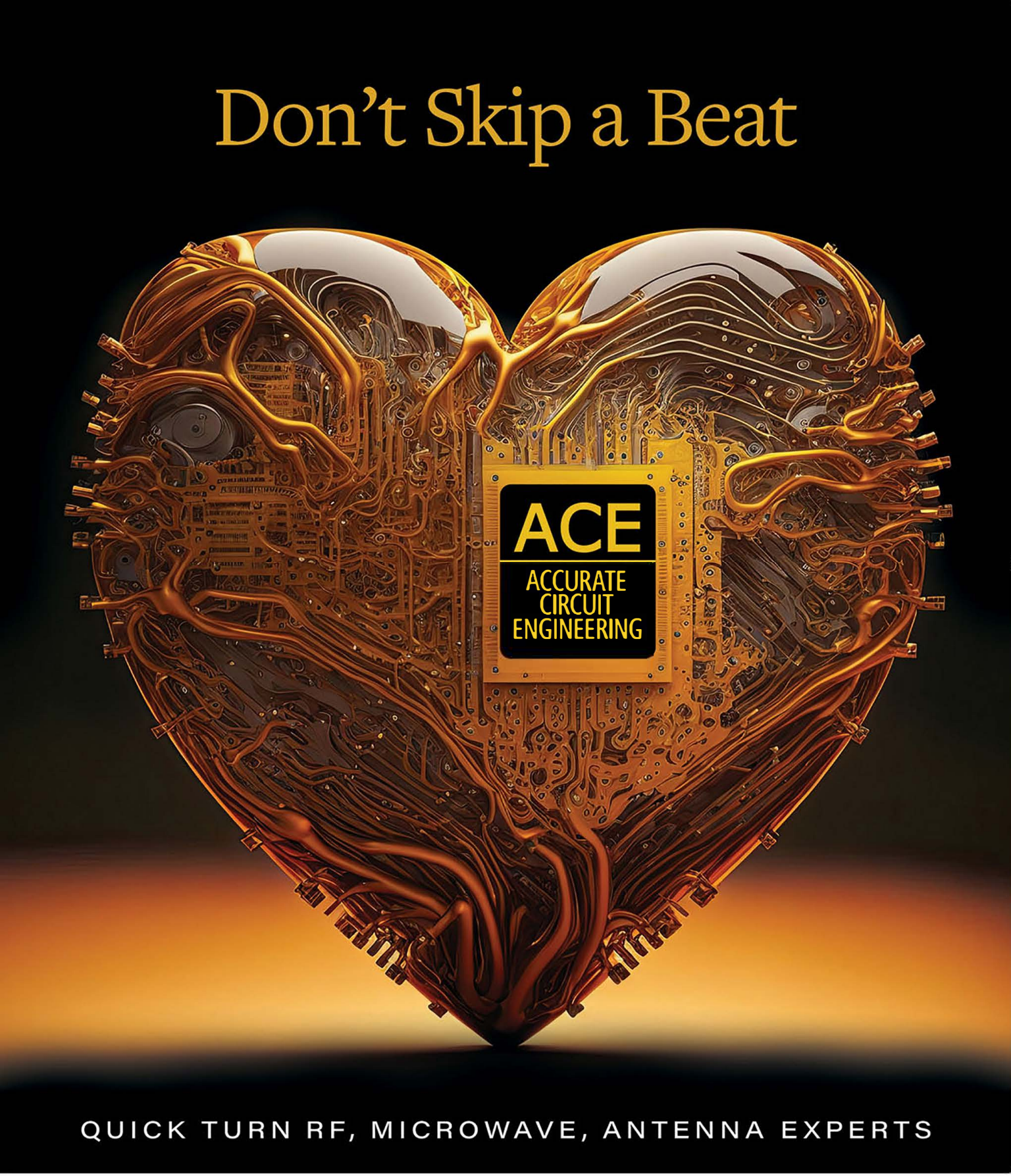
Feature Column by Tim Haag, FIRST PAGE SAGE

What does it take to become a PCB design instructor and be successful? This question is especially relevant in the rapidly changing landscape of printed circuit board design and layout. Our industry, like many others, is dealing with a lot of change. Turnover is chief among those changes as newcomers step into the positions left open by those retiring. There is the constant escalation of technology that triggers continual enhancements of standards and processes that designers must keep up with. It is evident that our industry needs good technical instructors to carry this load.

Over the years, I've seen many discussions about bringing in new PCB layout designers to replace those transitioning out of the field. But it's rare to see discussions acknowledging the need to promote good design instructors who teach the new PCB layout artists, and update the rest of us. I look forward to reading the ideas and insights from the other contributors while also sharing some of my own experiences. You see, once upon a time in the not-too-distant past, I logged some time as a PCB CAD instructor. To be clear, I wasn't a design instructor as there are differences. But as Andy Shaughnessy says about these differences, it's close enough for jazz.

When you're sitting in a technical class at a high-profile event like a design conference, it

Don't Skip a Beat



QUICK TURN RF, MICROWAVE, ANTENNA EXPERTS

ACE
ACCURATE
CIRCUIT
ENGINEERING

Accurate Circuit Engineering
3019 Kilson Drive
Santa Ana, CA 92707
(714) 546-2162 • sales@ace-pcb.com

For technical assistance call:
James Hofer
(714) 425-8950
www.ace-pcb.com

may seem like the instructor has the easy life. After all, the instructors fly in from their last gig, spend the night at a ritzy hotel, babble for a while to a roomful of people, and then fly out again the next day after a steak and lobster dinner. However, the truth isn't nearly so glamorous; there's a lot more to being a technical instructor than just that. Here are some of the tasks that occupied my time when I was conducting PCB CAD training classes:

- **Research:** I would spend time going through the latest versions of my company's software to update the training materials and databases.
- **Content creation:** Sometimes I even rewrote the materials from the ground up if the changes were significant enough.
- **Travel agent:** Since I conducted most of my classes at the client's site, I researched and booked all my own travel arrangements and hotel accommodations.
- **Publisher:** Teaching onsite meant arranging for the publication and distribution of the training materials and databases for the class.
- **System administrator:** I ensured the classroom CAD systems were correctly configured because often these systems wouldn't have the correct software installed.
- **Racing the clock:** Sometimes I arrived the morning of the class and jumped right into teaching without a chance to catch my breath.

There are also a few misadventures. There were times that clients were, let's say, less than satisfied with my efforts. Thankfully, I was never run out of town on a rail, but there were a couple of times when I left a training session with my ears burning. I've had people fall asleep during the class, get up, and storm out because "it was too hard." They would glare at me with such red faces and bulging eyes that I nervously looked for a quick escape route out a back door.

(Right about now, Andy is probably thinking, "What are you doing, Haag? We're supposed to create interest in being a design instructor, not scare everyone away." Be patient, Andy! I'm getting to that.)

However, those misadventures, although worthy of a good story shared around the campfire, were only a small slice of my experiences as an instructor. The other 98% of my time was filled with some genuinely great moments. Here are some examples of those times in no particular order:

Travel

In my time as an instructor, I journeyed from coast to coast and spent some time up in Can-

ada as well. I visited places and saw sites that I probably never would have experienced without the benefit of traveling as a CAD instructor. One thing that I enjoyed a lot was asking the people I was working with what the best local spots were for dinner, and I was usually rewarded with a great and unique place to eat. I even sampled a "Garbage Plate" in upstate New York, which was a next-level adventure. A word of warning: Be careful of the practical jokers who send you to a place not necessarily considered "family dining." I can neither confirm nor deny that something like this ever happened to me. Just be on your guard.

People

I am very grateful for everyone I met while teaching CAD classes. Those in the PCB design business are some of the greatest folks you will ever know, and it was a privilege to spend time with them. It has been said that teachers learn just as much or more than they can impart. That was certainly the case for me. Not only did I learn about my students' businesses and the products they were designing with our software, but getting to know them and finding out what makes them tick added another layer of richness to my own life.

Technology

My company created software to design electronic circuitry and lay out printed circuit boards. However, in the field I discovered how diverse the applications were for those PCBs our clients were designing and how unique their CAD workflows could be. Addressing the specialized needs of our clients often required new software specifications and enhancements that we "in the factory" had not originally thought of. A few that come to mind are library tools, database translators, and plug-ins for different third-party tools. It was an absolute pleasure for me to take these needs back to the factory and be part of the change and enhancement process that ultimately led to increased functionality and better design solutions.

When you conduct technical training, you can count on being faced with many demanding challenges. There's a lot of work that goes into technical instruction, and most of it takes place long before you stand in front of a class. However, the great thing is when you come to understand just how much of an effect you are having on the lives of others and how you are contributing to the growth of our industry. So, how do you become a great technical instructor? Here are some simple takeaway points that can help:

- Know your topic inside and out.
- Study how to be a speaker and teacher; don't simply assume it will happen organically.
- Plan ahead for your trip and teaching engagement; don't wait until the last moment to prepare.
- Be armed with materials, data, and whatever else you need to make a great presentation.
- Be prepared for the unexpected because, like death and taxes, it will happen.
- Develop a mindset that puts yourself in your students' shoes so you can better see what they need. Teach toward that.

Perhaps the most important key to successful teaching is not something that easily fits into a bullet point. To be a great technical instructor you must have a passion for wanting to see others succeed. This perspective will allow you to truly connect with those you are instructing, and help them to get the most out of what you are presenting.

All right PCB designers and instructors, until next time, keep on designing. **DESIGN007**

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

UHDI FUNDAMENTALS:

ASC Sponsors Ultra High Density Interconnect Symposium

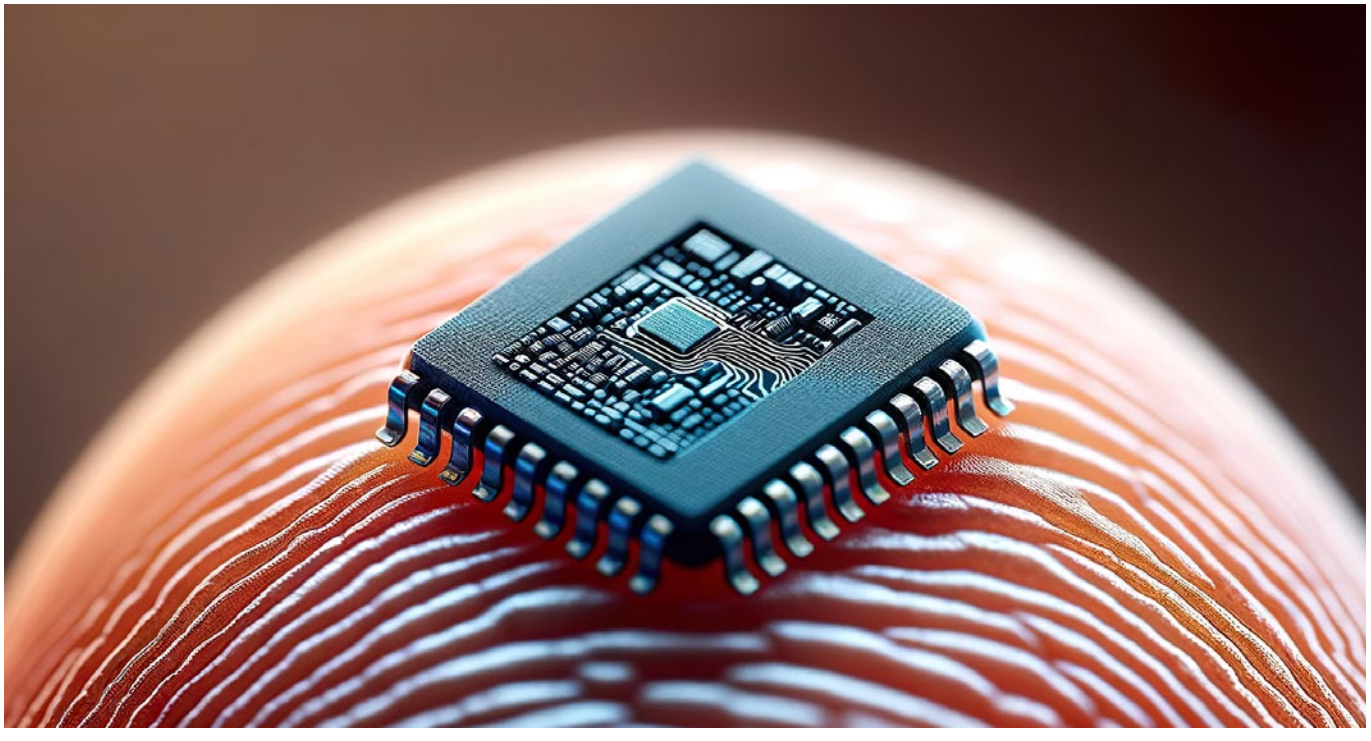
Article by Anaya Vardya
ASC SUNSTONE

American Standard Circuits (ASC) is a major sponsor of an upcoming symposium on one of the most talked about new technologies that has come along in quite a while: ultra high density interconnect (UHDI).

As an early adopter of Averatek's A-SAP™ process, ASC is leading the industry in advancing this technology and working with PCB designers and OEM/ODMs on developing new applications that can take advantage of the sub-25-micron line width and space capability. The event is organized by SMTA (Surface Mount Technology Association), which has been dedicated to sharing practical knowledge in the electronics manufacturing industry for 40 years.

Tara Dunn, SMTA director of education and training, and Kat Erdahl, SMTA events and administrative manager, have put together an agenda of the preeminent industry experts in UHDI who will be speaking on various topics. In addition to the speakers, there will be a roundtable discussion moderated by Mike Marshall of NCAB Group.

John Johnson of American Standard, Altium's David Haboud, and Chrys Shea of Shea Engineering will speak on the design of a UHDI PCB test vehicle that ASC will be build-

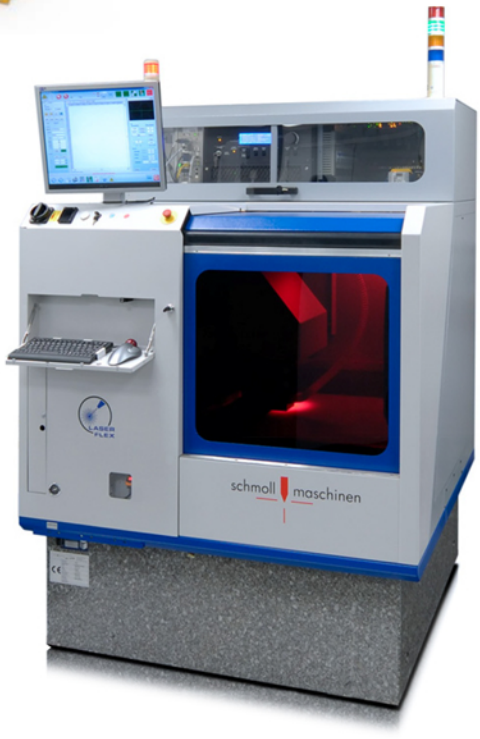


Focused on Flex

schmoll optronics

LaserμFlex series

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



Typical applications:

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

BÜRKLE
NORTH AMERICA, INC.

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

ing to optimize performance through UHDI design, fabrication, and assembly processes. The PCB test vehicle design will be the topic of the next issue of this UHDI series. Other key discussions will include:

1. **UHDI journey:** How we got here from the early subtractive processes to the various current technologies that allow the game-changing sub-25-micron trace and space width.
2. **Material selection:** The choice of materials for the interconnects is crucial. Materials with high conductivity, low resistance, and good thermal properties are preferred. Copper and gold are commonly used for their excellent electrical conductivity, while materials like silicon and diamond are used for their thermal properties.
3. **Design and layout:** The design and layout of the interconnects play a significant role in optimizing their performance. Factors such as the width, length, and spacing of the interconnects, as well as the number of layers, need to be carefully considered to minimize resistance, capacitance, and signal delay.
4. **Reliability and durability:** UHDIs must be reliable and durable to withstand the rigors of operation. Techniques such as stress testing, accelerated aging, and failure analysis are used to ensure the reliability and durability of UHDIs.
5. **Technology availability:** The availability of both domestic and global capability of organizations that can produce this technology is a concern as UHDI applications continue to increase and the ability to meet demand requirements are currently somewhat limited.

Title: Ultra High Density Interconnect Symposium
>> FULL PROGRAM INCLUDED ON NEXT PAGE >>
Date: March 26, 2024
Location: Peoria Sports Complex, Peoria, Arizona
Registration Link: Click here. DESIGN007

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

2024 UHDI Symposium Program

Tuesday, March 26, 2024

TIME	
7:00 AM	Registration Opens
7:15 AM - 8:15 AM	**WFD Breakfast** **Building Tomorrow's Expertise:** A workforce development discussion identifying key competencies for semiconductor technology and PCB electronics.
8:20 AM	Opening Remarks
	Session 1
8:30 AM	**The Way to UHDI - From Subtractive to mSAP To SAP Processes** Nava Shpaisman, Ph.D., KLA
9:00 AM	**Optimizing Performance: Insights Into the Design, Fabrication, and Assembly of Ultra HDI Assembly Test Boards** David Haboud, *Altium* John Johnson, *American Standard Circuits* Chrys Shea, *Shea Engineering*
10:00 AM	**Addressing the Domestic Capability Gap for Ultra HDI in High Reliability Applications** Roger Smith, *NSWC Crane*
10:30 AM	Break
	Session 2
10:45 AM	**Achieving High Reliability With Non Reinforced Materials** Paul Cooke and Chris Hanson, *Ventec*
11:15 AM	**Critical via Reliability Using Advanced Materials To Support High Performance UHDI Designs** Steve Karas, *GreenSource Fabrication*
11:45 AM	**A Comprehensive Approach To Ensuring Reliability in Ultra HDI Solutions** Daniel McCormick, *NSWC Crane*
12:15 PM	Lunch
	Session 3
1:00 PM	**Panel: Navigating the Paradigm Shift and Driving Ultra HDI Technology** **Moderator:** Mike Marshall, NCAB Group Dan Blass, *Lockheed Martin* Tina Landon, *NSWC Crane* Islam Salama, Ph.D., *Hyperion Technologies* Gene Weiner, *Weiner International Associates*
2:00 PM	**Metal Complex Inks and Films for Additive Manufacturing** Mike Vinson, *Electroninks, Inc.*
2:30 PM	**Novel Sustainable Surface Finish for Optimum Signal Integrity and Better Reliability of Ultra HDI Printed Circuit Boards** Kunal Shah, Ph.D., *LiloTree*
3:00 PM	Break
	Session 4
3:30 PM	**Configurable Automation in Medical Electronics Manufacturing, and Its Impact on Product Design, Quality and Reliability** Alok Sharan, Ph.D., *Micro Systems Engineering, Inc.*
4:00 PM	***Design and Verification of Ultra-HDI Topologies*** Stephen Chavez, *Siemens*
4:30 PM	**Heterogeneous Integration, Ultra-HDI and Onshoring - One Material Manufacturer's Response to the Challenge Trifecta** Brian Sinclair, *AGC Multi Material America, Inc.*
5:00 PM - 6:00 PM	Wrap Up - Next Action Steps and Cocktail Reception
6:00 PM	Event Concludes

IPC Mourns Loss of Former Vice President of Industry Programs, Tony Hilvers

It is with sadness that IPC announces the passing of Tony Hilvers, former IPC vice president of industry programs, on Tuesday, Feb. 6. Hilvers left IPC in 2012 after 29 years of service.

During his long tenure, Hilvers was responsible for the association's market research, government relations and environmental policy, meetings, and professional development departments.

In addition, he was responsible for industry segments including the PWB and EMS Management Councils, the PWB Suppliers Council, the Surface Mount Equipment Manufacturers (SMEMA) Council, the Solder Products Value Council, and associated events, including IPC APEX EXPO. While he was vice president of industry programs, IPC APEX EXPO was named one of the top 25 fastest-growing U.S. trade shows in attendance by Trade Show News Network.

While serving as director of educational services and marketing communications, he formed the EMS Management Council and published the first market research study on the EMS industry in 1984. Hilvers was also instrumental in launching IPC Printed Circuits Expo, IPC APEX Conference and Exhibition and the co-located IPC APEX EXPO and worked with the Hong Kong Printed Circuit Association to launch the HKPCA/IPC International Printed Circuits and Assembly Fair.

Said David Bergman, IPC's vice president of standards and technology and long-time colleague of Hilvers, "I had the privilege of working with Tony for nearly 30 years. I did sales visits, traveled internationally, collaborated, supported and was supported by Tony and his team. I always admired his ability to listen to a group of business leaders, figure out a program to solve a problem they were faced with, and then convince others that this program was worth their support. Tony's passion for the industry and his natural sales ability facilitated his creation of many new programs for IPC."

IPC extends its sincere condolences to Tony's family, friends, and former work colleagues.

(Source: IPC)

Integrated Tools to Process PCB Designs into Physical PCBs

Visualize

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

Verify

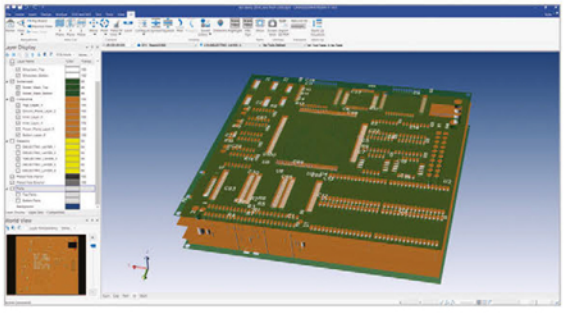
Ensure that manufacturing data is accurate for PCB construciton.

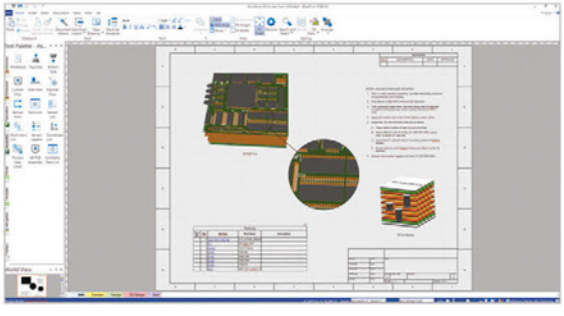
Panelize

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

Document

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





CAM350®

Verify and Optimize PCB Designs for Successful Manufacturing

BluePrint-PCB®

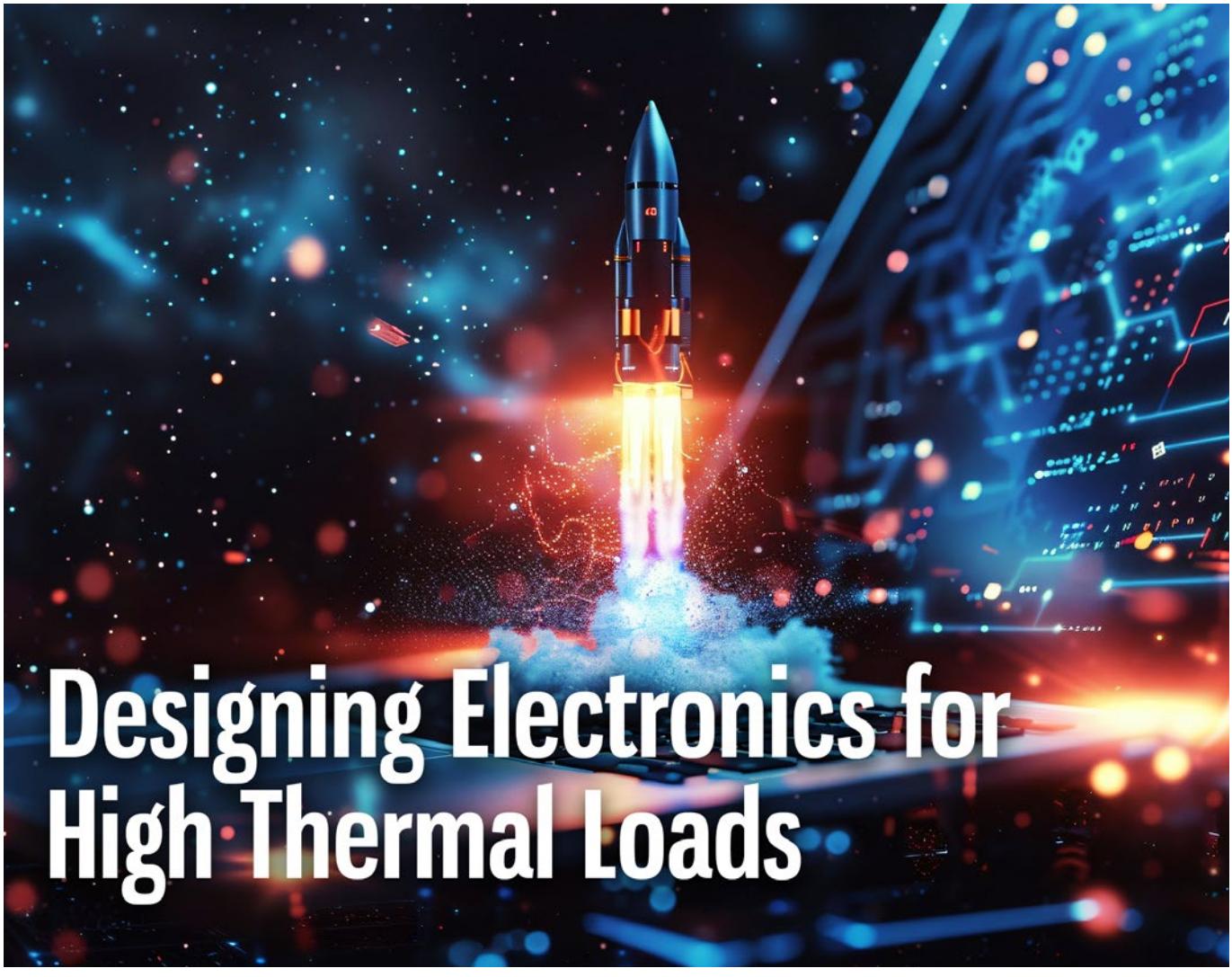
Create Comprehensive Documentation to Drive Fabrication, Assembly & Inspection

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

DownStream Technologies

800-535-3226 | downstreamtech.com

NOW WITH 3D



Designing Electronics for High Thermal Loads

Article by Akber Roy
RUSH PCB

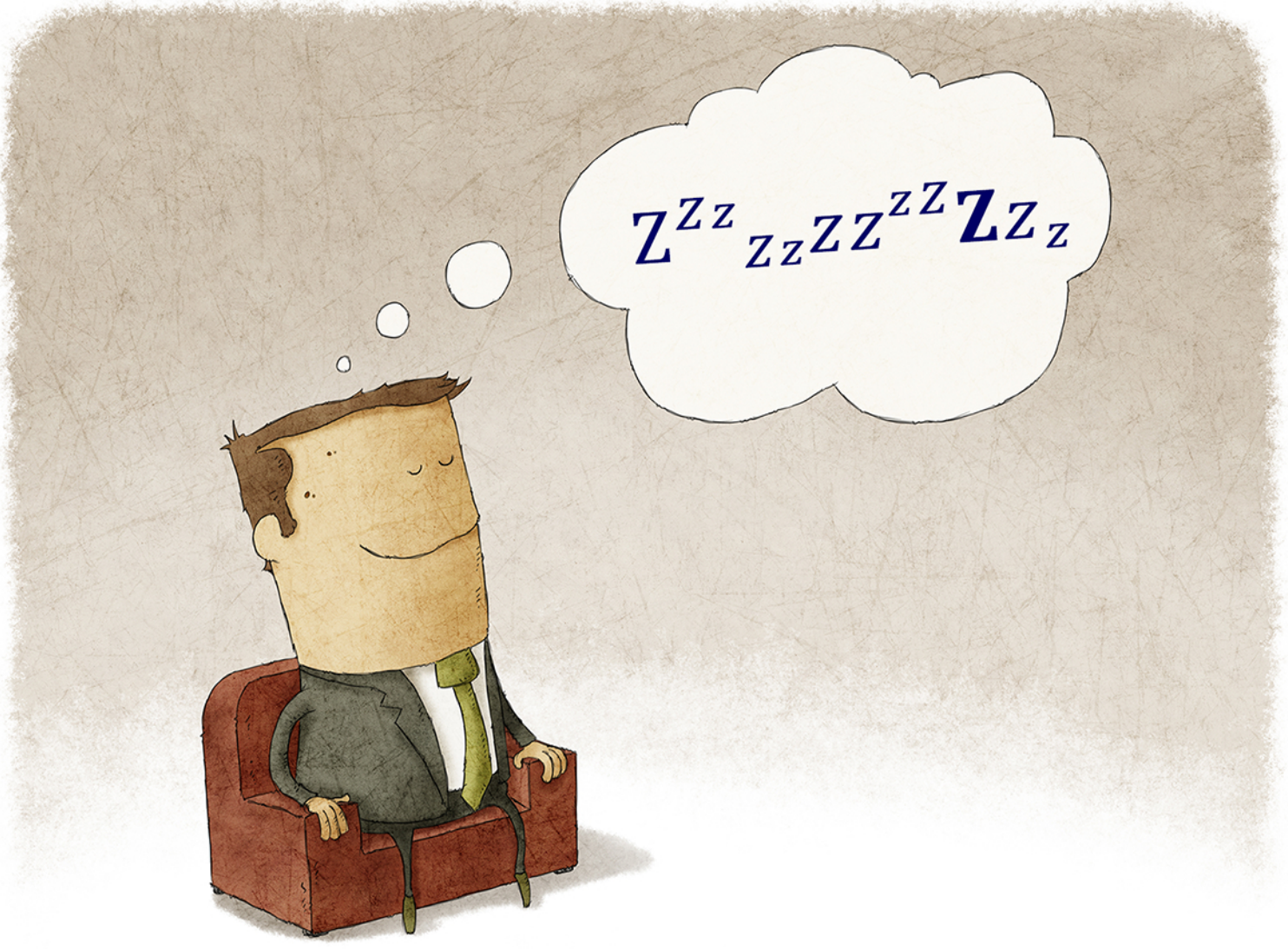
Developing proactive thermal management strategies is important in the early stages of the PCB design cycle to minimize costly redesign iterations. Here, I delve into key aspects of electronic design that hold particular relevance for managing heat in electronic systems. Each of these considerations plays a pivotal role in enhancing the reliability and performance of the overall system.

Thermal Management: A Pinnacle Challenge to Reliability

All electronics generate heat due to energy conduction inefficiencies. This phenomenon arises from inherent electrical resistance in all metals and semiconductors, leading to some energy in the circuit converting to heat. Over time, that heat, if not properly managed, can accumulate and cause damage to sensitive components.

In my capacity, I've witnessed an increase in thermal management issues as electronics trend toward higher functionality in smaller packages. Although many thermal management methods and copious amounts of component thermal data are available to assist engineers in understanding thermal loads, thermal issues continue to be difficult to predict, creating unwanted surprises. Unfortunately, these surprises often arise during functional test-

We DREAM Impedance!



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

- Transmission line impedance and
- Power Distribution Network impedance

DISCOVER MORE

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

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

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

iCD
In-Circuit Design Pty Ltd

www.icd.com.au

ing, not during the front-end design process. I firmly believe that today's compact, high-performance electronics must be developed with robust thermal management strategies early in the design cycle. This proactive approach not only reduces development time and costs but also helps product designers ensure that crucial electronic components operate in their optimal temperature range, ultimately enabling long-term product safety and reliably.

To help designers achieve these goals, I've developed the following overview of best practices and key considerations for managing heat in electronic systems.

Understand the Factors Affecting Thermal Load

Many factors contribute to the overall thermal load in electronic systems, from power consumption and I/O speeds to component package designs, features, and functions. Drawing on decades of experience supporting PCB fabrication and assembly, it becomes apparent that advanced and simple systems alike can grapple with thermal issues, and there is no one-size-fits-all thermal solution.

I strongly recommend using thermal modeling and simulation tools at the beginning of the design process...

I strongly recommend using thermal modeling and simulation tools at the beginning of the design process to aid in understanding the many factors that affect thermal load, identify potential hot spots, and assess the potential effects of different thermal management strategies. This approach makes it possible to accurately predict thermal behavior in complex electronic systems prior to building physical prototypes. With thermal modeling, designers can efficiently optimize their designs, simulating and analyzing multiple design iterations, while experimenting with different components and materials.

By leveraging thermal modeling, designers can proactively address thermal concerns early in the design process, paving the way for optimal performance and longevity.

Component Selection and Thermal Resistance (R_t)

The components are central to all electronic devices. While all components generate heat, the predominant source of thermal load typically originates from power electronic components such as MOSFETs and other transistors, digital processors like CPUs and GPUs, and RF components including amplifiers and transceivers.

When selecting such components, it is important to consult technical datasheets (TDS) for thermal resistance values. Thermal resistance (R_t), a measure of a component's resistance to heat flow, is influenced by the material and structure of the component's packaging. Higher R_t values signify the potential for a greater internal temperature rise per unit of power dissipated, posing increased risks of premature failures. Therefore, lower R_t values are preferable.

While not every component requires selection based solely on the lowest R_t value, understanding these values aids in guiding component selection and provides valuable insights into areas where thermal management solutions may be necessary.

Material Selection and Coefficient of Thermal Expansion (CTE)

Material choices are pivotal to managing heat in electronic systems, especially as they relate to each other in terms of their CTE values, another key value on a component's TDS. Substrates, interposers, components, and solder alloy materials with very different CTE values will expand and contract at different rates as they heat up and cool down. This can

cause mechanical stresses that weaken bonds between these elements and, ultimately, lead to warpage, breakage, and premature failures.

Effectively managing CTE mismatch is especially important when implementing 2.5D and 3D semiconductor packaging technology. These innovative packages involve vertically stacking heterogeneous dies to achieve substantial increases in functionality and performance within smaller form factors. To realize the full potential of 2.5D and 3D packaging, advanced multi-layer high-density interconnect (HDI) PCB fabrication technology is needed to create very fine and precisely placed die-to-die interconnections. To ensure success, modeling and simulation of all elements in these three-dimensional systems is highly recommended to gauge the effects of power output, chip placement, and CTE disparities.

Metal Heat Sinks and Thermal Interface Materials (TIMs)

When a component is susceptible to overheating, a common practice is to employ a heat sink or a thermally conductive TIM to assist with heat dissipation. These passive cooling devices, characterized by their ability to cool without moving parts, play a crucial role in thermal management of electronic systems.

Heat sinks, typically crafted from aluminum alloys or copper, feature fins or pins that increase their surface area to facilitate efficient heat transfer to the surrounding air or fluid. Whether off-the-shelf or custom fabricated, heat sinks are typically attached to the top of heat-generating components like CPUs or GPUs using a thermally conductive paste. While highly effective at enhancing heat dissipation, heat sinks come with the drawback of occupying a considerable amount of space—an aspect that contradicts the prevailing market trend favoring smaller and more compact electronic devices.

In addition to heat sinks, thermally conductive TIMs are widely used to draw heat from hot components. Composed of composite materials (polymers with thermally conductive fillers), TIMs are formulated to adhere and conform to a variety of surfaces. Strategically positioned between the top surface of a heat-generating component and a heat-spreading substrate, such as a metal enclosure, TIMs provide an effective heat transfer path. Because they are conformable, they offer additional benefits such as stress relief and sealing.

PCB Materials as Heat Sinks or Heat Spreaders

Substrates and conductive materials in PCBs can be designed to dissipate heat similar to a heat sink or heat spreader. To use a board as a heat sink or heat spreader, specialized materials and construction methods may be needed. Some options include:

- **Polyimide:** Widely employed in the fabrication of flexible and rigid-flex circuit boards due to its exceptional flexibility, polyimide is best known for achieving an optimal balance between flexibility and rigidity. Polyimide PCBs demonstrate exceptional thermal endurance and resilience, showcasing stability across a wide range of temperatures, supporting operations up to 260°C. They have very good thermal conductivity, and in certain applications, bonding the polyimide PCB to a metal base further improves heat dissipation.
- **PTFE-based laminates:** Polytetrafluoroethylene (PTFE), known by the trade name of Teflon®, is a synthetic fluoropolymer highly favored for RF electronics due to its high frequency and heat resistance. PTFE-based laminates are specialized composite structures composed of PTFE with the inclusion of carefully selected additives and fillers that determine the electrical, mechanical, and thermal behavior of the laminate. Ceramic powders are the preferred filler materials for RF applications due to their superior thermal

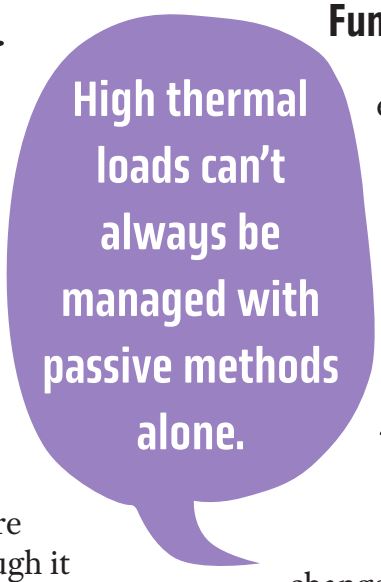
conductivity and low CTE mismatch compared to copper.

- **Heavy copper:** As power and temperature requirements increase, the incorporation of thicker copper layers and traces emerges as a strategic approach to efficiently disperse heat across the board. Heavy copper PCBs, characterized by copper thicknesses in the 4-to-10-ounce range, not only enhance dissipation, but also increase current carrying capacity and bolster mechanical strength. These PCBs empower the use of high-performance, high-temperature components, such as those built on silicon carbide (SiC) or gallium nitride (GaN), ensuring the realization of their full high-temperature potential without the risk of circuit failure. Heavy copper PCBs, while larger and heavier than standard PCBs, deliver markedly superior thermal performance.
- **Insulated metal substrate (IMS):** Also called metal core PCBs, these boards are built with a metal base or core layer that serves as a heat sink or heat spreader. Aluminum is the prevalent choice, being more cost-effective than copper, though it is essential to note that aluminum is less resistant to corrosion compared to copper. IMS PCBs are constructed with thermal vias, which enable heat to move from top-side components to the bottom-side base metal. Moreover, the metal layer imparts exceptional hardness and strength to the PCB, while also functioning as an electromagnetic shield and a ground layer.

Choosing a heat spreading PCB technology for a specific high-thermal application involves carefully balancing material properties with design requirements. Most often, high thermal loads can be effectively managed through the utilization of one of these specialty PCB construction methods.

Active Cooling

Until now, I have exclusively explored passive measures of heat management in electronic systems. However, high thermal loads can't always be managed with passive methods alone. In such cases, adding an active cooling approach is required. This may entail the use of fans, water cooling systems, or heat pumps. While active methods contribute to increased size and complexity of the system, they can offer significant temperature reductions precisely when and where they are needed most.



Functional Testing

The final step in the design of electronic systems is functional testing of physical prototypes. During this phase of the process the entire PCB assembly is validated, verifying that the device performs as intended and meets all of the design and regulatory requirements. Thermal cycling and thermal shock tests are performed to determine reliability in environments with sudden extreme changes in temperature. Data gathered helps engineers understand the product's operating temperature limits and provides information that can be used to extrapolate the product's potential lifespan.

Given the fast pace and extreme competition in the electronics industry, I strongly advocate selecting a manufacturer capable of producing rapid prototypes. This strategic choice can condense prototype fabrication time from several weeks to a mere few hours. Expedited prototyping accelerates the final stages of product development, enabling an agile release of the product to the market.

LOOK INSIDE

IPC BUILD ELECTRONICS BETTER

Contact us to become an advertiser

Conclusion

Thermal issues will likely continue to challenge the reliability of modern electronics, particularly as devices become more powerful, compact, and feature-rich. Effectively addressing these challenges demands a heightened emphasis on thermal management throughout the entire design process. From thermal modeling and simulation in the initial design stages to careful selection of components, materials, and PCB fabrication technologies, numerous factors play pivotal roles in achieving success, often creating a sense of being overwhelmed. In this intricate process, designers can derive substantial benefits by tapping into the extensive experience and expertise offered by PCB manufacturers. Serving not only as valuable resources but also as collaborative partners, these manufacturers can provide crucial support in navigating the complexities of designing electronics for high thermal loads. **DESIGN007**

Akber Roy is CEO of RUSH PCB Inc.

Siemens Joins Semiconductor Education Alliance to Address Skills and Talent Shortage

Siemens Digital Industries Software announced it has joined the Semiconductor Education Alliance to help build and nurture thriving communities of practice across the integrated circuit (IC) design and electronic design automation (EDA) industries, from teachers and schools to universities, publishers, educational technology companies and research organizations.

Founded by Arm in 2023 with a mission to help close education and skills gaps in the global semiconductor space, the Semiconductor Education Alliance brings together key stakeholders from across industry, academia, and government, to provide resources that help teachers, researchers, engineers and learners access new, accelerated educational pathways.

"Siemens joining the Semiconductor Education Alliance is a significant step forward in our collective efforts to promote communities of practice in STEM education and research with the involvement of academia and industry partners throughout the EDA industry," said Mike Ellow, Executive Vice President, Electronic Design Automation, Siemens Digital Industries Software.

"The global semiconductor industry is facing a shortage of skills and talent that requires industry-wide action," said Khaled Benkrid, senior director, Education and Research at Arm. "The Semiconductor Education Alliance was created to address semiconductor skills challenges and we welcome the capabilities Siemens brings to the alliance as we come together as an industry to nurture the talent pipeline."

"The semiconductor industry needs more than one million additional skilled workers by 2030 to keep up with global demand, according to Deloitte," said Dora Smith, senior director of the Global Academic Program, Siemens Digital Industries Software. "Partnering with Arm through the Semiconductor Education Alliance helps us collectively bridge pathways to address both the quality and quantity of talent needed to drive innovation and meet market growth. We look forward to collaborating with this ecosystem of expertise to futureproof the workforce."

(Source: Siemens)

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!

The *Art* of Presenting PCB Design Courses

Beyond Design

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

In the early days of my career, I was a typical backroom geek more comfortable with technology than engaging in conversation. My obsession lay in my work, fueled by the exhilaration of mastering new technologies. The notion of standing before a class of 50 or more individuals to deliver a solo weeklong course seemed utterly terrifying. But necessity gives birth to innovation. When confronted with challenges, we have two choices: step up or fade into oblivion.

Thirty years ago, my journey into public speaking commenced with an unexpected plunge. I found myself at the helm of PCB and software training courses, sales demonstrations, and trade shows. Additionally, I was appointed as president of the Australasian Chapter of the IPC Designers Council. Despite my initial reservations, I embraced these challenges head-on. I soon realized that technical training and sales demonstrations share common ground. They complement each other. To empower people with technology, you must adeptly convey its intricacies. 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. Understanding fosters confidence, and confidence fuels adoption. As a presenter, your role extends beyond information delivery; it's about igniting curiosity and competence. In this month's

Experience Tomorrow's Technology Today!

FREE CAD TOOLS

DIGITAL

FLEX/ RIGID-FLEX

RF/METAL/ MICROWAVE

HDI/ULTRA HDI

QUICKTURN ASSEMBLY

THE WHOLE PCB SPECTRUM – PROTOTYPE TO PRODUCTION

American Standard Circuits

Sunstone Circuits

Join us at the forefront of innovation and witness the power of progress. Let's shape a brighter future together!

www.ascsunstone.com | 800-228-8198

column, I'll divulge a few essential techniques I've learned.

1. Know Your Subject Matter.

Avoid discussing topics you're not well-versed in. Instead, focus on what you know and love. Be yourself during presentations. Authenticity resonates with the audience. A great presentation doesn't rely on reading slides. Use concise bullet points to prompt concepts. When you truly understand the content, your delivery will flow naturally. Remember, confidence and passion go a long way toward captivating the audience.

2. You Don't Need to be a Performer.

While a touch of humor can break the ice, don't stress about being overly charismatic or a comedian. Instead, be genuine and discuss what you're knowledgeable about. Your authenticity will shine through, allowing your audience to connect with you. Be yourself, and you'll feel more at ease, enabling a stronger and more relaxed presentation. Your unique perspective and expertise are your greatest assets.

3. Don't Think. Just Do It. Up There, You Don't Have Time to Think.

Thanks to Maverick from "Top Gun" for giving me this line. Before I start a presentation, I don't even think about the first word I'll say. Face it, you can't memorize the entire course (unless you have a Neuralink embedded in your brain). Start by introducing yourself. Validate your experience and your background. Briefly explain why the audience should listen to you. Ask the participants to share their background to get a feel for where they are coming from. Knowing your audience is crucial for any presentation, but it's even more important for a technical one. You may need to adjust the presentation to suit their needs. I often get requested to focus on particular topics when running in-house courses. Before diving into slides, share a relevant short story. It helps establish rapport with the audience and sets a positive tone.

4. Rehearse, Rehearse, Rehearse.

After practicing your presentation multiple times, it becomes second nature. Just turn up and start the show. Avoid using "um" or "ah." Instead, focus on volume and diction. Speak clearly and confidently, projecting your voice to reach the back of the room. I initially choose someone at the back of the room to focus on and then shift my attention to others. This technique eases nerves and enhances engagement. The best speakers don't even need microphones. We are not rock stars and don't have the benefit of fold-back. When you talk into a microphone, you hear an echo a few milliseconds later and it's very distracting. It is better just to project your voice. Keep a bottle of water handy; you will need it as your throat dries out.

5. Never Say, "I Don't Know."

Promote audience engagement by encouraging questions throughout your presentation. When unsure, assure them, "I'll find out and get back to you." Make a note of the question and the person who asked it, then provide accurate information later. This approach maintains the audience's confidence in your expertise. If one person consistently asks questions, acknowledge their enthusiasm but mention that we're falling behind schedule. Assure them that their queries will be addressed during the break. Fostering interaction enhances the overall presentation experience.

6. Pace the Course Content.

Starting with the basics, even in advanced courses, ensures everyone is on the same page. Make sure they fully understand each section as you progress. If in doubt, go back over the key points at the end of each section to reinforce the ideas. Understanding is crucial for retention. Pace the course by dividing the content into sections spaced by breaks. Gradually introduce more complex topics. A well-structured presentation keeps the audience engaged and facilitates effective learning.

A well-structured presentation keeps the audience engaged and facilitates effective learning.

7. Picture the Topic.

Leverage the power of visuals. A single image can convey multiple complex ideas more effectively than words. Include plenty of graphics to illustrate the point. Use the whiteboard to assist in explaining any queries. In PCB design courses, I relate concepts to real-life scenarios. Demonstrations of complex techniques can clarify issues and make the content more relatable. Simulations also convey the message when discussing high-speed design. I get an awe-inspiring response when I graphically show the results of what-if analysis; it becomes more memorable. Always keep a backup set of files so that you can quickly reset the database after showing a particular point. Technical glitches happen, and having a reset option ensures smooth transitions. A well-prepared presentation with compelling visuals can leave a lasting impression.

8. Feedback Matters.

Finally, ask the attendees to fill out a course evaluation form. Important questions to ask (good, bad, or ugly):

- Did the seminar meet your expectations?
- How do you rate the presenter?
- How valuable was the content?
- Are there any comments or suggestions you would make to improve the seminar?

Feedback is important for ensuring relevant course content and gauging your presentation skills. There is always room for improvement, and you get better as you gain experience. I normally fare pretty well, but there is always one guy who sits in the back, falls asleep, and gives you a bad rap. If you don't listen, you don't learn.

Don't be apprehensive. Change is good, and if you know the subject well, then presenting ideas to others should come naturally. Give it a try. You may discover a hidden talent waiting to evolve. **DESIGN007**

Barry Olney is managing director of In-Circuit Design Pty Ltd (iCD), Australia, a PCB design service bureau that specializes in board-level simulation. The company developed the iCD Design Integrity software incorporating the iCD Stackup, PDN, and CPW Planner. The software can be downloaded at www.icd.com.au. To read past columns, click here.

PCB Designers Still Wanted

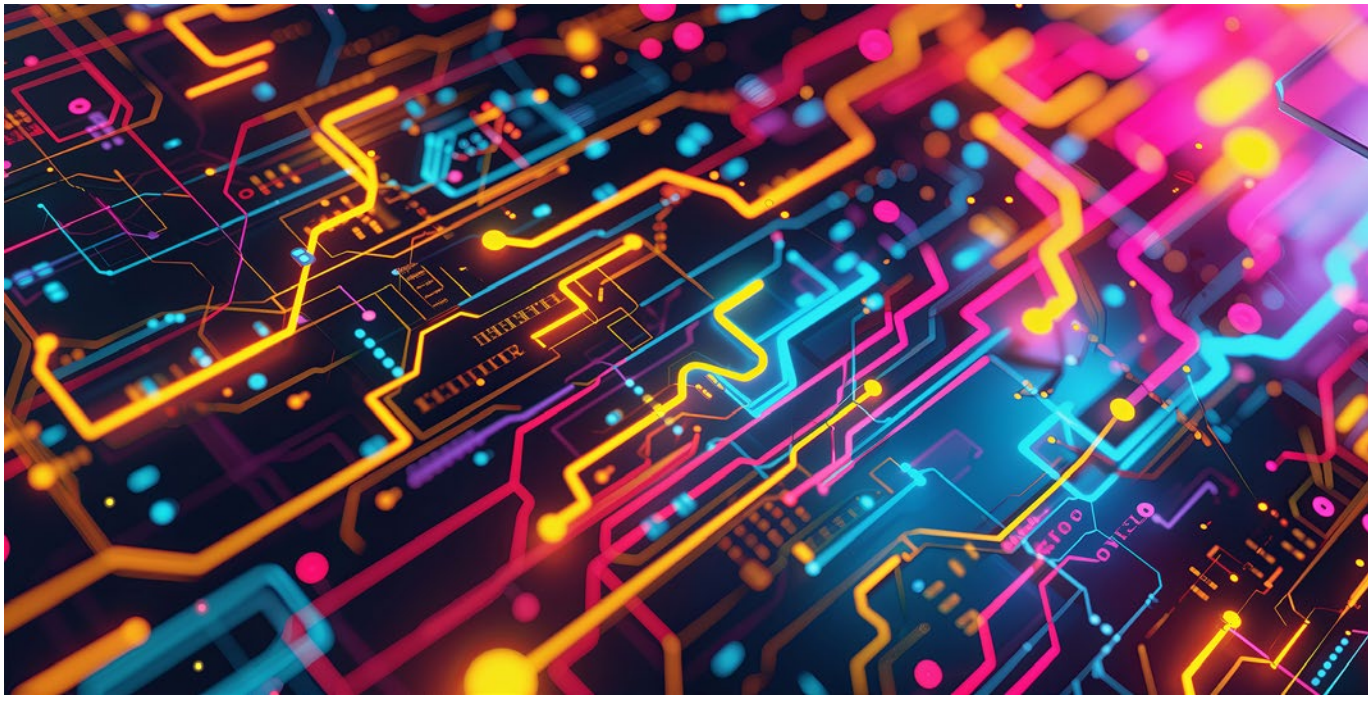
Flexible Thinking

Feature Column by Joe Fjelstad, VERDANT ELECTRONICS

From their relatively simple beginnings, PCBs were largely structured using through-components, including axial leaded resistors, capacitors and diodes, and the iconic dual inline bug-like IC packages. They were often assembled using breadboard and wire-wrapped connections that enabled the electronics hobbyists to prototype their imaginative circuit designs. Current generation PCBs rival their integrated circuit siblings in complexity as they have become even more closely associated with creating today's mind-numbing products.

As you can see, designing printed circuits requires a diverse combination of skills that cover basic electronics, fundamentals of circuit design, and an appreciation for the various common components mentioned earlier (as well as how they behave in circuit function). Along with some basic problem-solving skills, one should have proficiency with one or more electronic design automation/EDA (PCB design automation) software tools such as those offered by Altium, Cadence, Mentor, and others. Another good skill is using a good PCB simulation tool such as that offered by Ansys. Designers should limit themselves to a single design spin to prevent wasting time; simulation will go a long way toward preventing such a wasted effort.

Mastery of any PCB design software normally requires a working knowledge of PCB



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

FLEXIBLE CIRCUITS | EMS/ASSEMBLY | CONTRACT MANUFACTURING

Industry leading Flex Design Support

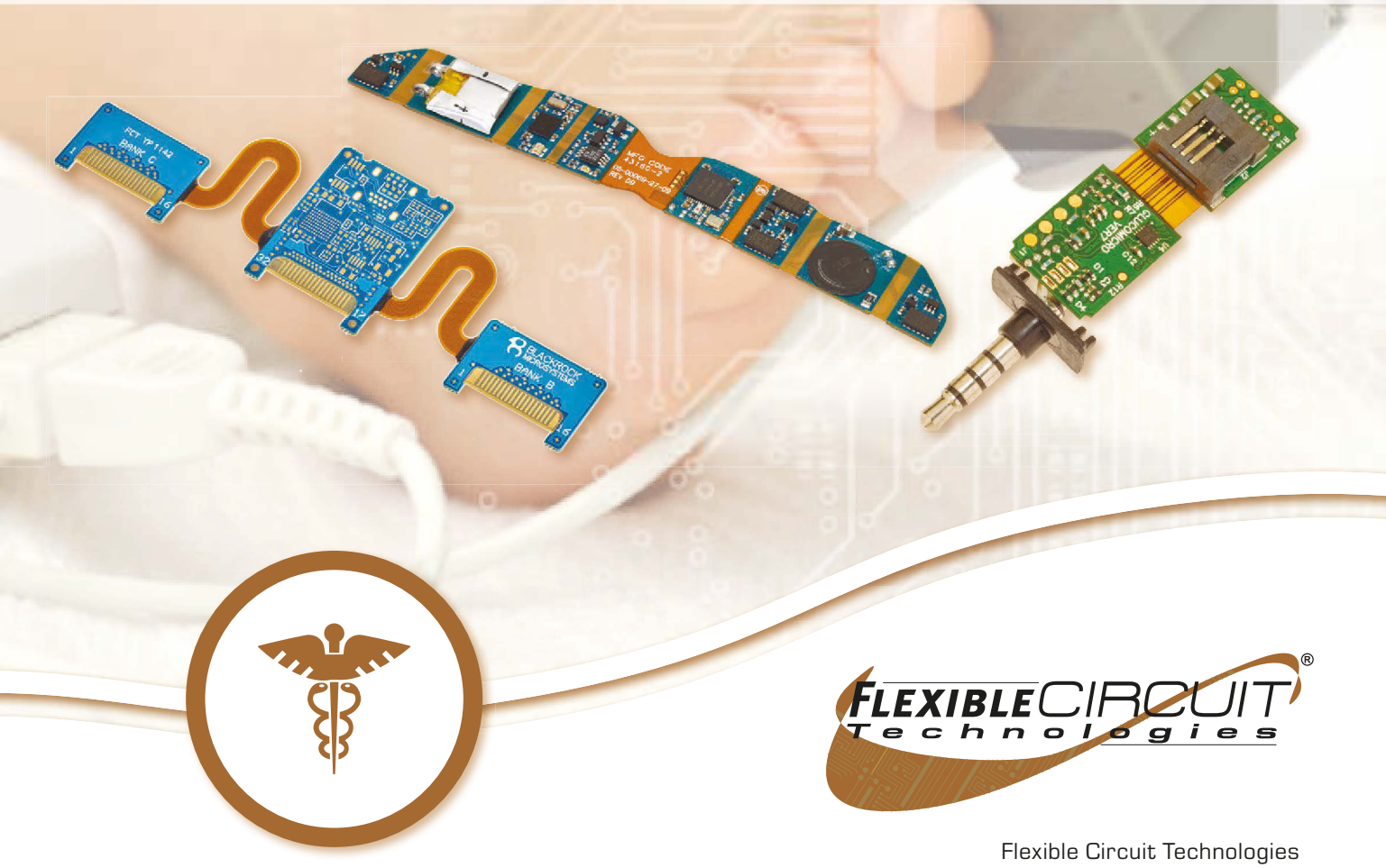
Flexible Circuits, Rigid Flex, Flexible Heaters Membrane Switches, Plastic Moldings

Specialized EMS/Assembly Services

Product Module to Complete Product Box Builds

ISO 13485 – FDA registered Box Builds

EXPLORE the POSSIBILITIES!



FLEXIBLE CIRCUIT® Technologies

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

layout rules, including component placement, routing, and layer stackup guidelines codified in IPC design standards. These include:

- IPC-2221, *Generic Standard on Printed Board Design,* which provides guidelines for the design of PCBs, including requirements for copper thickness, minimum annular ring, conductor spacing, and other design considerations.
- IPC-2222, *Sectional Design Standard for Rigid Organic Printed Boards,* covers specific design considerations for rigid PCBs and includes topics like thermal stress, thermal conductivity, and mechanical considerations.
- IPC-2223, *Sectional Design Standard for Flexible/Rigid-Flexible Printed Boards,* reviews the unique design challenges associated with flexible and rigid-flex PCBs, including bend radius, material selection, and dynamic flexing considerations.
- IPC-6012, *Qualification and Performance Specification for Rigid Printed Boards,* covers qualification and performance requirements for rigid PCBs, including criteria for base materials, conductor thickness, hole plating, and other fabrication processes.
- IPC-6013, *Qualification and Performance Specification for Flexible Printed Boards,* which is like IPC-6012 but is specific to flexible PCBs, covering material properties, design considerations, and fabrication requirements.
- IPC-6018, *Microwave End Product Board Inspection and Test,* focuses on requirements for microwave PCBs, addressing high-frequency design considerations and inspection/test requirements.
- IPC-2226, *Sectional Design Standard for High Density Interconnect (HDI) Printed Boards,* provides guidelines for the design of HDI PCBs, and involves advanced technologies like microvias, fine-line traces, and high-density component placement.
- IPC-7351, *Generic Requirements for Surface Mount Design and Land Pattern Standard,* gives guidelines for the design of surface mount components, including recommended land patterns and component dimensions.

Another highly valuable skill is related to actual board manufacturing and assembly. The designer should invest as much time as possible in learning about the actual manufacturing process used in PCB fabrication and assembly, the many materials used in manufacturing, and the circuit features, including line and space, and via dimensions. The decisions made by the designer have far-reaching consequences. Making informed decisions is vital.

As signal speeds continue to climb, a designer should have a solid understanding of signal integrity principles, including impedance matching, trace length matching, and noise reduction techniques, along with electromagnetic interference (EMI) regulatory requirements and standards. This includes paying attention to ESD, which can destroy a board if not well managed.

In the realm of the actual product, there is an increasing need to understand and apply thermal management techniques to prevent overheating of components on the PCB. It's a growing problem as the industry moves toward ever-higher performance and physically hotter products.

I would be remiss not to mention the elephant standing in the corner getting ready to take the stage: PCB design enabled or assisted by artificial intelligence (AI). This will likely be here quicker than most folks can imagine. I'm very intrigued, for example, by Benchmark's recent investment in Quilter, which intends to do just that. However, as Quilter CEO Sergiy Nesterenko said in an interview with Reuters[1], "Just because (software) got to 90% comple-

tion doesn't mean it did 90% of the work. That remaining 10% of the work that is left is really, really difficult." On the bright side, it's likely PCB designers will still be needed, just as autonomous vehicles will require drivers for some time to avoid crashes along the way.

The last skill (perhaps it should have been the first) is having well-polished documentation and communication skills. The long list of technicians and engineers who will be using the design information product can produce an electronic circuit no better than the information presented will allow. Check your work and then check it again. It is critical to stay updated. Even though it is infinite, the most valuable entity on the planet is time. Unfortunately, it only runs in one direction (though physicists point out that physics works regardless of the direction of time), so don't waste it. **DESIGN007**

Reference

1. "Quilter raises $10 million for AI-powered circuit board design, led by Benchmark," Quilter.ai.

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

Keysight Technologies Chosen as Test Partner for Deutsche Telekom Satellite NB-IoT Early Adopter Program

Keysight Technologies, Inc. has been selected as the Test Partner for the Deutsche Telekom Satellite NB-IoT Early Adopter Program, providing an end-to-end narrowband internet of things (NB-IoT) non-terrestrial network (NTN) testbed to enable designers and developers to validate reference designs for solutions using 3GPP Release 17 (Rel-17) NTN standards.

Led by Deutsche Telekom in collaboration with Skylo and Murata, the Satellite NB-IoT Early Adopter Program addresses critical challenges posed by limited cellular coverage and enables data transmission across vast expanses. By incorporating satellite communication, the program accelerates the development and deployment of NB-IoT solutions, offering reliable, ubiquitous, and secure data transmission.

Program participants will gain exclusive access to cutting-edge solutions from Deutsche Telekom, Skylo, and Murata, including early development access to the first Rel-17 standards-based NB-IoT modules and a global satellite network. In addition, participants will receive state-of-the-art hardware, connectivity services, dedicated support, and the opportunity to tailor NB-IoT NTN solutions to their specific needs.

As the Test Partner, Keysight provides program developers with access to the NB-IoT NTN testbed, which is based on Keysight's industry leading NTN test solutions, and to Skylo certification test plans. This ensures that certified NB-IoT-capable devices seamlessly integrate with the Skylo satellite network, offering unmatched flexibility and efficiency.

Jens Olejak, head of Deutsche Telekom's Satellite IoT Program, said, "Keysight was chosen as Test Partner for the Satellite NB-IoT Early Adopter Program based on its extensive collaboration with all market makers, providing unparalleled NTN solutions from the earliest stages of development."

Lucas Hansen, Vice President, Wireless Devices and Operators, Keysight, added, "Keysight's unique position lies in our comprehensive expertise, which seamlessly leverages both cellular and satellite technologies, and by providing end-to-end solutions, including network modeling, protocol emulation, field testing, and application testing. Our NB-IoT NTN solutions set the standard for validating the dynamic capabilities of NB-NTN devices and systems, exemplified by our pioneering role in defining Skylo's test plan."

(Source: Keysight Technologie)

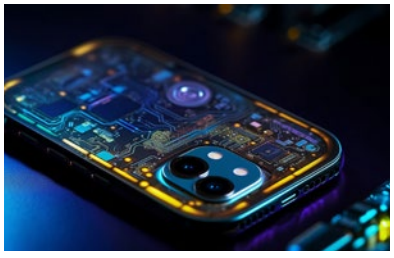
TOP TEN

EDITOR'S PICKS

Cadence Continues 'Left Shift' of SI Functions

At DesignCon, the I-Connect007 Editorial Team met with Brad Griffin, product marketing group director for the System Analysis Group at Cadence Design Systems. In this interview, Brad explains how the Sigrity Aurora analysis tool enables designers to address SI and PI issues early in the design process, before they become costly errors, and why he believes his late co-worker Dennis Nagel would be proud to see his efforts come to fruition.

Every Designer Needs to Understand Embedded



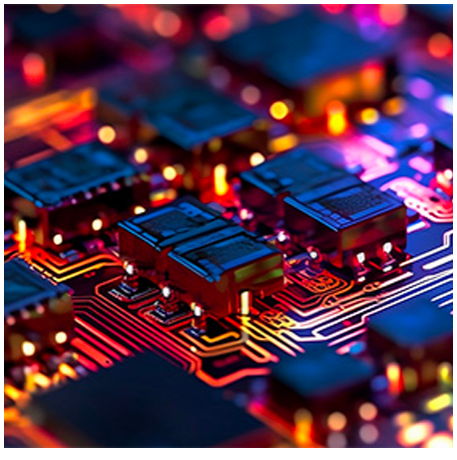
IPC's Kris Moyer teaches design techniques for embedding components, and he's noticed an upswing in his students' interest in embedded component design. We recently spoke with him about embedding component design: best practices, pros and cons, and when it makes sense for designers to start embedding.

Renesas to Acquire PCB Design Software Leader Altium

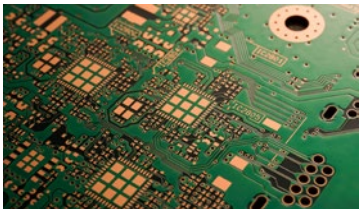
Renesas Electronics Corporation, a supplier of advanced semiconductor solutions, and Altium Limited, a global leader in electronics design systems, announced they have entered into a scheme implementation agreement (SIA) for Renesas to acquire Altium by way of a scheme of arrangement under Australian law.

ASC Sunstone Proudly Sponsors SMTA's Ultra High-Density Interconnects (UHDI) Event

ASC Sunstone, a leading provider of high-quality printed circuit board (PCB) solutions, is pleased to announce its sponsorship of the upcoming Ultra High-Density Interconnects (UHDI) event hosted by the Surface Mount Technology Association (SMTA). The event, titled "Advancements in HDI: Driving Innovation Forward," will serve as a platform for industry professionals to explore the latest trends and technologies in high-density interconnects.



Embedded Design: Materials Matter



The rapid advance of mobile technologies has sparked an insatiable demand for radio spectrum bandwidth. The rush to capitalize on wider bandwidths, higher data rates, and lower latency offered by frequency bands like 5G and millimeter wave is evident across industries.

Imagineering Driven by Customer Demand

During DesignCon, I met with Amir Roy, vice president of business development for Imagineering. Amir discussed the company's high-reliability manufacturing and focus on listening to the needs of their customers. Business is booming and Imagineering is adding new lines to increase capacity.

Beyond Design: Embedded Capacitance Material

Embedding components into the multilayer PCB substrate can have many benefits, including reduced board size and improved signal integrity. However, embedded capacitance material (which is not really a component but rather part of the substrate) can improve power integrity dramatically by reducing AC impedance and generally enhancing the performance of the product. It takes up no additional space, is easy to implement (because it is compatible with standard FR-4 processes), and can be cost-effective.

Elementary, Mr. Watson: Know the Tradeoffs With Embedded Designs

John Watson

One of my great joys as a grandfather of eight is spending time with them at the park. It doesn't take too long until I'm getting stuck on a slide that is too small for me or on the seesaw, with me on one side and them trying to lift me. At that point, they learn some harsh lessons in physics and how heavy Grandpa really is.

Flexible Thinking: Embedded Design—A Term With Multiple Meanings

The concept of distributed capacitance evolved with the introduction of multilayer PCBs in the 1960s and '70s when it was realized that the stacking of multiple layers of circuits, including ground and power, provided an opportunity to incorporate distributed capacitance intrinsically embedded and made possible by the built-in parallelism of adjacent layers which naturally contributed to the capacitance between them.

EMA Design Automation and Hawk Ridge Systems Reshape Landscape with Full ECAD/MCAD Convergence

EMA Design Automation and Hawk Ridge Systems entered a strategic partnership to give customers a full ECAD/MCAD design experience, leveraging best-in-class tools and support for the entire electronic product design flow.

Find Industry-experienced Candidates at jobConnect007

For just $975, your 200-word, full-column ad will appear in the Career Opportunities section of all three of our monthly magazines, reaching circuit board designers, fabricators, assemblers, OEMs, suppliers and the academic community.

In addition, your ad will:

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

Potential candidates can click on your ad and submit a resume directly to the email address you provide, or be directed to the URL of your choice.

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

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

+1 916.365.1727

I-Connect007
GOOD FOR THE INDUSTRY

smt007 pcb007 design007

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

GARMIN

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

GARMIN

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

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

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

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

YOUR JOB AD HERE

For information, please contact:
BARB HOCKADAY
barb@iconnect007.com
+1 916.365.1727 (PACIFIC)

I·Connect007
GOOD FOR THE INDUSTRY

GIVE YOUR TRAINING THE IPC EDGE

Online Instructor-Led

Design Tomorrow's Technology Today

More than 500 students experienced the personalized instruction of IPC's instructor-led courses. Start the year mastering Mil-Aero Applications, Radio Frequency Boards or learning the fundamentals with Introduction to Design I & II.

2024 COURSE SCHEDULE FOR FEBRUARY TO APRIL

PCB Design for Manufacturability	Feb. 20–Mar. 7	T/TH	9 am PT/12 pm ET/6 pm CET	3
Certified Electronics Program Manager	Feb. 27–Apr. 4	T/TH	2:30 pm PT/5:30 pm ET	6
PCB Design II section 1	Mar. 18–May 15	M/W	8 am PT/11 am ET/5 pm CET	8
PCB Advanced Design Concepts	Mar. 18–May 15	M/W	3:30 pm PT/6:30 pm ET	8
PCB Design II section 2	Mar. 19–May 16	T/TH	3:30 pm PT/6:30 pm ET	8
PCB Design I (Brazil)	Apr. 22–May 29	M/W	7 pm BST/ 6 pm ET	6
Top Lead-free Production Defects & Issues – Causes, Remedies & Prevention	Apr. 23–May 2	T/TH	8 am PT/11 am ET/5 pm CET	2

WHAT STUDENTS ARE SAYING!

"The live interaction facilitated asking questions that helped clarify the information."

"The material of this course was great."

"The instructor explained the course in detail, in a way that can be understood by everyone."

"I liked the approach the instructor took for full participation of all students."

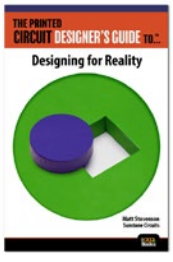
"The recorded lectures help me to review the training materials at my convenient time."

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

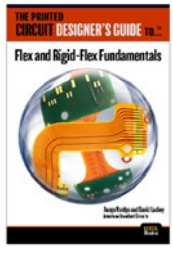
by Matt Stevenson, Sunstone Circuits

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

BUSINESS DEVELOPMENT REP: **JOCELYN DARLAND**
jocelyn@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 © GAGA AI**

DESIGN007 MAGAZINE

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

March 2024, Volume 13, Number 3
DESIGN007 MAGAZINE is published monthly
by IPC Publishing Group, Inc., dba I-Connect007

ADVERTISER INDEX

Problems solved!

SUBSCRIBE NOW