I-Connect007

DESIGN007 MAGAZINE

OCTOBER 2023

Take your flex game to the next level

This guide provides additional insights and best practices for those who design or utilize flex and/or rigid-flex circuit boards.

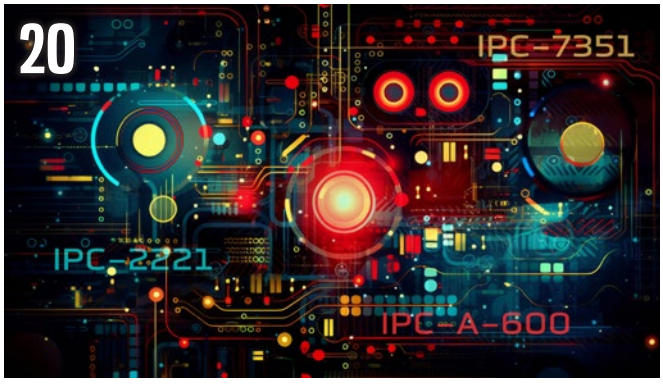
Download

I-007e Books

DESIGN007 MAGAZINE

The Standards of Design

In this issue, our expert contributors discuss how PCB designers can utilize standards to save time and money, not to mention frustration. We also discuss the updated version of the *IPC Checklist,* a handy guide that illustrates which standards cover which topics, from front-end design through assembly.



asc-i.com

WE ARE READY TO BRING YOU THE FUTURE

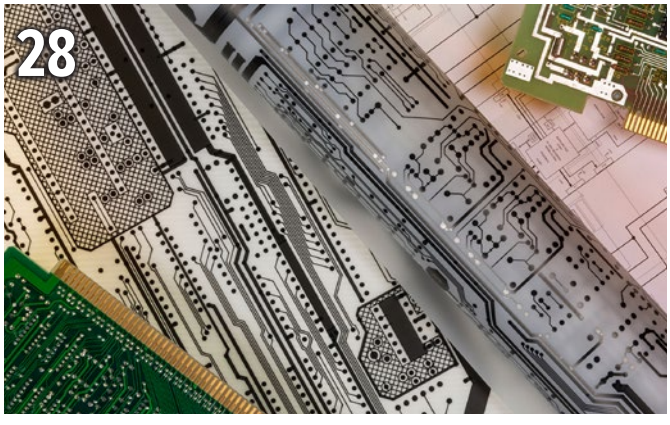
Our new Ultra HDI technology allows us to produce PCBs with parameters never seen before in our industry. With lines down to 1 mil with a line aspect ratio of 1:1 at production volumes, the future is here!

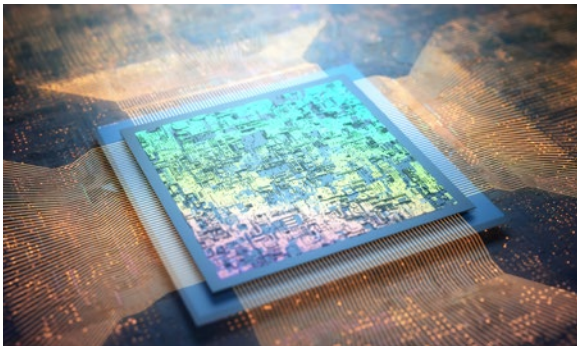
Check out our UHDI capabilities

American Standard Circuits

Creative Innovations In Flex, Digital & Microwave Circuits

DESIGN007 MAGAZINE





EMC

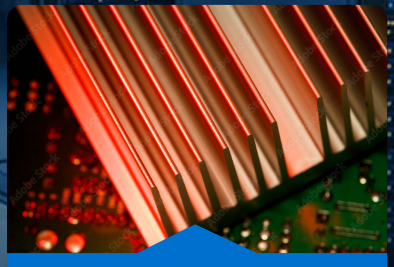
STRIVE FOR **EXCELLENCE**

www.emctw.com | EMC.INFS@mail.emctw.com

Enable 800GbE Ethernet Speed with EM-892K and EM-892K2 Materials

Industry's First Halogen-free Extreme Low Loss (ELL) Laminates

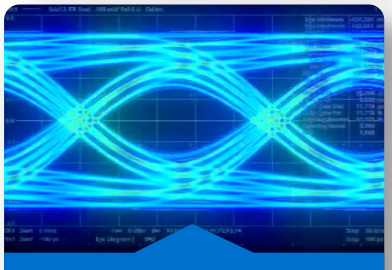
Key Material Features:
EM-892K @70%RC:
Dk: 2.84/ Df: 0.0017 @10GHz
EM-892K2 @70%RC:
Dk: 2.76/ Df: 0.0013 @10GHz
with Cavity Resonator Method



Lead-Free Soldering Test:
PASS LF-260°C/ 10X
with 32L 0.6mm hole pitch

Anti-CAF Test:
Pass 100V/ 85°C/ 85% RH
with 32L 0.6mm hole pitch
for 1K Hours

IST Test:
Pass Room temperature
to 150°C for 3K Cycles
with 32L 0.6mm hole pitch



Advanced Foil Availability:
Both HVLP3 and HVLP4 are
compatible with "Glicap"
non-etch oxide treatment

Hybrid Compatibility:
Fully compatible with EM-370(Z) (Mid. Loss) material for cost reduction and pass LF-260 10X with 32L 0.6mm hole pitch

INSULECTRO

North American Master Distributor
(949) 587-3200 • insulectro.com

What Are the Standards of Design?

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

Some people think standards are boring, but I disagree. If you mention "standards" to a group of PCB designers—or fabricators and assembly providers, for that matter—you'll trigger a lively discussion every time.

Everyone has something to say about standards. As you'll see in this month's issue of *Design007 Magazine,* some designers don't use IPC standards at all, preferring to blaze their own trail. But as our contributors point out, that's like building a house without any plans; why not take advantage of the contributions of subject matter experts and avoid mistakes?

IPC standards can seem overwhelming to anyone who is new to the industry. How many standards are there, anyway? It can be confusing for new designers to determine which standards hold the answers they need. Plus, most recent grads say they get almost no information about standards in school. In this issue,

our experts discuss some ideas for making standards more accessible to new PCB designers and design engineers at the college level.

To help shed some light on the world of standards, IPC has released an updated version of the *IPC Checklist*, a handy guide that illustrates which standards cover which topics, from front-end design through assembly. One section explains the entire chain of production, while another details every single step of a typical PCB design cycle. Managers can hand this to new hires on their first day and say, "Read this."

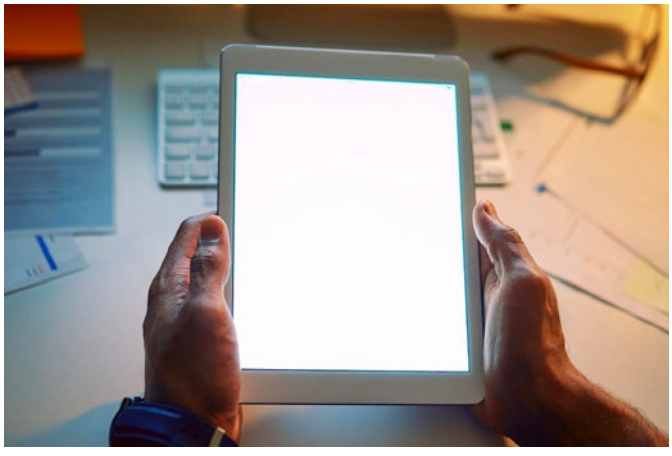
We think the *IPC Checklist* is so useful that we have published it as an I-Connect007 eBook, available by clicking here. Check out the checklist.

This month, we begin by talking to John Watson, who is working to put IPC standards into the hands of his PCB design students at Palomar College. Columnist Dana Korf discusses the new standards for additive manufacturing. Jen Kolar and Cory Grunwald discuss how Monsoon Solutions built its internal documentation process around the relevant IPC standards. Gerry Partida discusses the repercussions when designers don't follow IPC standards. Kelly Dack explores the *IPC Checklist*, and Barry Olney discusses how standards can give an extra advantage to high-speed designers. IPC's Patrick Crawford explains which standards should be top of mind for PCB designers, and Matt Stevenson breaks down his criteria for selecting the right PCB design tool.

It's show time, and we'll soon be covering SMTA International and productronica. 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.

Game-changing New Tech to Make AR/VR, Cellphone and Laptop Displays Better



Two researchers at the University of Waterloo have combined their innovations to create a suite of new technologies that will increase battery life by 30 percent and lower the cost of manufacturing. The new technology could be used in smart glasses, watches, tablets and other products.

The game-changing technology developed by Dr. William Wong and Dr. Manoj Sachdev, professors in Waterloo's Department of Electrical and Computer Engineering, allow for the creation of MicroLED and flexible displays for next-generation AR and VR headsets and micro-displays. This new technology will increase optical brightness as the energy-efficient displays provide up to 30 percent power savings while lowering the cost of manufacturing.

The technology has also led to the launch of a new start-up, Lumatus Semiconductor Inc., led by Wong, Sachdev and Mario Montana, an engineering alum. The technology has broad applicability in various display types such as LCD, OLED, and MicroLED. To date, a family of foundational patent applications in various jurisdictions have been filed.

Both the Canadian and U.S. governments have announced significant funding initiatives to support the development of robust North American semiconductor manufacturing and supply chains to underpin national security resiliency.

(Source: University of Waterloo)

IPC Standards: Ticket to a Safe Passage

Feature Interview by the I-Connect007 Editorial Team

John Watson, CID, is a customer success manager with Altium who also teaches PCB design at Palomar College in Carlsbad, California. He's noticed that most of his students are primarily new to the industry and not familiar with IPC standards. In his classes, he teaches how to use these standards while trying to help students access the documents more easily.

When we met with John, we asked him to talk about the importance of understanding IPC standards, the price you'll pay if you decide not to follow them, and his plans for getting new designers on board with standards as early as possible in their careers.

Andy Shaughnessy: John, many of your students have never had an introduction to IPC standards and don't know why they could help them with their designs. Where is the disconnect?

True, but there's an underlying issue. There's another group of designers who don't follow any standards. When I ask them what standard they're following, they respond that they don't follow standards. But that's like building a house with no measuring tape. I look at PCB design and manufacturing as an all-in-one package, where it's combined with a fabrication drawing and an assembly drawing; it's a contract and bill for services.

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

IPC standards are the drivers of the fabrication process. They guide designers and explain what they should be doing. When I ask these designers why they don't use them, often they'll say, "I want to reinvent the wheel." I'll tell them, "Do you know how difficult that is? You have an entire system and structure here of standards that have been developed through experts in the industry, and you want to ignore them?"

Barry Matties: At the very least, that unnecessarily elevates risk.

That's right. You're asking a fab house to build a board not based on any guidelines. If you don't aim for anything, you're bound to hit it. That's what can happen if you don't have a guideline to direct you when you're designing the PCB. It's a recipe for failure.

Matties: Why aren't these designers following the standards? Do they see them as being too expensive, or maybe they don't know how to access what they need?

Pricing may be an issue, but it doesn't explain the whole problem. We must do a better job of exposing new designers to IPC standards. When I started my PCB design career 25 years ago, I took a course at Palomar College (where I'm now teaching), and the very first thing my fantastic instructor Bill Brooks did was hand us IPC standards. It was my introduction to the whole area of standards, and everything developed from there.

Let's be honest: IPC standards are not exciting reading. It's not like you curl up on the sofa with a glass of wine and an IPC standard, but it is important reference material; designers must understand what's in there and where to find what they need. The standards will guide you.

When a designer says he's going to reinvent the wheel and not follow standards, I ask, "Do you understand that the fab house is following that standard? That's their guideline for building your board and you need to understand those standards, so you know what the board shops are expecting." If we help new designers understand these standards early on, that would be a big step.

Matties: Essentially, they're trying to create their own standard.

They may get their design fabricated, but that doesn't make it right. I was talking to a company recently who told me, "We don't really have a standard that we follow. We tried IPC standards, but it's not our thing."

We must do a better job of exposing new designers to IPC standards.

Shaughnessy: Where should a new designer start with IPC standards? What's the "starter pack," so to speak?

Start with IPC T-50, which covers terms and definitions, because you must understand the terminology involved. It's vital that everyone is on the same page about terms we're using and what those definitions are. Next is IPC-2221, the basic PCB design standards. Then I would loop in IPC-7351 for libraries and IPC-7251 for footprints.

They're both vital, but I would lean toward implementing IPC-7351 because the library is everything. I learned this years ago when I set up a library and didn't follow any standards. If I had followed IPC-7351, I could have avoided a lot of problems. It gave me the answers I needed and explained how I should have done things in the first place.

One company I worked with was having library issues with only 300 components. I asked them what they would do when they got to 50,000 components. I got a view into their library, and they had set things up based on default designators. I couldn't tell what was what. It was a great example of a company that wanted to do things their own way. I feel like I'm back in the '60s: "Hey, man, we can go do our own thing. Cool!"

Shaughnessy: After Woodstock, they should have all started following standards.

Right. I had the pleasure this past year of sitting on the steering committee for IPC-2221, and it was an interesting experience to work with a roomful of experts. As we went through the standard, the input from these individuals was so amazing; we're capturing real-world experience and documenting it in the standards. Everyone should attend one of these meetings, even if you don't contribute; just sit and listen because you can learn a lot from these experts.

Matties: How do you know which standards you should be aware of and which ones to investigate? What's the process for finding the correct standard?

I use the IPC Standards Tree, which is available on the website. There are branches for PCB design, materials, and assembly. This guide directs and helps me find the standards for that topic. It's all there: components, embedded technology, cleaning, solderability, and support.

Matties: If you were coming up on a job that uses a new material, would you first go to IPC to look at material standards, or would you go to the vendors? What standards play into that equation?

I would start with the material itself. What are the specifics? Then go into, say, slash sheets, followed by speaking with the vendor. It's all part of one big ball of wax that we call information. It's also important to understand the

John Watson

changes that have happened in the industry because the standard may be a few years old. Take that into consideration. IPC committees are updating the standards as quickly as they can, but it's a combination of standards and vendor information.

Shaughnessy: Maybe IPC could offer a class called "Intro to IPC Standards?" Now, when someone first starts designing boards, how do they know what to make of the standards? How do you get your foot in the door with them? Most EE grads say they don't learn much about standards in college.

Warning, here's where I talk negatively about my own industry. I see that often, as designers, we just shoot from the hip. We look at it completely wrong by telling ourselves, "We're just connecting the dots and the board works." Just because it works doesn't mean it's right. There may be problems inside the PCB design that haven't raised their ugly heads yet. That's where standards come in.

Shaughnessy: Seems that designers don't know because most of them don't visit board shops.

They don't. How can they design PCBs without visiting a fab house? I asked one of our really

good fab houses here in the San Diego area, "What percentage of your PCB designer customers have ever toured your facility?" Their answer: About 10%. It illustrates how important it is to get standards into designers' hands by any means possible.

Matties: Maybe a designer who attends a session at IPC APEX EXPO gets a free standard?

That would be great. The second issue is letting the industry know about the importance of IPC and their standards. IPC is not just a standards organization; they're a teaching organization with their own design and manufacturing certifications. We need more people to volunteer to work with a committee; I guarantee you'll learn a lot.

We need more people to volunteer to work with a committee; I guarantee you'll learn a lot.

Matties: How do we get the industry to participate more in driving the use of standards, especially for new hires?

That's a tough question. You have to volunteer to participate in this industry, and that gets down to your motivation. Why do we do this? What do we like about this job and this industry? As a professor, I want to get the standards into my students' hands; every student should have a standard. Maybe for $40 a student could get three standards? That would be a good start. As a professor, I'm trying to develop a chapter of IPC Design sponsored by the college. With 20 students, the college would cover the chapter fee, and we could put three standards into their hands.

Matties: Who are your typical students? Are they all brand-new designers?

It's mostly brand-new, first-time designers, with some coming from other industries who want a career change.

Matties: That's interesting. Why do they find your class so appealing?

I've been spreading the word through marketing, messaging on LinkedIn, and several national podcasts. But, honestly, a lot of the interest comes from the support from I-Connect007 for me and my classes.

Matties: Beyond the programs, what's the appeal for a young person to want to be a circuit designer today?

I find that they often want a career change to something that will make them more money. That's a real attraction. I have conversations with students the first week about what they should expect when they leave this class, including income. But they also see it as an exciting career; they get to solve a puzzle and they like the intrigue of design.

Shaughnessy: Do you have any final thoughts on this topic?

Don't dismiss the importance of these standards. Without standards, you're trying to cross the ocean without a boat. Take advantage of the experience inside the standards, and don't try to reinvent the wheel. It's all there.

Matties: This has been really good, John. Thanks so much.

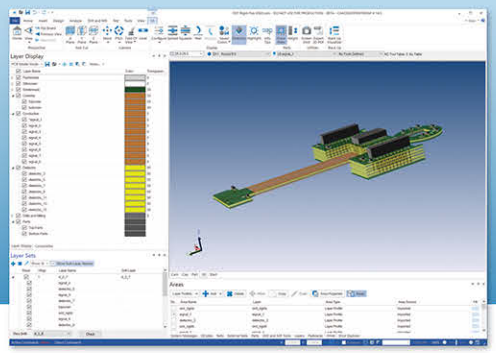
Thank you. Always a pleasure to speak with you all. **DESIGN007**

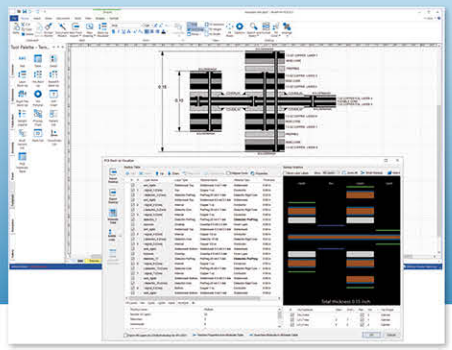
Support For Flex, Rigid Flex and Embedded Component Designs Now Available.

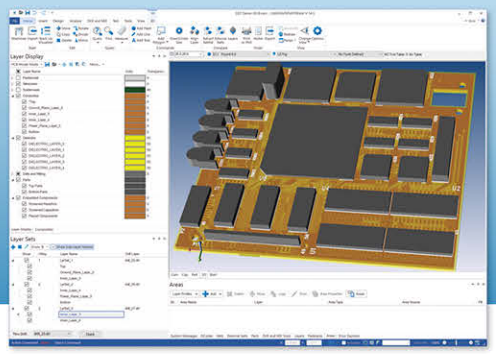
BluePrint-PCB®

CAM350®

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







DownStream Technologies

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

IPC Launches Additive Standards

Dana on Data

Feature Column by Dana Korf, NANO DIMENSION

An important element of design requirement determination and manufacturing requirements involves qualification, performance, and inspection standards. These are critical to ensure that the product's electrical, mechanical, and reliability requirements are met without expensive over/under specification.

Additively manufactured electronics (AME) technology is progressing from proof-of-concept, quick-turn circuit board production into volume shipped circuits. These are true pure additively manufactured (AM) PCBs which may incorporate formed 3D passive components, antennas, coax, and twisted pair interconnect.

As volumes increase, users are asking what industry specifications these can comply with and how production volume quality and reliability will be validated. For example, they are looking at how to populate the following drawing note (see Figure 1).

"This PCB shall comply with the Class (1, 2, or 3) requirements of IPC-ABCD and IPC-A-XYZ, unless otherwise noted."

There currently aren't any AME performance, qualification, or acceptance standards published by any international standards organization.

There are very good IPC standards in place for rigid, flexible, and rigid-flex boards for design, performance, qualification, and acceptance. So, why doesn't AME just use them? The reason is there are many substantial differences between how AME and traditional PCBs

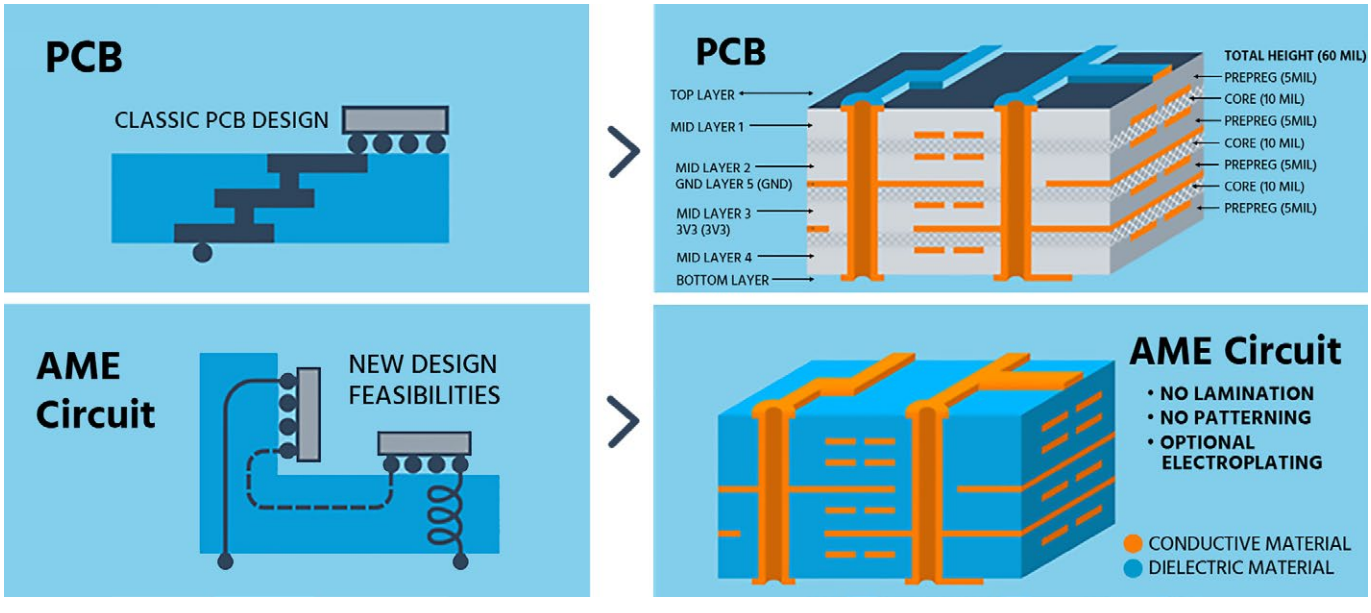


Figure 1.

APCT
Passion | Commitment | Trust

ADVANCED CIRCUITS
A Division of APCT

One Company | One Team

THE APCT DIFFERENTIATORS

TECHNOLOGY CAPABILITY

- **Rigid Through-Hole**
  ~ Up to 40+ Layers
- **HDI; Blind/Buried/Stacked Vias**
  ~ Up to 8x Sequential Lamination Cycles
- **Flex Applications**
  ~ Up to 6 Layers
- **Rigid Flex Applications**
  ~ Book Binder Capability

LEAD TIME CAPABILITY

- **Corporate Cycle Times**
  ~ Consistently faster than industry standards
- **Premium Cycle Times**
  ~ Quick-Turn capabilities are “World-Class”

APCT: WHERE OUR PEOPLE ARE THE DIFFERENCE

CAPABILITY | CAPACITY | EXPERTISE

Give Us The Opportunity To Earn Your Trust And Gain Your Loyalty

APCT Leading the Printed Circuit Board Industry

APCT.com | 4PCB.com

are manufactured; Table 1 presents some key differences.

New IPC Standards Development

The IPC TAEC recently approved the request for the formation of a new subcommittee (D-67) and three new task groups to create three initial standards. As their basis, these standards are derived from the existing IPC-6012, IPC-6013, IPC-A-600, and IPC-2221 standards.

IPC-6905 Qualification and Performance Specification for AME

Many users of AME boards consider them to be a traditional PCB or as a component to potentially mount onto a PCB. IPC-6905 requirements are derived from the existing PCB IPC-6012 and IPC-6013 rigid, flexible, and rigid-flexible qualification and performance specifications. Requirements are removed that are not applicable for AME boards (Table 1). Requirements for potential AM-specific failure mechanisms are added, such as slice-to-slice voids and delamination.

Embedded formed, packaged, and unpackaged components, along with surface components, are typically fabricated in these structures. The goal will be to reference the IPC-6017 Embedded Standard requirements and test methods as much as possible.

IPC-6911 Acceptability of AME

This is an important document as it will provide visual surface and cross-sectional acceptance and rejection requirements. Its structure is derived from the IPC-A-600 Acceptability standard. The task group may potentially acknowledge AI inspection and real time process adjustments. As with IPC-6905, views have been removed from the IPC-A-600 standard that are not applicable for additively manufactured boards. Additional views will be included to match the new IPC-6905 requirements.

IPC-B-XX AME Coupons

IPC has been using 2D Gerber file-based coupons for decades. These are documented in IPC-2221 along with associated data files. These coupons are based on manufacturing processes for horizontally routed conductors and vertically drilled/ablated plated holes. AME does not remove material and has

Table 1: Key differences between how AME and traditional PCBs are manufactured

Function	Traditional PCB	AME Technology
Create traces and planes	Add copper foil, then etch unwanted copper	Print conductive ink just where required
Vertical interconnect (vias)	Drilled/ablated vertical plated holes	Print conductive ink on required angle
Interconnect routing	Horizontal layers + vertical holes	3D (start, rout, and end anywhere)
Add dielectric	Add core and prepreg	Print and cure/sinter inks
Back-drilled holes	Drill out unwanted hole section and pads	Don't print conductor
Via-in-pad microvias	Laser drill and plate from target to capture pad	Rout trace from under component pad
Protect traces during assembly	Add solder mask	Print dielectric over conductors
Board form factor	Top, bottom, and edges	3D surfaces

3D-formed passive components, such as coils, 3D antennas, coax, and twisted pair wiring along with vertical traces routed in the X, Y, and Z axes. This standard will create new 3D coupons that will be utilized as the test vehicles confirming AME performance and used as qualification test vehicles. AME boards may consist of functional components and interconnect which present a challenge because testing will advance from interconnect reliability testing to incorporation for functional component parametric validation.

The Teams

The subcommittees and task groups have a wide breadth of volunteers from Europe, North America, and Asia to form these standards. Industries such as military, PCB fabricators, consumer, AME equipment manufacturers, and AM material suppliers are represented. IPC is always looking for more volunteers to review and develop standards to ensure they accurately specify industry requirements for electronic interconnects. Join the teams that are developing these new standards for this exciting AME technology. **DESIGN007**

Dana Korf is the AME standards manager at Nano Dimension where he is responsible for working with the industry to create additively manufactured electronics (AME) standards. He is chair of IPC D-67 AME subcommittee and vice chair of the ASTM F24-07-06 AM Applications, Electronics subcommittee.

Dana is the principal consultant at Korf Consultancy LLC, working with companies to improve PCB fabricator front-end engineering processes. He works with OEMs and ODMs to create design rules and technology roadmaps, assist with supplier selection/qualification, and reduce DFM cycles. Dana previously worked for more than seven years in China at Multek in Zhuhai as senior director of manufacturing engineering and NPI and at Huawei Technologies in Shenzhen as director of PCB technology, where he was responsible for PCB technology ranging from mobile phones to RF antennas, base stations, and high-speed digital servers and switches.

Before that, he worked for Samina-SCI, HADCO, and Zycon as director of product engineering. He has been awarded the IPC President's Award, chaired many high-speed IPC committees, and was a co-chair for the iNEMI Data Convergence project, which became IPC-2581. Dana graduated from Washington State University with a BSEE, and he enjoys following college and professional football and golf. To read past columns, click here.

A New Twist on Rechargeable Battery Performance

For decades, researchers have assumed that the inevitable filmy buildup on electrodes inside rechargeable batteries is the driver of performance loss. Now, we know that view is backward.

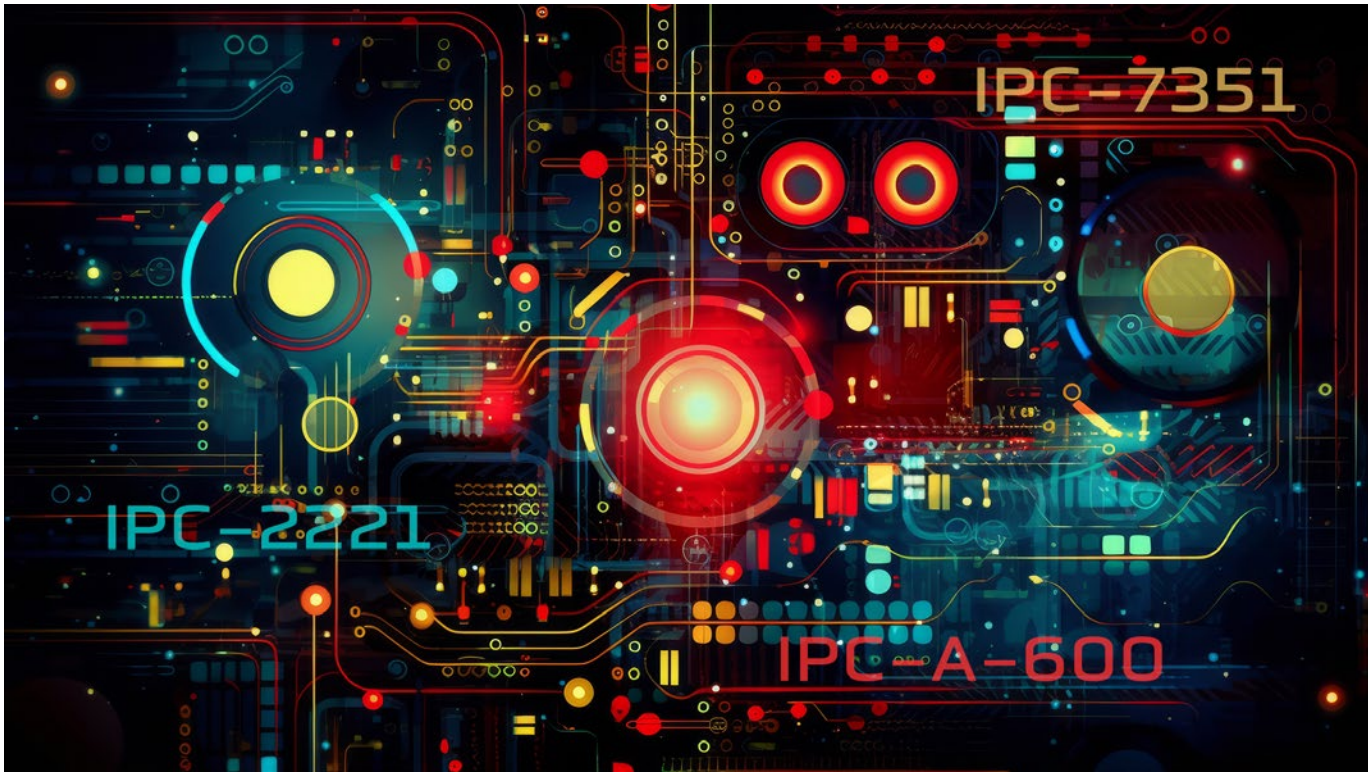
A study, led by a research team at the Department of Energy's Pacific Northwest National Laboratory, shows that the so-called solid electrolyte interphase (SEI) is not an electronic insulator, as previously thought, but instead behaves like a semiconductor.

The first direct measurement of the electrical properties at the boundary between the solid electrode and the liquid electrolyte inside a rechargeable battery is reported today in Nature Energy. The research solves the long-standing mystery of how SEI functions electrically during battery operation.

Researchers focus on this SEI layer, which is thinner than a sheet of tissue paper, because of its outsized role in battery performance. When batteries are new, the SEI forms on the first charging cycle and ideally remains stable during the battery's expected lifespan. But a look inside an aging rechargeable battery often reveals substantial buildup of solid lithium on the negative electrodes. Battery researchers have assumed that this buildup causes the performance losses.

The group's measurements revealed that as voltage increases in the battery, the SEI layer in all cases leaks electrons, making it semi-conductive.

(Source: Pacific Northwest National Laboratory)



A Standards-driven Design Environment

Feature Interview by Andy Shaughnessy
I-CONNECT007

For this issue about standards, we interviewed Jen Kolar, VP of engineering, and Cory Grunwald, director of designer development, at Monsoon Solutions, a design bureau in Bellevue, Washington. In this conversation, Jen and Cory explain how they created the company's internal documentation process around the requirements of IPC standards. They also discuss their onboarding system for new hires, including the standards that the new designers should become familiar with.

How about if you start by discussing your protocol for new hires, and the standards that your onboarding process is built around?

Cory Grunwald: Sure. When bringing on a new designer, we have a lot of internal Monsoon documentation and standards that tend to work specifically for ourselves and our customers. If we're teaching our new designers how to do footprints, we've got our own quick reference guide on footprint creation, as well as references to IPC-7351. The designers know that if they can't find it in our internal documentation, they can go to the IPC standard. We usually provide the quick reference guide for our documentation and a handful of good-to-know IPC specs for designers, such as IPC-7351, *Generic Requirements for Surface Mount Design and Land Pattern,* and IPC-2221, *Generic Stan-*

OrCAD X

NEXT GENERATION PCB DESIGN

THIS EVOLUTION OF PCB DESIGN EXPANDS OUR SERVICES TO CREATE A TOOL THAT IS FASTER, MORE AUTOMATED, AND MORE CONNECTED THAN EVER.
DESIGN FAST, CORRECT, CONNECTED.

GET STARTED TODAY >>

EMA Design Automation®

Cory Grunwald

dard on Printed Board Design. Another standard we provide is IPC-A-600, *Acceptability of Printed Boards Endorsement.* It covers class 2, class 3, and what's acceptable in board production. While we don't expect new hires to read through all the standards, we do expect them to know what's in them and where they can go to find more detailed information.

Your internal documentation is largely based around IPC standards?

Jen Kolar: Yes, because reading through standards isn't necessarily user-friendly. So, we will have one document that includes the standard's image and numbers, and then we can say, "Here's exactly what you're looking for." We condense it down to what we really need to think about day to day.

Grunwald: That's right. There's not a lot of direct design-related information in the IPC standards, like how to handle impedances, so much of our internal documentation is more design-focused than manufacturing-focused.

We were joking before the interview about whether there was a "starter pack" of standards for new designers. That seems to be IPC-7351, IPC-2221, and IPC-A-600, right?

Grunwald: Yes, those are definitely the ones I look at the most as a designer.

Kolar: All our designers earn IPC's Certified Interconnect Designer (CID). We usually have them wait a couple of years to get their feet under them before they get their CID+, but it is a requirement that everybody has their basic CID. That said, you don't really see designers walking around the halls saying, "This standard is for this or that." Usually, designers are asking things like, "What's the distance I need to have for my copper to be pulled back from the edge for this particular fab shop?" Most people don't know everything in the standards, but we need to know where to look if we have questions.

We have libraries that we've gone through and built for certain components, which are kind of "golden libraries" that designers can follow. We've essentially taken the standards and put together documentation that distills them down and gives designers what are essentially checklists and user guides.

Cory, I understand you created Monsoon's internal documentation. Tell us about that.

Grunwald: I gathered all the information and set up a training plan. Much of that internal documentation was compiled before I got here; basically, we go into more detail about tools and specifics in our documentation, such as footprints, specific layers, and naming styles that we use. We've set up smaller internal groups of designers, so new hires have someone they can check with and ask questions.

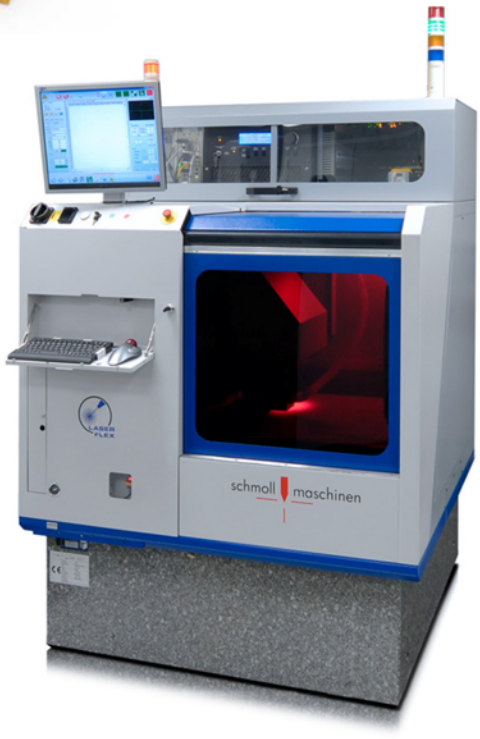
Kolar: In addition to general manuals and working documents, we have tools with specific guides, and those significantly speed up the ramp-up time for designers learning new tools. We also have designers give presentations regularly—at the weekly design meetings, for example—on ways to do things more efficiently and different ways to use the tools.

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

Jen Kolar

Cross-pollination really helps with our efficiency.

Grunwald: I prepped for this call today by reviewing the IPC standards in our system. But they're only numbers; it's up to whoever is looking at them to know which ones have the information they need: "Here is this document for assembly, and here are the documents for design and manufacturing." But if you don't know, they're just numbers. IPC focuses very much on the assembly and manufacturing processes, and those specifications roll down to designers to help them make something that is manufacturable.

You have configured the relevant standards into your internal documentation system. I imagine other service bureaus, or even bigger customers, would have some similar sort of documentation processes in place.

Kolar: Sure. For some of our bigger customers with large design teams, they have documented standards around their libraries. I have seen multiple incarnations of documentation standards, specific to libraries, for instance.

All this begins with onboarding on the first day? Are your new hires generally experienced designers or complete newbies?

Kolar: It's very hard to find experienced designers who work well in the world that we're in, where we're constantly changing the kind of cutting-edge work we do over a wide variety of designs. If somebody has been doing design for a specific type of thing for a long time at one company, they may not have kept up with what's happening in the industry. It can be hard for them to transition from the way they had been working internally to a service bureau mindset. We've seen many really struggle with it.

We've ended up hiring designers right out of school with just a little bit of experience. Maybe they've had one other job, have messed some with design, and then they discovered that they really liked electrical engineering work. We'll train them because that's where we've had the most success vs. trying to bring in someone who's really seasoned.

In the past, it had typically been either me or our COO saying, "Here's a practice project." They would go through, work on it, ask us questions about what's in the documentation, and then somebody would mentor them as they work with the customer, learning as they go.

Let's talk about acquiring IPC standards. How does this work as an IPC member?

Kolar: There are certain standards that are "packaged," and we prepaid for the standards that we care about. All of this happens digitally. It has helped that we have a new project checklist that includes questions that you should ask your customer when you're starting a project. This helps designers avoid saying, "Oh, by the way, I just realized that this is class 3 and 100 volts."

Grunwald: It's a good idea to go through the specs listed on IPC's website, where you can usually get the first couple of pages of the stan-

dard at no charge with a breakdown of the specs and who the subject matter experts are. We're lucky enough to have good relationships with the fabricators and assembly companies that we work with so that we can ask them questions.

Kolar: A lot of times, our questions are usually about fabrication capability because of some limit that we're trying to push to meet. We also work with assemblers on a process that's non-standard, or one that has really stringent requirements, maybe for flight or space, and we see that the customers is really worried about reliability.

Sounds like you are dialed in on the whole standards process. What improvements would you like to see as far as standards and specs from the industry or IPC?

Grunwald: It would be nice to know more manufacturing requirements as far as the copper tracing or copper spacing, part sizes, etc., but those are so technology-based by the fab shop that you're working with, so I don't know if that's even possible. But if I could have anything, it would be to have a spec sheet that lists, "If you have half-ounce copper, here's the spacing you should go with," and so forth.

What advice would you give new designers when using standards?

Grunwald: Get familiar with them. You don't need to memorize them, and you don't need to read every page, but get familiar with the handful that I mentioned. You need to at least know where to find the information.

Kolar: It's worth familiarizing yourself with the more important standards. Be sure you have physical copies of the standards; that's definitely valuable. New designers should talk to fabricators whenever they have questions. I've seen these newer designers go down rabbit holes on slash sheets or materials. It's far better to just ask your fabricator than this rabbit hole of trying to specify everything and understand all the different material properties; that can be a waste of time.

This has been great. We really appreciate it.

Kolar: Thank you. Good to speak with you all. **DESIGN007**

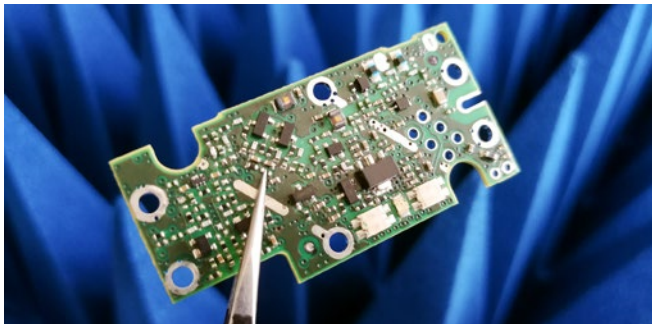
IDTechEx Explores How EMI Shielding is Enabling Compact Electronics

It is no secret that electronic devices are becoming increasingly compact, with greater functionality contained in smaller volumes. As such, increasing efforts are being made to mount integrated circuits (ICs) and other components such as antennas closer together, sometimes within the same semiconductor package.

IDTechEx's report "EMI Shielding for Electronics 2024-2034: Forecasts, Technologies, Applications" explores the current status and technology trends within this essential aspect of many electronic circuits.

Conformal package-level shielding is especially important for consumer devices where both ness and wireless communications are needed. These include smartphones, smartwatches, and AR/VR headsets.

At present, sputtering is the dominant method of creating conformal EMI shields. Emerging methods such as spraying and printing are gaining traction and offer much lower equipment costs since no vacuum chamber is required, along with additional benefits such as reduced variation in package top and side coating thickness and fewer process steps. (Source: IDTechEx)



MilAero007 Highlights

BAE Systems Signs Agreement to Acquire Ball Aerospace ▸

BAE Systems announced a definitive agreement for the proposed acquisition of Ball Aerospace for approximately $5.55 billion.

Flexible Thinking: Growth of Flex and Flex-hybrid Electronics in Mil-Aero Applications ▸

Over the past several years, flexible electronics (FE) and flex-hybrid electronics (FHE) have enjoyed heightened attention in the electronics industry and have seen special interest and attention given by mil-aero companies. This is evidenced by June's NextFlex conference titled "Hybrid Electronics Commercialization Path for Aerospace Applications," an event at Boeing's Seattle facility. It is thus worth considering some specific examples of FHE applications in the mil-aero environment.

Lockheed Martin Australia Selected as Australian Defence Force's Strategic Partner for AIR6500 ▸

Lockheed Martin welcomed the announcement by the Department of Defence on being selected as the strategic partner to steward AIR6500 Phase 1 (AIR6500-1).

Jet Appeal ▸

In 2007, Stratos Aircraft set out to design and manufacture an exciting, high-flying new product, which they estimate is still at least five years away from going to market. With a hefty capital investment, they are now working on their second prototype: a six-passenger, single jet engine plane that cruises at 41,000 feet, with a range of around 1,500 nautical miles, and a comfortable cabin pressured to an altitude of 7,600 feet.

IPC Welcomes U.S. House Veteran Rich Cappetto as Senior Director of North American Government Relations ▸

IPC welcomes Richard Cappetto, a 14-year veteran of the U.S. House of Representatives, as its new senior director for North American government relations.

Northrop Grumman's B-2 Capabilities Enhance its Digital Communications ▸

Northrop Grumman Corporation, in partnership with the U.S. Air Force, successfully completed an integrated airborne mission transfer (IAMT) demonstration with the B-2 Spirit at Whiteman Air Force Base as part of the ongoing modernization efforts incorporating digital engineering.

NASA Selects Small Businesses for Orbital Debris, Surface Dust Tech ▸

NASA has selected six U.S. small businesses to receive nearly $20 million in total to advance technologies to address two challenges in space exploration: orbital debris and surface dust.

Boeing Partners With the World Energy Council to Advance Aerospace Energy Transitions ▸

Boeing has become a Patron of the World Energy Council, marking the first aerospace company to partner with the organization to drive sustainable energy transitions.

In a hurry?
We're the fastest in the business!

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

Prototron Circuits

America's Board Source

Serving the industry for over 35 years | prototron.com | (520) 745-8515

Choosing the Right CAD Tool

Connect the Dots

by **Matt Stevenson,** SUNSTONE CIRCUITS

Every profession has its tools. Painters use brushes, mechanics have socket wrenches, and parlor magicians have long sleeves. For PCB designers, the tool of the trade is CAD software. Designers spend hours, days, or weeks in their CAD tool for each circuit board design. They thoroughly check each connection, plot every through-hole, and carefully place components. This process inevitably reveals that not all CAD tools are created equally.

Each designer has different needs from a CAD tool. Some only produce a few designs per year, which means there is limited opportunity to learn the ins and outs of complex CAD software. Others constantly work on PCB design, moving from one design to the next with barely enough time to catch their breath.

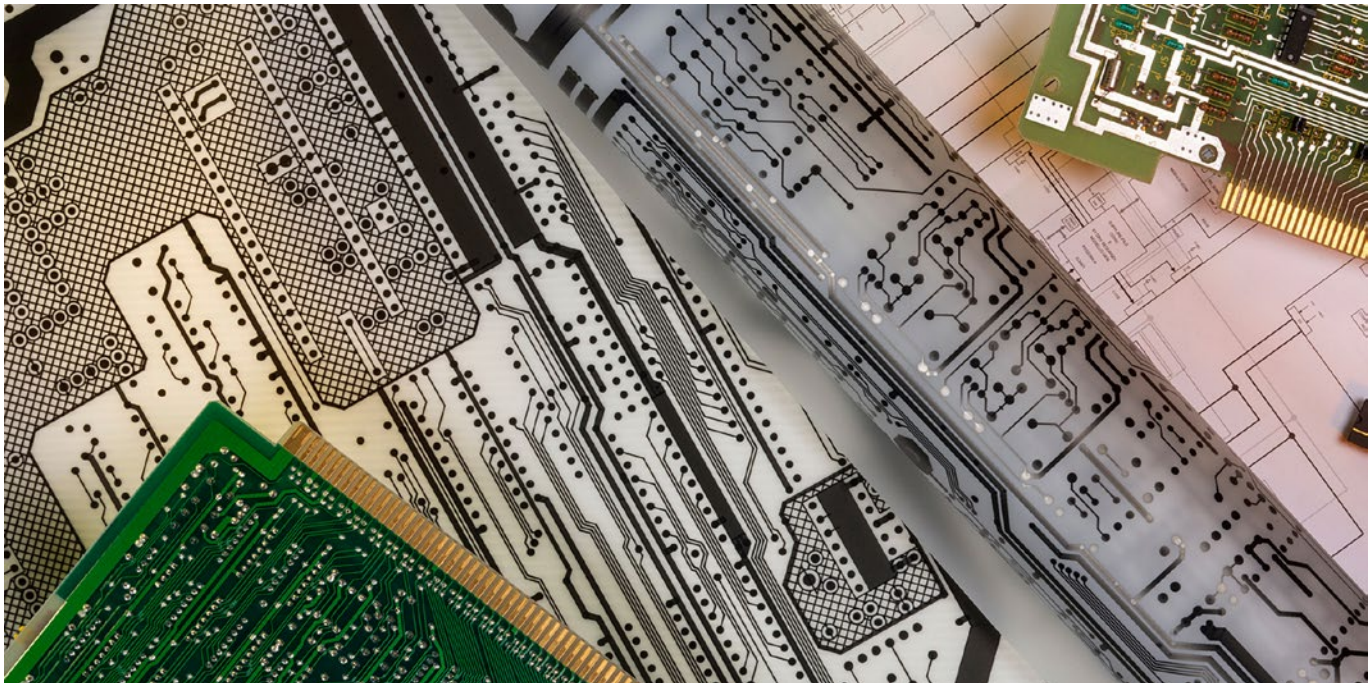
How can designers find the right CAD tools to fit their design needs?

Making Sense of the CAD Tool Ecosystem

A quick search reveals that the world of PCB design is flooded with tools, but evaluating more than a handful can be daunting. To narrow the field, examine some basic criteria.

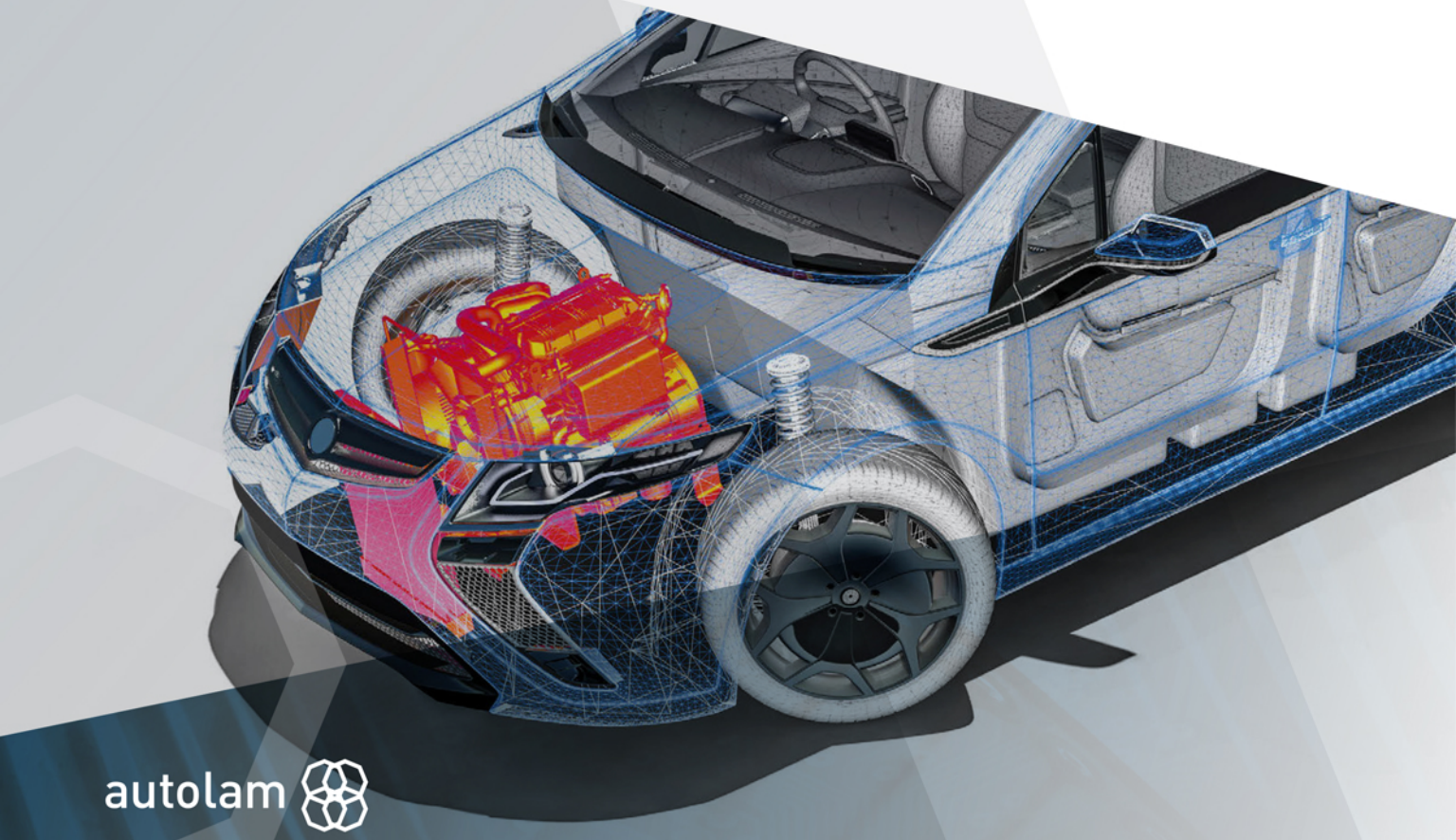
Price

For those who design PCBs intermittently, high-priced tools may not make sense, and there are many low-cost and free CAD tools



ventec
INTERNATIONAL GROUP
騰輝電子

autolam: Base-Material Solutions for Automotive Electronics

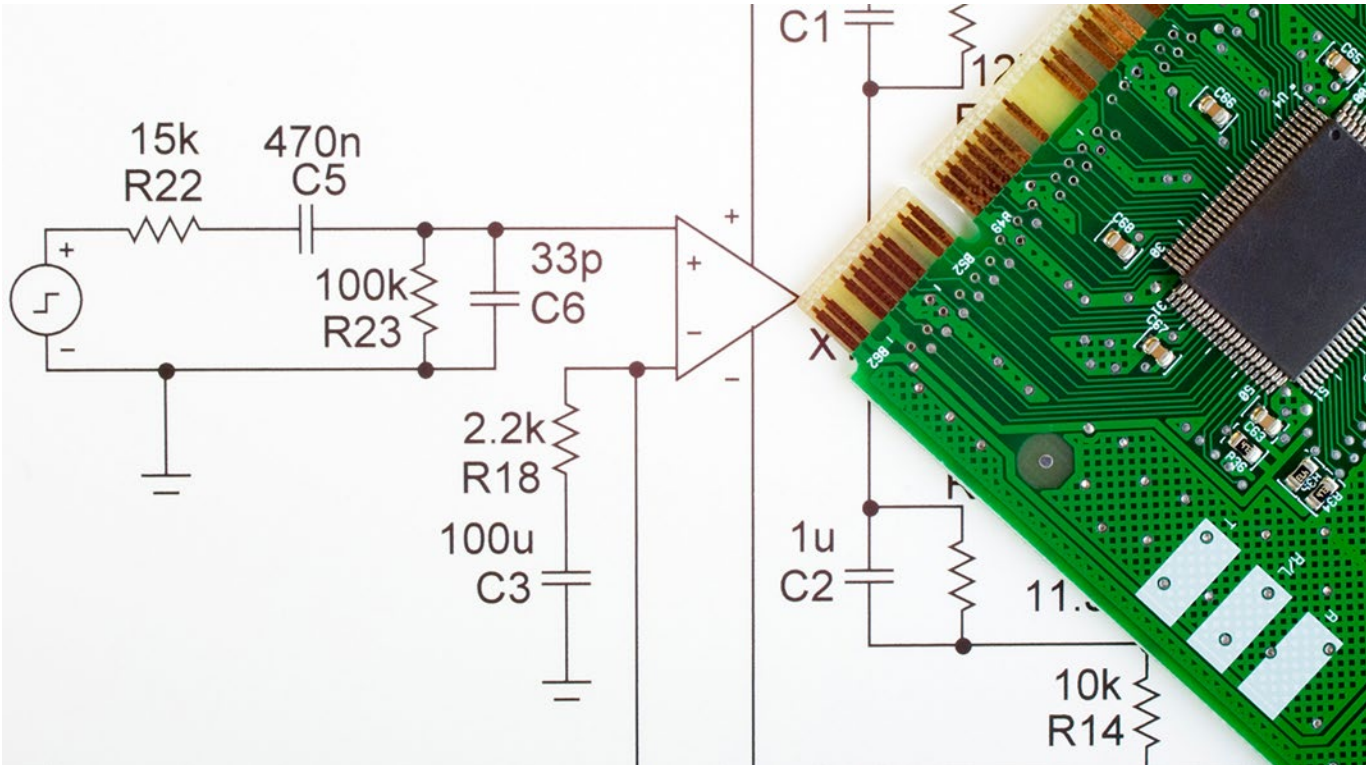


autolam

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



available. When evaluating free tools, pay close attention. Some will be open source or community-run projects, and while the philosophy behind them might be attractive, make sure those 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. However, this choice can also lock in a specific manufacturer path, so choose carefully. Does the manufacturer's CAD tool cost money? If so, how much? Can it provide Gerber files?

Commercial packages can quickly become expensive. There are a few CAD tools available that charge a monthly subscription price, but most offer prices ranging from about $75 all the way to over $2,000. Pay attention to the limits placed on products with multiple tiers of pricing and be particularly careful not to get caught in a "freemium" trap, where a piece of software seems great until that really important feature is locked behind an expensive paywall.

Component Libraries

CAD tools are almost always customizable with libraries full of parts and components. It is important to be sure that the tool being considered can support the parts required by the design. Many tools like to brag about the size of their parts libraries, but don't be fooled by a big number.

A typical designer might need a library of about 10,000 parts, but when looking at the library offered by a CAD tool, confirm that the requisite parts are available. Look for commonly used parts and check whether the tool comes with proprietary libraries or access to specialized libraries required for the project.

No matter the CAD tool, eventually designers end up creating components themselves. Make sure the tool provides the capabilities needed to both create new components and add them from external sources.

Complexity

CAD tools exist to tackle the complex problem of designing PCBs. Unfortunately, it's inevitable that the CAD tool will itself be somewhat

complex, but even so, it should be intuitive and easy to use as it is.

Ease of use is an area where less expensive, open-source software tends to lag behind. If price is an important factor in the decision-making process, the result can be a balancing act between price and usability.

Help and Documentation

There will come a time when expert help is needed to move a PCB from design to manufacture. Make sure that the CAD tool provider offers user support early in the process. If there is a phone number for support, call and ask some questions about the product. Live support can come in very handy with more complex designs.

CAD tool software providers should make their documentation available online. Quality documentation can be a huge help to designers who need questions answered in a hurry. Even YouTube tutorials and user forums can offer solutions to common issues with a CAD tool.

Checking the tool's documentation and looking for video tutorials can also help you evaluate a CAD tool when deciding what to use. If there are too many people reporting problems, that might mean the tool is too difficult to use. Likewise, a YouTube tutorial that takes the viewer through too many convoluted steps can be a red flag toward usability.

Additional Considerations

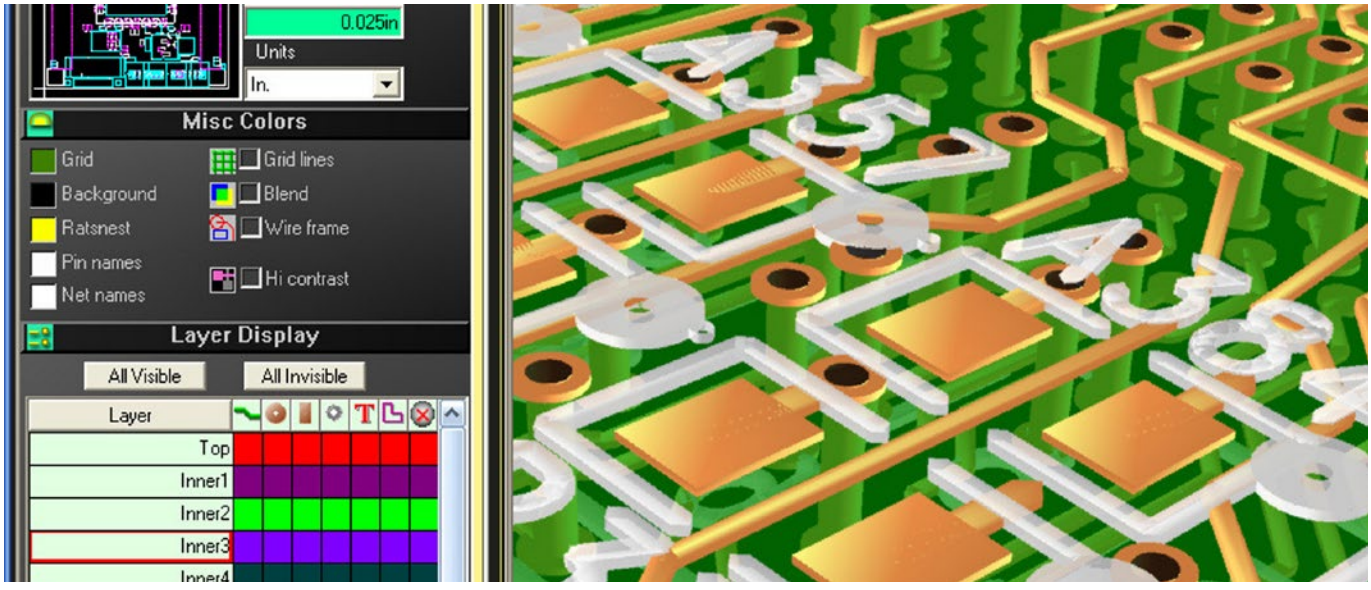
CAD software is almost always highly customizable with design rules, favorite libraries, shortcuts, and modules to increase productivity and make the design process personalized and special.

Pay attention to limitations, such as board size, and the number of pins or layers a tool can support. Watch for limits on minimum component pitch and the supported sizes for traces, spacing, and vias. Once you have narrowed the field, download the CAD tools that look the most promising and try them out. Fifteen minutes or so should be enough to tell whether the tool is intuitive and will fit your PCB project.

CAD Tools Must Be Easy to Use

The most important part about choosing a CAD tool, though, is that it shouldn't make designing too much of a chore. You want designing to be productive and efficient and, in the end, maybe that's the most important element to evaluate when you're trying out different tools. **DESIGN007**

Matt Stevenson is vice president at Sunstone Circuits, a division of American Standard Circuits, and the author of *The Printed Circuit Designer's Guide to... Designing for Reality.* To read past columns, click here.



Got Questions? Standards Have the Answers

Feature Interview by the I-Connect007 Editorial Team

The I-Connect007 Editorial Team recently spoke with Gerry Partida, vice president of technology at Summit Interconnect, and co-chair of the IPC-6012 committee. Gerry has been involved with standards development for years, and he believes that adhering to IPC's standards and specifications has been a big part of Summit's success.

In this interview, Gerry explains how Summit utilizes standards, manufacturing data, and information to reduce respins and scrap while increasing yield and profitability. He also discusses the repercussions that can occur when designers don't follow IPC standards. As Gerry points out, everything you need to know is in the standards, so why not follow them?

Nolan Johnson: Gerry, we've spoken before about Summit's use of manufacturing data, including microvia modeling. Tell us about how this all works with DFM rules and IPC standards.

Yes, we try to use data and information to make good decisions. We've been talking over the last couple of years about modeling microvia reliability. But we also use our ERP systems to

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

Gerry Partida

collect data, and then look at that data for our yields. If there's a defect, we can track it to see what has caused it. Then if we buy new equipment to solve and bring that defect rate down, we can track it to see whether our investment is going through to work it out. COO Sean Patterson believes that we are a technology company that owns board shops, so we want to use technology and the tools available to manage and do things in an intelligent manner.

Traditionally, the board industry would just do another re-spin and not analyze what is working and what is not, what the yields are, etc. At times I'll get a very challenging design from a customer who didn't follow IPC rules, and we'll say, "No, this is not to the reliability that you require." They'll reply, "We've been getting these boards built for years," and I'm thinking, "I understand what you're saying. You've been receiving good boards, but you do not know what is happening to make the quantity of your good boards." Their fabricator may have had to throw away 50% or 75% of the boards that they produced to get that 100% deliverable.

I said, "Let's redesign it, but let's follow the IPC design rules." Often, people will say that the IPC rules are old and out of date, but they probably haven't read them recently, because the rules have been modified and updated. Every year, we work to improve all the standards. If you follow them, and design it that way, you'll find that you get two things: a much more reliable product in the field, and the yields for the fabricator will probably be better too. In this instance, we simply followed the IPC rules. We looked at the challenges, how they wanted the boards built, and we changed certain things like the surface finish. We didn't move any components. We didn't change the number of laminations—I guess it was still five. At the end, their yield went from 25% to 88%.

Of course, customers may not know about this, because a lot of the time fabricators don't share their struggles and pains. But this is reality for PCB manufacturers. If they drill down, they ask, "Why is the yield so bad?" That's where it becomes a problem within the fabrication segment. Fabricators look at their processes for documenting collected data to analyze it, and when you do that, you can see, "The fallout is here, and it's because of your design." Then the customer asks, "What can we do about the design? The cost is too high. What can we do?"

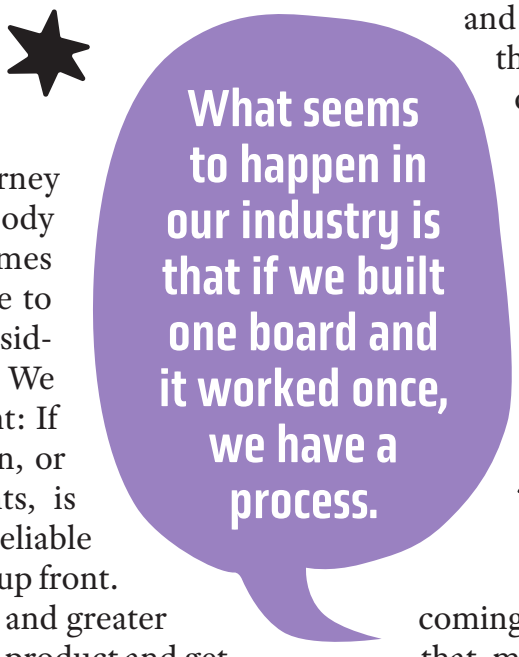
Barry Matties: Do you have an example of a rule they weren't following that you did follow?

Yes, here's one of my favorite rules to share with designers: removing the non-functional pads and routing the traces closer to the drilled hole where there's more room, because when they're routing it, they don't necessarily see the hole. They remove the pad, and now there's all this room. Within IPC-2221 and IPC-2222, there's a rule that says, "Do not remove the pads to make more room." You route with the pads in and remove them afterward. It will take care of a lot of design problems. Some design guidelines from component manufactur-

ers will say that this is what you do. I've seen design guidelines and app notes that literally say to remove the non-functional pads and run the traced formulas away from the drilled hole. But app notes are not reliable.

Matties: It sounds like we might just be in a rut because it works the way we've always done it. There's often no one challenging the status quo or coming back like Summit and saying, "Hold it, you need to look at this differently."

What seems to happen in our industry is that if we built one board and it worked once, we have a process. Somebody once gave me a great explanation: Sometimes you have to cross Death Valley, and as long as you get there, the journey was successful. But nobody considers how our part comes out exactly right. You have to take these things into consideration. How do we do it? We have collaboration up front: If we put this into the design, or change these requirements, is that a good functional, reliable product? We all agree to it up front. We will have higher yields and greater success when we finish the product and get it into the field.



Matties: IPC has the standards. What can IPC do beyond just providing standards to help designers better understand this?

That's the challenge that we face within the standards community. This is where you all do a great job of socializing these things and making people aware that there are rules. Designers need to understand that there are reasons that we have rules, and if you break the rules, here's what happens. When we write in IPC-A-600, for instance, that this is the minimum dielectric, this is how you measure, this is acceptance, and this is non-conforming, there's a reason. When we socialize this stuff and share that the yield went from 25% to 88%, then that rule starts to have some weight and impact. You know, if you've suffered with poor yields and gone through multiple suppliers and you still run into the same things, maybe it's not the supplier.

Matties: That's an interesting perspective.

We've posted some blogs on our website to make the customers aware of these things: This rule is allowed, and here's what happens when we go outside the rule. Here's the fallout and here is the reward if you follow the rules. But people read the rules differently.

I had a phone call yesterday from a customer asking if we had changed the rules. No, we clarified a rule because there was confusion about what was right. It was a clarification of what existed. It's hard for people to understand, so we blog about this on our website to help socialize this incident.

By the way, IPC-6012F is coming out soon, with many changes that make it easier for us to clarify a lot of things and make them easier to understand. For example, with cavity boards, there were no documents on edge plating. Is it acceptable or not? Now, in IPC-6012F, we have rules for different types of cavities. You have to design certain things a certain way to make cavities successful. But there's no perfect situation; if you do a cavity board, there's usually bleed-out, and you have to accept bleed-out.

Matties: This type of thing needs to be communicated to the industry.

Exactly. When I was first blogging on LinkedIn,

I had to explain something so many times out of frustration, "Hey, these are the rules, and if you don't follow them, I'll give you examples of what happens." Now, when somebody calls me with a question, I can reply with, "Let me give you this write-up." It helps them and gives them something to fall back on.

Matties: We know that many designs go through respin after respin. How do you help your customers reduce respins?

It's important for them to collaborate with the fabricator, whoever it is. It's important for them to understand their IPC design rules. One classic mistake is not having enough annular ring for the class 3 that you said you want. You may have a 7-mil annular ring, and yes, that's fine for the finished hole size, but the rule is based on the mechanical drill that I have to use to finish at that size. If it's immersion gold or ENIG, you've lost two mils of annular ring. If we can have that collaboration and share our expertise with them as they're laying it out, it reduces failures and respins.

If we can have that collaboration and share our expertise with them as they're laying it out, it reduces failures and respins.

For instance, with rigid-flex boards, IPC-2223 says that vias have to stay 125 mils away from the transition area from the rigid to flex. Customers ask if they can go down to 100 mils or 30 mils. No, but you can go to 200 mils all day long. Here's what happens: Designers see the big strip of open area, and they think, "Let's put vias there." But land voids exist there, and if you drill and hit a void, the chemistry will go in and short out to another plane. If the clearance was just another 20 or 30 mils away, this would never have happened. Or you end up with final drill in the transition area that's too close to the bend area, and those are the ones that fail. The whole landscape gets affected. But if you know the rules and apply them, it's like butter.

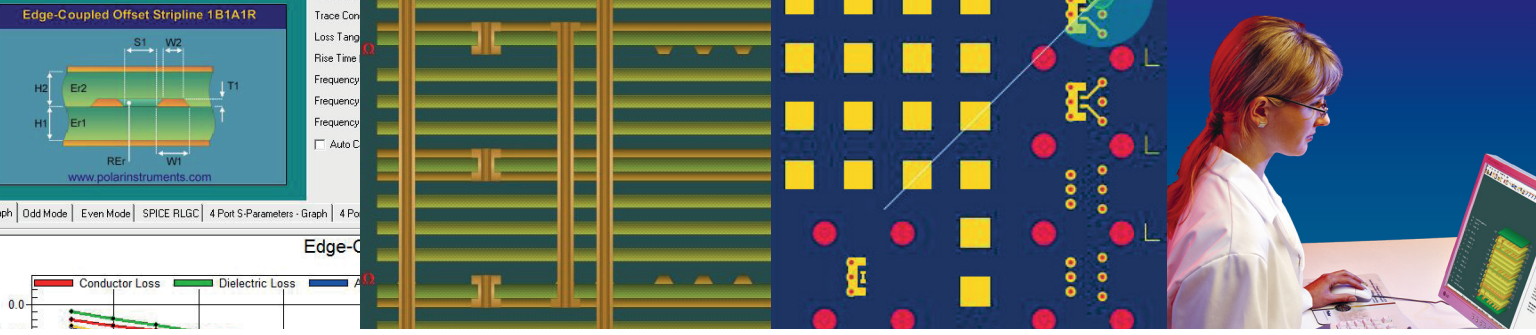
Designers would benefit greatly if they read the design specs too. They don't have to memorize them; just know the sections. When they think they need to bend a rule here or there, they can go read what the rule says, possibly even look at IPC-6012, and see what the fabricator has to say. If it looks like a challenge, then call a fabricator and ask if they can do this.

Matties: Do your customers come to you requesting feedback about their designs?

Sure. I had a customer who asked if we could review six part numbers that had been under development for three years. They were going to production now. This was our last chance to change anything, because after that it's going to be flight-certified and then it would be impossible to change. I thought, "Man, now I have to do more homework."

This is where we use intelligent data. I wanted to look at stackup. What was the major fallout cross-section? But our automation team can provide all of this, and it was very valuable to the customer when I said that we could do this systematically. Our IT guy said that we are really programmers who data-mine our ERP software. On a post-DFM build, I just put in the tool number, hit "Make a report," and I get all that information in about a minute: Yields are 95%. Here are the questions, and make sure you fix these so we don't have these same questions again. Go back and fix your drawing, because a lot of times, they don't, and we'll have the same question on the next rev.

Matties: That's one of the greatest challenges the industry faces. Obviously, you have resources, but when you consider the smaller companies, many of them won't have resources for systems like this.

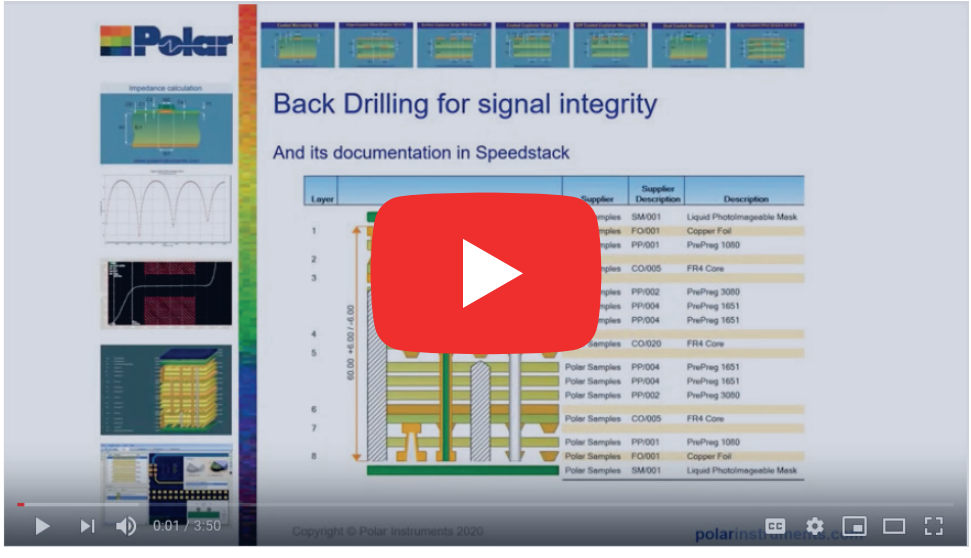


SUBSCRIBE

Subscribe to the Polarinstruments YouTube channel for helpful impedance and stackup videos

PCB Signal integrity tools for design & fabrication

- Impedance & insertion loss modeling with Si9000e
- PCB stackup design & documentation
- Test systems for controlled impedance & insertion loss
- Application notes on a wide range of Si topics



polarinstruments.com

That's where it's going to be tough for smaller operations. If you can use the data, knowledge, and intelligence about what will work and not work, you can find out that you never should have gone through Death Valley with that design because it was just going to crash and burn. If we have information and data that tells us a certain outcome, then we can make intelligent decisions during the quoting process and the collaboration process. This is what we can do to make it work.

The most fun part for us is being approached by and working with PCB design tool suppliers. They're very active, but their worldview is a little further upstream. They have tools that take DFM rules and apply them to a design. But they'll have like 50 rules on a ledger, and they're killing me trying to set it up. I think the PCB design tool companies need to have a better understanding that there are different rules for a class 2 board and a class 3 board. There are rules for half-ounce copper, one-ounce copper, and two-ounce copper. You can't use one rule for all of them. This is because they're not seeing how the board is actually manufactured, and it becomes a very huge attribute list.

The most fun part for us is being approached by and working with PCB design tool suppliers.

Matties: What do you think is the preferred design data transfer format?

Gerber is old and archaic, dating back to 1973. The ODB++ format is a good solution that the industry has been using for at least three decades. We're in the third revision of using IPC-2581, which is an open format. There have been two other attempts at a standard not owned by anybody. People seem to want to use 2581 for stackups, outlines, and CMM downloading. From my experience as a CAM application engineer, when there's an open format, people have different interpretations of the protocols. Different implementations can interpret things differently, and when that happens, we create scrap. We're now in rev C of IPC-2581. Every time a new revision comes out, it takes a year or two before everybody updates their tools.

Matties: It's my understanding, from a small survey that we did a few years back, that manufacturers prefer 2581 over any of those other formats.

To my knowledge, out of our seven factories, I don't think we've built one board with 2581. Yes, the designers still primarily use Gerber or ODB++.

Matties: I've really enjoyed this conversation. Is there anything else you'd like to add?

This is my 39th year in the industry, and I've never had as much fun as I'm having right now. It's fun and it's a challenge. Every day, I use everything I've ever learned, starting with my first electronics job in cable TV amplifiers, where you had to make amplifiers that went from 50 to 550 megahertz, which is low frequency now. But trying to get circuitry to behave the same way between that range of frequency was not easy. Now we're into gigahertz, and your RF signals are all over the map. It's challenging, but it's fun every day.

Matties: Gerry, thank you.

Any time. I enjoyed it. **DESIGN007**

IPC BUILD ELECTRONICS BETTER®

PCB DESIGN COURSES

DRIVE QUALITY AND INNOVATION

PCB Design for Manufacturability 9/12 to 9/28	**Introduction to PCB Design I** 10/10 to 11/16	**Introduction to PCB Design II** 10/9 to 11/29
Introdução a projeto de PCB II IN PORTUGUESE 9/25 to 11/20	**PCB Advanced Design Concepts** 10/9 to 11/29	**PCB Design for Military, Aerospace & Other Extreme Applications** 10/10 to 11/16

PCB Fab and Assembly Troubleshooting

- PCB Troubleshooting & Defect Analysis
  Led by Mike Carano — September 12 to October 5

NEW

- Troubleshooting & Defect Analysis for Electronics Assembly
  Led by Jim Hall & Phil Zarrow — September 19 to September 28

Learn more about our PCB Design Courses.

What the Heck? A PCB Tech Spec Check

Target Condition

Feature Column by Kelly Dack, CIT, CID+

In 1972, learned the adage "Measure twice, cut once" from my seventh-grade woodshop teacher, Mr. Fenoglio. To this day, I hear his voice every time I use a pencil to mark a piece of wood that I'm ready to cut. I mark it and then re-measure the edge to which it will join before sawing or drilling. Over time, I've saved a lot of money avoiding costly personal woodworking and building project errors.

I wish I could say that my PCB design record over the years was similarly as error free in regard to form, fit, and function. Why didn't I hear Mr. Fenoglio's voice when I was dimensioning that CAD drawing? How could I have overlooked that I was drafting in a different scale? What about when I imported a "metric" DXF outline into my "inches" PCB layout and thought there was still plenty of room to add more circuitry? How about that time I built that card-edge connector library part and reversed the A and B sides? I've lost count of how many times I didn't catch the reversed pin numbering on some of those little three-pin SOT23 transistors.

The fact is that effective PCB design and manufacturing is not done sitting alone in a cave. Stakeholders in PCB project development must have ways of checking each other's work before the chips are cut, so to speak. Printed circuit assemblies contain thousands of geometric points which must successfully interface with unseen "next assemblies" on a worldwide scale in order to fit together correctly while running down the manufacturing conveyors. Designing a PCB form to fit is only part of the project equation. After the design is manufactured and assembled, it must perform its function electrically. The characteristics

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

Every day we get questions like those. And we love it. We have more than 500 PCB experts on 3 continents speaking 19 languages at your service. **Regardless where you are or whenever you have a question**, contact us!

What's your PCB question?
www.ncabgroup.com/pcb-design-mistakes/

Reliable answers. Reliable PCBs.

NCAB GROUP

of signal integrity are affected by the physical geometry. The form and fit of the selected materials must be calculated, measured, and adjusted for their various compositions before any material is printed, etched, pressed, bonded, drilled, plated, coated, or soldered.

Designers Must Look Outside of Their Caves, But Where?

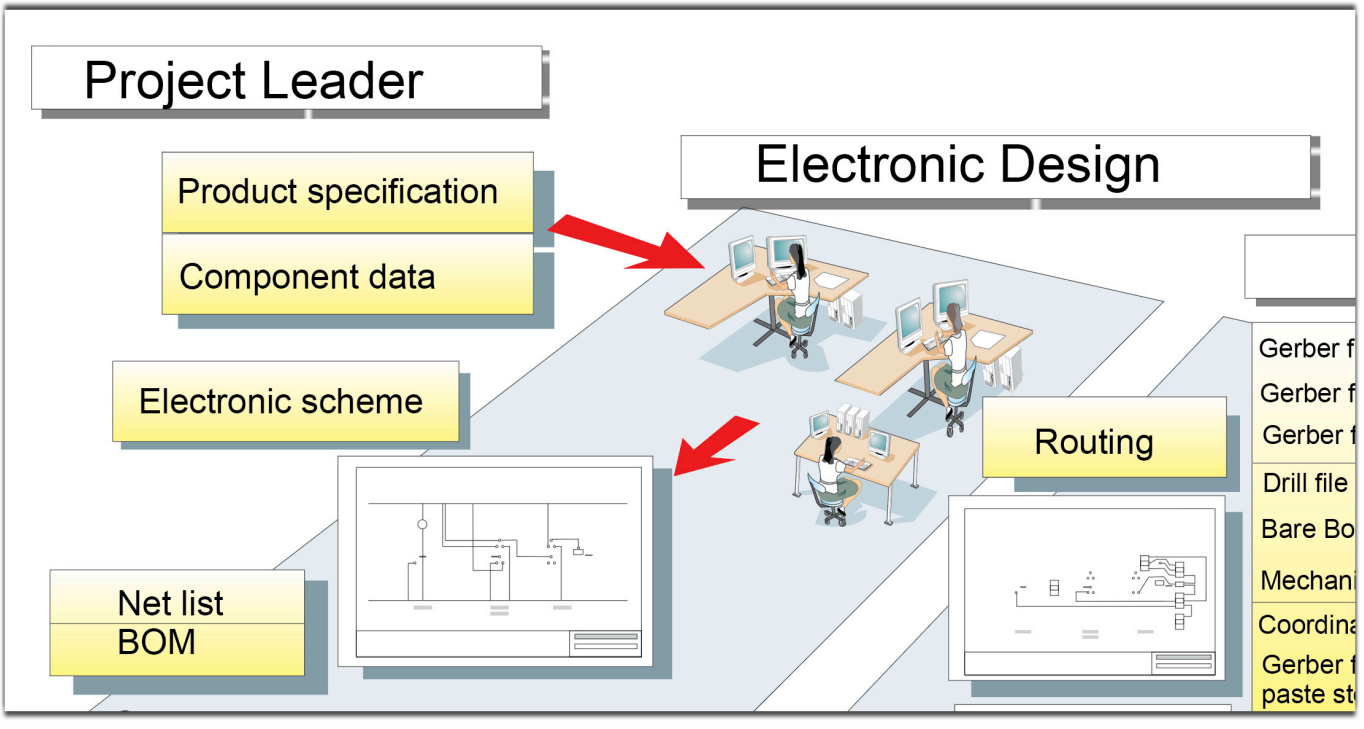
Now more than 40 years later, my woodshop teacher's maxim seems to be drowned out by a new wave of PCB industry social media influencers who constantly remind me to "always check with my fabricator." Some bluntly warn me, "Never trust component datasheets." Still others will push their own recipe for the "perfect" footprint to prevent an 0201 chip capacitor from tombstoning. With all due respect, can these sources for design wisdom be trusted? Do their methodologies consider all the manufacturing process stakeholders? Does their content only show how to get their design portion of the job done with the caveat, "But check with your fabricator," as an admission that the content you've been watching for the last 15 minutes may not be producible under many circumstances? Just search a few PCB design topics on social media. How can there be dozens of "perfect" tiny chip capacitor footprints when the geometries all vary, and the critical considerations of solder deposition and thermal profiling are not even mentioned? How can a PCB designer check in with a fabricator when it is unknown where the design will be produced in volume?

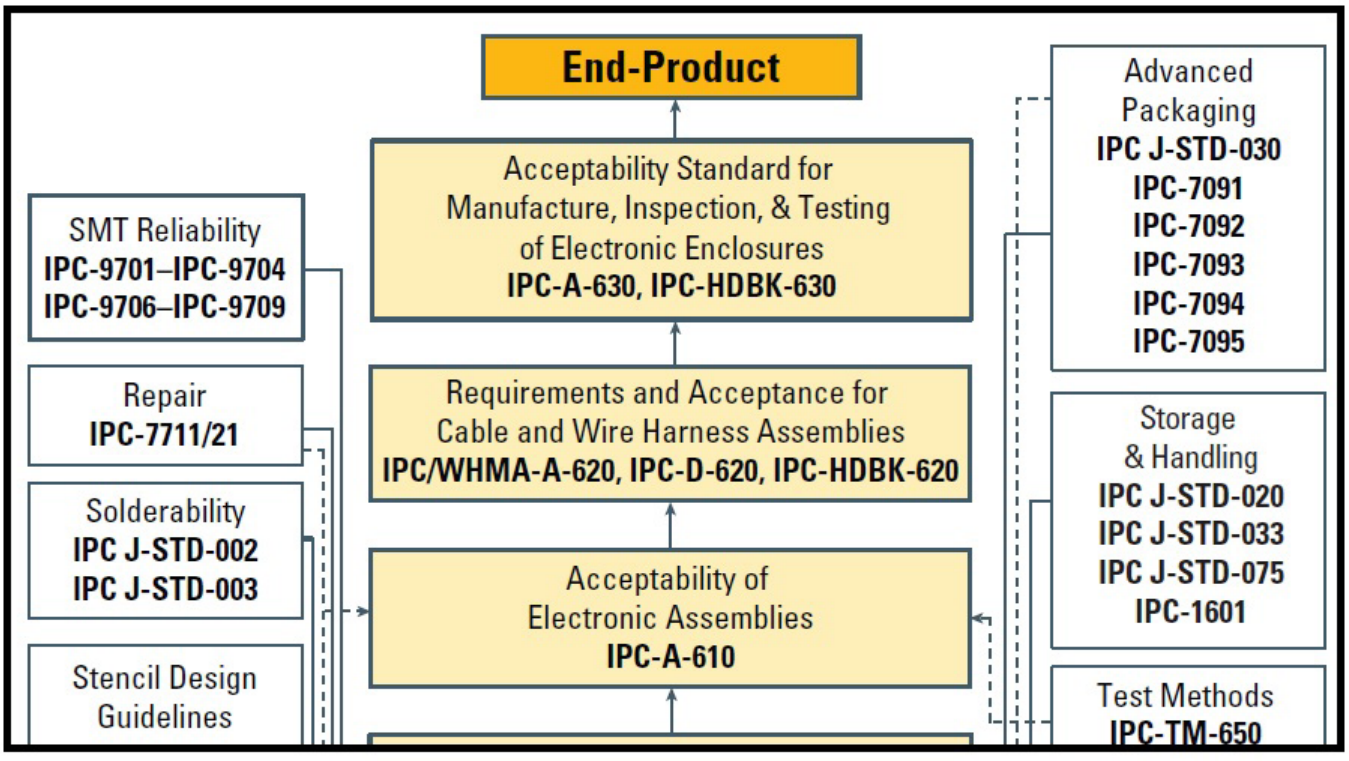
When it comes to a successful PCB design for production, the first step is for designers to admit that, in most cases, they do not know where the boards they design will be fabricated or assembled. The next step is realizing that for the world to stay in business, all the manufacturing process stakeholders have built their process success around the ability to meet IPC standards.

The final step is to become knowledgeable about IPC standards—not only those which apply to the PCBA design in process, but the ones which need to be appropriately specified in the manufacturing process notes.

PCB Tech Spec Check

Most every day I come across customer PCB design documentation showing severe gaps in PCB designer familiarity with outside materials and manufacturing processing. To make matters worse, the fabrication and assembly





notes often reference specifications intended for design, not manufacturing. The numbers and class designations are often inappropriately assigned or non-existent. I often make the point that every other stakeholder in the PCB industry adheres to a process and checks their output to a spec. Why should the process of PCB design be any different?

PCB design must be considered a process by the PCB designer and must therefore be checked to some appropriate specifications, but which ones? Based on the wide variety of designs I review from our customers across North America, designers could use some help in this area.

I've recently come across a useful key to help solve this issue. IPC has published a PCB checklist guideline titled *IPC Checklist for Producing Printed Board Assemblies.* This document not only serves as an effective roadmap pointing designers to existing PCB design and manufacturing specifications, but who is responsible during the various stages of engineering, manufacturing, and test. After previewing this checklist, let me tell you a story about how I put it to use right away.

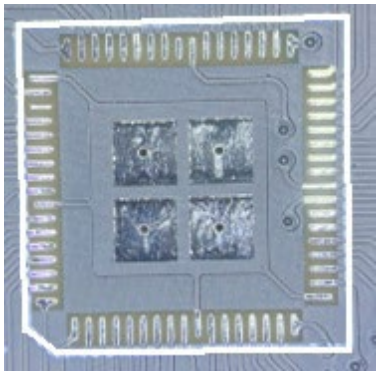
Our assembly line was having a challenge assembling a QFN part in which the PCB designer must have "dreamed up" a land pattern to attach the part to the PCB using a complicated window array, presumably to reduce the solder deposition. The footprint was "designed" but to what standard and with what manufacturing parameters in mind? The land pattern incorporated several "creative" technologies, including via-in-pad, solder-on-pad, and special "windows paning" features to attach the part. DFM rules appeared to be ignored for every technology addressed on this footprint. Our manufacturing engineering team exclaimed, "What the heck?" My response was to perform a PCB

Checklist at the Project Start Level

E	Responsible	Demand on Assembly	IPC Standard	Information	Check
24	Electronic Designer	QFN on the Assembly?	IPC-7093		

design "PCB tech spec check" using this newfound document. I first needed to find the category for the technology in order to check the guidelines.

I quickly found QFN in the "Demand on Assembly" column and proceeded to look up the specs relevant to issues found by our manufacturing engineers. First, I could see that this challenge had to do with a quad flat pack-style component including a large heat sink "slug" and otherwise referred to as a "bottom termination component" or quad flat no-lead (QFN). I went directly to the "Project Start Level" checklist to see how the responsible electronic designer could have easily referenced IPC-7093 to see more information on how to design this part footprint for better DFM.



Example of a problematic footprint.

I won't address every DFM improvement here that I was able to suggest to our manufacturing engineering team. But they were appreciative for the list of detailed IPC based improvement I was able to publish to be forwarded to our customer.

I found this checklist knowledge very empowering. As a designer of QFN footprints, I must admit I didn't know that IPC-7093 exists. But with this checklist as a guide, I am led to the IPC-based knowledge I need for help. I can read the spec and research what other well-known contributors have written on similar design challenges when dealing with these parts. I was amazed when I searched this standard and found a very helpful presentation on QFN prinicples by Vern Solberg, which will prove very effective in "selling" these IPC spec-based DFM ideas to our customer.

IPC Checklist for Producing Printed Board Assemblies

- Checklist at the Project Start Level
- Checklist at the CAD Level (map)
- Checklist at the Printed Board Ordering Level (map)
- Checklist at the Assembly Ordering Level (map)
- Checklist at the Cleaning and Conformal Coating Level (map)
- Checklist at the Quality Check Level (map)

Conclusion

The void of knowledge about how to collaboratively design within the electronics industry continues across the PCB design organizations I've participated in. It seems as though 12 different designers will always give you 12 different ways to approach a design challenge. That's not necessarily a bad thing. However, when choosing to adhere or reject to any of the 12 ways, I always want to hear first about how doing so will help the project meet an industry specification. I always say that a good PCB designer needs to see a dozen different ways of solving a design problem. However, great designers—those whose eyes are set on volume production—will draw only from ways allowing them to create and specify design solutions conforming to an inspectable IPC standard or collaborate with others to create one if it doesn't to exist.

When I think about the need to learn more about PCB industry standards—where to find them, how to access them, and how helpful the layout of this checklist has been for me—another one of Mr. Fenoglio's woodshop sayings comes to mind. This one is about storing valuable tools: "A place for everything, everything in its place." DESIGN007

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

To read past columns, click here.

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

Every day we get questions like those. And we love it. We have more than 500 PCB experts on 3 continents speaking 19 languages at your service. **Regardless where you are or whenever you have a question**, contact us!

What's your PCB question?
www.ncabgroup.com/pcb-design-mistakes/

Reliable answers. Reliable PCBs.

NCAB GROUP

Design Success with IPC Standards

Beyond Design

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

PCB design is all about knowing your EDA software tool backward. There is no point in designing a complex board if it doesn't perform to expectations or isn't manufacturable. The goal is to create a board that is functional, reliable, and cost-effective. Design rules are important because they help optimize the design of a product for its manufacturing and assembly praocess. This is called design for manufacturing (DFM). It aims to minimize the risk of errors and delays when a product is transitioning from prototyping to mass manufacturing. It also optimizes design and cost and is vital to meet time to market. Employing DFM strategies reduces the cost and difficulty of producing a product while maintaining its quality.

I started laying out single- and double-sided boards way back in 1975 using Bishop Graphics tape. The thinnest tape was 12 mils with 50-mil vias and 60-mil component pads. It was too easy; I did not have to worry about design rules for traces and footprints. Clearance was always the same as the trace width. A few years later, I progressed to the first version of Protel PCB layout software. No schematics or libraries to consider; just drop a few DIP footprints and passives and start connecting

Integrated Tools to Process PCB Designs into Physical PCBs

Visualize

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

Verify

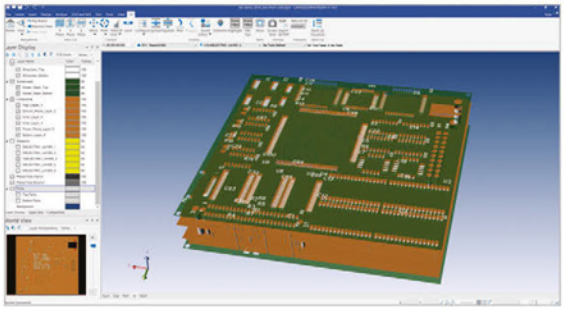
Ensure that manufacturing data is accurate for PCB construciton.

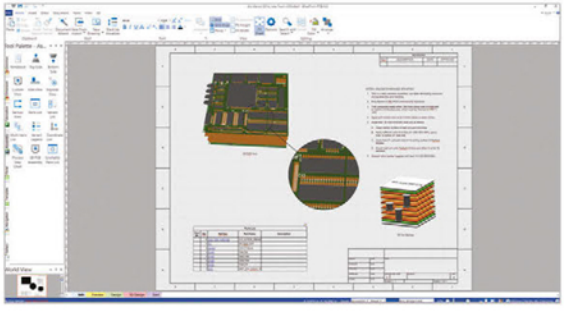
Panelize

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

Figure 1: IPC standards.

the dots. Again, the rules were all done with standard 60-mil-wide round and oblong pads, but I could push the envelope to 8-mil traces which was the narrowest ink pen for the XY plotter. The challenge was to etch the 8-mil traces without a break. But a touch-up pen on the photoresist fixed that. This was sufficient for prototype boards we were producing for university research projects. The clock speed was 10 MHz max, so signal integrity was not an issue and planes were not considered.

In 1987, I was thrown in the deep end. The R&D company I was working for purchased IC and PCB design software from Daisy (later acquired by Intergraph Electronics). BoardMaster PCB was a complex UNIX package that had a steep learning curve so the focus was on learning the tool so that we could produce our new fiber broadband network. The prototype network took 18 months of colossal effort to produce and it proved the concept. However, it was not manufacturable in quantities.

Our technology partners—Siemens and Alcatel in Europe, and US West and Bell Atlantic in the U.S.—provided us with their design rules so that all our equipment could be assembled in any modern production facility worldwide. I also obtained the IPC standards for PWB design and manufacture, and merged them into a complete set of design rules. Three years later, we were ready for mass production. The lesson here is that anyone can produce a prototype but it takes much, much more effort (at first) to produce a manufacturable product. However once set up, the constraints become part of the design process and take little extra effort to enforce.

Where Does a Newbie Designer Start?

IPC was founded in 1957, and has provided the worldwide electronics industry with standards for the design and manufacture of printed circuit boards, compiled over the years with the support of both committee and industry members. IPC has more than 300 active multilingual industry standards that cover nearly every stage of the electronics product development cycle.

IPC-2221B is the foundation design standard for all documents in the IPC-2220 series. The series is built around the IPC-2221, *Generic Standard on Printed Board Design,* the base document that covers all generic requirements for printed board design, regardless of materials. From here, the designer chooses the appropriate sectional standard for a specific technology.

All five sectional standards are included in the series:

- IPC-2222B, *Sectional Design Standard for Rigid Organic Printed Boards*
- IPC-2223E, *Sectional Design Standard for Flexible Printed Boards*
- IPC-2224, *Sectional Standard for Design of PWBs for PC Cards*
- IPC-2225, *Sectional Design Standard for Organic Multichip Modules (MCM-L) and MCM-L Assemblies*
- IPC-2226, *Sectional Design Standard for High-Density Interconnect (HDI) Printed Boards*

This series provides coverage on material and final finish selection, current-carrying capacity and minimum electrical clearances, test-specimen design, guidelines for V-groove scoring, dimensioning requirements, and conductor thickness requirements. Some of these standards are now published in Chinese and German.


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

Also, several documents apply to high-speed and land-pattern design:

- IPC-2228, *Sectional Design Standard for High Frequency (RF/Microwave) Printed Boards*
- IPC-2251, *Design Guide for Electronic Packaging Utilizing High-Speed Techniques*
- IPC-2141A, *Design Guide for High-Speed Controlled Impedance Circuit Boards*
- IPC-7351B, *Generic Requirements for Surface Mount Design and Land Pattern Standard*

These standards (and their predecessors) have been part of a well-used section of my technical library since 1987. They provide excellent reading and reference material for all PCB designers. These documents are available for purchase from IPC. A complete list of standards with their latest revision is available from the IPC Document Revision Table. Design rules must keep up with the latest devices and fabrication processes without losing sight of DFM. If you follow the above IPC guidelines, you will be designing for both manufacturability and mass production.

IPC standards are important for PCB design because they standardize requirements for the installation and fabrication of electronic equipment and assemblies. IPC standards ensure that manufacturers produce safe, reliable, and high-performance PCB boards by focusing on manufacturing details, and are committed to maintaining quality throughout the manufacturing process. Adhering to IPC standards issued by the industry body can ensure that you produce high-quality PCBs consistently. IPC certification ensures that vendor and PCB manufacturer use the same terminology so there is not any scope for miscommunication.

For the latest trends in PCB and high-speed design, both the newbie and the seasoned designer should read industry magazines such as *Design007*, *PCB007*, and *SMT007* to keep up to date with new techniques for design and manufacture. **DESIGN007**

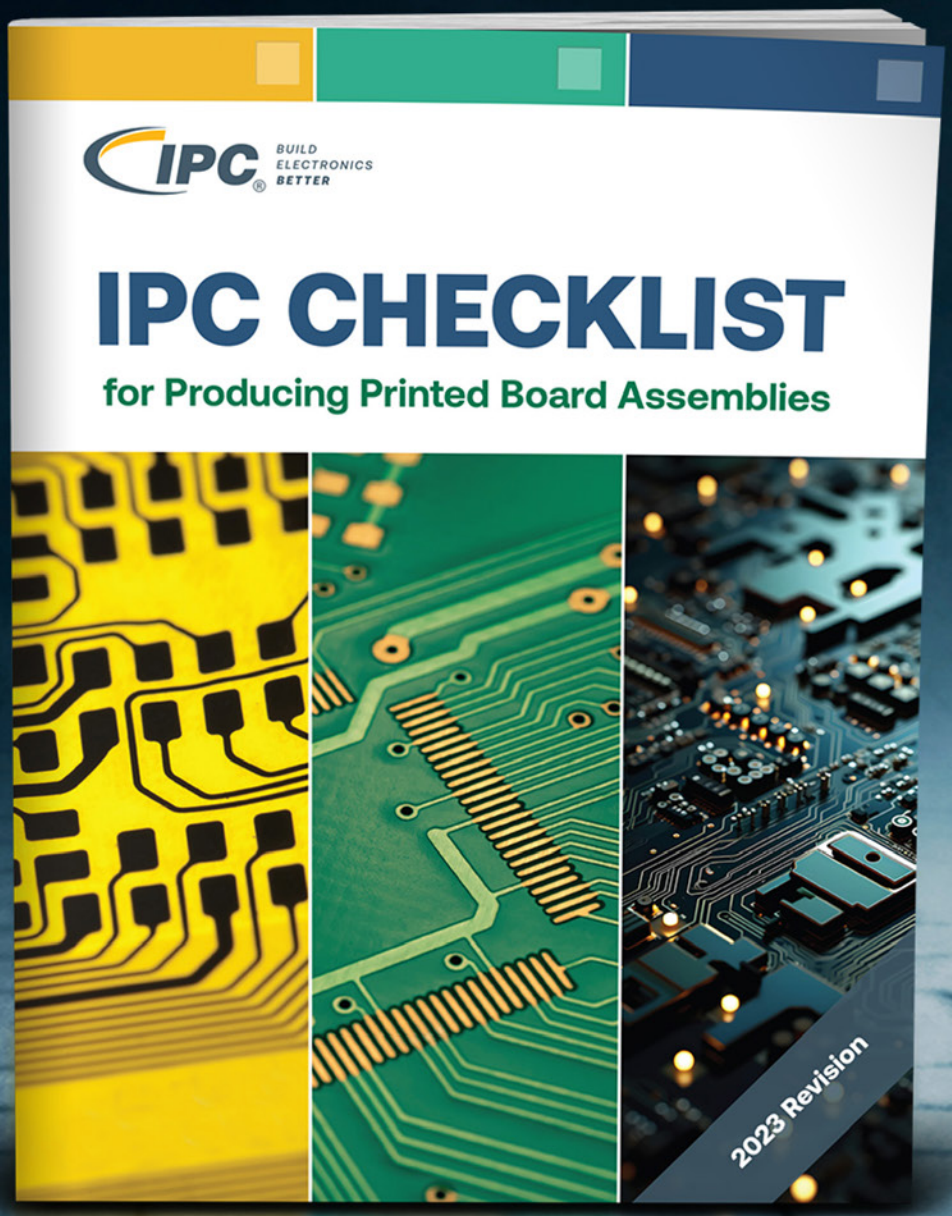
Resources

1. Beyond Design: Design Rules & DFM for High-Speed Design by Barry Olney

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.

A must-have for every company in the supply chain.

The IPC Checklist shows you where and which standards are used.



Get yours here

IPC BUILD ELECTRONICS BETTER

Flex007 Highlights

Rigid-flex Design Guidelines ▸

Rigid-flex boards use the same materials as any rigid board. One way to do this is by creating the flex portion along with the layers of the rigid board, and use a delamination process to remove any unwanted layers. No components are mounted on the flex portion of the board. One layer is used for the return and the other layer is used for your signals. The return is commonly a GND plane and hatched in the flex area of the board. Vias are not usually placed in this area either but may be used if needed.

Flexible Thinking: Unlocking the Key to Rigid-flex Design Success ▸

Despite what some seem to believe, rigid-flex circuits are not a new technology. In fact, they are more than a half-century old. At the time of the invention, my friend Thomas Sterns was working at Sanders Associates, the pioneering flex circuit manufacturer. Like many products in the first decades of printed circuit technology, they were working on a military application. The objective was to provide a reliable method for reducing the size, bulk, weight, and questionable reliability of wire harness assemblies while simultaneously reducing cost and assuring that human error might be minimized.

Rigid-flex Stackup: It's a 3D World ▸

The rigid-flex structure carries a lot of advantages—reliability, dynamic flexure, and the ability to get things done in tight spaces. But there's the additional burden of needing to manage a mechanical world that has additional dimensions or "degrees of freedom." A rigid stackup, for example, can easily be viewed in 2D, and that world is relatively easy for most electrical hardware engineers to understand and manage. You have impedance, frequency, and loss and then physical and electrical parameters that drive them. When flex substacks come into the picture, the fact that the flex portion needs to bend turns it into a 3D mechanical engineering problem that takes some time to learn.

Rigid-flex DFA ▸

Rigid-flex assembly brings its own set of issues, but designers can do quite a bit to make things easier on their downstream brethren. We asked IPC instructor Kris Moyer to give us the lowdown on DFA for rigid-flex circuitry.

Nan Ya PCB Reports Slight Growth in August Revenue ▸

Nan Ya Printed Circuit Board Corp. (Nan Ya PCB), a manufacturer of single-sided PCBs, HDI PCBs, and rigid-flex PCBs based in Taiwan, has posted unaudited sales of NT$3.37 billion ($105.52 million at $1=NT$31.97) for August 2023, up by 1.2% from the previous month, but down by 41.4% year-on-year.

IDTechEx Asks Where Flexible Hybrid Electronics Add the Most Value ▸

Can digital and/or high throughput manufacturing be applied to circuit boards? Can stretchable electronics be produced without sacrificing processing capabilities? Flexible hybrid electronics (FHE) is an emerging manufacturing methodology that aims to resolve both of these questions.

PREMIER GLOBAL SUPPLIER for **WEARABLES**

FLEXIBLE CIRCUITS | EMS/ASSEMBLY | CONTRACT MANUFACTURING

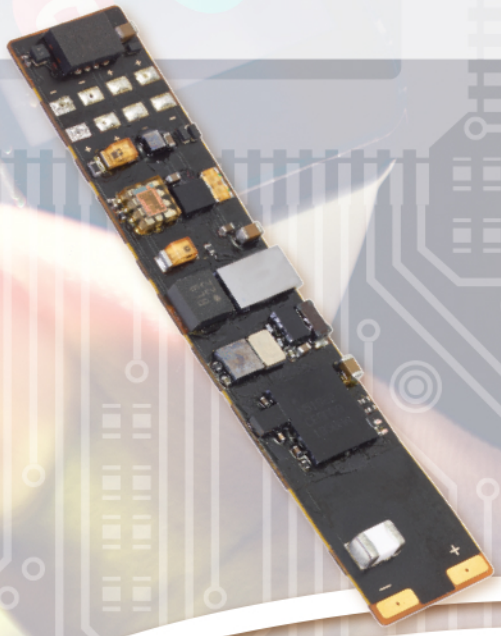
Industry leading Flex Design Support

Flexible Circuits, Rigid Flex, Membrane Switches, Plastic Moldings

Specialized EMS/Assembly Services

Product Module to Complete Product Box Builds

EXPLORE the POSSIBILITIES!



FLEXIBLE CIRCUIT® Technologies

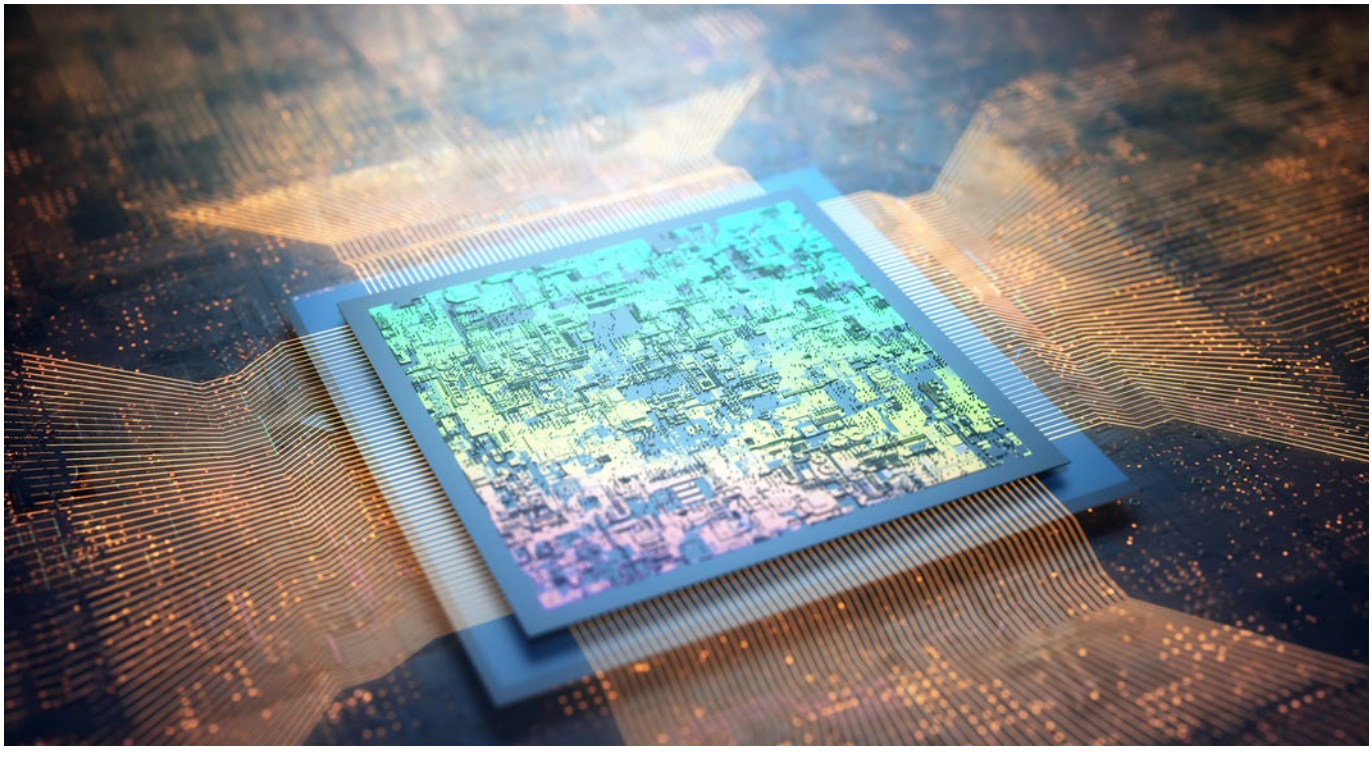
Flexible Circuit Technologies
9850 51st Ave. N. | Plymouth, MN 55442
www.flexiblecircuit.com | +1-763-545-3333

UHDI Fundamentals:
Ultra HDI Pushes PCB Manufacturing Capabilities

Article by Anaya Vardya
AMERICAN STANDARD CIRCUITS

Ultra high density interconnect (UHDI) is a term used in the electronics industry to describe a cutting-edge technology that pushes the limits of fabrication capabilities for printed circuit boards (PCBs) and semiconductor devices. UHDI represents an advancement in miniaturization and integration, allowing for the creation of electronic components and systems with extremely high levels of functionality in a smaller footprint. UHDIs are sub-1-mil (0.001") line widths and spaces, which necessitate that we change the unit of measurement from mils to microns. For reference, a 1-mil trace is 25 microns. In general terms, UHDI refers to traces and spaces on a printed circuit board that are sub-25 micron. As electronics continue to shrink, so does the printed circuit board, not only in the X-axis, but also the Y-axis. Designers are challenged with reducing the form factor as well as the thickness of printed circuit boards to meet these demands. This is where UHDI comes in.

With every major advancement in technology comes manufacturing challenges. UHDI is



cadence®

Transform Your PCB Design with

Allegro X AI

Collaboration
Complete solution inclusive of mechanical and supply chain with industry-leading partners

Unprecedented Productivity
Empowering EEs with a unified design and floorplanning solution

Next-Gen Routing
Reimagines heterogeneous integration with auto-routing for today's challenges

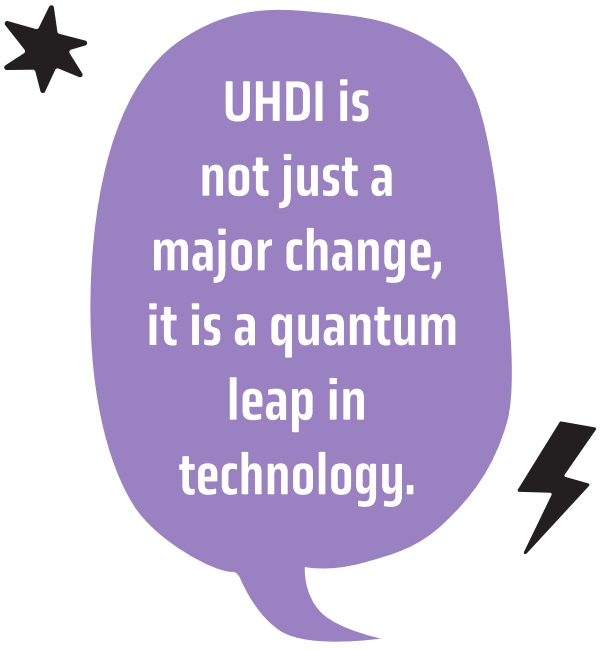
AI Innovation
Order of magnitude design productivity with AI-driven place and route

Analytics and Analysis
Fusion of design intelligence, data, and enabling realization of PCBs faster

FIND OUT HOW

not just a major change, it is a quantum leap in technology. It represents a change in the fundamental method of manufacturing printed circuit boards, moving from the traditional subtractive process to an additive one. UHDI technology requires not only new manufacturing methods, but new manufacturing equipment, chemistry, materials, and inspection capabilities. While there are some crossover processes, it is definitely not a plug-and-play implementation. PCB manufacturers that want to take on the challenge of producing ultra HDI boards will need to assess the more stringent requirements with regard to equipment and their manufacturing environment.

Here are some key aspects of UHDI and how it pushes fabrication capability limits:



1. **Miniaturization:** UHDI involves the fabrication of electronic components, such as microprocessors, memory chips, and PCBs, with features and interconnections that are significantly smaller than what was previously possible. This enables the creation of smaller and more compact electronic devices.
2. **High-density interconnects:** UHDI technology focuses on achieving a higher density of interconnections between components on a PCB or within a semiconductor device. This requires advanced fabrication techniques like multilayer PCBs, fine-pitch surface-mount technology, and advanced semiconductor manufacturing processes.
3. **Advanced materials:** UHDI often relies on the use of advanced materials, such as high-performance substrates, specialized dielectric materials, and advanced metallization processes. These materials are crucial for achieving the desired electrical and thermal performance in densely packed electronic systems.
4. **Precision manufacturing:** The fabrication of UHDI components and systems requires extremely precise manufacturing processes. This includes advanced LDI equipment that can create features at the nanometer scale, as well as advanced etching and deposition methods. It also requires new methods like A-SAP for subtractive processing.
5. **3D integration:** UHDI may involve 3D integration techniques, where multiple layers of components are stacked vertically to save space and improve performance. This requires sophisticated manufacturing processes for aligning and connecting these stacked layers.
6. **Thermal management:** With components packed so densely, managing heat becomes a significant challenge in UHDI systems. Advanced thermal management solutions, such as microfluidic cooling and heat spreaders, may be employed to dissipate heat efficiently.
7. **Increased complexity:** As UHDI technology allows for more functionality in a smaller space, the complexity of electronic systems can increase significantly. This necessitates advances in design and simulation tools to ensure the reliability and performance of these complex systems.

LOOK INSIDE

IPC BUILD ELECTRONICS BETTER

Contact us to become an advertiser

8. **Applications:** UHDI technology finds applications in a wide range of industries, including consumer electronics, aerospace, automotive, telecommunications, and medical devices. It enables the development of smaller and more powerful devices with improved performance.

9. **Inspection:** Electrical test and automatic optical inspection equipment will need to be able to accurately test and inspect sub-25-micron features on the PCB.

Contrary to popular belief, UHDI is not a new technology; it has been employed in Asia for decades for extremely dense PCBs in products (like smartphones) and in IC packaging. Historically in the U.S., it has been cost prohibitive in most applications except high volume manufacturing. It's important to note that UHDI is an evolving field, and advancements in fabrication techniques continue to push the boundaries of what is possible. As technology evolves, we can expect even more compact and powerful electronic devices and systems to be developed, thanks to innovations in ultra high density interconnect technology. **DESIGN007**

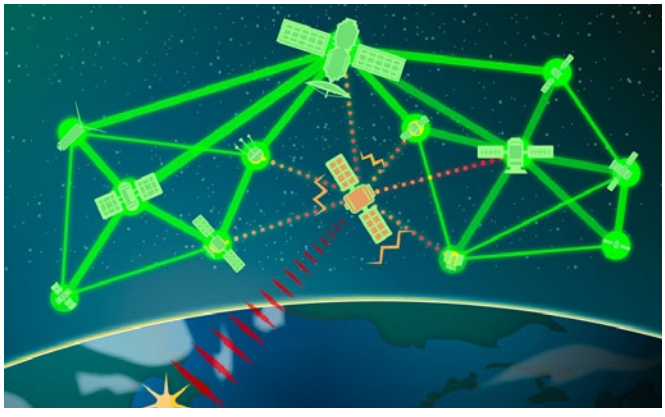
Anaya Vardya is president and CEO of American Standard Circuits; co-author of *The Printed Circuit Designer's Guide to... Fundamentals of RF/Microwave PCBs and Flex and Rigid-Flex Fundamentals.* He is the author of *Thermal Management: A Fabricator's Perspective* and *The Companion Guide to Flex and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

Increasing National Security with Satellites That Team Together

Imagine a satellite observing ships on the ocean. As it takes pictures of each ship, an algorithm decides what kind of vessel it is. But one sneaky sailor paints a pattern on the deck that confuses the satellite, so it can't decide what it's looking at.

How can the satellite work around its problem? According to new research, the answer could someday be: with a little help from its friends.

For the past six years, Sandia has been working on an autonomy project led by the Air Force Research Laboratory to enable a cluster of relatively small and inexpensive satellites to work together as a single, autonomous unit. The project could improve the nation's ability to conduct national security missions, including intelligence, surveillance, reconnaissance, climate monitoring and emergency response.



A confused satellite, for example, could communicate its problem to others in the network, which could straight away point their sensors to the same spot, combine their data and make a positive identification.

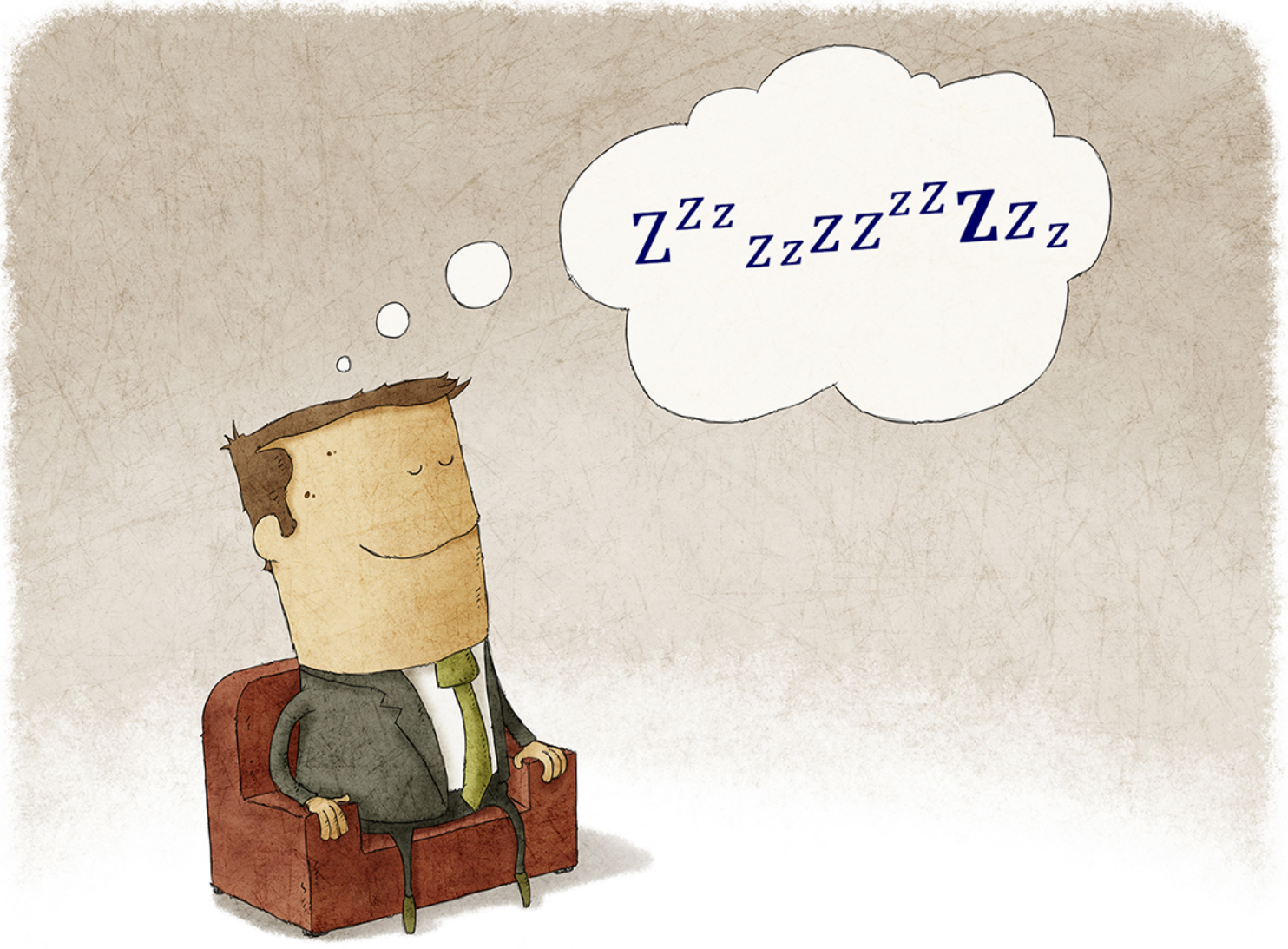
"This is a paradigm shift from large, exquisite, billion-dollar satellites to talking about multiple million-dollar satellites," said Sandia project manager Jered Mitchell.

While current research aims to connect up to about a dozen satellites, researchers say that further development could lead to connecting much larger networks.

"The ultimate idea is to scale up to constellations that might cover virtually the entire globe — and to potentially coordinate data collection even with non-space assets," like sensors on the ground or on planes, said Sandia's Drew Woodbury, a senior manager in Sandia's space programs.

(Source: Sandia)

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

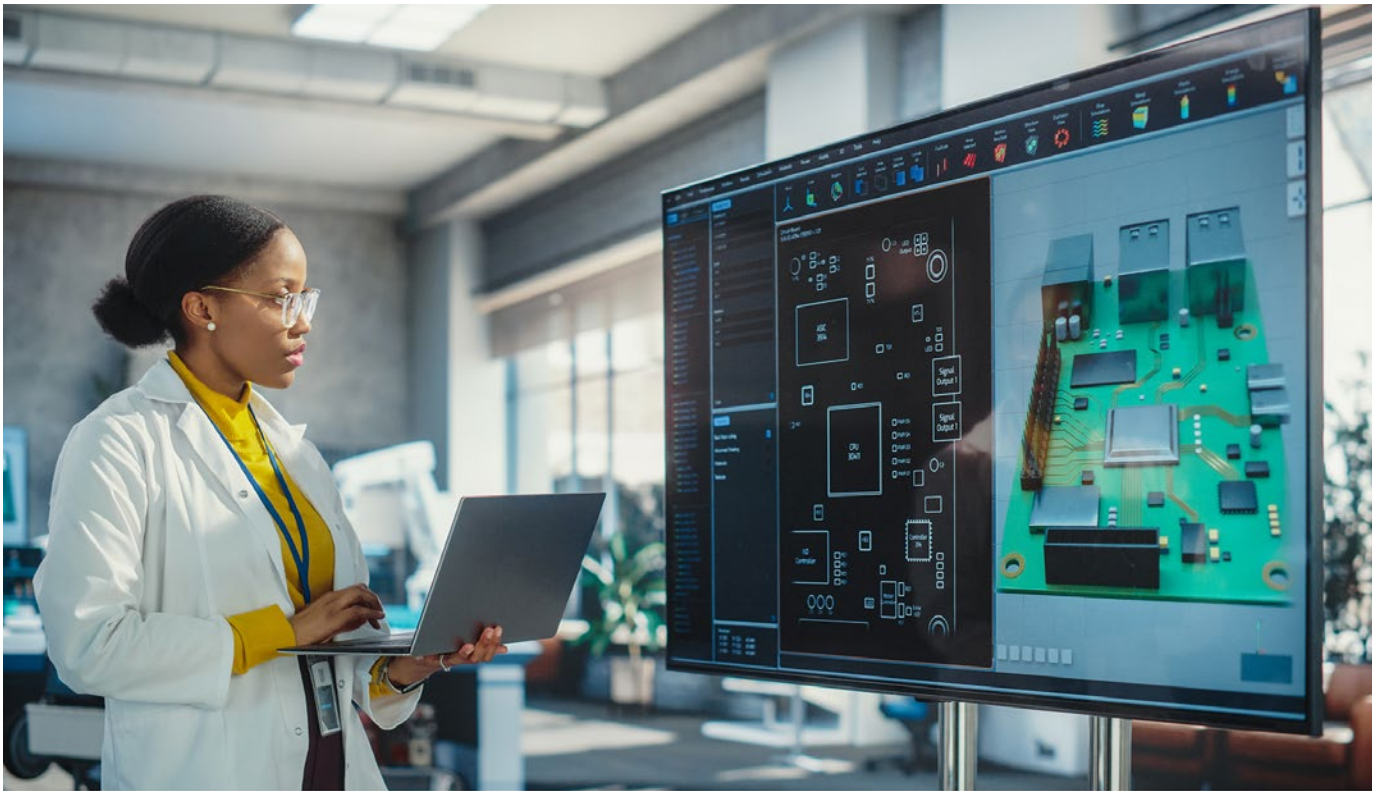
iCD software quickly and accurately analyzes impedance so you can sleep at night.

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



Managing Design Standards

Feature Interview by Andy Shaughnessy
I-CONNECT007

IPC provides a wide range of standards that cover PCB design, fabrication, assembly, materials, components, land patterns, and much more. Design standards fall under the purview of Patrick Crawford, so I asked him to share some background on the processes for developing and updating standards. Patrick explained which design, fabrication, and assembly standards PCB designers should be familiar with, and how standards can help designers avoid making costly mistakes.

Patrick, as the manager of design standards for IPC, what standards do you think every new PCB designer needs to understand?

I would consider the IPC-222x series of documents—IPC-2221, *Generic Standard on Printed Board Design,* and its sectional design standards—to be essential reading, although which sectional standard you invest in will depend on exactly what you're working on. For example, if you're working on a rigid-flex board, then a copy of IPC-2223 is probably required, and so on. I would also highly recommend a copy of IPC-7352, *Generic Guidelines for Land Pattern Design,* and IPC-2231 DFX Guidelines. The land pattern document will be invaluable as you begin to design lands for SMT and through-hole components, especially when designing in compact form—factors with electrical and mechanical clearances to consider. The DFX guidelines document is a higher-level design guide, in that it provides an overview of the entire design process and what to consider when designing or manufacturing, testing, end-of-life, etc. It's written to be understood

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

by designers of any level of experience, which is good for folks who are new to the industry.

I believe that even if you never pick up a soldering iron in your life—which you should, by the way—it is important to understand how components are joined to boards, how those joints are assessed for acceptance, and so on. Having a copy of IPC-J-STD-001, IPC-A-600, IPC-A-610, and the like is a good idea to have around the office.

Beyond that, building an understanding of design needs to start before standards come into the picture. In my opinion, even a cursory understanding of the materials science and physics of electronics can help a new design engineer form an intuitive grasp on what they're working with. IPC's PCB Design Fundamentals courses provide a great primer in the physics of electronics, and there's a wealth of knowledge freely available on YouTube that can help designers bolster their background knowledge on electronics.

I know you're working with committees to update some of the older standards. How does the updating process work?

We call this process "reaffirmation," and it is essentially working with volunteer subject matter experts (SMEs) to determine whether an old document (i.e., a document published more than five years ago that has not been revised or amended since) is still valid for new products. In this case, "valid" means that every requirement and/or recommendation therein is still sound for contemporary design, fabrication, assembly, etc.

For any document being reaffirmed, we first convene a meeting of either the originating group (i.e., the committee, subcommittee, or task group that first developed the document, revision, etc.) or an ad-hoc reaffirmation group (in cases where the originating group no longer exists) and fill them with as many relevant SMEs as possible. Then, they are given a period to review the document. From there, it's a ballot vote on whether the document should be reaffirmed or retired.

If a document is reaffirmed, it will be given a new title page indicating as such and remain available in our store. It can also be revised or amended if the committee so desires. If it is retired, we will remove it from the store, but will still make it for sale upon request. Critically—and this is a concern I hear voiced frequently—a retired document is not obsolete. It is perfectly valid for old products that originally cite it as a requirement.

We occasionally hear from designers who brag about not following IPC standards. What would you tell them to change their mind?

This is akin to bragging about using the wrong gauge nail to frame a house. There's a reason why building associations (read: manufacturers) require contractors (read: suppliers, design houses, etc.) to build to code (read: standards). Great, you saved some money by skimping on nails, and the house should stand for a while. But when the hurricane comes, or the ground settles, or the everyday dynamic loads push those under-gauged nails beyond their sheer limits and the whole thing comes crashing down, was it worth it to save a bit of cash in the first place? I don't think so.

Analogies aside, there are plenty of reasons why standards exist. They are a wonderful example of collective wisdom learned from iterations of scientific rigor, advancement,

and failure. Maybe that's another term for standards: iterative excellence.

So, failing to "build to code" because of ignorance is one thing, and that's why we at IPC are focusing so much on education about our standards and printed board education in general. But to willfully ignore standards, and the thousands of years' worth of collective knowledge that they represent? In other words, to willfully build your house to fail?

Patrick Crawford

Forgive me for being frank, but that's a whole universe of dumbness that I don't want to live in. I would implore any rogue designers to re-evaluate their knowledge with respect to the greater cosmos of collective understanding that standards represent.

We're publishing the IPC Checklist in our I-Connect007 library very soon. Why should designers—and manufacturers—download this document?

I worked on the Checklist revision and it's very comprehensive. It provides a thorough overview of every manufacturing process from soup to nuts, and it details which standards and guidelines can help manufacturers build their products, replete with handy diagrams of every area of a shop floor and which specific processes correlate to which IPC document. I personally think we should sell posters of those diagrams because they're very helpful.

For designers who are on the front lines of a product lifecycle, the stages of manufacturing can seem opaque, especially for design bureaus that are completely detached from a shop, or for larger companies with (hopefully unintentionally) siloed departments. Having a resource that outlines every step in the lifecycle can help designers understand their impact on that lifecycle, and how their choices during the design phase ripple through manufacturing. (Another great primer is to read the IPC-2231 DFX Guidelines document.) Of course, not even the best brochure is a complete education, but this is a great launching pad for a designer to seek out more information and become involved in IPC standards.

Also, the IPC Checklist includes an exhaustive table of every standard, guideline, and white paper that IPC has published since 2010 (unless they've been obsoleted or superseded). This table also lists the IPC committee responsible for the maintenance and development of a given document, which we hope will inspire readers to volunteer their time. We welcome volunteers, and I guarantee you will learn a lot.

What's your goal for standards soon, and maybe in five to 10 years?

I'd like to at least get through IPC APEX EXPO in April, and then maybe we can talk. But seriously, I want to make sure that the standards I am personally responsible for stay as up to date as possible, while becoming more usable and relevant in a modern context. For example, there are quite a few documentation standards (such as the entire IPC-2610 family) that are still used extensively but haven't been touched in more than a decade. We believe that these standards can be consolidated or revised to be more applicable to modern ECAD environments. I am working on reaffirming those standards so they can be revised as needed. I also would love to see IPC-2581 take a much more prominent role in design data transfer, because it's so powerful as a smart data format, and I don't think designers are taking full advantage of it yet.

My moonshot goal is that in 10 years, every ounce of product data—design and manufacturing data, relevant shop floor data that hooks into CFX, sustainability and compliance data, end-of-life and recycling instructions, traceability, and provenance information, etc.—can be found in a single IPC product file that follows it throughout its lifecycle, to be modified and ingested by manufacturers, suppliers, customers (both within the supply chain and end-user), regulators, and waste-handlers, etc. Data gathering is a massive time and money sink for companies, especially as regulators (and therefore end-tier manufacturers) worldwide begin to ask for more product information for compliance reasons. Having a single source of truth for this kind of information, as well as the usual manufacturing information, can save everyone a headache.

If you aren't sure whether you're designing to spec, just ask your colleagues.

Of course, that's easier said than done, and I'm sure the reader is screaming, "Confidential information!" or "One standard to rule them all!" But there are already solutions to these problems being circulated within the IPC community, and a decade is a long time to achieve a robust, workable solution to the growing complexity of the global supply chain and associated governmental and contractual requirements thereof.

What advice would you give young PCB designers regarding standards?

Read them and understand how they relate to and should (or "shall," as we say in standard speak) be incorporated into your designs, but also make sure that you understand, as I detailed earlier, the mechanisms and principles driving them.

If you aren't sure whether you're designing to spec, just ask your colleagues. If they don't know, take it as an opportunity to learn together or as a department; grab a copy of the relevant standard, study it, and reach out to the development committee for guidance if you have any questions on implementation. If your colleague doesn't know how to implement the spec but thinks they don't need to (remember, houses and nails), be an advocate for doing things the right way.

If you're brand-new to designing boards or need to shake off the dust, it's a good idea to investigate IPC's PCB Design Fundamentals courses, or any of the related advanced design topics. If you have time and your company allows for it, I also highly recommend becoming involved in standards development in general. IPC committees are always free to join and participate in, and you will help shape the future of the industry.

That isn't just flashy marketing speak; some of the most important clauses in the most used standards in our industry came from hearts, minds, and laboratories just like yours. Yes, you can join even if you don't have a keycard for the lab. Plus, you'll learn so much from working and hanging out with a diverse community of designers, some of whom have decades of experience to share. Trust me: you'll teach them a thing or two as well.

Aside from standards, find a mentor in your company or community who will help you as you grow as an engineer. A one-on-one professional relationship can be invaluable. If you need help with this sort of thing, check out IPC's Emerging Engineer program.

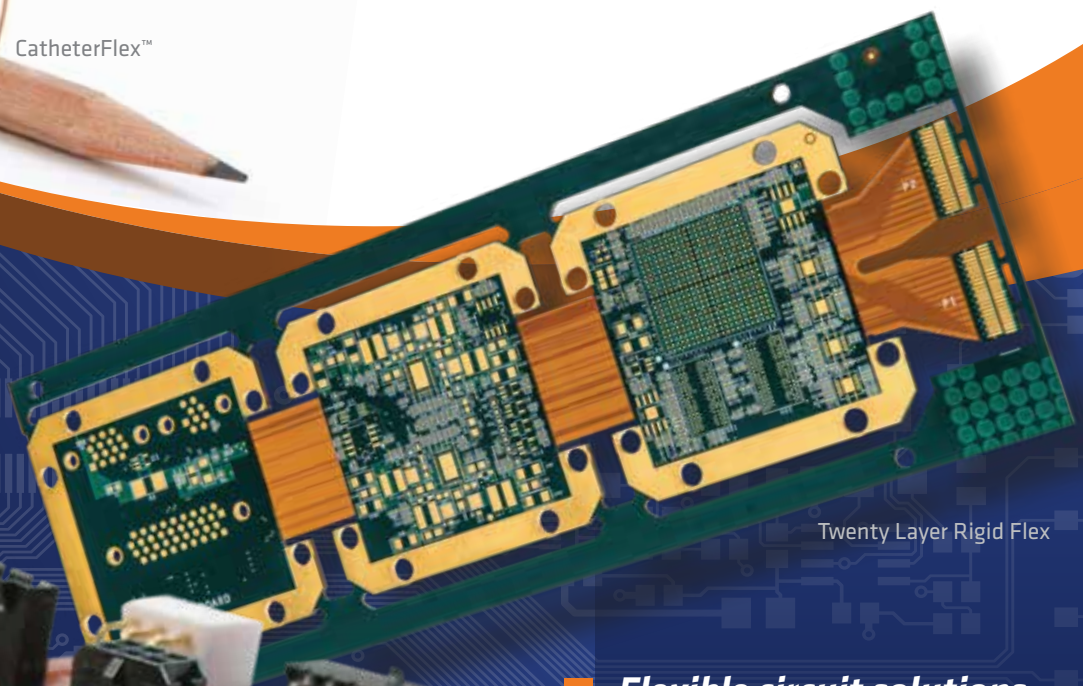
What else? Look to nature for some design inspiration if you need it. Don't let the initial cost of the standards outweigh the many benefits. Coffee can't replace sleep, as much as we would like it to, so take a break if you need it. Take a deep breath; you're doing great.

Thanks, Patrick. Great talking with you.

Thank you, Andy. Always a pleasure. **DESIGN007**

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!*

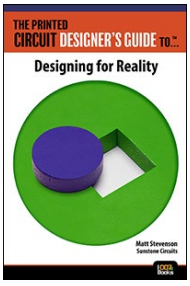
All Flex Solutions
1601 Cannon Lane
Northfield, MN 55057

(507) 663-7162
AllFlexInc.com

all flex
SOLUTIONS

TOP TEN

EDITOR'S PICKS

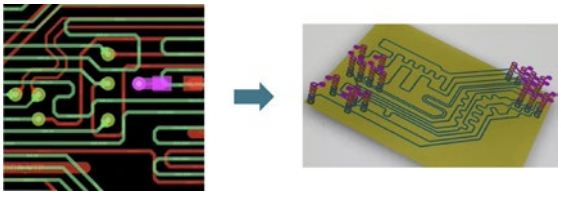


Book Excerpt: *'The Printed Circuit Designer's Guide to... Designing for Reality,'* Introduction

With more of us working to gain board layout experience on the job, Designing for Reality becomes a very pertinent discussion topic in the PCB industry. But what does that phrase even mean and why should we focus on such a topic?

Three Things to Improve High-Speed PCB Signoff

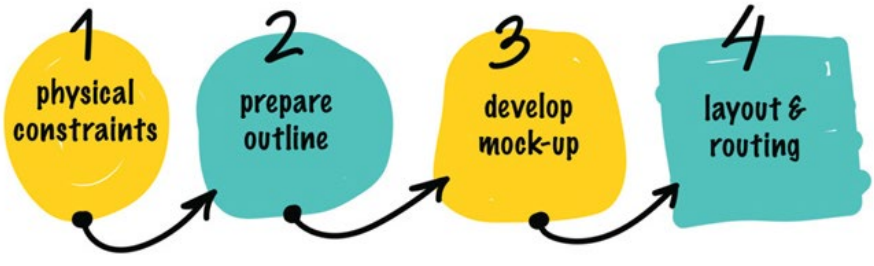
Another challenge for SerDes is losses within the channel design. At high speeds, dielectric material can be very lossy, making the appropriate selection of the right material, length, etc., critical for the channel. Many questions about stackup, trace widths, and height from the ground plane need to be defined up front.



Keysight EDA 2024 Integrated Software Tools Shift Left Design Cycles to Increase Engineering Productivity

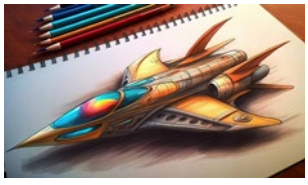
Keysight Technologies Inc. introduces Keysight EDA 2024, a tightly integrated suite of electronic design automation (EDA) software tools that ensures first pass success. This new integrated EDA software facilitates a "shift left" approach to increase productivity for engineers developing high speed, high frequency products in multiple applications.

Designer's Notebook: DFM Principles for Flexible Circuits



Flexible circuit applications can be as basic as furnishing electrical interconnect between two conventional circuit board assemblies, or to provide a platform for placing and interconnecting electronic components. During the planning and pre-design phase of the flexible circuit, there will be several material and process related questions that need to be addressed.

Connect the Dots: Best Practices for Prototyping



PCB prototyping is a critical juncture during an electronic device's journey from concept to reality. Regardless of a project's complexity, the process of transforming a design into a working board is often enlightening in terms of how a design can be improved before a PCB is ready for full production. But the PCB design needs to be carefully created and thoroughly tested before moving to prototype.

A Primer on UHDI

There has always been pressure to reduce line and space as we have seen the bleeding edge technology go from 8 to 5 mils and then to 3 mils. The difference between "then" and "now" is that the prior advancements, for the most part, used the same processes, chemistry, and equipment going from 8 mils to 3 mils. But going from 3-mil to sub 1-mil trace and space is a quantum leap in printed circuit board (PCB) technology that requires a whole new set of processes and materials.

Ansys Named to Newsweek's List of Top 100 Most Loved Workplaces in America for 2023

Newsweek announced their annual rankings for the Top 100 Most Loved Workplaces list and Ansys was ranked among the top 20. The 2023 Top 100 Most Loved Workplaces are the result of a collaboration with the Best Practice Institute (BPI), a leadership development and benchmark research company.

Altair, EMA Design Automation Announce Integration of Ultra Librarian into Altair ECAD Verification and Multiphysics Solutions

The integration of Ultra Librarian into Altair electronics solutions gives users the ability to search, preview, and place components quickly through the web or with native CAD integrations, saving time and eliminating errors.

Zuken Introduces AIPR for CR-8000: Empowering PCB Designers with Innovative AI-Powered Place and Route Technology

Zuken Inc., an industry-leading provider of electronic design automation solutions, is pleased to introduce an innovative three-stage approach to AI-powered PCB design within its flagship CR-8000 platform. This forward-looking approach demonstrates Zuken's commitment to delivering heightened efficiency, accuracy, and technological progress to address the ever-increasing challenges of electronic design.

PCB West Review: EDA Tool Companies Finally Embrace AI

The weather couldn't have been better for PCB West, and PCB designers and manufacturers packed the show floor for much of the exhibition on Wednesday, Sept. 20 in Santa Clara, California. The show floor was sold out, and it was great seeing old friends and meeting new ones.

Career Opportunities

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 pcb design007

Career Opportunities

ventec INTERNATIONAL GROUP 騰輝電子

Senior Sales Representative

Ventec Central Europe

Location: Kirchheimbolanden, Germany/Remote

We are looking for a self-motivated Senior Sales Representative—Ventec Central Europe, ideally with experience in the PCB industry. This position requires significant selling experience (15+ years) in the electronics and PCB industries. Candidates must possess a proven & consistent history of proactive sales growth with OEM customers. Most notably, they must be able to connect with OEM contacts that have decision-making capabilities.

Key Responsibilities

- Promote, sell, and close business for all Ventec product lines with focus on key OEM and PCB manufacturing customers.
- Track projects and submit monthly updates to management.
- Coordinate cross-functional resources when applicable.
- Assist in coordination and set-up of relevant trade show events.
- Assist in strategic planning initiatives.
- Assist in market and customer intelligence gathering.
- Recommend pricing strategies.

Job Requirements

- Entrepreneurial spirit, positive, high energy, and desire to win.
- Proactive and self-motivated work strategy to develop and win business for all business units.
- Excellent written and oral communication skills in German and English
- Excellent computer skills (Microsoft Office, especially Excel).
- Proven track record securing new business at OEM accounts.

Please apply in the strictest confidence, enclosing your CV, to: accountingde@ventec-europe.com

apply now

ViTrox®

Technical Support Engineer

USA Region

ViTrox aims to be the world's most trusted technology company in providing innovative, advanced, and cost-effective automated Machine Vision Inspection Solutions for the semiconductor and electronics packaging industries. Located in Hayward, California, ViTrox Americas Inc. is actively looking for talent to join our expanding team.

Key Responsibilities:

- Delivering excellent and creative problem-solving skills for servicing, maintaining, machine buy-off, and troubleshooting advanced vision inspection machines at customer sites. Providing remote customer support to minimize machine downtime.
- Cultivating strong customer relationships and ensuring comprehensive customer service to drive repeat orders and support business development in machine evaluation.
- Proactively understanding customer needs and feedback to drive continuous improvement in existing technologies and new product development.

Qualifications & Requirements:

- A recognized diploma/advanced diploma/degree in Science and Engineering, preferably in Electrical & Electronics/Computer Science/Computer Studies or equivalent.
- 3+ years of relevant experience in servicing automated inspection equipment (SPI, AOI, and AXI).
- Strong communication and troubleshooting skills.
- Willingness to travel extensively across the USA.
- Positive attitude and flexibility to accommodate conference calls with headquarters.
- Applicants from the USA and Canada are welcome to apply.
- Training will be provided at our headquarters in Penang, Malaysia.

apply now

Career Opportunities

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

TAIYO AMERICA

Europe Technical Sales Engineer

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

PRIMARY FUNCTION:

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

ESSENTIAL DUTIES:

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

QUALIFICATIONS / SKILLS:

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

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

apply now

Career Opportunities

BLACKFOX
Premier Training & Certification

IPC Instructor
Longmont, CO

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

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

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

Salary: Starting at $30 per hour depending on experience

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

Schedule: Monday thru Friday, 8–5

Experience: Electronics Manufacturing: 5+ years (Required)

License/Certification: IPC Certification–Preferred, Not Required

Willingness to travel: 25% (Required)

apply now

Prototron Circuits

Sales Representatives

Prototron Circuits, a market-leading, quick-turn PCB manufacturer located in Tucson, AZ, is looking for sales representatives for the Southeastern U.S. territory. With 35+ years of experience, our PCB manufacturing capabilities reach far beyond that of your typical fabricator.

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

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

apply now

Career Opportunities

KYZEN
Where Science and Care Converge.

Regional Manager

Southwest Region

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

DETAILS OF FUNCTION:

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

QUALIFICATIONS:

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

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

fernando_rueda@kyzen.com

apply now

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

pluritec®

MACHINES FOR PRINTED CIRCUIT BOARDS

Field Service Engineer

Location: West Coast, Midwest

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

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

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

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

Valid driver's license is required, as well as a passport for travel.

Must be able to travel extensively.

apply now

KOH YOUNG AMERICA

Technical Service & Applications Engineer

Full-Time — Flexible Location

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

Responsibilities

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

Skills and Qualifications

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

Benefits

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

apply now

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, panalization 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

APCT
Passion | Commitment | Trust

APCT, Printed Circuit Board Solutions: Opportunities Await

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

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

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

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

apply now

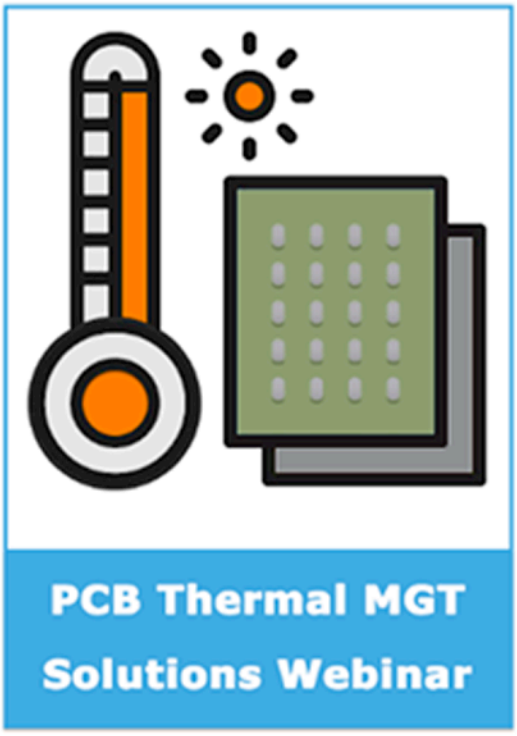
NEXT GENERATION THERMAL MANAGEMENT SOLUTIONS

This webinar will go beyond basic LED technology and delve into advanced methods of thermal management that are also applicable in other industries such as high-speed networking and telecom.

We further the knowledge base by detailing alternative methods that provide cost savings with limited performance loss.

And last, but not least, we will explore technologies currently under development for future thermal management solutions.

Save the date and come join us for this discussion at 11am, Thursday November 16th, 2023.

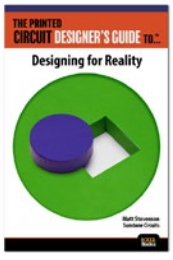


I-007eBooks The Printed Circuit Designer's Guide to...

NEW! Manufacturing Driven Design

by Max Clark, Siemens

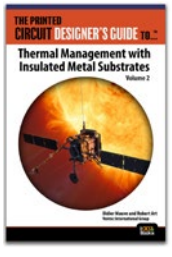
This book introduces a new process workflow for optimizing your design called Manufacturing Driven Design (MDD) and is a distinct evolution from DFM. When defining Manufacturing Driven Design, it is important to recognize that this is, foremost, an element of the design stage. Manufacturing certainly 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. **Read Here.**



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!**

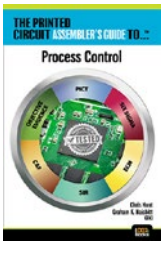


Thermal Management with Insulated Metal Substrates, Vol. 2

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.**

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

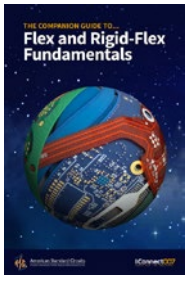


Process Control

by Chris Hunt and Graham K. Naisbitt, Gen3

In this book, the authors examine the role of SEC test and how it is used in maintaining process control and support for objective evidence (OE.) Issues, including solution choices, solution sensitivities, and test duration are explored. **Take a look inside!**

The Companion Guide to... Flex and Rigid-Flex Fundamentals



This compact guide, written as a companion to The Printed Circuit Designer's Guide to... Flex and Rigid-Flex Fundamentals by topic experts at American Standard Circuits, is designed to provide additional insights and best practices for those who design or utilize flexible and/or rigid-flex circuit boards. Topics covered include trace routing options, guidelines for process optimization, dynamic flexing applications, rigid-to-flex transition and more. **Find out what it's all about.**

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

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

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

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

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

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

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

SALES MANAGER: **BARB HOCKADAY**
(916) 365-1727; barb@iconnect007.com

BUSINESS DEVELOPMENT: **TAMARA SITES**
tamara@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: **SHELLY STEIN & FIREFLY AI**

DESIGN007 MAGAZINE

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

October 2023, Volume 12, Number 10
DESIGN007 MAGAZINE is published monthly
by IPC Publishing Group, Inc., dba I-Connect007

ADVERTISER INDEX

Problems solved!

SUBSCRIBE NOW