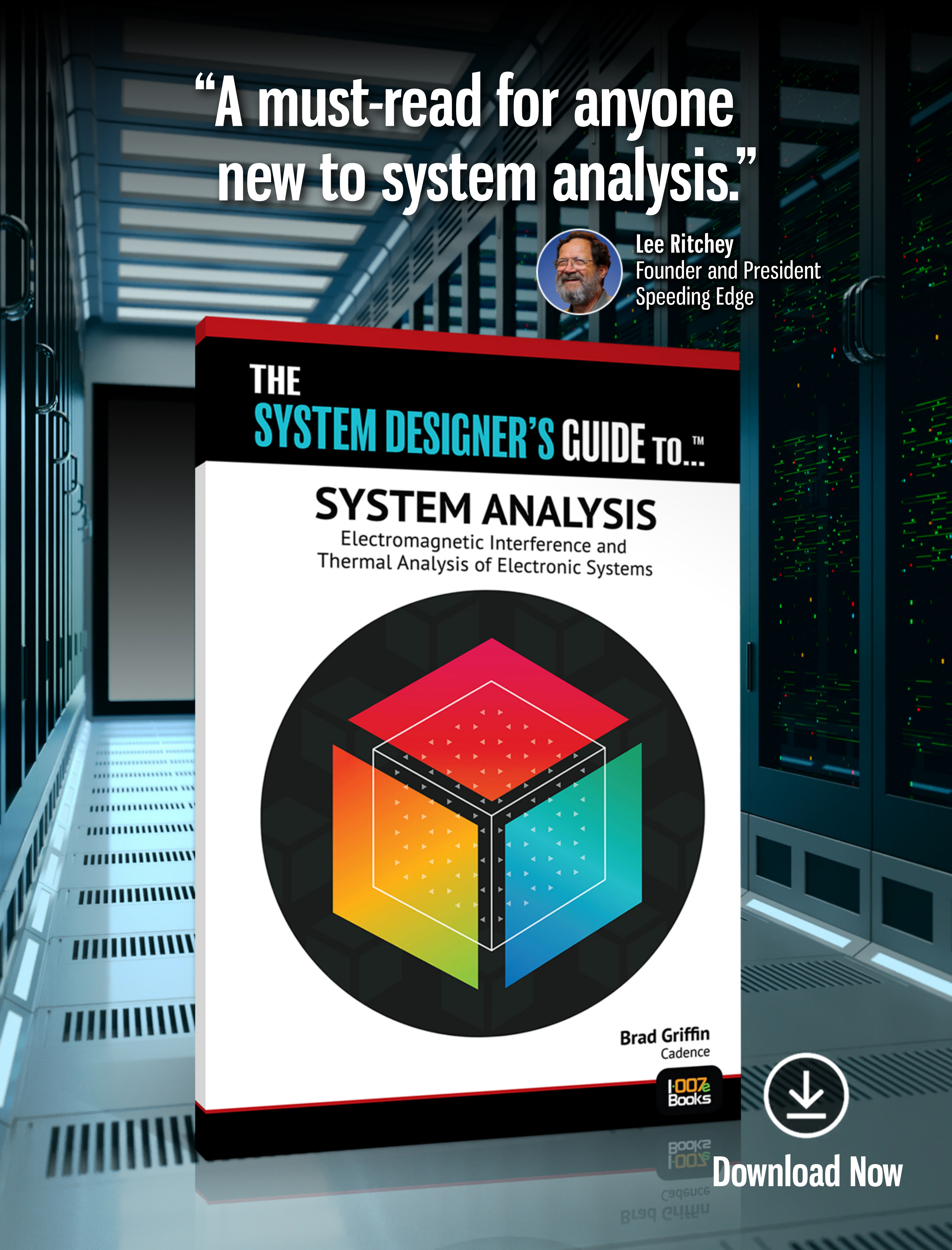
DECEMBER 2021

I-Connect007

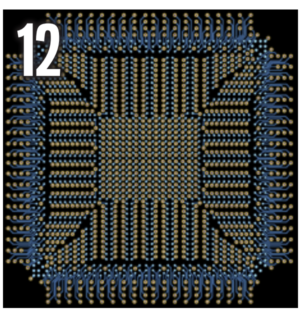
DESIGN007 MAGAZINE


"A must-read for anyone new to system analysis."
Lee Ritchey
Founder and President
Speeding Edge
THE SYSTEM DESIGNER'S GUIDE TO...™
SYSTEM ANALYSIS
Electromagnetic Interference and Thermal Analysis of Electronic Systems
Brad Griffin
Cadence
I-007e Books
Download Now

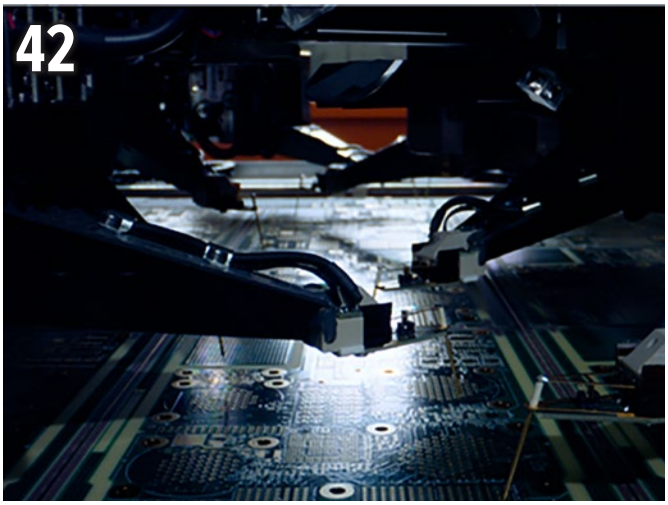
DESIGN007 MAGAZINE

The Art and Science of PCB Design

There is definitely an artistic angle to this job. PCB design requires the perfect mix of artistic and technical skills. PCB designers and design engineers are graphic artists, but unlike Renaissance sculptors, their masterpieces don't just have to look nice—they also have to function. These PCBs may enable devices that help feed humanity, save lives, or carry us to other countries or planets.



DESIGN007 MAGAZINE



ARTICLES

SHORT

DEPARTMENTS

COLUMNS

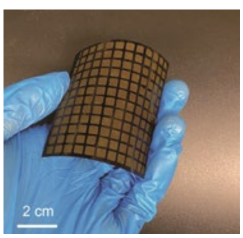
HIGHLIGHTS

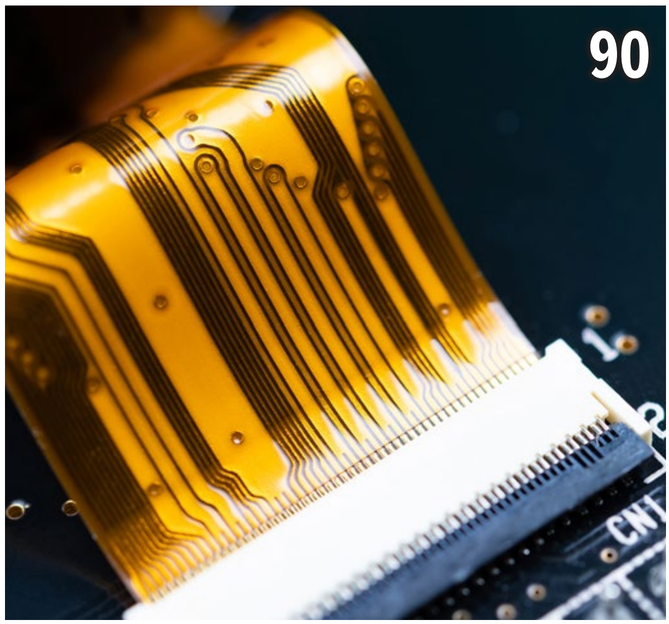

Valor NPI
shift ←

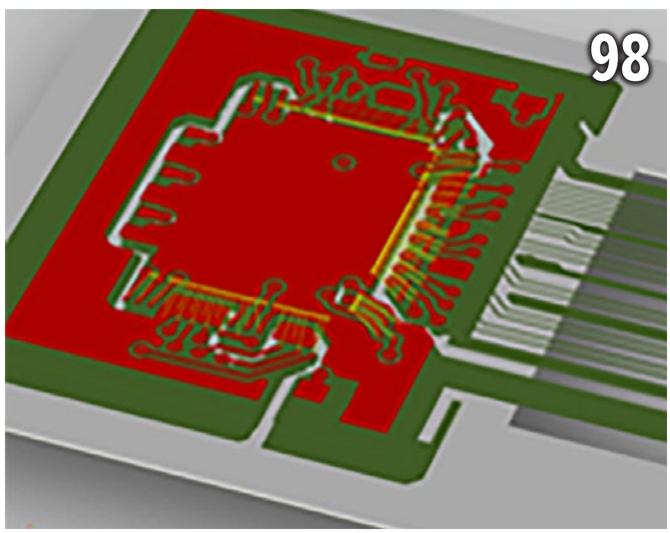
FLEX007

This Month in Flex

Flexible and rigid-flex circuits offer even more opportunities for the artistic designer—and more possibilities for miscues and missteps. This month, Cadence columnist Brad Griffin discusses some of the challenges related to simulating wirebonded chip-on-board on rigid-flex. And columnist Joe Fjelstad explains the art and science of creating perfect flexible and rigid-flex circuits.








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

The Art of PCB Design: I Know It When I See It

The Shaughnessy Report

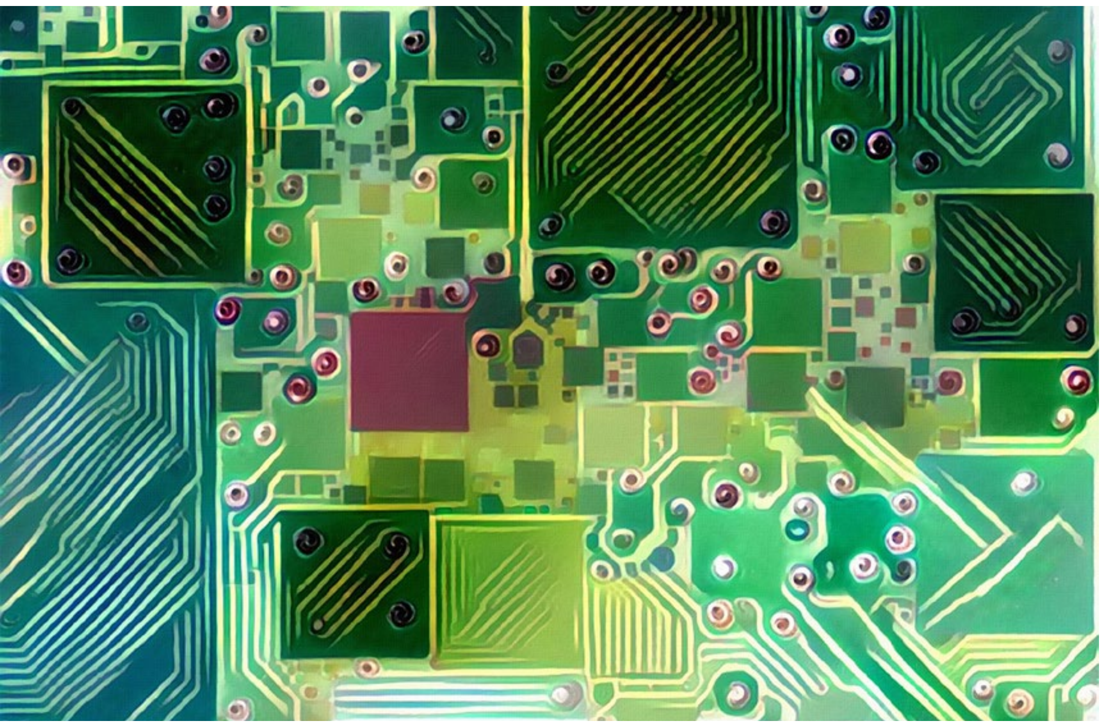
by Andy Shaughnessy, I-CONNECT007

When we first started planning this issue, we looked back over topics that we've covered for the past few years. We noticed that our contributors spend most of their time discussing the technical side of PCB design.

That's to be expected. When we discuss "best practices" for PCB design, we're typically looking at it from a technical viewpoint. After all, *Design007 Magazine* is a technical publication. And in PCB design, what is "design," exactly? Perhaps we could quote Supreme Court Justice Potter Stewart, who said in a 1964 case about obscenity, "I know it when I see it."

But there is definitely an artistic angle to this job. PCB design requires the perfect mix of artistic and technical skills. PCB designers and design engineers are graphic artists, but unlike Renaissance sculptors, their masterpieces don't just have to look nice—they also have to function. These PCBs may enable devices that help feed humanity, save lives, or carry us to other countries or planets.

At every PCB design competition, from the old Top Gun at PCB West to IPC's design contest at IPC APEX EXPO next month, the entries will be judged, in part, on artistic layout. Form follows function, and the art of the design is dependent upon the requirements of each board.



In many ways, today's PCBs represent the height of industrial art. As some of our contributors point out this month, an eye-pleasing, symmetrical layout is less likely to warp, twist, and turn into a "potato chip" during manufacturing. Plus, a beautiful PCB design will be much simpler for every stakeholder downstream to deal with. And if it's an expensive board, rework and repair in the field will be much easier if everything on the board is placed in a logical fashion.

The PCB designer reminds me of an old-world painter whose palette contains an ever-changing set of paints. But while Renaissance painters needed only to please their rich patrons, the designers of today must serve a variety of masters: managers, downstream stakeholders, and customers. They must do so quickly, and without going over budget.

So, in this issue, we salute you—the Da Vincis of design, the Lombardos of layout, and the Picassos of the PCB!

This month, our contributors explore the symbiotic relationship between art and science in PCB design, and why "right brain" input is beneficial for an elegant, robust design.

We start off with a conversation with three PCB designers who are all artists in their own right: photographer Charles Pfeil, painter Barry Olney, and sculptor Bill Brooks. They share their thoughts on the intersection between art and science in PCB design, and why both disciplines are necessary to produce great design. In a separate article, Steph Chavez discusses the move in EDA tools toward more artificial intelligence and the effect this could have on designers' artistic inclinations. Dan Beeker focuses on the art of designing PCBs with an emphasis on electromagnetic field theory. Cherie Litson explains why much of PCB design involves the art of applying the science correctly, while staying within constraints and budgets. Chris Young has a great feature on the need for symmetry—and asymmetry—in PCB designs, and how a well-designed PCB draws our eyes into the focal points of the layout.

Vince Mazur delves into the need for an artist's touch in design for manufacturing, and why it's human nature to prefer a pleasing form as well as function. Tomas Chester points out that for all the focus on EDA tools, the best tool is still the designer and the designer's knowledge of how to mix art and science perfectly. And Tara Dunn explains how so many of the electronic industry's innovations are the result of technologists allowing their creativity to push the existing boundaries to their breaking point.

Speaking of breaking points, let's hear it for 2021. It's been quite an improvement over 2020. Last year was a rough one for all of us, but our industry came out of it stronger than ever.

Have a great holiday, and I hope to see you on the road next year. DESIGN007

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

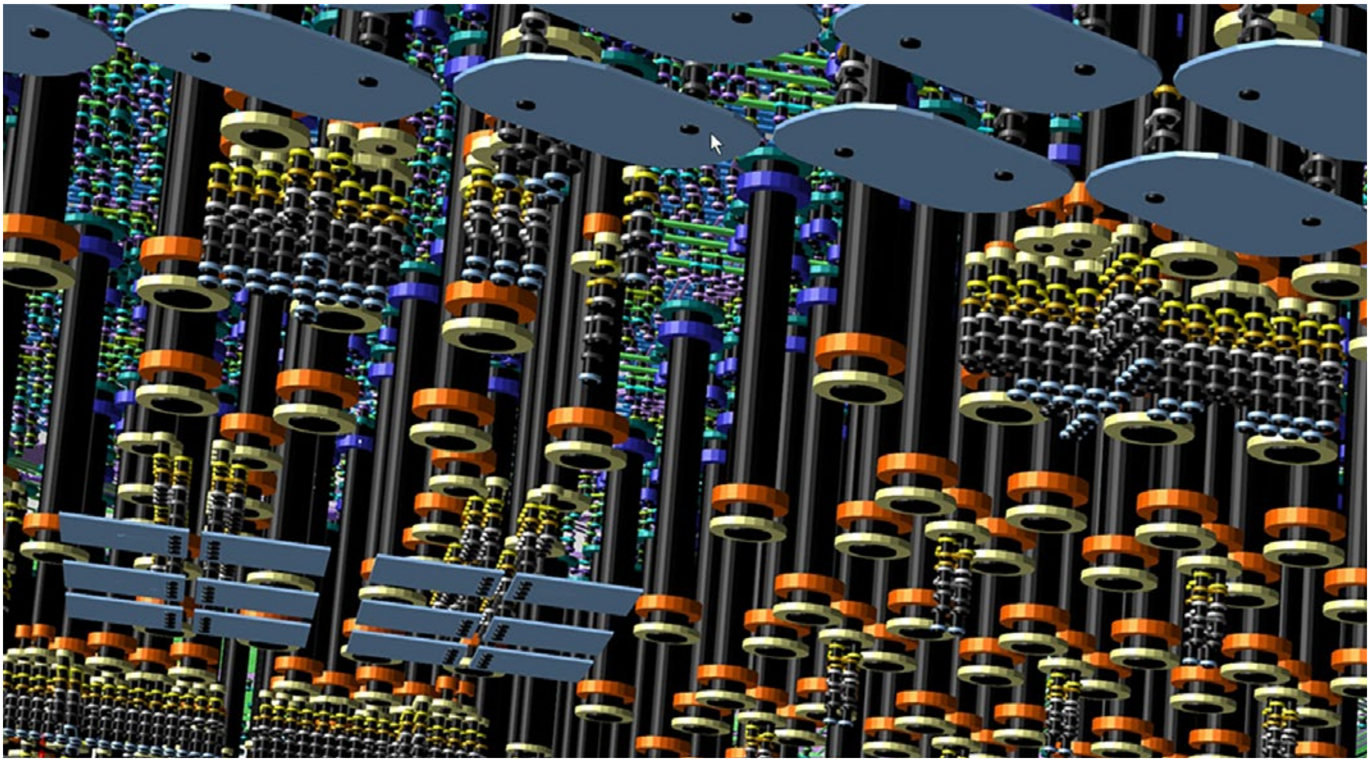


Image by Charles Pfeil

PCB Design: Where Art and Science Intersect

Feature Interview by the I-Connect007 Editorial Team

When we started planning this issue on the art of PCB design, we knew we'd have to speak with artists who are also PCB designers. Fortunately, we know several such designers. For this issue, we spoke with Mentor Graphics (now Siemens) and Altium alum Charles Pfeil, a great photographer; iCD founder Barry Olney, a wonderful painter; and Nordson Asymtek's Bill Brooks, whose sculptures rival those of Remington.

In this lively, wide-ranging conversation, we asked this trio to discuss the artistic side of PCB design, how art intersects with science, and whether a young designer can learn to be artistic. If you already had an idea on this topic, we venture this interview might change your mind.

Andy Shaughnessy: Many designers say PCB design is as much an art as it is a science; sometimes maybe more, or maybe not. We decided to gather some artists in our industry—in this case a painter, a sculptor, and a photographer—to talk about it. Is PCB design more art than science?

Charles Pfeil: Sure, there are artistic aspects to it, but I think there are some others too. For example, a PCB designer has the knowledge, experience, and ability to use the tools they have, and their intent is to create a printed circuit design that functions as desired. They don't start out wanting to create a work of art. A photographer also has the knowledge and ability to use tools that create a picture to fulfill the photographer's imagination. So, in a sense, they both have similar talents. I would call it a creative talent, but their goals are slightly different. To be a printed circuit board designer with artistic talent does indeed help solve the problems with placement routing. They can use imagination when they make decisions on which approach is the best. The photog-

Charles Pfeil

rapher goes through that as well with all the available options on the camera. You use one tool and another, there are conflicts, and you've got to compromise. So, I think that being able to use your imagination to come up with solutions is a fundamental, artistic ability.

Barry Matties: If you had two photographers and you set out to photograph a sunset, you would wind up with two very different photographs, and the same would probably happen with circuit design, right?

Bill Brooks: I would agree with that.

Barry Olney: Exactly. You can't get two designs that are totally the same. They'll always be different. That's just the way you see it. It starts with placement, and then you're routing on top of that.

Brooks: Depending on the skill level of the designer the boards may also be functionally different. The designer may do things that would compromise or potentially enhance performance based on their understanding of the design rules. Artists can play by rules too, but you must start by defining what art is. I find there are many things called art that don't all meet the same criteria. Somebody taking an art medium, pouring colored alcohol inks and paints on it, and coming up with a pleasing pattern is called art; they might be called an artist, but I would say the skill level to do that is not so high. On the other hand, there are those who can copy Monet or Renoir. I've even attempted to copy Bernini, who was a phenomenal sculptor. So, different levels of skill sets come into play as well as being able to use the imagination.

Olney: You mentioned, Bill, that it's the way you visualize the design, and I think that's true. You must look at their design with an electromagnetic field perspective to understand transmission lines, transmission line propagation, and controlled impedance, recognize how signals and currents flow, and how the coupling occurs at high frequencies. If you take all that in, you change your perspective of the board.

Now, a beginner who just starts designing a board will just look at the physical aspects. He'll place the chips and route it as best he can, but he won't visualize the electromagnetic fields and the propagation of the signal. It's very important from a high-speed design perspective to get all those fields working together, and your signal integrity and power integrity is all nailed during the process. It's a common process that you follow. You get a similar outcome every time, although the board looks totally different.

Pfeil: Take a designer who doesn't have the knowledge and experience of what is required to manage the high-speed condition; if that per-

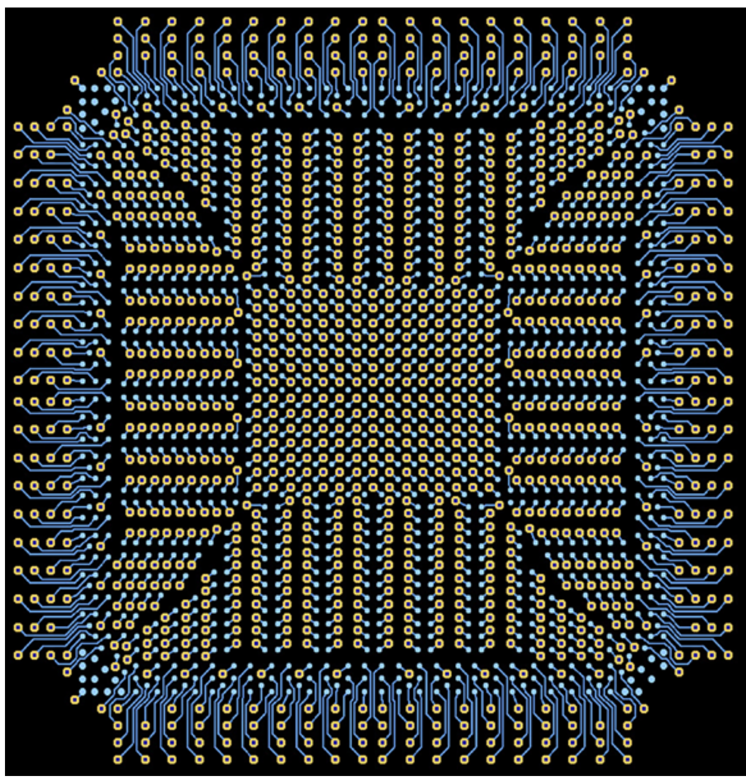


Image by Charles Pfeil.

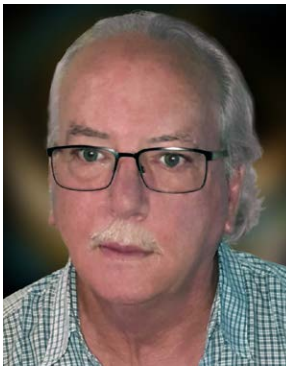
son has some artistic talent, it is likely there will be a lot of strictly parallel lines bunched together and creating crosstalk. It may look nice, but it won't function properly. But a designer who has the knowledge and experience, plus a little bit of OCD, will also want to have the lines running nice and straight with adequate clearances to avoid signal integrity problems.

It's a matter of efficiency. If I fulfill the design rules of a layout and it simulates properly, using efficient meandering as necessary, and using traces as short as possible to make routing less than the critical length, that is good. It can look artistic and will be easy to edit in the future.

Shaughnessy: It sounds like an artistic eye is a requirement for this job?

Olney: Yes. You must be symmetrical as well. I do everything symmetrically. I hate things that are out of balance. That's true with a PCB stackup from top layer to bottom layer. All the planes must be symmetrical about the center, and the way you route things, you choose strip line combinations that are symmetrical about the center. It's an artistic talent, but there's also attention to detail and you must be a perfectionist.

If I see a little kink in a trace, I'll fix it. I won't just leave and say, "Oh, that'll be all right." I can't leave it; it just annoys me. You must be a real perfectionist to design a board properly and you need to have the capacity to accept change because a board designer is in a constant state of flux. You're given a schematic, and you start placing and routing it. Then someone comes along and says, "Okay, here are some more changes. Add these five chips to the board." Now you must move everything around. You anticipate changes until the end of the design, so another ability you must possess is to deliver on time and under constant pressure. The board layout is the last process in the design cycle. For example, with Christmas coming up, there's always this mad rush



Barry Olney

from customers who want things finished by Christmas. It's just human nature.

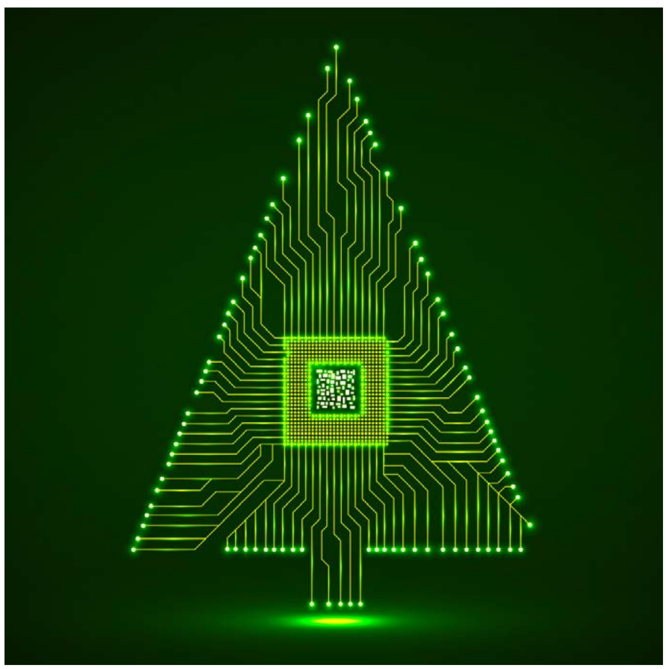
Matties: We surveyed designers and asked about the skill sets that are required to be a designer. The answers led us to see that circuit design seems to be more of an art of communication, an art of problem-solving, etc. All these disciplines are skills, if you will, but you must be able to creatively put them together to come up with something that is right the first time and optimally efficient.

Olney: You must have a thorough knowledge of your EDA tool. You must know how to drive it forward and backward, no matter what tool it is. Then, you must understand the fabrication process, assembly, and testing of the board. If you've got a high-speed board, you're also considering signal and power integrity, crosstalk, and EMI. There are these other issues that come into it apart from just taking the trace from one point to another on a board.

You must understand how the board is manufactured and how close that trace can go to certain obstacles. You must allow for the reflow process, depending on which soldering process you're using. Back in the old days, there was wave soldering and you had to worry about shadowing of two components. They all had to be in the same direction so the wave comes over both pins of a chip at the same time. It's simplified these days with reflow, especially double-sided reflow, which makes it really easy. But all those manufacturing concepts must be considered during design.

Pfeil: People like us have had a long career with printed circuit board design, and part of the attraction was the artist experience. Of course, we had to learn how to design boards that fulfill the engineer's requirements; but as more designers come from the engineering side, they must learn how to do layout and solve the problems that prevent successful performance. What's going to happen? Will they care about the efficiency of artistic solutions? Probably not so much. Maybe the automatic tools will do that automatically. I can tell you that since I've worked on the software side for many years, when we develop autorouting and interactive routing tools, we must come up with solutions that current artistic designers will also accept. If the results do not produce results that the designer would have produced themselves, they're not going to use it. This has been a problem for the software industry since the 1980s.

Trace glossing routines are intended to improve the routing, which can be rather chaotic. These routines will minimize lengths and smooth out unnecessary jogs, while hopefully fulfilling all the rules. Automatic routines will need to continually evolve to make the design—especially with new circuit technologies—function as desired and be easier to edit when it's done. These are the tools that the people from the engineering background will want to use, and current artistic designers will



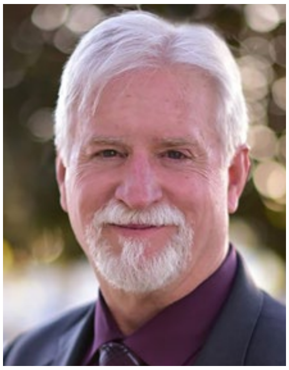
be less likely to say, "Well, it doesn't look like I would have done it, therefore I'm going to do it myself."

Brooks: Charles, which CAD tool were you writing software for?

Pfeil: I wasn't writing it. Primarily, I was the architect for Mentor Graphics Xpedition PCB (initially VeriBest PCB and now Siemens' Xpedition), and after that, I worked on Altium Designer. I did a lot of work on XtremePCB, Sketch Router, and Active Route, with a focus on route quality and efficiency.

Brooks: That's very cool. The art of glossing new routes to make them look human-created is an intuition thing. When we talk about how an artist has an intuitive mind, they get inspired by something to create art. We get inspired by being able to visualize or see things that haven't yet been created. That part of the artist brain is what we use when we're initially looking at a design and trying to assemble all these various requirements from our customers. As designers we have many customers. We have people who build and test it, and who want to make sure it survives in the environment that we are trying to build for.

The machines that I work on now have high vibration and things like that, so we need to be able to survive that. Look at space applications. I've done some aerospace design and microwave circuits, and each have their own special requirements. Being able to visualize that and compensate or plan for them takes that ability to kind of visualize or see, like we were talking about. I like to say people don't see things until they see them. By the way, I like what you've done with the glossing. I've been using Altium for quite some time and it's improved. It keeps getting better as time goes on. And it takes the excess effort and cleanup after working on a board and reduces it quite a bit and allows you to go in and make changes on the fly that you can't really plan for. An engineer gives you a



Bill Brooks

change in the middle of a design, like Barry was saying.

When I design a piece of art, it has an evolution. It starts as a big lump of clay, and I start changing it until it becomes the thing I'm creating. Oftentimes as I go along, I'll see a way to make it better, to make it more expressive, or to make it more exciting. The same thing happens with board design. I'll put down a circuit, maybe I've got a group of components that really want to be a family. They want to stay together. They don't want to be very far apart. I'll do a lot of that in the beginning when I'm designing the board and then I'll take those groups and find the best way to put them into the package so that they maximize all the other criteria that are part of the design. And the two processes are very similar.

I think we tend to use the tool we like the best to get the results we want. It may not be the fastest but I'm not working in a service bureau, with somebody standing over my shoulder, so I don't have that problem.

Now I can spend more time making sure I meet all the criteria of the design and it's not just slapped together, out the door, and then, "Oh gosh, we got a problem in environmental, or we've got a problem in assembly, or we've got a problem here, problem there." You don't want to find those when they get out to the customers. You want to have it all thought up ahead of time, checked, and ready to rock and roll. Most of my boards are right the first time, and every so often the second time, but usually the first time.

Olney: Bill, a good designer prevents problems. If it works beautifully, it keeps working and it's reliable, then you've done a good job.

Brooks: Yep.

Pfeil: One of the problems with software development is that it's always behind the technology curve. If you have new technology, new circuits, new kinds of components, and much higher speed circuits, when the software developed over the previous two or three years comes out, it is automatically behind the technology curve. Designers who use their imagination and their problem-solving capability will still be essential because the automation capability will always be behind the curve.

Brooks: Good point.

Shaughnessy: We've found that you can teach the science part of this. Can you teach the artistic side too? You know, EEs get this rap that they're not artistic enough, and you know how it goes. Can you teach the artistic side to someone who's more science-oriented?

Olney: No. I think people either have it or they don't. I've taught a lot of PCB design and these guys are all very capable. You chose them because they're smart people, but most of them are very logical and they'll do things properly, but there's a difference between peo-

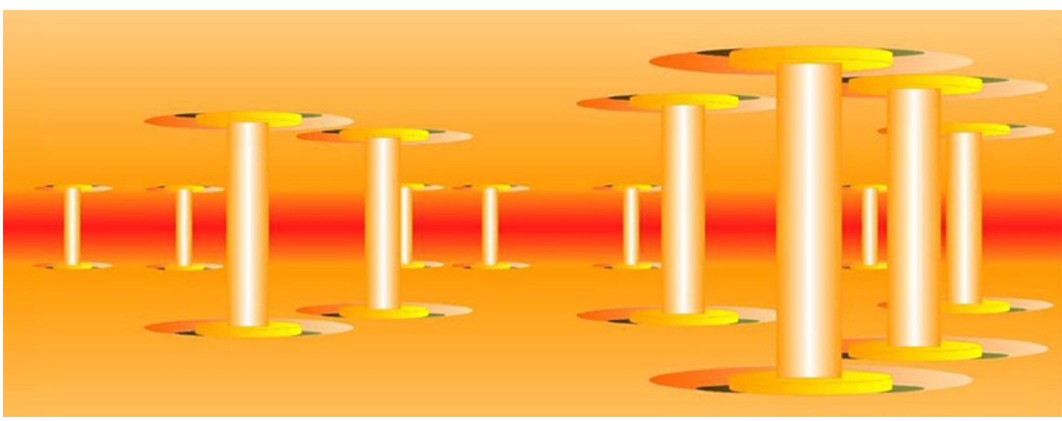


Image by Barry Olney.

ple with artistic talent and those who just do the job as a job. You've got to really love the job to do it well. I was talking to a guy who runs the service bureau here in Sydney a few months ago and he said, "I've tried to retire, but I just love the job so much, I just keep doing it." I feel that way too. I just keep going because I like doing what I do. If you don't like doing what you do, then you don't do it properly. You can't teach that.

Pfeil: I think it's interesting that advertising agencies for many different products will have an image of a PC board or PCB routing in the background. I don't know if you've noticed that. It's not really a printed circuit board design, it's the artist's rendition of what a printed circuit board looks like, and often you'll see multiple traces running through an area and then they all connect at one tab somewhere, or they'll connect to each other, or there will be traces that start and end just in space. The art attraction is that you see the parallel lines running through a maze of pads and it just kind of looks cool. Because of that, they see it as art, and they use it as a background for their advertising campaigns.

Brooks: Actually, my introduction to art happened later. I designed printed circuit boards a long time before I discovered art. I don't know too many people who were artist types that became printed circuit board designers. I know technical people with a lot of left-brain activity. But I think the right-brain part of it helps with spatial relationships between things and being able to see patterns. More like puzzle solving than art.

For example, look at a memory circuit; we used to place them in a such a way that you could get a bus going through the whole memory circuit, and it was very compact. It was recognizing those patterns, instead of somebody sitting all day, running a router, and getting a bunch of jobs. After the software was smart enough to recognize that too, then we

Painting by Barry Olney.

have that more easily. It's what you're able to see or visualize, or to be inspired on how to solve a problem. Those same characteristics help with art.

Brooks: When I was a kid, my dad was teaching me about printed circuit boards using a 10-watt audio amplifier. The schematic was inside the Texas Instruments data manual. It was one of the example circuits in there. I built it up and made it work, and that was a great experience. It got me started. I think I was about 14 years old.

Shaughnessy: What else would you like to share about this topic?

Brooks: I have a short story. My dad used to work for Aerospace Incorporated, and when he was down at UCSD working on the Pioneer 10 space program, he made all his own printed circuit boards; like an artist, he wanted to sign them. He would put his initials or his name on the lower part of the board. Dr. Walker Fillius, the PhD running the project, was a little concerned because when you're doing aerospace and rockets, every ounce that you send up costs a lot of money.

Well, one of his boards ended up getting on the Pioneer 10. After my dad passed away, his boss, Walker Fillius, told me, "Someday long after we're all gone, that probe is going to be very close to the eye of the constellation Taurus." I think the name of that star is Antares. Walker continued, "So maybe someday the little green men up there will be puzzling over what those initials were there for." That was a cute story about artists on the art side. I knew a lot of guys who wanted to put their initials on their boards because they were very proud of what they did.

Olney: I used to put a little red spider on mine.

Brooks: Ha, ha. I love it.

Olney: But the CAM guys used to delete it all the time because they thought it was just a piece of something that didn't belong there.

Brooks: Yes. After I did microwave boards, I realized that that loose copper tacks could also act like a really good antenna and reflect unwanted signals around the board at those higher frequencies.

Shaughnessy: Really good, guys. Thanks for talking about this.

Pfeil: Thank you. I enjoyed it.

Olney: Thanks, guys. DESIGN007

Charles Pfeil is a retired PCB layout software architect. **Bill Brooks** is a senior printed designer at Nordson ASYMTEK. **Barry Olney** is the founder of In-Circuit Design and an I-Connect007 columnist.

William “Bill” Brooks
sculptor, painter,
PCB designer.

FROM ELECTRONICS TO ART, VISTA MAN FINDS SOLACE IN SCULPTURE

BY ALBERT H. FULCHER

She sits tall in the saddle as her horse makes the last turn on their way to the finish line. With determination written across her face, her horse's muscles are taut as they begin their final leg to the finish line. She is a barrel racer.

William "Bill" Brooks' sculpture "Passion and Heart" captures this moment in time, already commissioned for finishing in bronze. After only four years of study, Brooks' works are on display at Art Beat on Main Street in Vista. With three in bronze and one in clay, people are taking notice of his innate ability to bring sculpture to life.

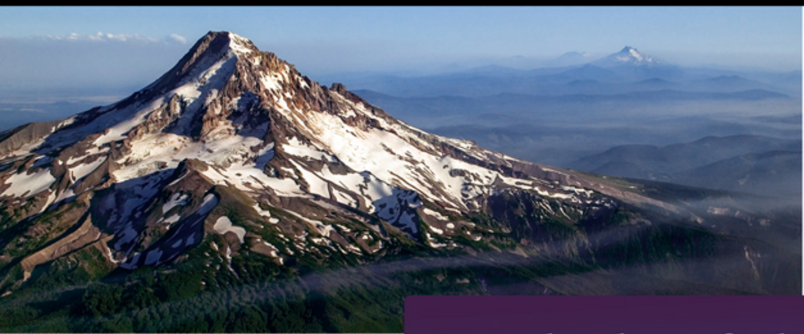
This is something new for the electronics technician who spends his days working on circuit boards. That was until a friend helped him discover Lynn Forbes Sculpture in Carlsbad and, after one visit, he decided it was too

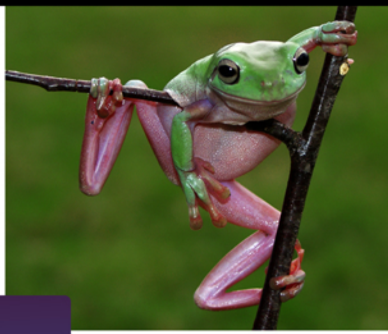
You put energy into it, so there's an energy that is never lost. It spreads to other people."

He said Forbes is a master at what she does and the serenity of her studio is ideal for bringing out the best of his artistry.

"She nurtures each artist who comes in and helps them find their voice through art," he said. "There's n competition between people to creat a negative environment. Here you ar only in competition with yourself. Eve thing you do is to try to improve you self as you go along."

Brooks said Forbes guided him,





Charles Pfeil, Photography

‘Dealing’ With PCB Design

Target Condition

by Kelly Dack, CIT, CID+

Welcome to my new column, where I will be pursuing the need for stakeholder advocacy in the printed circuit board industry. I’ll be deliberating topics essential to helping all of us understand what each PCB project stakeholder needs to achieve 100% acceptability for their stake in a PCB design project. These requirements will be expressed using the same helpful, graphic target condition example methodology which is utilized in the IPC-A-600 and A-610 specifications. I’m looking forward to exploring along with you, the PCB stakeholder, the target condition requirements for PCB sales and business development, engineering, design, procurement, manufacturing, test, inspection, and customer satisfaction.

Stakeholder—There is Power in the Term

Regarding the overall success of the PCB industry, it has been written in the pages of this publication that “Everything starts with design.”[1] I agree with this statement as long as the definition of design is based upon a project team approach, one which recognizes all the players or “stakeholders” who have a vested interest in the overall success of a PCB assembly project. There is power associated with the role of a stakeholder. Most PCB project stakeholders are not designers per se, but they have just as much of a vested interest in winning. They are empowered by the knowledge, capabilities, and skill they have managed to acquire over time, but they are too often disempowered by being overlooked and misunderstood by other stakeholders due to project team oversights which do not consider their

unique requirements far enough in advance.

Winning in the business of electronics must include all aspects of producing a successful printed circuit assembly. Every aspect is represented by a skillful stakeholder who, together with the project team, step up to play their hands against "the house"—the customer or entity which makes all the rules. Individual stakeholders must consider the odds together as a project team before going all in. This is far from a scenario of winner takes all because to get a PCB project successfully to market it takes all to win.

What's at Stake?

"What's at stake?" This crucial question has been asked around casino gaming tables long before printed circuit project proposals ever existed. When powerful players stepped up to a table, they first had to ante up with something of value—cash, a promissory note or even a deed to property—to offer as a stake in the game.

Any PCB stakeholder—a designer, contractor, supplier management professional, bare board supplier, or electronic manufacturing services (EMS) company—must consider what is at stake before engaging in business. The savvy stakeholder realizes there is risk involved in doing business. Failure to count the cost of the project and consider all the available stakeholder resources required to play in the game risks the valuable time and financial resources of the associated stakeholders if unforeseen odds or circumstances should arise. Too many projects are put at risk of folding due to blatant disregard for stakeholder requirements. If something is at stake, it might be lost or damaged if the players are not successful playing their hand at the table.

After the Dealin' is Done

Your company has won the business and is revving up to produce the new project. What could go wrong? Plenty, if your fellow-stakeholders don't get a chance to play their hands. This continues to be a sad fact cited in many a PCB project team's post-mortem review. Some examples:

- The PCB design routing and stackup was approved and released before the production supplier ever had a chance to verify material availability
- The layout was populated with parts which were not verified for cost
- The layout was sent out for fabrication before the test engineers had a chance to evaluate for accessibility
- The EE forgot to tell supplier management that certain features on the PCB could only be manufactured by a sole source

The list goes on and on. I am a rabid advocate for DFX in printed circuit design. I've read, written, and taught extensively on the subject. But now, going into my seventh year working within an EMS organization as a design and manufacturing liaison, I am continuing to see a steady disregard for basic manufacturability, materials sourcing, assembly, and test. Has the predicted tsunami of new EE graduates been too quick to learn the layout tools? Too quick in outputting data "for" their downstream stakeholders before learning what they really need? In many cases, yes. But fresh EE designer stakeholders don't know what they don't know. They'll continue leaning forward, independently running with scissors, so-to-speak, until their fellow stakeholders can kindly and gently introduce them to an extended project team that compels them to learn to provide manufacturing data feasible for helping them to hit their target conditions.

The Thanksgiving I Overplayed My Hand: A DFT Story

I prepared a turkey for my family one year at Thanksgiving. I sourced the bird. I obtained the propane-powered base, frying kettle, and the peanut oil. I watched the "how to" video on YouTube to become a quick expert on how it was done. I lowered the bird into the hot oil and watched it boil to a crispy doneness. I carved it up, plated the meat and placed it upon our kitchen island for all to enjoy. While I had processed this endeavor alone in the safety of the backyard, taking just a quick 30 minutes, my family had been working together all day inside, making stuffing, seasoning, and creating all the side dishes for a traditional turkey dinner which they savored for hours as it roasted in the oven.

To make a long story short, my DFT (deep fried turkey) project was a flop. It missed everyone's target condition. Good thing there was a diverse project team of family stakeholders working together on a viable alternative. I spent all afternoon working on a deliverable which I alone had determined my family needed. In the end, they didn't need it. It wasn't working for them. The crispy skin had a hankering of rancidity from the peanut oil. The meat tasted like peanut oil. It was a Thanksgiving turkey project only a mother could love. My mom did, in fact, empathetically take a slice of breast meat and compliment me on my efforts while she laboriously chewed away on it. Later in the evening she had to pass on the pumpkin pie—her favorite—because her stomach had begun to hurt.

Have we arrived at a time in which there is entirely too much written each month on the topics of designing "for" someone? Is the "design for" message being misconstrued by some, as in my DFT example?

On that day, harmony and fulfillment were experienced by all who worked in the kitchen on the roasted turkey project. Everyone seemed to know what to do and who to call for help. Convection oven profiles were set to ramp up, preheat/soak times were minimal, and peak temperature was set to an even 350°F, calculated and set at 13 minutes per pound "based" on an unstuffed "turn-key." The succulent meat was successfully excised from its carcass and each side of the breast was gently lifted out by one family stakeholder, set upon the cutting board and precisely cross-cut against the grain by another. This project was a success because the family stakeholders worked together throughout the process by communicating and agreeing on what it takes to meet the target conditions for the entire project team.

Working Together, We Can Beat the House

But can our entire industry of PCB project stakeholders be helped to meet their target conditions? Perhaps it is time for the PCB industry to start designing "with."

Part of the success of this column will involve reaching out to you regularly to make contact. I want to talk to and hear about your unique experiences as you strive to achieve the target conditions of your specialized requirements for PCB product development. With your help and permission, I want to collect and re-tell your accounts of stakeholder success and failure. You can remain anonymous if you wish, but the idea here is to share your story so we can learn from it. Second, I want to connect you with other potential project stakeholders who can become as familiar as family to you. Who knows? You and that new stakeholder contact might end up working together someday on the same project and be able to help each other reach your target condition requirements.

Who Says You Can't Choose Your Family?

I know a lot of wonderful people in the PCB industry—stakeholders—each with their own set of target conditions which I want to help them hit. But a quick search on LinkedIn shows there are thousands more of you whom I do not yet know. Perhaps we can connect through this column to share and help others.

I don't presume to know every PCB project stakeholder's job inside and out. Over the course of the past few decades, I've spent time in my industry roles as a PCB designer, and more recently as a PCB design and manufacturing liaison, learning a lot about stakeholder challenges. But I want our readers to hear these challenges from you so we can process this and learn together. Again, please contact me on LinkedIn or through my column page if you would like the PCB project world to know more about your role in the industry so we might help.

The Denouement: Read 'Em and Weep

Concurrent engineering (DWF or design with fabrication) and training for evolving manufacturing practices are needed now more than ever. For the short term, our PCB project stakeholder community must become more "woke" to stakeholder limitations and more supportive of present stakeholder capabilities for their short-term projects. Connected, automated, smart factories and the software and systems which support them are here and more are coming quickly. PCB stakeholders must become aware of what is coming so they can adjust, prepare, and tool up because competition and advancements in technology without first building stakeholder relationships and awareness can cause project teams to call the bets before these stakes are raised considerably.

Thankfully, the I-Connect007 family of PCB industry publications provides a voice for PCB project stakeholders to share this information. Month after month, we read about what to watch out for in design in order to avoid overplaying a fellow stakeholder's hand. May we all become tuned in to one another's target condition requirements for success and strive to help one another achieve it.

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

References

1. *Everything Starts With Design, Design007 Magazine,* June 2019.

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 contact Dack or visit his column online, click here.

Will AI Replace the Art of PCB Design?

Feature Article by Stephen V. Chavez
PCEA

With this issue of *Design007 Magazine* focusing on the art and science of PCB design, let's pause for a moment and think about how PCB design has advanced over the past few decades. As I step back and think about this in the grand scheme of things, I am amazed at how this segment has evolved.

In today's world, PCB design has become much more complex in nature, with so many details to account for. At the same time, many of today's PCB designers have evolved into much more than the "ECAD draftsmen" of the not-so-distant past. We don't just simply "connect the dots" or "push the magic button" as some may imagine.

The profession of designing PCBs, once known as electrical drafting, is more accurately known as printed circuit engineering. Today, not only are we designing complex PCBs that contain physical packages smaller than ever before, but we are addressing electrical, mechanical, and thermal variables with a much higher level of complexity. And don't forget—we need to design each board faster than the last, all while cutting cost and resources. That is an art in itself.

Today's PCB designer must be part technologist and part artist. Think about it—it takes a lot more than technical knowledge and ability to design PCBs now, many with blazing fast signal speeds and edge rates. As if the electrical and mechanical design complexity of a PCB weren't enough of a challenge, let's add in the manufacturing complexities. All this makes designing entire complex systems a true challenge indeed. Simply put, to be successful in PCB design, you really need to know and understand what you are doing and how the decisions you make and implement upstream impact your stakeholders downstream.

Everything a designer does, right or wrong, has ramifications downstream. We are witnessing a trend in this industry: Many new PCB designers are recently degreed electrical engineers. As the industry continues to change, so do our EDA tools, which have more horsepower and capabilities than ever before. We are doing things better and faster than ever, but is there still opportunity to be even better and faster? I believe so, but we must be willing to trust our respective EDA tools.


EMA Design Automation®
Enhance Your PCB Design Flow
Check Out Our Year-End Offers While They Last
View Offers
PCB Design | Data Management | Supply Chain Integration | Simulation & Analysis | Training
www.ema-eda.com

Here's where the "art vs. science" discussion comes in, because knowing how much automation to use requires an artist's touch. Many designers still do things manually instead of using the tool's automation to save time and costs, while reducing the number of errors. Don't get me wrong; many designers have great success with, for example, routing boards by hand. While doing things manually may be acceptable within one company's ecosystem, there are many engineering teams and companies who are taking advantage of today's automation and horsepower to be better, faster, and ultimately more successful.

I've read a lot recently about the development of AI and IoT, and I believe AI offers true potential for PCB designers. But we must be willing to think and act outside the box. Industry research suggests that there are crucial electronic systems design areas where AI technologies, including machine learning, can be applied to minimize or eliminate the complexity of electronic system design work.

This topic had my internal gears turning. I reflected on a recent project design in which we implemented hardware IP design re-use with Xpedition Enterprise, and I realized that today's EDA tools already utilize some forms of AI. Implementing a complex schematic-driven constraint approach while creating the schematic, and automating component cluster placement while implementing IP design re-use blocks, is possible because of AI. Autorouting and sketch routing are also the results of limited AI functionality.

These capabilities lead us to significantly reduce our overall design cycle time for three designs vs. the typical manual approach. The potential for long-term cost and labor savings from creating and unitizing the hardware IP re-use blocks alone is huge. But despite the high level of horsepower and automation in our EDA tools, they are still limited because of the human limitation factor.

What if we removed the human limitations factors by applying AI to the actual design and layout of these PCBs? If we were to replace the human limitations factor with an AI functionality that could design boards based on rules and constraints from algorithms of best practices and lessons learned from human designers, can you see the potential for success?

AI has been implemented for many years in the IC and software worlds with great success. I don't see why PCB design would not fall in line with this. We already have high-level automation built into today's tools, and AI might not be such a big step after all. But allowing AI to design PCBs and electronic systems is a scary thought because it will remove much of the human touch from PCB design.

I see the art and science of PCB design intertwined with AI. Will we lose the art form, artistic side, or craftsmanship of PCB design? I don't think so. We will just continue to evolve. Whether a design is completed manually or with automation, they both look great. If the mechanical and electrical constraints are implemented and adhered to, and then the final design meets the final requirements, looks are irrelevant of the final design. All we care about is that it works and that it can be built with higher yields, on time, and under budget.

In the end, utilizing AI in PCB design will require us to trust the tool and utilize its horsepower to our advantage. And even the most "intelligent" AI tool will always require a human touch.

Are we ready to take that step? Many senior designers with many decades of experience would probably say No. But the PCB designers of tomorrow are a different story. These "digital natives" have grown up with cellphones and mp3s, and they might wind up embracing AI in PCB design. We'll know soon enough. DESIGN007

Stephen V. Chavez, MIT, CID+, is the chairman of PCEA.

PegaClad Series

for Sub-6 GHz Antenna (Dk 3.00, 3.88, 3.45, 3.65)

PegaClad 1&2

for mmWave

Developed for Sub-6 GHz antenna applications, TUC's PegaClad 300, PegaClad 338, PegaClad 345, and PegaClad 365 - with respective Dks of 3.00, 3.38, 3.45, and 3.65 - are advanced materials designed to minimize passive intermodulation and meet designers' needs. PegaClad 300, PegaClad 338, PegaClad 345, and PegaClad 365 can be used in multilayer and double-sided circuit board designs requiring excellent thermal reliability.

TUC's PegaClad 1 and 2 are designed for high frequency, very / super low loss applications, including mmWave printed circuit board designs. Their electrical properties are equivalent to PTFE laminates, yet they perform in multilayer circuit boards requiring excellent thermal reliability in harsh environments.

TUC's PegaClad Series and PegaClad 1 and 2 laminates also exhibit excellent moisture resistance, improved CTE, superior chemical resistance, and thermal stability, and are compatible with modified FR-4 processing.

tuc

Taiwan Union Technology Corporation

www.tuc.com.tw

TUC Asia
Raymond Foo
TEL : +886 3 5551103 x 129
E-mail :
raymond.foo@tuc.com.tw

TUC USA
Dana Korf
TEL : +1 408 643 1144
E-mail :
dana.korf@tuc.com.tw

TUC Europe
Marko Holappa
TEL : +358 40 770 9618
E-mail :
marko.holappa@tuc.com.tw

The Rise of Resin in IoT Applications

Sensible Design

by Beth Turner, ELECTROLUBE

The Internet of Things (IoT) is a platform enabling embedded devices that are connected to the internet to collect and exchange data with each other. Devices can begin to interact and work with each other, even learning from each other's experience as humans do. The potential for IoT and connectivity is endless, as everyday objects can connect and share intelligence and knowledge. Some current examples deploying IoT include one of the earliest examples of this technology, wearable technology, which has now vastly extended to encompass the healthcare sector, sports, athletics, and even implantable devices.

Some other areas where IoT applications are experiencing rapid growth include the smart home, health care, smart cities, smart grids, agriculture, connectivity within vehicles, and industrial automation. In this month's column, I will be exploring how resin chemistries can be incorporated by design engineers facing ever increasing new challenges from IoT applications.

New technology is generally targeted at making our lives better. IoT is all about connecting physical devices wirelessly, using sensors and software to exchange data and facilitate connections and communication.

There are loads of exciting examples of how IoT is helping to improve and sustain the world around us, not just make our day-to-day lives simpler. Smart connected homes can prevent unnecessary energy use with the ability to turn off lights, or control heating/cooling activity remotely. Smart sensors are even more sophisticated, able to measure air quality and send alerts when pollution levels are high, which is a great way to drive communities and

governments to combat rising pollution levels and bring health benefits to asthma suffers.

Smart farming involves monitoring the external environment and agricultural processes to automate a response; this makes it possible to reduce the amount to water used to hydrate crops, thus reducing wasteful processes, and focusing on the value-added inputs.

In my work, we have developed potting and encapsulating resins for the protection of electronic devices from challenging environments. One recent collaboration involved protecting an IoT sensor within a fuel storage tank monitoring system.

Are Potting Compounds and Encapsulation Resins Identical?

The terms "potting" and "encapsulation" are used interchangeably. Both describe a simple process of covering a component to provide protection from the threats posed by the external environment. In the case of protecting electronics, typically a one- or two-part resin is poured in its liquid state over components of a circuit board. A curing mechanism takes place; this could be a chemical, thermal, UV, or a moisture cure process, to form a solid resin to sufficiently ruggedize the electronics.

However, if you want to be technical, there is a slight difference between potting and encapsulation processes. Potting describes the scenario where the electronic component, for example a PCBA, is housed within a unit, a "pot." It would normally be the case where the "potting compound" is poured in to fill the entire housing unit; once cured, the entire system—the electronic component, cured potting compound and the housing unit—become the finished component.

Encapsulation on the other hand, describes a process where the electronic component and cured encapsulation resin are removed from the housing (the pot), and placed into a different assembly; the housing is merely there to act as a temporary mould to pour the resin in place.

Resins for Wearable Technology

Wearable devices are becoming increasingly popular, and by utilising IoT technology, these gadgets are leading us to a better-connected lifestyle. Typically, these devices rely on sensors and use RF signals to transfer information, connect, and communicate with another device. The main challenge for wearable devices is ensuring the delicate sensors and components can continue to function in the environments they are subject to. Consider a smart watch. The sensor must survive several physical interactions; the watch is worn, moved, experiences physical shocks, and may be stretched, plus it is usually exposed to several elements, like water, chemicals, or oils—especially for fitness fanatics tracking all their extreme activities. Clearly these devices must be protected to ensure reliable performance.

IoT is an evolving market where the various performance criteria for components is constantly changing to keep up with the latest developments in technology. Encapsulation resins capable of dissipating heat away from hard-working components clearly add value to a variety of applications.

I hope this has been of interest to those of you involved in IoT and helps make life a bit easier for those who are responsible for making the decisions on protecting components and circuitry. Look for my next column, where I will be exploring resin systems in more detail. DESIGN007

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


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

Leveraging Digital Twins to Optimize Electronic Systems

Digital Transformation

by David Wiens, SIEMENS EDA

Since the beginning of PCB design, physical prototypes have played a heavy role in the verification and ultimate approval of a new product for market. The challenge posed by this approach is simple—most designs require more than one prototype spin to achieve approval.

A Lifecycle Insights survey[1] turned up an industry average of 2.8 respins per project at a cost of $46,000 per spin. Certainly, there are variables such as design size and complexity, number of boards in a system, bare vs. fully assembled, and target industry or application. These all impact the cost and number of typical spins. As design complexity increases, so do the unknowns; even if a team designs conservatively, they're bound to trip over a few new spin-inducing issues. This is so common that project managers often "bake in" three to four respins to project schedules; you could call that mitigating risk—or planning to fail.

The promise of the digital transformation of the electronics design process is "zero-spin," going directly from design into volume production. This requires that every existing check performed on a physical prototype has a digital equivalent, or better yet, constraints synthesized from requirements that ensure correct-by-design. The reality today is that confidence in digital verification isn't high enough for anyone to bet the farm on zero-spin; most consider a single, fully-tested prototype pass as the holy grail.

The heart of digital verification is the digital twin, a model of the design with enough fidelity to ensure that checks or simulations catch errors typically caught within the physical testing process. A model of the electronics system is hierarchically constructed of many smaller models representing the enclosure, the environment, the multiple boards, and each board's materials/stackups, components, and

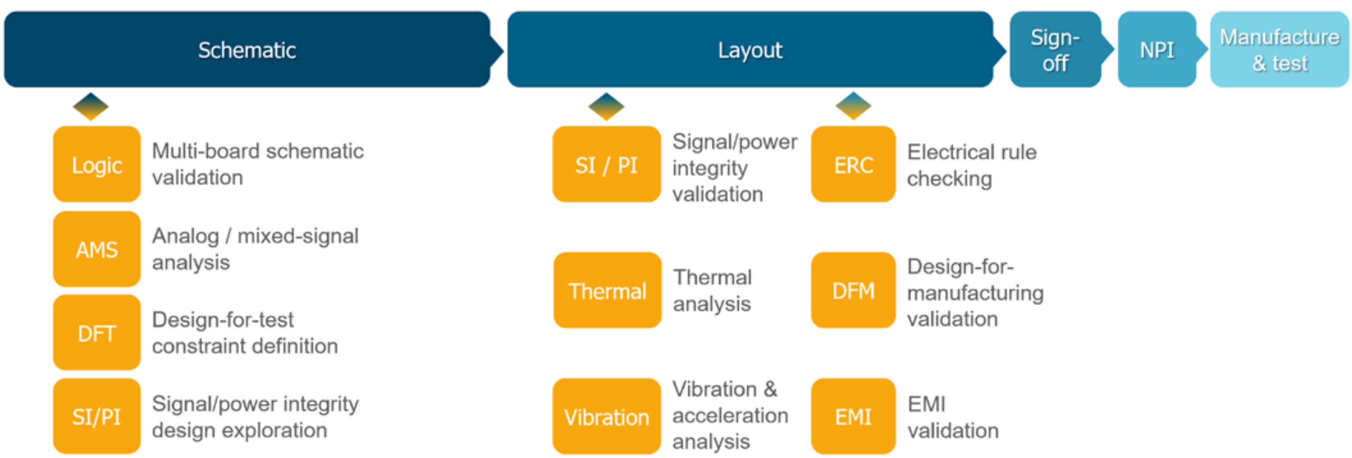


Figure 1: A strong digital twin facilitates design trade-offs and identifies problems early.

interconnects. Since models are necessary for an array of checks, they must also be multi-discipline. For instance, a PCB component model could be used to verify signal integrity (SI), power integrity (PI), thermal, form/fit, vibration, and more.

Today, while there are more models for PCB components than any other part of the electronics system, they are still far from ideal: there are multiple formats (e.g., IBIS, VHDL-AMS, BCI-ROM); there are many variants, each targeted at a single type of analysis; and quality varies widely. Yet, there is hope. Device vendors realize the value of models to help sell their product, so some are introducing models when they launch a new device. And there are industry efforts to standardize on neutral formats to represent multi-discipline models. In the meantime, creation and/or validation of models is often left to each engineering team that wants to utilize them.

> **If digital twins (models) are the fuel, then the simulators that consume them are the engine.**

If digital twins (models) are the fuel, then the simulators that consume them are the engine. These engines typically perform a single function (e.g., SI analysis, or even more specifically, DDRx or SerDes analysis). Significant energy has been spent over the last two decades to improve the accuracy of these simulations so that they mirror what's seen in the real world with the physical product. Of course, it's a moving target, as new signaling protocols are introduced along with new devices, materials, and manufacturing processes. Since the results of these simulations are typically one-dimensional, it's left to the engineer to manage trade-offs (e.g., between performance and manufacturability, or SI and thermal performance).

Today, the state-of-the-art is multi-physics simulators (e.g., SI and PI, PI and thermal, thermal and vibration) coupled with AI-enabled technologies capable of identifying the ideal solution given a set of target operating parameters. Cloud-based compute farms promise scalability from today's world where, even within one discipline like SI, only small portions of the system model can be analyzed at a time. As there are often far too many manual steps and opportunities for error and wasted time, there are efforts to streamline the digital thread from the core design database and authoring environment into these simulators. Yet we're still a ways off from the holy grail of a single, multi-dimensional model of the product that's consumed by a single, multi-dimensional simulator capable of emulating all real-world conditions.

I should note that, while many simulators have been validated in comparison to a physical prototype, the real target is the product in use in its end-market. That environment doesn't have test fixtures that artificially send signals, nor does it have mechanical jigs that hold a single board in a HALT chamber. Simulators can test corner cases on a digital twin—something that's impractical with physical prototypes (even if you build thousands of boards, you can't ensure that they represent every possible operating condition). And simulators offer the promise of automated multi-domain verification vs. the one-at-a-time manual testing processes for physical prototypes.

Over the past couple decades, verification has been slowly shifting earlier (to the left) in the design process. It moved first from a physical prototype-only approach to complex, hard-to-use simulators applied at the end of the design process by domain specialists. The second shift in verification has been from the end of design to the core authoring stage, enabled by tighter integration with the central digital twin of the design. While not replacing

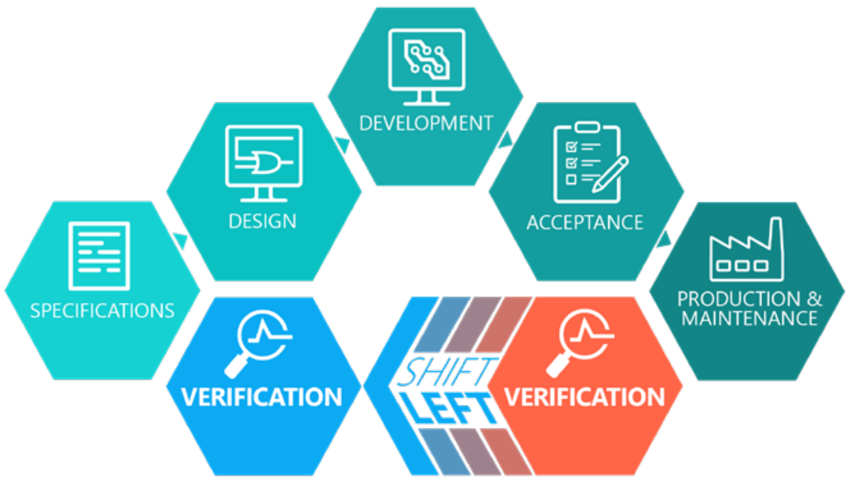


Figure 2: An optimal design process leverages digital verification to minimize respins.

the specialists, this shift enabled the design authors to make smarter decisions earlier, minimizing iterations with the specialist at the end of the design process. The third shift, currently in gestation, leverages AI to generatively synthesize designs to achieve optimal performance, based on a set of high-level product requirements. This generative technology has been applied to mechanical design, enabling additive manufacturing of optimal structures. Lest I cause anyone to fear for their jobs, I should note that electronics systems represent a far more complex, multi-domain, multi-discipline problem than purely mechanical structures.

An Aberdeen survey asked about processes for design verification. It classified the use of simulation by design engineers early in the design process as best-in-class for the following reasons:

- First, it enabled product innovation and optimization, because physical prototypes were reduced 27% using virtual prototypes and virtual testing. This allowed them to explore hundreds of design iterations, freeing them up to identify and focus on designs with the highest potential.
- It improved time to market. Best-in-class designers improved their length of development time by 29%—six times the rate of improvement by all others. Best-in-class organizations also met their time-to-market targets 76% of the time, a 17% higher rate than all others.
- It reduced product costs; 71% of best-in-class designers met their product cost targets.
- Higher product quality was achieved; 77% of best-in-class firms met their product quality targets. Plus, best-in-class products were more likely to work right the first time and less likely to require rework: the best-in-class teams improved their ECOs, after release to manufacturing, by 21%.

While I've pointed out plenty of obstacles to complete digital verification with an optimal digital twin, there are still many steps an engineering team can take today to leverage existing technology and minimize physical prototypes. Organizations are incorporating discrete simulators and checkers (e.g., for SI, PI, thermal, vibration, or manufacturability) and are reporting first-pass success. Incorporation earlier in the design process has improved engineering decision-making and minimized iterations with specialists. Fundamentally, leveraging a digital twin enables easier, faster verification which ensures higher quality products without time- and cost-consuming respins. **DESIGN007**

References

"Accelerating development with pervasive simulation," a white paper presented by Siemens.

David Wiens is Xpedition product manager for Siemens Digital Industries Software. To read past columns or contact Wiens, click here.

Field of Dreams: The Art and Science of Designing with Field Theory

Dan Beeker

Feature Interview by Andy Shaughnessy
I-CONNECT007

For this issue of *Design007 Magazine* focusing on the art and science of PCB design, we sought input from Dan Beeker, senior principal engineer of NXP Semiconductors. Dan has spent years teaching designers about the need to focus on the electromagnetic fields around the traces, not the traces themselves, as the late Ralph Morrison advised. We asked Dan to share his thoughts on the art of designing PCBs, and how art and science figure into his views on field theory.

Andy Shaughnessy: We often hear that PCB design is part art and part science, right brain and left brain. Do you think it's more of one than the other, or does it depend on the design job?

Dan Beeker: I think that the legacy has been more focused on the art side, to make things symmetrical and "pretty." Now, the science needs to dominate, and those grouped traces need to be separated as much as possible in order to improve signal integrity and reduce crosstalk. The same idea needs to apply to the urge to flood large areas of power conductors to fill in the entire layer. The correct way is to use the vias that connect the power "islands" to the ICs and capacitors, leading to ugly, asymmetrical shapes.

Shaughnessy: Can someone with a great mind for science learn the artistic side of design? That's one thing that non-degreed designers jokingly say about EEs—that they focus too much on science and not enough on the beauty of the design. What do you think?

Beeker: I am sure that is possible. It is about where you find beauty. I find beauty in visualizing the continuous transmission lines, perfectly controlling the flow of energy, the perspective of symmetry of the two conductors bounding the dielectric. (Yes, I know I am weird.) There is always some beauty resulting from the different colored shapes shown in the layout tools.

Shaughnessy: Does a PCB designer need to have an artistic side to apply the science?

Beeker: I think that one needs to have an abstract side, like Van Gogh or Delaunay—shapes that are defined by their function and connectivity result in the best performance.

Shaughnessy: You've been spreading the word about the need for designers and EEs to focus more on fields than on the traces, as many designers were taught. Does this require a designer to have artistic and scientific abilities in order to apply this concept correctly?

Beeker: I believe it goes to trust. It was hard for me to trust the shapes to manage the fields; plumbing connects the sources of energy to the places on the board where it will be used, with enough capacity to do the job. A one-gallon-a-minute hose cannot deliver five gallons a minute. You need a bigger hose or five hoses. The same is true for PCB design.

Shaughnessy: Steve Jobs demanded that the inside of the device look as beautiful as the outside. Are electronics designers and engineers "closet artists" looking for a chance to express themselves?

Beeker: As a goal in itself, that is not always going to yield a good design. Symmetry is a common standard for beauty, and that is rarely present in a good design.

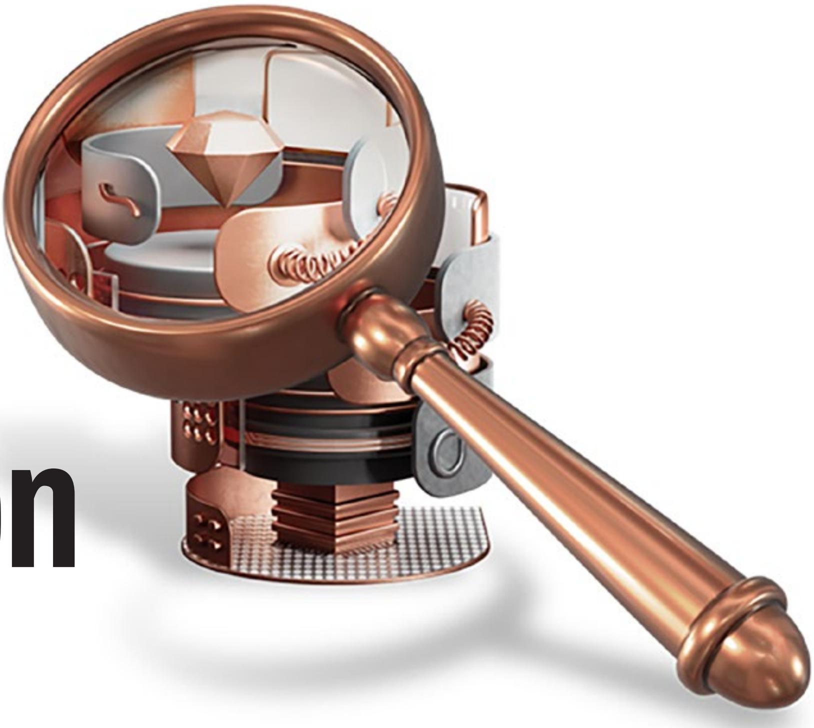
Shaughnessy: Is there anything else you'd like to add?

Beeker: It's all about the space, which still seems to be the final frontier. If you are not designing the spaces, with today's products, the cost is severe. Billions of dollars are spent every year by well-meaning design teams on redesigning PCBs, often with very little hope that the new version will be any better than the first. Ralph Morrison said it best:

"Buildings have walls and halls. People travel in the halls not the walls. Circuits have traces and spaces. Energy and signals travel in the spaces not the traces." **DESIGN007**

Dan Beeker, Principal Engineer, NXP Semiconductors chatting with Happy Holden.

Test and Inspection



Article by Ofer Manovitz and Isaak Golod
PCB TECHNOLOGIES

Hardware design and manufacturing of new electronic devices entail end-to-end quality inspection processes to ensure the delivery of defect-free products and systems. Traditionally, each step along the value stream (PCB design, PCB manufacturing, PCBA, and system integration) has its DFX carried by the respective engineering and QA teams.

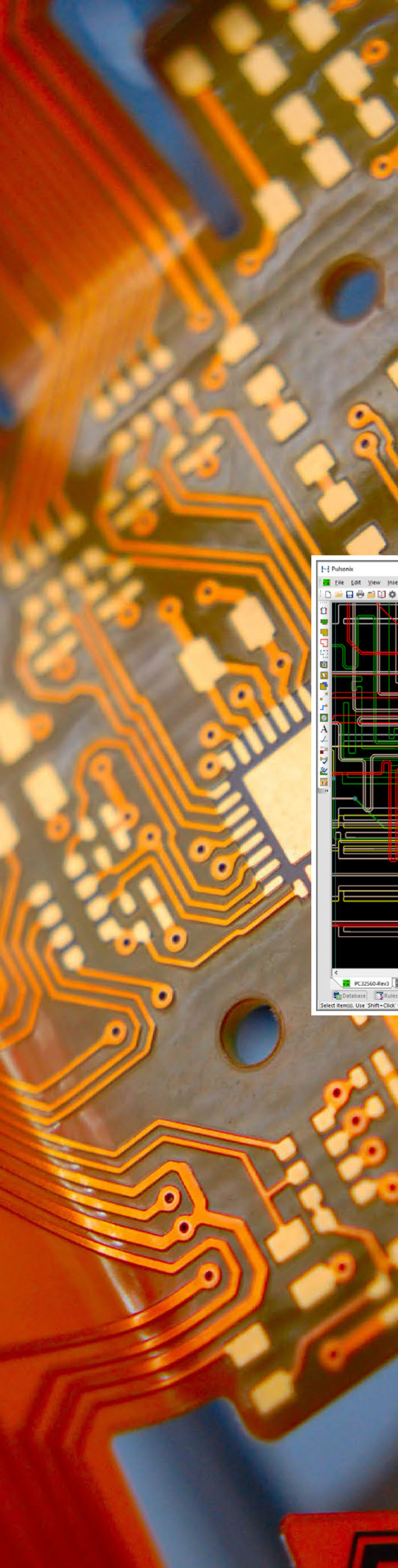
When DFX is carried out by one engineering team for the PCB design and manufacturing, assembly, and complete system integration, it allows the planning and execution of the optimal tests and inspections necessary for the product's design, quality, cost-effectiveness, and functionality.

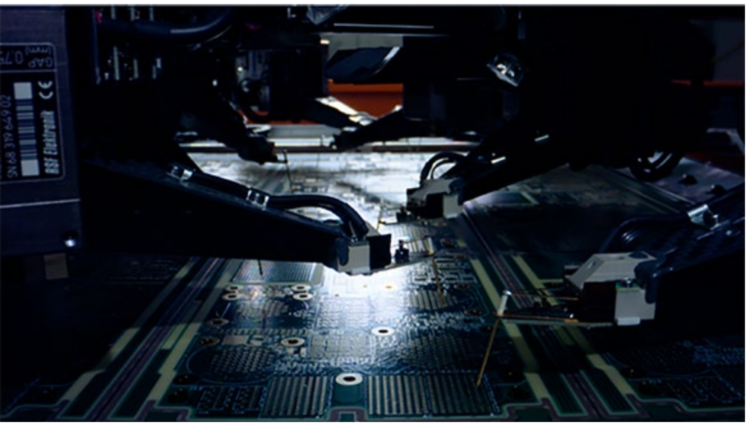
Designing PCBs, especially complex ones, must be intertwined with the test and inspection plan and management. When designing a new product, you need to consider all phases in the product's life cycle—from sketch to cradle—and carefully plan each phase. One of the critical elements in the design is the testability of the PCB and the final system long before its manufacturing to ensure a consecutive fabrication process, the highest yield possible, and zero defects—all aimed at securing your client's maximum profitability. For that, you need a solid validation strategy.

Three main verticals are required to get the optimal test and inspection end-to-end process:

1. A solid quality and inspection plan throughout the entire production process.
2. Detailed definition of the interfaces between all inspection steps enables optimized risk assessment and root cause analysis.
3. Continuous feedback to enable improved yield and system performance.

With long years of experience, we have concluded that it's best to get the engineering team involved in the PCB, PCBA, and system design, even at the schematic layout and design stages, to ensure their successful implementation. Another issue that needs special attention is the quality of the components. You need to make sure you're going to use qualified components or validate them yourself before you start the assembly to ensure no error arises due to their malfunction during or after the assembly process.





A tight, detailed, and carefully planned testing process will guarantee the following:

1. Cost savings: No unnecessary or double tests are conducted. At the same time, the right mix of tests allows for complementary results.
2. Enhanced test reliability while predicting the number of false negatives and improving the manufacturing process in real-time.
3. Tracing the root cause for any discrepancy once occurring and enabling an effective corrective action preventive action (CAPA) plan.
4. Peace of mind for the customer.

This evolutionary process intensifies the tests' reliability and even enables a decrease in the number of tests conducted while relying on sample testing only. The incentive is minimizing the expenses involved in the process and shortening it even further, making it as efficient as possible.

If you're looking for an optimal test process, you need to keep an open dialogue between the fabricator and the customer, covering as many iterations as required. Its main goal is to meet the criteria set by the customer.

Manufacturing of any electronic system shall embed many inspections; the most generic, yet important ones, are:

1. HATS (highly accelerated thermal shock): This test (IPC-9151D standard-based) evaluates the PCB's survivability after reflow simulation while inflicting extreme heat/cold conditions on its stackup. The main initiative is to determine how well it will operate under field conditions in which it will be deployed. The via's copper plating goes under dissemination and shrinking caused by heating or cooling of the PCB, respectively. This might result in cracks along the via or separation between its layers (delamination). Refraining from conducting this test in the early stages of the PCB design might also reflect upon the PCBA stage when you fail to trace faulty components and replace them (rework). That, in turn, might diminish the level of survivability of the whole system and damage the customer's prestige in the process. PCB Technologies has its in-house HATS lab, making the easy implementation of this test an ordinary course of action. The fabricator must have relevant experience with various raw materials and their dedicated application use to choose the most reliable ones.
2. Holes reliability test: Maintaining design crossing points between PCB and PCBA processes result in tight mechanical interconnection to the PCB PTH, thus providing a proven, reliable interface. For example, a PCB comprising of press-fit components will benefit from defining a hole's diameter and drilling, as well as tests to be conducted during copper plating of the hole (maximum and minimum copper volumes). That way, once the components are pressed in the holes the space allocated for them will match the initial plan.
3. In-circuit test (ICT): based on a separate bed of nails for every PCB, checking the electrical conductivity quality between each board's components. This offers:
   a. High testability coverage, up to 100% of boards

b. Indicates failures with all additional information
c. Highly efficient for volume production
d. Easy to implement ECOs

4. BSCAN (boundary scan) test: checking the components' boundaries and the quality of the interconnections between them, according to JTAG (Joint Test Agreement Group) standards. These standards are based on unique protocols checking the components' surroundings and the transmission of their signals to verify the reliance of their memory, propriety of their soldering to the boards, etc. This is all aimed at allowing the complex test conducted in the shortest time possible and the definite determination of the components' reliability and bill of materials (BOM) compliance. The test also offers:
a. Connection Scan TAP with the board via the bed of nails or via connectors
b. Superb tool for designers for prototypes and production
c. Easy tools to use
d. Digital coverage

5. Flying probe test: Robotic test points inspection utilizing a reduced number of nails. Its main advantage revolves around its cost-efficiency when the customer only requires a small number of PCBs and wishes to avoid the costs involved in creating a one-of-a-kind prototype. This method can allow you to reduce the number of test points. Its disadvantage derives from the relatively long time it takes to conduct the test due to its small number of nails and the reduced test options it offers. Additional advantages:
a. No need for jigs
b. Fast program development
c. Good for prototype boards
d. Fast testability and coverage reports

6. Cable testing: Designed to test the propriety of the resistance and isolation

between the PCB's connectors and electrical wires. The test is conducted in a high-voltage environment (100-200 V or even higher. In comparison, ICT tests can endure only 0.2 10 V). The test is of high importance, especially for the aerospace industry application.

7. Automatic optical inspection (AOI): Utilizing a camera to conduct an automated visual test of the board's components. This test requires the use of consistent and unified marking of the PCB.

8. Add solder paste inspection (SPI)

9. Automatic 3D X-ray test: Looking for soldering errors that are not traceable through the use of AOI. The test automatically covers every soldering point of any individual tin ball in a short time while scanning soldering quality along with the stackup. Common errors usually occur when using too much tin or not using enough of it. This can result in interconnections between components that were not meant to be connected, and in the long run cause an electrical short circuit.

10. Functional testing: Testing the various functions the final system is meant to perform. The test offers:
a. Real-time test
b. Full path and cluster test
c. Specific function test

d. Electrical test
e. Cycle testing that simulates the environment the system will operate in (thermal, electrical, etc.)

Reliance solely on functional testing might be due to reluctance on the customer's part to let the fabricator in on every aspect of the final system or due to security issues. In any case, it is highly recommended to conduct the most appropriate testing mix, even before the functional one, even though the process might take a bit longer. In the long run, it saves time since one can detect failures at a much earlier stage and can attend to them instantly. Even if these tests might result in design adjustments, they increase the process reliability and, hence, the final system.

Optimal, end-to-end test and inspection process saves the execution of unnecessary tests. For example, many of our production reports prove that conducting ICT before functional tests improved production failure rate—from 5–7% scrap to 0.1%.

Avoiding ICT might increase the risk of the entire production process and result in a high level of failures, which the functional test failed to trace. If, for instance, during the functional test, we find that one of the system's functions is not active, we won't be able to determine which capacitor or resistor has caused the problem without dismantling the entire system to check each one of its PCBs. Managing an effective test process prevents the need to replace components, alter design after production has begun, and invest large amounts of time and money, not to mention avoiding the embarrassment of delivering a malfunctioning device to the end-user. Using the right test mixture in the early stages of the design is vital for delivering quality systems. Here are three examples of problem-solving in the works:

In advance, you can make sure that test points are located in the right spot on the PCB surface and are adequately spaced from each other to allow for their effective conduct. Allocating the suitable space for the test points (not too close to one another or too far away) on the PCB contributes to the successful production process.

There are times when a component under inspection interrupts the testing of the one adjacent to it due to the release of electronic signals. Planning for additional test points in the early stages of the design enables setting the interrupting component as "disabled" during testing and activating it again once testing is concluded.

A PCB with a combination of misplaced resistors, which has gone through functional testing only (without adding an ICT to the test mixture), might result in no conclusion since the resistor's small size doesn't enable its marking or traceability. DESIGN007

Ofer Manovitz with vice president of quality assurance and safety at PCB Technologies.

Isaak Golod is manager of the ICT Test Development Group at PCB Technologies.

The Colorful World of PCB Design

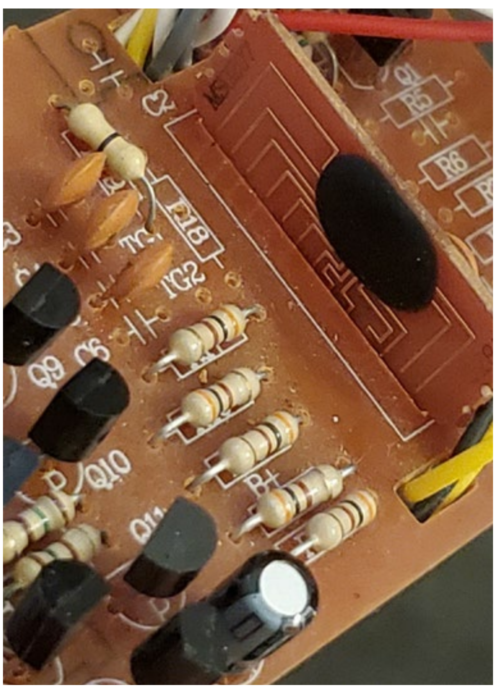
Feature Article by Cherie Litson
LITSON1 CONSULTING

Many people say that creating a PCB design is a mixture of both science and art. The science we understand; the art is not as obvious.

Let's break it down a little. What is a PCB design, generally? Most of the time I just tell people, "It's that green, blue, or tan board in your electronic product with all the parts on it." To be a little more specific, it is a product of multiple pieces of science—mechanical, electrical, physics, and chemistry—combined in such a way as to support electronic components using conductive pads in order to create a device that will perform one or more electrical functions. There's a lot of science in that mouthful.

But where's the art? Well, what is art? One definition reads, "The expression or application of human creative skill and imagination producing works to be appreciated for their beauty or emotional power." I don't know about you, but I get very emotional when a product works as planned, or when it doesn't. A product that doesn't perform as advertised becomes junk, not beautiful, even if the outside is pretty.

Art is also a form of communication. It's the combination of materials, techniques, and images that communicate the intentions of the design. With PCBs this becomes very important on many levels. From the creation of a schematic to the placement of the components and traces, we are communicating with all the disciplines of science and



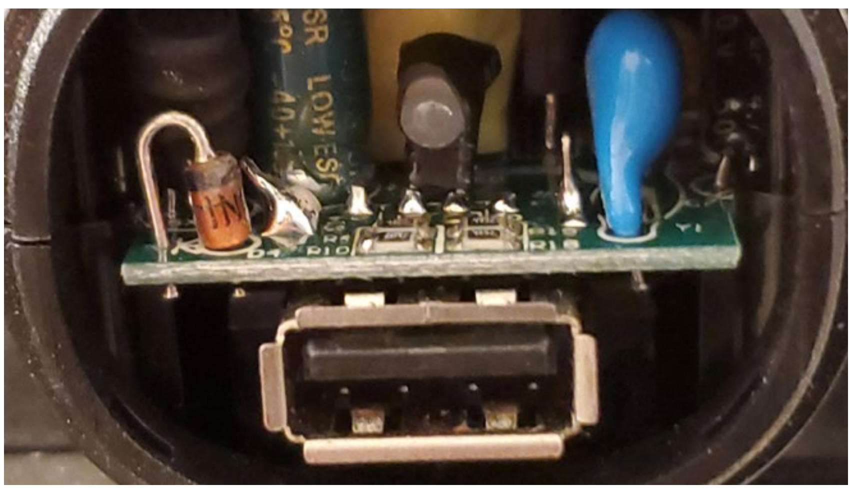


Figure 1: Many designers enjoy disassembling electronic devices. Here are a couple of examples of things I've taken apart.

manufacturing to end up with the envisioned product.

This always takes a great deal of creativity and imagination—creating balance when we have conflicting demands. Making sure that our design can be properly manufactured. Escher designs are great in a 2D plane, but difficult to produce in 3D. And that's one requirement of the "art" of PCB design: It has to perform as advertised. Because others need to interface with our designs, the designer has to be able to visualize the finished 3D form.

As an example, take an eight-layer board. It should be pretty straightforward, but it's not. When you mix multiple power rails, RF, and digital signals on one board, get ready to be very creative. I can't tell you what will work all of the time, but I can give guidelines to get you going.

The Art of Applying the Science

First, gather the facts. These become your building blocks. This includes component configurations, power rails, impedance requirements, current requirements, space available, keep-outs and keep-ins, deadlines, cost constraints, etc. Attend those planning meetings. That's the best place to pick up on the intent of the product design.

Then, learn from those who have tackled the issues before. This helps give you some guidelines. Read articles from *Design007 Magazine,* go to conferences, take a course or two online, attend PCEA Designer Council meetings, talk with others, etc. Talk with fabricators and assemblers so that you're aware of the terminology they use and issues they have to resolve with other designs.

Also, use the tools. There are many design tools out there; some are free. There are also many design tools built into your ECAD software. Learn how to take advantage of these. Other tools may not be as obvious, or they may be a little too expensive. Find others who have them and use their knowledge. Collaborate a little.

Now the real processing begins. Put those blocks together to fit in the space allowed. This is where you have to apply your visualization skills, imagination, creativity, and patience. Yes, this is an art in itself. It could take multiple arrangements before you find the one that will work for all the criteria you've gathered. Don't worry about making mistakes. Every artist makes mistakes on the way to their masterpiece. Each design you complete gets you closer to understanding what works and what doesn't work.

Being able to recognize that what you're doing isn't working is just as much of the process as finding out what does work. I have to remind myself of this all the time. Sometimes I find myself wanting the design to be perfect before I start. So many iterations go through my head before I start putting parts down, probably a holdover from my tape and decal days back when it was difficult to make changes. Once I get going, then things really start to move. I love the ECAD tools for that.

As with all art, no two trees look exactly the same, and no two designs will be exactly the same. So even when we learn how to do a thing, we have keep relearning how to do it slightly different each time. Don't get stuck in a rut. Keep learning new technologies and methods.

Lastly, there is one quality that 99% of all the designers I've met have in common with every artist I've met—passion for their work. Like creating anything, designing a PCB is a process of taking materials, manipulating them, and making a product. It is collaborative, uses all our senses and strategies, and ultimately leaves us knowing we have accomplished something that others can value. **DESIGN007**

Cherie Litson, MIT CID/CID+, is the principal of Litson1 Consulting. She is an instructor with EPTAC and Everett Community College.

PCB007 Highlights

Atotech Reports Q3 2021 Results, Narrows 2021 Full-Year Guidance Range ▸

Atotech, a leading specialty chemicals technology company and market leader in advanced electroplating solutions, reported financial results for the third quarter of 2021. The company maintained its revenue guidance and narrowed the Adjusted EBITDA guidance range for the full year 2021.

DuPont Announces Acquisition of Rogers Corporation ▸

Rogers Corporation announced that it has entered into a definitive merger agreement to be acquired by DuPont in an all-cash transaction that values Rogers at approximately $5.2 billion.

Trouble in Your Tank: Via Filling—Continued ▸

In a previous column, the author presented several options with which to accomplish blind and through-hole via filling. In this edition of "Trouble in Your Tank," filling blind and through holes with polymeric pastes will be presented.

Two-thirds of Engineers Entered the Career to Change People's Lives, According to New KLA Research ▸

KLA Corporation announced the KLA Engineering Inspiration Report, a new, multi-regional research that shows engineers are extremely passionate about possibilities that the field provides. The inaugural report notes roughly two-thirds (65%) of engineers and students indicate they entered the field to create something that might change people's lives.

Insulectro Advances Its Digital Transformation With All-New Website ▸

Insulectro, the largest distributor of materials for use in the manufacturing of printed circuit boards and printed electronics, has launched a new, "from the ground up" website.

KLA Opens AI-Advanced Computing Lab at Indian Institute of Technology ▸

KLA Corporation announced the opening of two important facilities in Chennai, India, supporting the company's investment in innovative research and talent development. KLA's Artificial Intelligence-Advanced Computing Lab (AI-ACL)—operated in partnership with the Indian Institute of Technology (IIT) Madras—will serve as a center of excellence for AI-focused research and development. In addition, a new office space in Chennai expands KLA's existing business operations as the company continues to grow and hire from the region.

Amphenol Reports Record Q3 2021 Results, Announces Dividend Increase ▸

Amphenol Corporation reported third-quarter 2021 results. Record sales of $2.818 billion, up 21% in U.S. dollars and 13% organically compared to the third quarter 2020.

The Big Picture: Cybersecurity and Hardware Security ▸

In his recent column, Mehul Davé says, "Wherever I go, I am pleasantly reminded of the role our industry plays in our everyday lives. From the sight of people texting and calling loved ones on their phones, to children laughing and playing with their high-tech toys.


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
DEPARTMENT OF DEFENSE
UNITED STATES OF AMERICA
ITAR
Registered International Traffic in Arms Regulations

The Art of Using Symmetry—and Asymmetry—in PCB Design

Feature Article by Chris Young
YOUNG ENGINEERING SERVICES LLC

An empty board outline is a PCB designer's empty canvas. Components are the designer's paint palette, and the traces are the brush strokes used to blend and mesh the components together on the canvas. The subject matter is defined by the schematic entry and the tone is often set according to the purpose of the design. The subject matter's form emerges during placement and takes shape when routed. The aesthetic nature of a PCB or PCBA is typically judged by the designer's use of symmetry, focal points, and centers of interest.

The enjoyment experienced by observing a bee (a bilaterally symmetric insect) symbiotically interact with a flower (a radially symmetric plant) is derived from the realization of two well-proportioned beings striking a mutually equitable existence, a classic win-win scenario. I surmise that our use of symmetry in our own creations is our sincerest form of flattery to these well-balanced relationships. Hence, we have embedded symmetry in nearly all aspects of our lives—from our homes, roads, and bridges, right down to the printed circuit board designs present in our modern-day electronics.

We are hard wired to identify symmetry, we tend to find it appealing, and the subject of PCB design is no exception.[1] Symmetry in PCB design is aesthetically pleasing to look at, and the physical balance of components, traces, and layers convey deeper meanings to the observer. A simple example of this can be seen in Figure 1.

Further observation will reveal that this board design is the physical representation of two identical circuits running vertically and

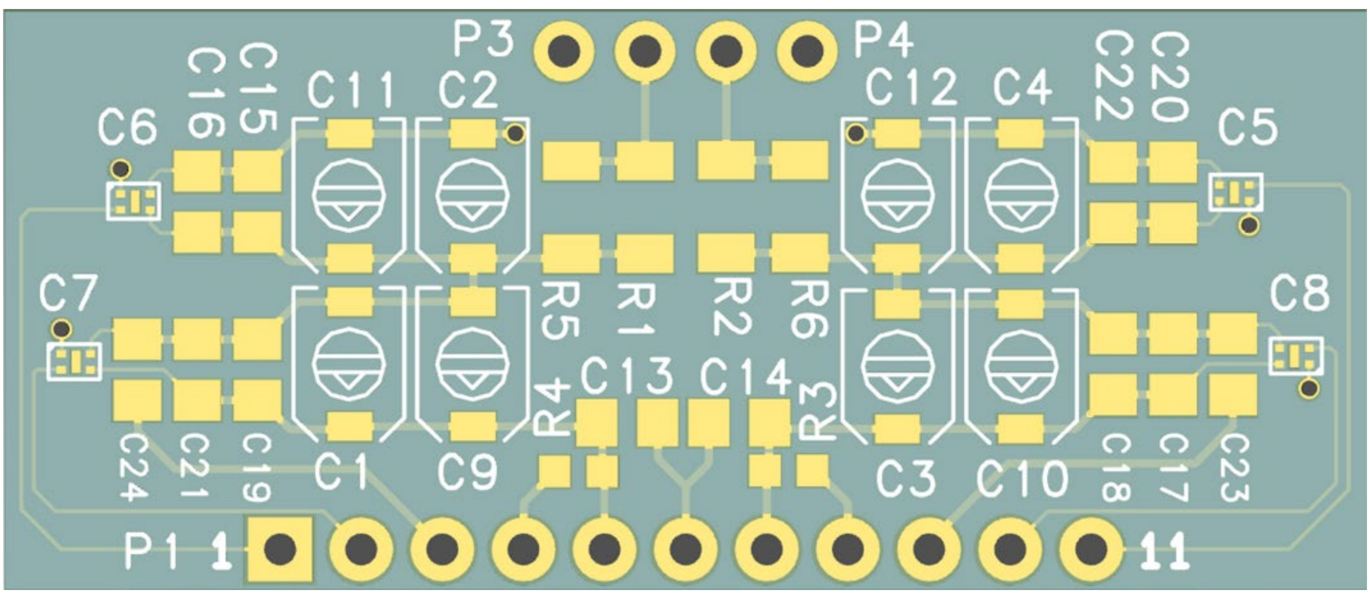


Figure 1: The mirroring across the vertical axis and proportional distribution across the horizontal axis convey a sense of organization and careful consideration. This is a board that has been architected to fit a purpose.

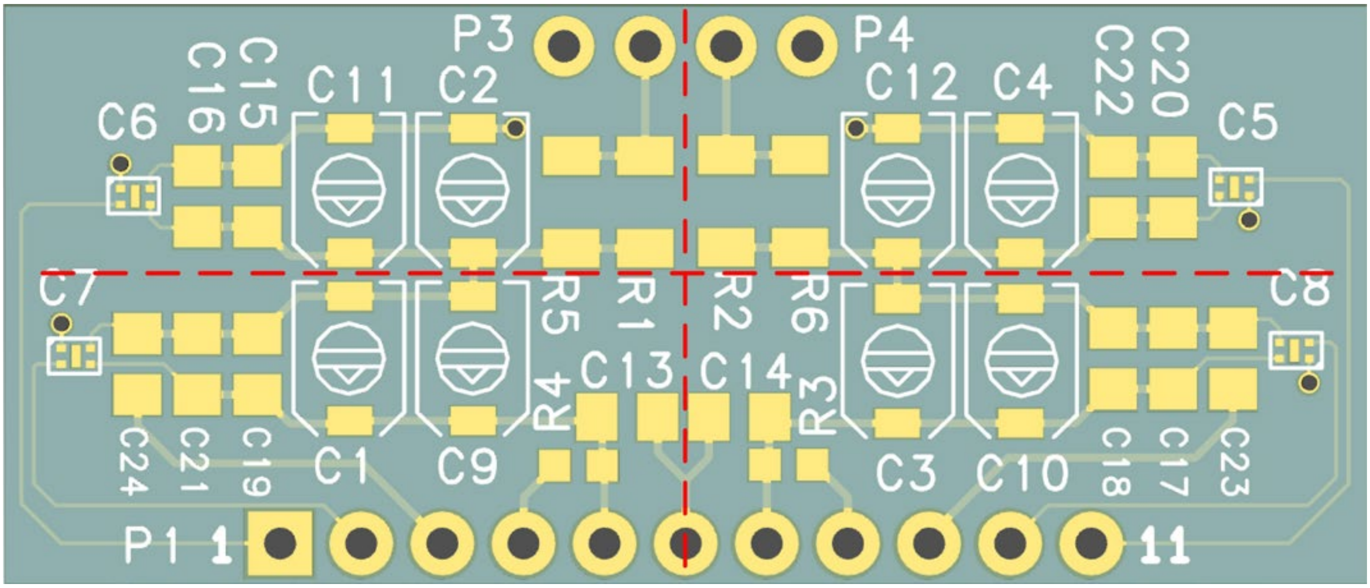


Figure 2: PCB designers use the artistic principals of symmetry to transform PCBs into functional works of art.

each circuit is composed of two sub-sections distinctly spaced apart horizontally (Figure 2).

These PCBs and circuit boards that exhibit symmetry are typically easier to troubleshoot and repair because defects that disrupt the symmetrical nature of the design are easy to identify.

In Figure 3, we see an example of symmetrical component population, and after the symmetry is broken due to a missing component (R3).

The use of symmetry attracts the attention of the observer and repeated patterns are more easily recognized.

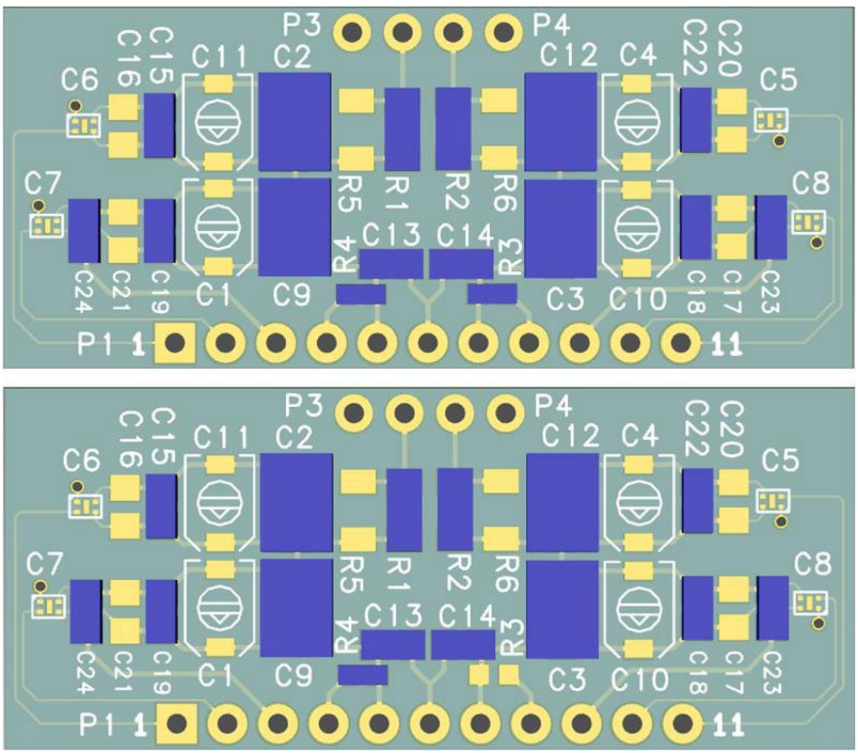


Figure 3: Pattern replication is often used hand in hand with symmetry as reinforcement of functional groupings of circuits in a design.

I designed the following voltage and current MUX board (Figure 4) for a customer. The connectors that provide the interface to the MUX (in/out, voltage/current) are level with each other and spaced relatively evenly across from each other on the board. The connector responsible for the support functions (power, control, monitoring) is placed asymmetrically to one side of the board. As much as we tend to like symmetry and patterns, we tend to dislike asymmetry just as much.

In Figure 4, off to either side are distinct grouping of circuits used which provide support functions to the main functions of the design (power, control,

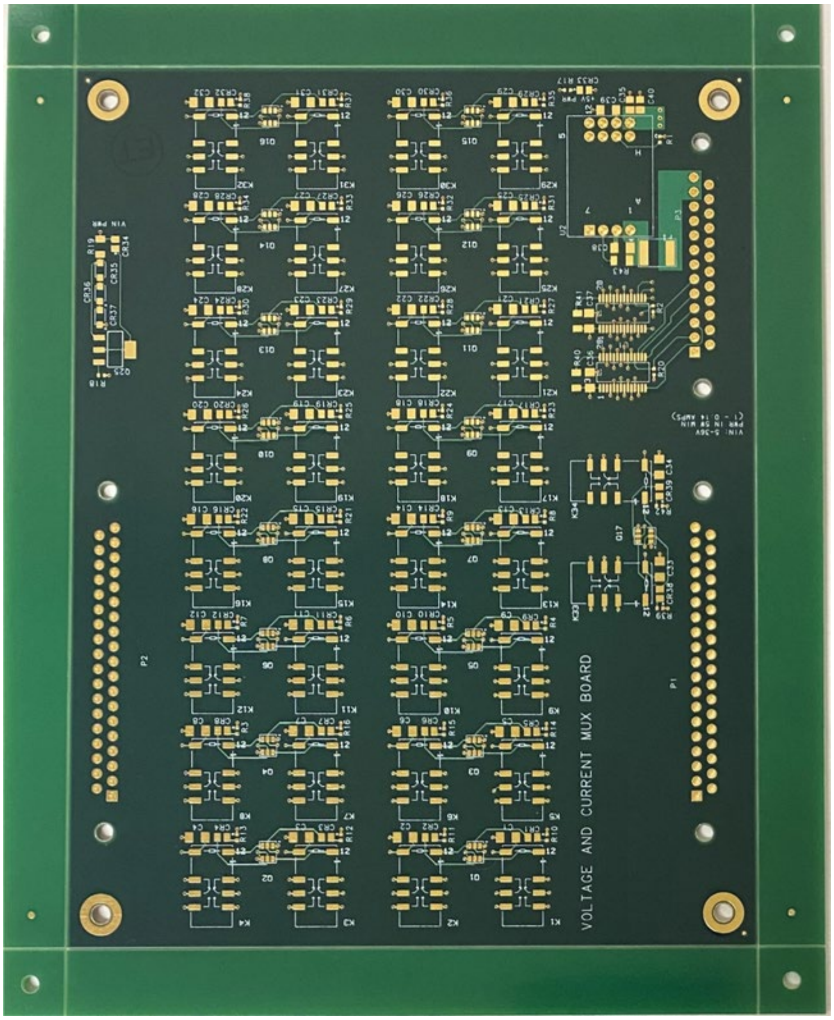


Figure 4: The primary focus of the board (voltage and current MUX) is the two vertically stacked patterns of relays spaced horizontally apart from each other on an offset vertical axis.

monitor, indicators). The purpose of using both symmetry and asymmetry in this situation is to visually draw the attention of the user to the primary aspects of the board design and keep attention away from the secondary support items. As the user spends more time with the board and is satiated with the primary focus of the board, the secondary items come into play. These distinct groupings of circuits off to the sides are actually sub-dominant focal points and centers of interest, exposing yet another set of artistic principles employed by PCB designers.

The use of focal points and points of interest are commonly used to organize PCB designs into coherent collections of functionalities. Focal points are areas that attract the eye and centers of interest and are meant to engage the mind.[2] To avoid conflicts between our visual and mental attention, centers of interest should also be focal points, however, focal points need not be centers of attention.

An example of an area that may only need to be a focus point is the part number etched or silkscreened onto the PCB. Focal points on a PCB design are commonly implemented by lumping components together in functional groups and using solder mask or silkscreen to create a color contrast. Centers of interest are about how you lump components together and use the solder mask/silkscreen to convey meaning or spark curiosity. It is important to understand that many PCBs are visually complex in nature and use of focal points should be approached in a hierarchical manner to avoid making the PCB/PCBA visually confusing and hard to understand. These hierarchies are referred to as dominant (primary), sub-dominant (secondary), and subordinate (tertiary) focal points.[3] There should be only one dominant focal point and it is the main focus of the design. Subordinate focal points should be used for support circuitry. Subordinate focal points should be used for items such as etched/silkscreened part numbers, quality assurance markings, or circuit arraignments focused on internal support/debugging. Figure 5 shows a complex board design utilizing hierarchical focal points.

The most important physical and electrical connections take place in this 7.5” x 5” area of the board and it needs to reliably interface with nearly 400 pogo pins applying an aggregate force of just under 200 pounds.

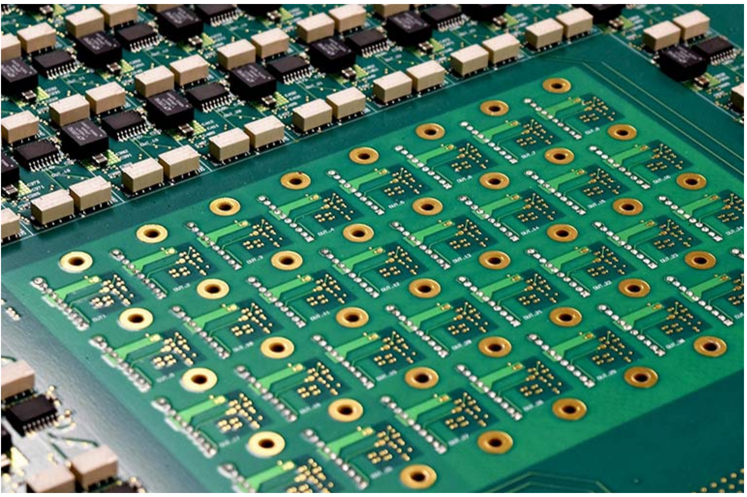


Figure 5: The main focal point and center of interest of the board design is a symmetrically-arranged array of 32 identical test point arrays vertically contained by five rows of mounting holes (non-plated with gold-plated pads top and bottom) and further emphasized by the contrast in the solder mask.

At the top level of the main center of interest, the symmetrical arrangement of 40 mounting holes is meant to convey that this area needs a significant amount of mechanical support. As we focus further inward, the individual test point arrays also use internal sub-dominant focal points meant to emphasize the criticality of electrical connections.

In Figure 6, the dark solder mask area (traced in yellow) contains the electrical support connections to the test site and are linked to the sub-dominant focal points located outside the dominant focal point. These external sub-dominant focal points contain the interface circuitry (power, communications, programming) for each test site and contain tertiary focal points (traced in blue) meant for system integration and debug.

Fundamental artistic principles of symmetry, focal points, and centers of interest are used to make PCB designs functional pieces of art in our modern electronic age. The symmetry draws us in, the focal points show us where to look, and the centers of interest engage our curiosity, prompting us to think more deeply about the subject matter we are observing. This is why I refer to Gerber files as artwork. DESIGN007

References

1. Human Preferences for Symmetry: Subjective Experience, Cognitive Conflict and Cortical Brain Activity, U.S. National Library of Medicine, National Institutes of Health.

2. "Focal Point vs. Center of Interest," Artists Network.

3. "Design Principles: Dominance, Focal Points, and Hierarchy," Smashing Magazine, Feb. 27, 2015.

Chris Young is owner/lead engineer at Young Engineering Services LLC, and an I-Connect007 columnist. To contact Young or read past columns, click here.

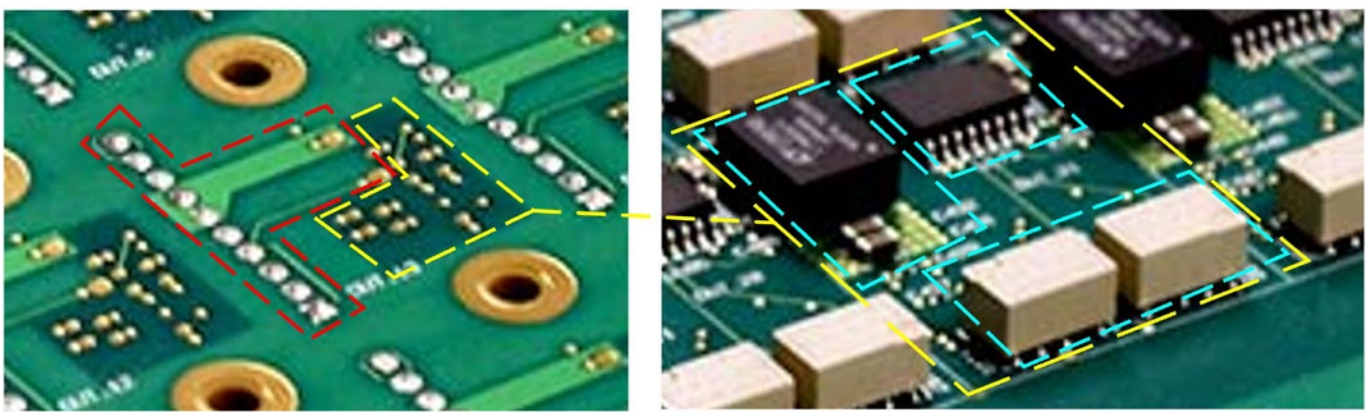


Figure 6: The silver-colored through-hole connector solder points and lighter solder mask area (traced in red) is the precision current measurement interface.

Best Practices for Solder Mask Application

Connect the Dots

by Matt Stevenson, SUNSTONE CIRCUITS

Today's PCBs increasingly must operate in challenging conditions. Whether it's an iPad hot to the touch after several hours of gaming or a drone slicing through smoke and debris to monitor a wildfire, boards need protection from the elements.

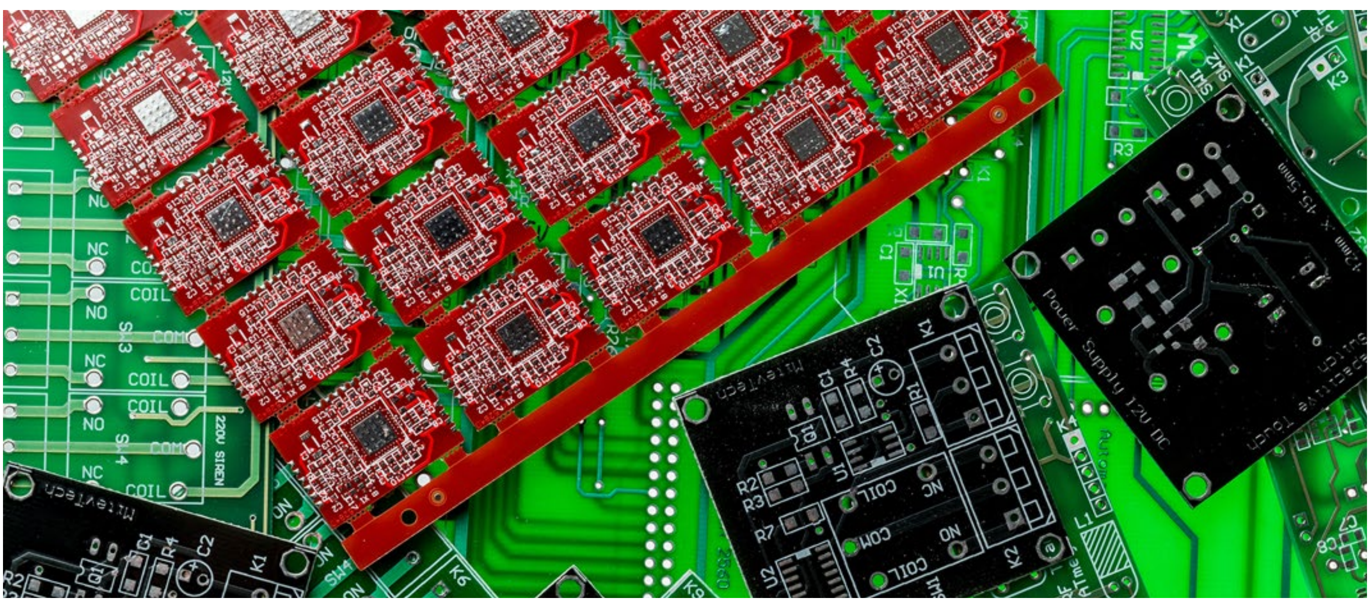
That's where solder mask comes in. Solder mask isn't just the stuff that makes PCBs green and protects the copper from the elements; it is an increasingly vital part of boards that have to do more in less hospitable environments.

Solder mask coats your whole board (apart from the solder pads) so the PCB doesn't react with the atmosphere and lose chemical properties through oxidation. It also prevents contamination from dust and debris that may settle on the board and create shorts. Solder mask prevents bridging between features during wave reflow assembly, limits external conductive influences, and helps ward off voltage spikes.

Don't Skimp on Solder Mask

Some board fabrication houses offer reduced prices if you order your boards without solder mask and that can work for some designs, but not always. If you choose to use a stencil to apply solder paste and run the board through a reflow oven, the solder will spread back on the exposed copper traces. This may leave your parts without enough solder on the pins to create a reliable connection or, worse yet, bridge adjacent conductors creating shorts. Solder mask will increase reliability and reduce cost in the long run.

All in all, solder mask is a critical element when it comes to sustained performance. How it gets on the PCB matters, making it



important to remember best practices for its application.

Breaking Down the Solder Mask Process

The solder mask process is much like the imaging process. First, a photoimageable polymer is applied to the manufacturing panel. There are several types of solder mask polymers, including liquid epoxy and dry film, but the most common in use today is liquid photoimageable solder mask (LPI). Liquid polymer can be applied through several different techniques including spraying, flood coating, and screening. Direct printing is another option that is gaining adoption in the industry, but it follows a different process than the other three methods and should not be considered as part of the process outlined here.

Once the panel is coated in the LPI on both sides, it is heated to remove some of the solvents, making the deposit illiquid and tacky to the touch, but not cured either. With the semi-hardened liquid polymer coating, the solderable surfaces can now be imaged onto the panel. The majority of the LPI is exposed to a high intensity light source, cross-linking the polymer, and chemically changing that portion of the solder mask.

During this part of the process, solderable surfaces such as pads, through-holes, and contact points are masked from the light source—leaving the LPI in its tacky and liquid form. The exposed panels are then "developed" through a chemical process that removes the unexposed LPI from the panels and leaves the parts that were exposed intact, creating all the pads and fiducials.

For most PCB designs there is much more solder mask left remaining on the board than the amount removed. This gives the PCB its familiar green coloring. Creative choice of solder mask color can also add some personality to your boards. Red, blue, yellow, and even pink are among the options for those who want a PCB to stand apart from the crowd. After developing the panels, they are again heated to drive off the remaining solvents from the LPI. This cures the solder mask coating into a hardened epoxy that protects the board.

Application of the legend or nomenclature to the panels before final thermal curing will enhance the bonding between the two epoxies. Doing so at this point in the process creates a very strong bond between the solder mask and the silkscreened nomenclature. The chemical bond created at this stage makes it much less likely that the nomenclature will wear off or chip away from the solder mask than if it was applied later in the process after the final thermal curing.

The silkscreen application process uses a printer to apply the silkscreen ink to a manufacturing panel in much the same way that an inkjet printer applies ink to paper. It comes directly from the digital data and is aligned directly to the panel image, improving registration and quality while lessening the process's impact on the waste treatment process.

More precise application of nomenclature can also improve component placement accuracy during automated assembly. Without nomenclature, machine assembly won't be as reliably accurate. Unfortunately, Gerber files do not tell the assembly machines exactly where each part is supposed to go and what angle and orientation is required. Footprint errors are common, as are components with ambiguous marking.

The addition of solder mask and legend to your PCB design will go a long way toward protecting your design from external factors and give your assembly team the best foundation to be successful: a clear roadmap of the parts and enough barriers to prevent solder shorting between components. DESIGN007

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

High-Performance 3D Printed Circuitry

Lightning Speed Laminates

by John Coonrod, ROGERS CORPORATION

3D printing has been around for many years and, as time has progressed, so has this technology. However, one aspect of 3D printing seems to have not been developed as much until recently. That aspect is related to the materials which are used for 3D printing and, specifically, materials which are compatible with RF or high-speed digital (HSD) applications. There are many materials that can be used for 3D printing, but until recently, none of these had properties which were conducive to printing circuits with good RF or HSD performance. Rogers recently released the first 3D printable material for digital light processing (DLP) and stereolithography (SLA) 3D printing, which has good properties for RF and HSD circuit considerations.

There are several different types of 3D printing technology and depending on the technology, there are different types of materials which can be used for printing. We teamed up with Fortify, a leading DLP printing platform for filled and reinforced photopolymer materials, to enable the manufacture of the new, high-performance material. For photopolymer 3D printing processes like DLP and SLA printing, the materials that have been historically used have a high dissipation factor (Df), a dielectric constant (Dk) that is not well controlled, and some of these materials have a high moisture absorption property. Table 1 has information that can be used for comparison between the traditional 3D printed materials and these new high performance 3D printable materials.

As shown in Table 1, most of these materials have very similar Dk values when tested at 10 GHz. What is not obvious is how consistent these Dk values are for potential Dk variation within a small area or Dk variations within a batch or variations within batch-to-batch. Rogers formulated low loss, high-performance circuit materials over the past five decades and now extends this expertise for consistent RF performance in DLP and SLA 3D printing processes.

No matter how well the electrical properties are controlled, if the moisture absorption

Material Type	3D Printing Process	Dk @ 10GHz	Df @ 10GHz
Urethane Methacrylate	DLP	2.9	0.039
Cyanate Ester	DLP	3.1	0.017
Clear Acrylic	SLA	2.6	0.019
Black Acrylic	SLA	2.6	0.021
Grey Engineering Resin	SLA	2.9	0.032
High Temp Acrylic	SLA	2.6	0.020
Rogers Corp 2.8 Dk Printable Dielectric	DLP/SLA	2.8	0.0044

Table 1: This information can be used for comparison between the traditional 3D printed materials and the new, high-performance 3D printable materials from Rogers.

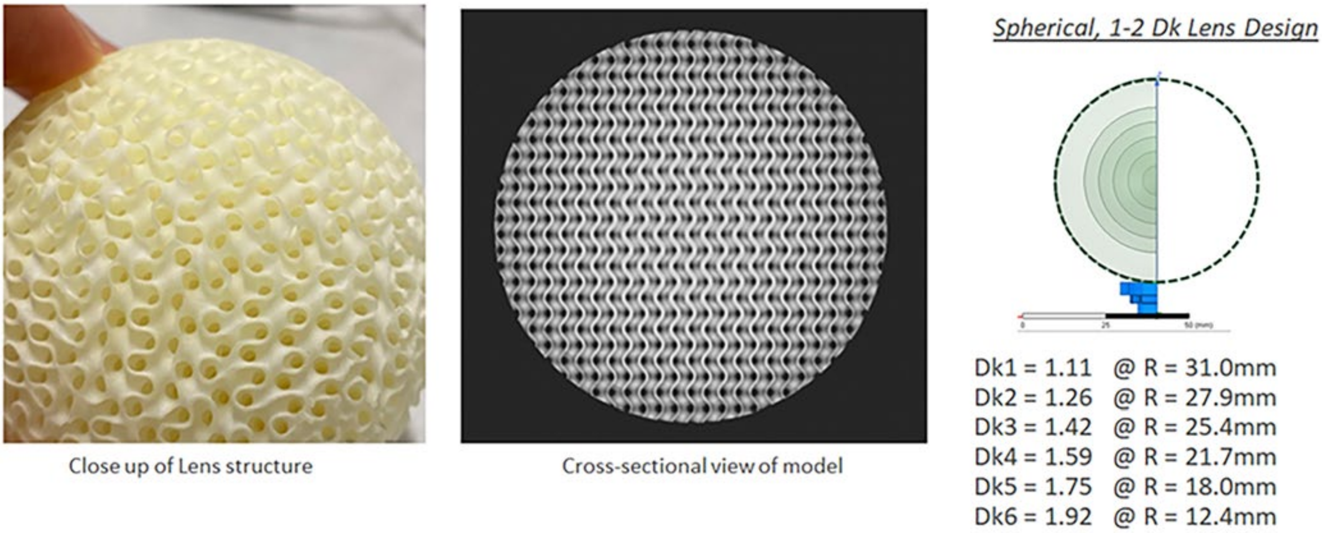


Figure 1: A Luneburg lens that was generated with the new 3D-printable RF material. The basic concept is shown in the upper right where the lens has higher density in the middle and the layers progressing out from the middle have decreasing density. The spatially variant Dk necessary for the Luneburg lens can be applied to other applications.

property is not low, the RF benefits of controlled Dk and Df can be negated. If the moisture absorption for a circuit material is high, that will allow the circuit to absorb moisture from the atmosphere as related to humidity. When there are significant humidity changes in the atmosphere, that can also cause changes in the amount of absorbed moisture in the circuit.

The absorbed moisture, water vapor, has become embedded in the circuit material and will increase the Dk and the Df of the material. The new 3D printable dielectric material has an extremely low moisture absorption rate at 0.08% by volume. Generally, good moisture absorption for a high-frequency circuit material is 0.3% by volume or less and moisture absorption below 0.1% considered excellent.

Having a 3D-printable material with good high frequency or HSD properties will allow the engineer to make and evaluate prototype structures much faster and cheaper than before. This could certainly open tremendous possibilities to evaluate structures that are 3D and were previously very difficult to make with PCB technology. Additionally, 3D printed technology with good RF properties can be used to make planar circuits which are similar to PCBs but also combine 2D circuit features with 3D areas, such as cavities, as an example.

We have investigated many different RF structures that can be achieved with using the RF printable 3D technology and one very interesting example is a Luneburg lens. The Luneburg lens is often used as an antenna lens and it uses different layers of slightly different Dk. This can be done easily with the 3D-printable RF material technology, where specific layers are printed to have a specific density lattice structure, while other layers have a higher or lower density than the previous layer. The difference in density will cause a difference in the effective Dk for that layer. An example of this technology is shown in Figure 1.

Historically, prototype circuitry that are built to prove-out concepts can be expensive for time and resources. Having the ability to 3D print an RF structure, or multiples of the same RF structure with slight differences, has the potential to significantly speed the development of RF and HSD projects. **DESIGN007**

John Coonrod is technical marketing manager at Rogers Corporation. To read past columns or contact Coonrod, click here.

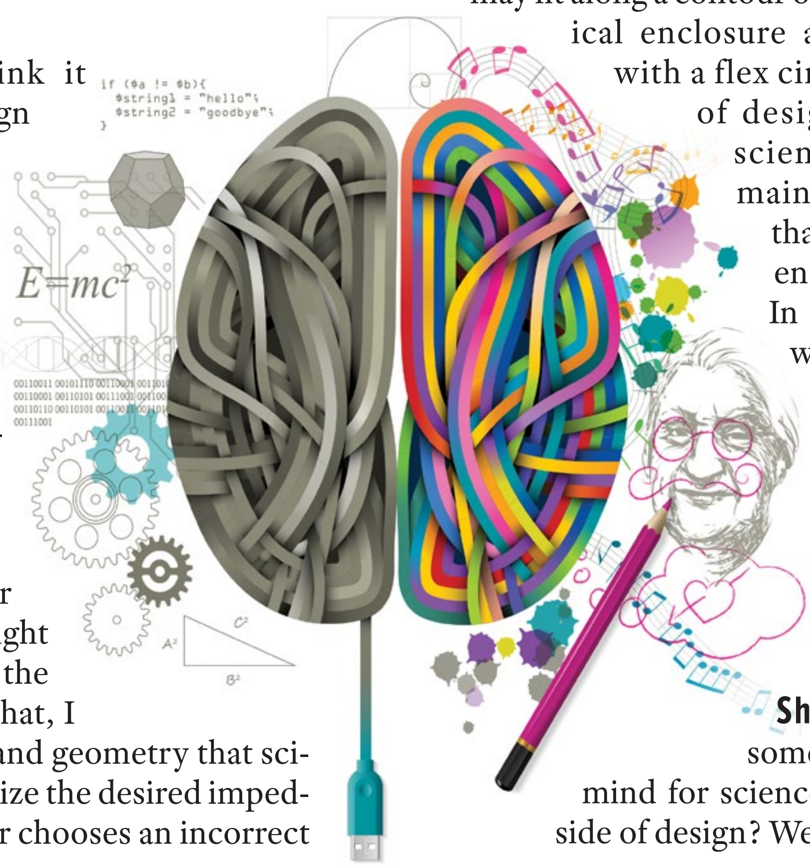
The Art and Science of DFM

Feature Interview by Andy Shaughnessy
I-CONNECT007

Our industry has been focusing on DFM for decades, but in our reader surveys, DFM issues are always among the biggest challenges cited by respondents. I recently spoke with Vince Mazur, product and persona marketing engineer at Altium, about the art and science of PCB design, and particularly design for manufacturing. Is DFM an art form in itself?

Andy Shaughnessy: We often hear that PCB design is part art, part science, a matter of right brain and left brain. Do you think it's more of one than the other, or does it depend on the design job?

Vince Mazur: I think it depends on the design job. For example, for impedance-controlled, high-speed PCB design, the priority is performance-to-spec, not the expression or application of creative skill and imagination, which is a common definition of art. The designer must choose the right materials and define the "right" stackup. By that, I mean the materials and geometry that science tells us will realize the desired impedances. If the designer chooses an incorrect stackup, it doesn't matter how pleasant the PCB looks, how optimized the placement and routing is, or how much expression or creative skill and imagination was deployed—it will not function as intended. The idea that science is the driver in this case is not to say that art is not present. Art most certainly is present in the all-important placement of components and routing of traces. Rather, I mean that the scientific aspect takes precedence. Art will always be present.



On the other hand, the design specifications may lend themselves to or even require more expression and creativity, especially when the design is closer to the user experience. Consider a display or series of visual indicators that may fit along a contour of a curved mechanical enclosure and implemented with a flex circuit. These types of designs are art first, science second. But I maintain, once again, that both art and science will be present. In this example, one would pay attention to bend radii and other mechanical and materials science to assure long term quality of the flex implementation.

Shaughnessy: Can someone with a great mind for science learn the artistic side of design? We hear non-degreed

designers say this about EEs—that they're too focused on the science and not enough on the art.

Vince Mazur

Mazur: Yes, I think this is possible provided the designer desires to learn more of the artistic side. Pablo Picasso's father was a professor at a school of fine arts. Education is widely available in art, which implies that it is something that can be taught and learned. However, I believe there is a limit to what a scientifically focused person can do in art. While anything is possible, one can pursue fluency in art, but they likely would have a difficult time becoming a Picasso or a Rodin. Just as either of these artists could likely learn more about science, they likely would not become an Einstein or a Tesla. But then there are those outliers that have extreme talent in both domains, such as Galileo. So yes, it is possible for someone with a great mind for science to learn the artistic side of design.

Yes, it is possible for someone with a great mind for science to learn the artistic side of design.

Tom Scholz of the rock band Boston comes to mind. He went to MIT and majored in mechanical engineering. Scholz, founder of Boston and inventor of the Rockman guitar amplifier, would later be profiled in the PBS web series *The Secret Life of Scientists and Engineers*.

The good news, particularly for non-artistic types like myself, is that over the years, the PCB CAD tools have improved their ability to create clean, pleasing-to-the-eye PCB designs, with identical spacing, parallel curved traces, silkscreen text formatted with publishing-grade fonts, and more. Add to this the ability to press a single key and the designer can see a 3D, realistic view of the board which provides a different platform for artistic assessment.

Good designers learn what they need to get the job done. However, the talent sets of individual designers vary. Some will have a more artistic disposition; some are more talented in design. The industry needs both skill sets.

Shaughnessy: Is DFM an art in itself? A lot of designers have a hard time communicating effectively with stakeholders up and down the stream. Can the art of DFM be taught the same way we teach the science?

Mazur: Sure, DFM can be considered art, but it is also rooted in science in that DFM is constrained by the limitations of the manufacturing process. Clearances, minimum trace widths, via types, etc., are predicated upon the capabilities of fabrication and assembly machines, such as a pick-and-place head size and spacing limitations.

I do believe that the industry can teach DFM in ways similar to science, but tools can help by providing a way to easily embed DFM-related rules and constraints directly into the PCB design environment to make it DFM-aware. Additionally, whenever one gets ready to pass a design downstream, communication becomes essential. For demanding designs, early and often collaboration with downstream stakeholders can pay dividends in assuring trouble-free manufacturing. Fortunately, improved design environments are making it easier to share designs with fabrication and assembly firms so that these processes have a higher probability of trouble-free, first pass success.

While component sourcing is not a conventional DFM aspect, today's supply chain issues are proving that component scrutiny throughout the design process, and particularly at the time of manufacturing release, is no longer a "nice-to-have," but rather a "must-have." Again, PCB design tools are available today, instrumented with supply-chain awareness throughout all design domains, which helps to assure that the availability of parts does not delay manufacturing. DFM, in my opinion, at least during these times, has a new attribute called design for availability.

Shaughnessy: Steve Jobs demanded that the inside of the device look as beautiful as the outside. Are electronics designers and engineers "closet artists" looking for a chance to express themselves?

Mazur: Some may indeed be closet artists, as each designer has their own artistic propensities. This may simply be a subconscious desire to make things that are evenly spaced, symmetrical, balanced, and pleasing to the eye, or a conscious objective of the designer. Like many things in life, it varies across individuals. I do think there is an opportunity for designers to express themselves artistically as they attempt to meet the specification objectives of designs and, in fact, many choose to do so.

I think Steve Jobs can be considered the father of modern human factor engineering. From the very beginning, his products were simply different than any other, especially compared to the more sharp-cornered and boxy computers of the early PC era. When Apple II arrived in 1977, it just looked different, and while most companies used traditional fasteners for their enclosures, Apple used Velcro in a practical manner, to hold the cover in place. Just as aficionados of Picasso, Monet, or Rembrandt can pick out each of these artists' works without looking at the signature, Apple has had its own look that is immediately differentiable to aficionados of computer hardware, and this trend continues today.

I have not studied the key players at Apple, but when it came to product development, it seems that Steve Jobs was more of the artistic type, while Steve Wozniak addressed more of the science. This was a balanced team that did some amazing things and trying to replicate these balanced perspectives on product design teams is not without merit.

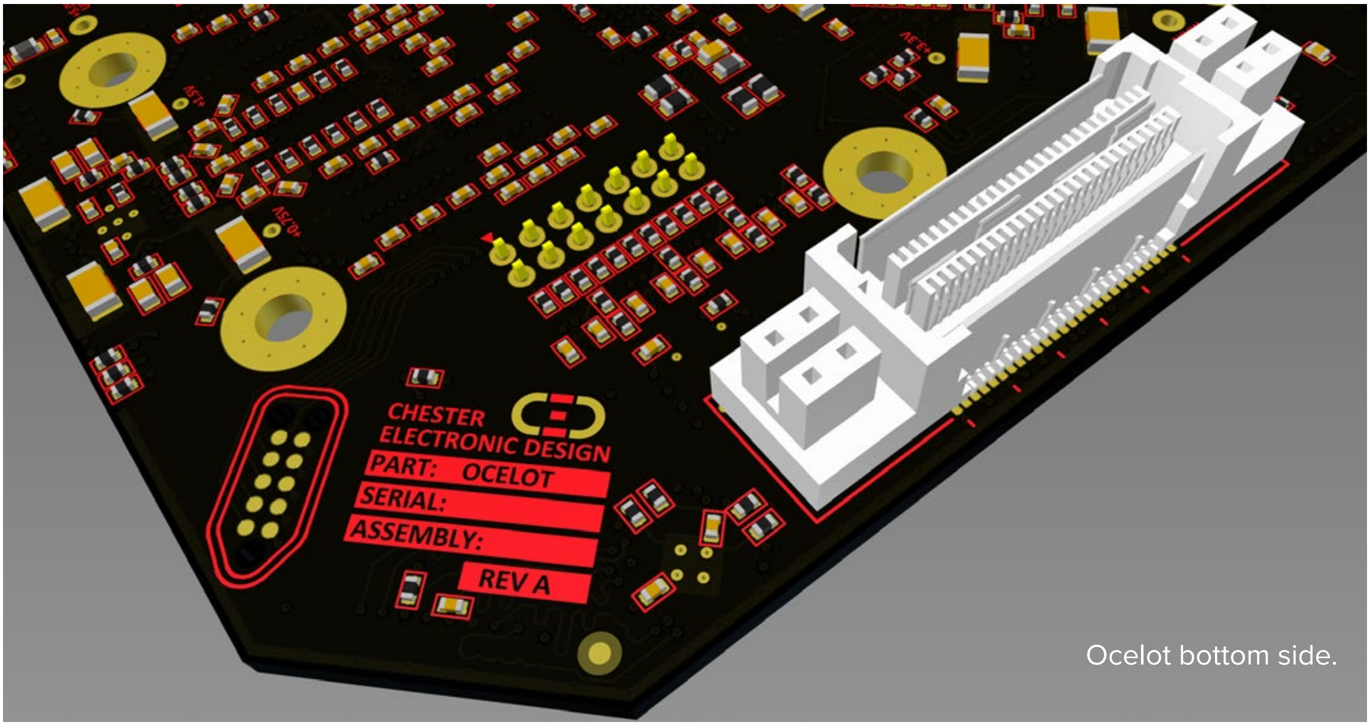
Apple has had its own look that is immediately differentiable to aficionados of computer hardware, and this trend continues today.

Shaughnessy: Is there anything else you'd like to add?

Mazur: In today's "tear-down video" world, paying attention to the artistic presentation throughout a design cannot hurt. Humans have a natural tendency to prefer things that are pleasing to the eye. It seems that all the classics—be it early radios, automobiles, and even computers—were heralded for not only how they performed and how they looked on the outside, but also how they looked under the hood. The art aspect both inside and outside can also give OEMs the opportunity to differentiate their products based on something more than specifications.

Shaughnessy: Thanks for your time, Vince.

Mazur: Thank you, Andy. DESIGN007



Ocelot bottom side.

The Art and Science of PCB Design—You Are the Best Tool

Feature Interview by Andy Shaughnessy
I-CONNECT007

For this issue on the art and science of PCB design, we spoke with Tomas Chester, a PCB design engineer and founder of Chester Electronic Design in Guelph, Ontario, Canada. I asked Tomas to share his thoughts on the art of PCB design, and how art and science must merge for a successful PCB design.

Andy Shaughnessy: We often hear that PCB design is part art and part science, right brain and left brain. Do you think it's more of one than the other, or does it depend on the design job?

Tomas Chester: I think it is always a combination of the two. Depending on the requirements of the design you are working on, it may shift to slightly favor one side or the other. Being able to stay organized, focused, and aware of your design parameters, while allowing the creative side to explore and find innovative solutions, will lead to the best design.

Shaughnessy: Can someone with a great mind for science learn the artistic side of design? That's one thing that non-degreed designers jokingly say about EEs—that they may focus too much on science and not enough on the art of the design. Does a PCB designer need to have an artistic side to apply the science to a design?

Chester: Anyone with enough determination and focus can learn and expand their abilities. A great scientific mind can grasp the art side of design, while a creative PCB designer can acquire the technical background. I would continue by saying that once you have added that skill set to your quiver, the learning and development does not stop. The more special-

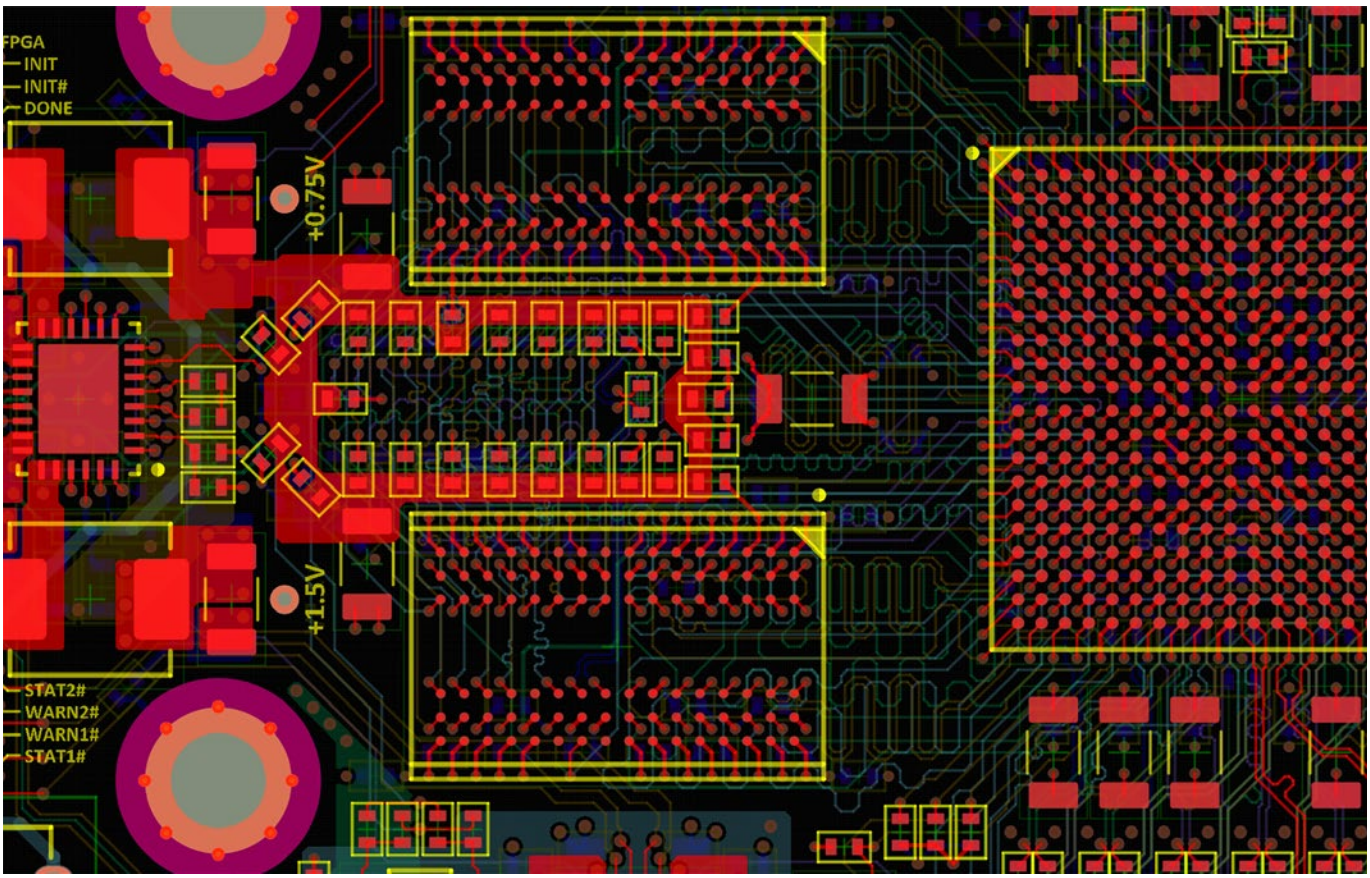
ized or focused in one area you are, the more difficult it will be to learn the opposite skill, but it is never impossible.

As someone with an analytical background, I know that I am still learning in both of these areas and I find as a design becomes increasingly complex, I have a greater appreciation for the art of the creative process and the aesthetic of the design itself. Sometimes I am inspired by something in a design; a novel or creative solution that works outside the typical configurations to solve a difficult problem. Other times that inspiration comes from facing a new technical challenge and applying a different method to solve the madness.

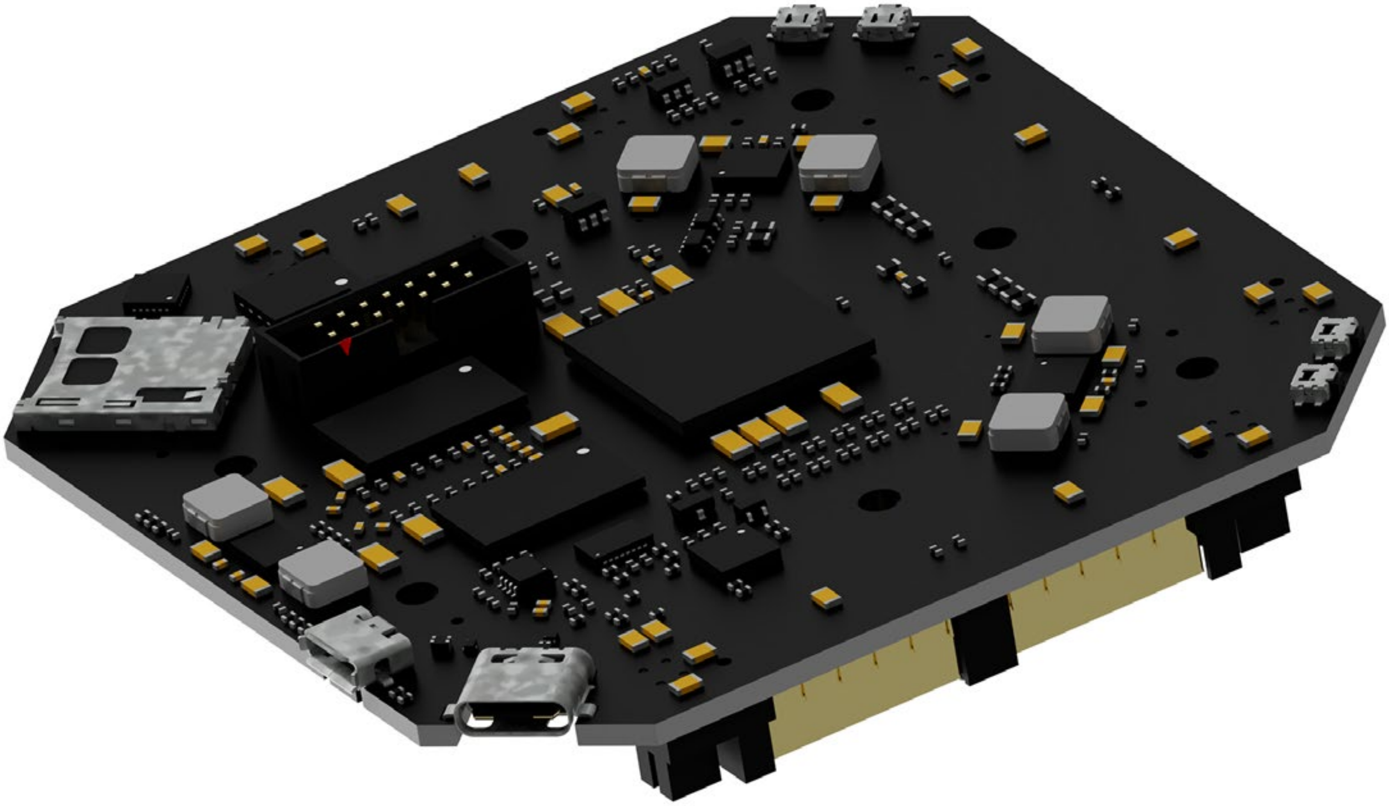
Shaughnessy: Your company's philosophy is "Design for the Future." Since this involves looking down the road at upcoming technology, how does art and/or science figure into your approach?

Chester: Technology is becoming the centerpiece of our lives more every day, from the phones in our pockets, to the smart systems in our homes, to the new electrified modes of transportation. In all of these areas we are seeing a blending of art and science—either to make those devices and/or technologies stand out and capture our attention, or to make them integrate seamlessly into our lives. My approach is focused on more of the science aspect of up-and-coming technology. Typically, future tech is announced as specification details, but is normally rather light on the art side. The other unfortunate aspect is that the imaginative design or layout is covered by enclosures, heatsinks, components, or even simply the outer design layers of the PCB.

Shaughnessy: Steve Jobs demanded that the inside of the device look as beautiful as the outside. Do you think electronics designers and



Ocelot DDR top layer.



Ocelot render top view A.

engineers are actually “closet artists,” looking for a chance to express themselves?

Chester: As a leader, Steve Jobs had many talents, but one of the most important was having focus and instilling a similar focus into others. He pushed the designers and engineers working with him toward more than the goal of completing a product. Being asked to design something beautiful gives people more ownership and investment in what they are working on. So, yes, this gives those designers and engineers a chance to express themselves, not as a “closet artist,” but as someone who can be proud of their work and as someone who wants to share what they have worked on.

Shaughnessy: Is there anything else you’d like to add?

Chester: The best design tool is you. Your creativity, your problem-solving skills, and your understanding of the circuit is what will ensure success. However, you can never remain stationary and should always be doing the following:

- **Question:** Always ask questions; this can be to clarify or expand your knowledge.
- **Contribute:** Where possible, share your knowledge and expertise with others. This helps advance the technical understanding of others, and also serves to reinforce the information in your own mind.
- **Explore:** Never stop exploring.

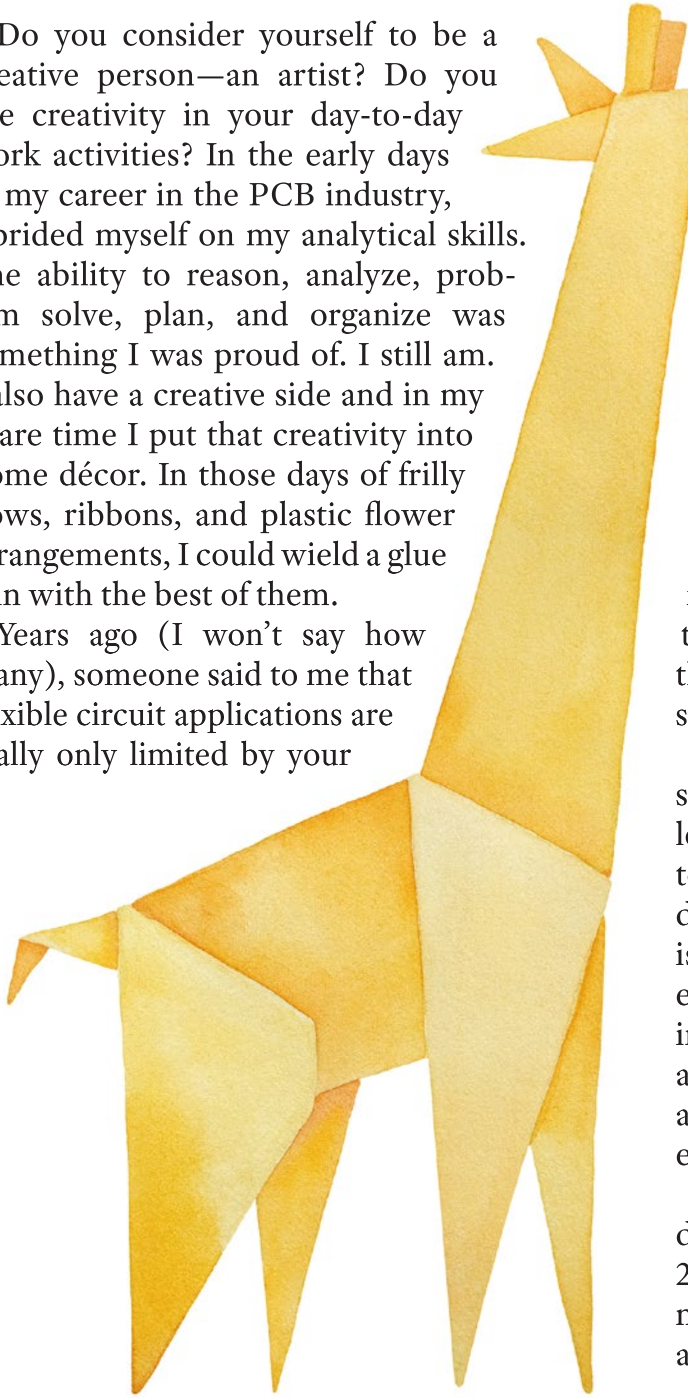
Shaughnessy: Thanks for your time, Tomas.

Chester: Thank you, Andy. DESIGN007

Creative Minds Pushing Boundaries

PCB Talk

Feature Column by Tara Dunn, AVERATEK



Do you consider yourself to be a creative person—an artist? Do you use creativity in your day-to-day work activities? In the early days of my career in the PCB industry, I prided myself on my analytical skills. The ability to reason, analyze, problem solve, plan, and organize was something I was proud of. I still am. I also have a creative side and in my spare time I put that creativity into home décor. In those days of frilly bows, ribbons, and plastic flower arrangements, I could wield a glue gun with the best of them.

Years ago (I won't say how many), someone said to me that flexible circuit applications are really only limited by your imagination. That resonated with me and is a phrase that I often repeat when talking with engineers and printed circuit board designers. The ability to use unusual shapes, bend and fold to replace wires, and wrap around fixed areas in the electronics are all significant benefits of flexible circuits. This also challenges us to think outside the box, (pun intended). Once a design is no longer limited to the traditional rigid board shapes, creative ideas are needed to take full advantage of these benefits. Side note, I wonder how many designers and fabricators have named their designs based on these unusual shapes. Personally, I remember the tire spoke boards, the giraffe, and the plus sign, amongst others.

But once this creativity is engaged, the left side of the brain must also be engaged. I have learned many tips and tricks over the years to help ensure "revision A" success with flex design. One that has always stood out to me is to start the layout of the flex first. That helps eliminate one of the inherent risks of designing a circuit that is going to bend, fold and go around corners. It is very easy to get finished and have those connections be on the opposite end of what you actually intended.

Today, there is a new tool being added to designers' toolboxes: the ability to design with 25-micron trace and space and below. While mSAP has been a solution for high volume applications, most notably the smartphone

market, it has not been something truly available for low-volume, high-mix applications. That has changed. Fabricators are installing the A-SAP™ process, which starts with a very thin electroless copper and uses semi-additive techniques to form traces much smaller than traditional subtractive-etch features. Fabricators are also looking at mSAP, which is a semi-additive process that uses thin copper foil.

These new capabilities are exciting from a design perspective—and also challenging. This is a new design space that most of us are not used to or familiar with. Unlike someone new to flexible circuit design, who can easily rely on their fabricators knowledge to help guide them through the learning curve, in this case the fabricators are also going through a learning curve.

This is a new design space that most of us are not used to or familiar with.

What a great opportunity to change the way we look at PCB design and fabrication. One of the first questions I am asked when talking about SAP (semi-additive PCB processes) is design rules. That is industry default, and it is completely understandable that a PCB designer would want to understand the limitations as they start to think through how to best apply these new capabilities to their designs.

I am advocating instead that we re-frame the typical communication cycle between PCB design and PCB fabricationl as one SAP expert has stated, let's design with manufacturing, not for manufacturing. My opinion is that we as an industry would be remiss by putting parameters around these capabilities without fully understanding how they can be applied. We are just starting to scratch the surface of reviewing this for a wide variety of applications. Some designs will benefit most from miniaturization. Some designs will benefit most from reducing higher layer count and dependence on multiple lamination cycles to accomplish tight pitch BGAs. Some designs will benefit by shrinking the routing area and increasing hole size to eliminate microvias altogether.

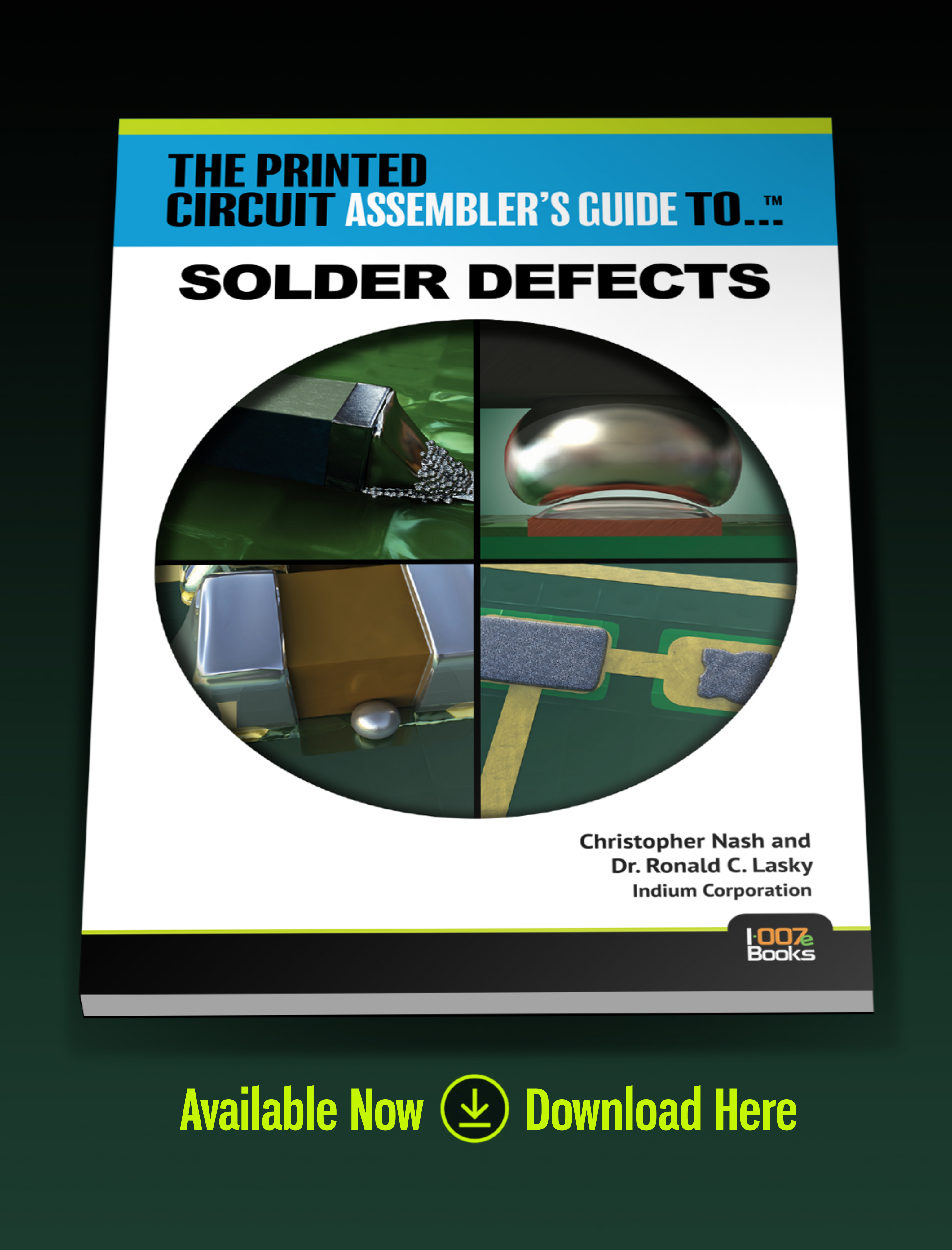
I am sure you get the point that ultra-high-density feature sizes can be used in so many different ways. This is the time for the PCB industry to challenge the creative side of PCB design to learn to identify all the different ways this technology can be used to solve today's packaging problems. At the same time, our left-brain traits must be put to full use.

Here are few thought starter questions to help kick-start things:

- How many layers could I eliminate with tighter pitch routing and what is the impact on impedance if I do that?
- How can I combine SAP layers and subtractive etch layers to the greatest benefit? Is that using layer pairs of SAP to simplify for manufacturing? Or does it benefit the overall design to increase the cost and use subtractive etch on one side of the core and SAP on the other?
- Do I use SAP layers or subtractive etch layers as outer layers? What are the benefits and drawbacks?

The list could go on and we will keep addressing these types of questions in future columns. What I want to close with today is a common message you often read in my columns. I will always recommend strong communication between design and fabrication and as early in the design cycle as possible. This is going to be even more important as we all collectively move through this exciting learning curve. **DESIGN007**

Tara Dunn is the vice president of marketing and business development for Averatek. To read past columns or contact Dunn, click here.


THE PRINTED CIRCUIT ASSEMBLER'S GUIDE TO...™
SOLDER DEFECTS
Christopher Nash and
Dr. Ronald C. Lasky
Indium Corporation
I-007e Books
Available Now
Download Here

MilAero007 Highlights

Monitoring the High Seas at High Altitude ▶

Raytheon Intelligence & Space demonstrated the SeaVue MR and DAS-4 on the SeaGuardian for the United Kingdom, The Netherlands and 12 other NATO member and partner nations as part of the Joint Warrior demonstration in the fall of 2021.

Honoring Those Who Served ▶

On this Veteran's Day holiday, the I-Connect007 staff takes a moment to honor those men and women over the years who served to protect and defend their country. Just on the I-Connect007 staff, we have loved ones who served in the military, ranging from World War I to present-day active-duty. The holiday isn't intended to be a political statement, but rather a reflection on service, duty, and personal sacrifice.

Defense Speak Interpreted: What Happened to Our Defense JEDI? ▶

When columnist Dennis Fritz last wrote about the Defense's JEDI program (not JEDI knight) back in June, he had high hopes for its success. JEDI stands for Joint Enterprise Defense Infrastructure and is the backbone cloud computer system for Defense to tie the service branches together.

Boeing Forecasts Africa's 20-year Commercial Aviation Market Opportunity ▶

Boeing forecasts that Africa's airlines will require 1,030 new airplanes by 2040 valued at $160 billion and aftermarket services such as manufacturing and repair worth $235 billion, enabling growth for air travel and economies across the continent.

The Government Circuit: How Can Government Help or Hurt You in 2022? ▶

The seasons may be changing, but IPC's commitment to advocating for the electronics manufacturing industry remains the same.

Nanoracks, Voyager Space, Lockheed Martin Teaming to Develop Commercial Space Station ▶

Nanoracks, in collaboration with Voyager Space and Lockheed Martin, has formed a team to develop the first-ever free flying commercial space station.

Blue Canyon Delivers CubeSats to NASA for Starling Technology Demonstration ▶

Small satellite manufacturer and mission services provider Blue Canyon Technologies LLC, a wholly owned subsidiary of Raytheon Technologies, delivered the first of four 6U CubeSats to NASA's Ames Research Center in California's Silicon Valley.

Army Grants to Bolster Unique, New Semiconductor Fab Facility ▶

More than $5 million in total funding from the Army Research Office and the Army Research Laboratory will go toward a unique silicon carbide semiconductor fabrication facility at the University of Arkansas. The grants—$4.5 million from the Army Research Office and $900,000 from the Army Research Laboratory—come on the heels of an $18 million grant from the National Science Foundation to fund construction and operation of the unique national fabrication facility.

Live or via Zoom, Knowledge Is All Around Us

The Digital Layout

by Mike Buetow, PCEA

As you may know, the Printed Circuit Engineering Association is acquiring the assets of UP Media Group. Once that happens, I'll become president of PCEA. So, for those who know me as the editor of a different industry publication, I can assure you are in the right place.

For the past year-plus, this column has been written by Kelly Dack, our erstwhile communications director, and Stephen Chavez, our chairman. In the next couple of months, the PCEA is transitioning from an all-volunteer organization to one with a fulltime staff, which will allow the board of directors to focus on higher-level strategy. Steph's role, then, will no longer be tied to monthly communications but rather leading the board in charting the goals and direction of the association. Filling the gaps is where I come in.

Think of the Printed Circuit Engineering Association as a "ground up" organization. We aim to advance the careers of professional engineers. We do this primarily through a peer-to-peer network where we offer training, technical knowledge, and career advice across the printed circuit engineering spectrum: design, fabrication, assembly, test.

We have nearly 20 local chapters around the world[1] and we are affiliated with multiple other trade associations, including SMTA and the European Institute of Printed Circuits (EIPC).

Those chapters engage in periodic meetings—in person and online—where they supplement their on-the-job training with tailored presentations from a host of industry experts. As a benefit of membership, PCEA is setting up a platform to make some of these presenta-

tions available on-demand. (More on that in a future issue.)

In the meantime, here's a sampling of what members have been treated to:

- Fundamentals of creating reliable PCBs
- How dielectric and conductor properties can affect high-speed PCB design performance
- PCB materials and their applications
- Designing for RF: Tips and tricks from the PCB pros
- Next gen line/space capability for PCB designs
- An overview of via fill
- Achieve optimal stackup design considering process and electrical performance
- Microvias: Have you designed for reliability?
- How fabrication processes determine DFM guidelines
- The top five symbol and footprint mistakes that even professional engineers make
- Signal integrity effects of different PCB structures
- Artificial intelligence and machine learning basics and how it will affect PCB design
- Addressing the challenges of multi-board design
- Design and manufacturing developments to lower insertion loss and digital pair skew
- Rigid-flex PCB design
- Flex for 5G—why materials matter
- Model-based software design
- Modelado y simulacion de fuentes commutadas (Modeling and simulation of switched fonts)

No, your eyes aren't playing tricks on you. Those last two presentations are among those given at our chapters in Mexico, which are based in Monterrey and Nogales. Other chapters outside the U.S. are in Toronto and the UK, and we are starting to affiliate with organizations in Australia and elsewhere. They mesh with chapters in 11 states (so far) in the U.S., some of which (California, for one) have multiple chapters.

At this point, you might be asking what all this costs. The answer is: nothing. Individual membership in PCEA is free.

At this point, you might be asking what all this costs. The answer is: nothing.

Some chapters have plans to resume live meetings. Others remain virtual. Either way, PCEA meetings remain one of the most cost-effective ways to expand your professional network. Time and again, I hear from engineers that their opportunities for career advancement are limited, or their companies do not support outside training (or for that matter, inside training). While I believe that approach is short-sighted, after 30 years in the industry, I have found the most successful folks are the ones who focus less on talking about how things should be and more on carving out a solution that works for them.

PCEA meetings are exactly that solution.

If you are interested in presenting to one of our chapters, please let me know.

Until then, we wish all readers a warm holiday season and the best for a healthy and prosperous new year. **DESIGN007**

Mike Buetow is a director of the Printed Circuit Engineering Association. To read past columns or contact him, click here.

Better Electronics Start With Better Training

One World, One Industry

by Dr. John Mitchell, IPC PRESIDENT AND CEO

While I've often heard the term "skills gap" used as a catch-all for training issues, it always seemed like an oversimplification of the problem. A skills gap problem implies that the skills the industry possesses are different than the skills the industry needs. The workforce challenges we are seeing today extend beyond a skills gap to also encompass the identification and onboarding of new talent, providing ongoing, job-based training to existing workers, and a means of upskilling to utilize the latest techniques and technologies. IPC is committed to addressing all aspects of the workforce challenge by collaborating with our membership and the industry.

According to a study from Deloitte and The Manufacturing Institute, the U.S. manufacturing skills gap could leave as many as 2.1 million jobs unfilled by 2030[1], costing the U.S. economy more than $1 trillion in 2030 alone. Finding skilled workers is becoming more difficult on a daily basis, as competition for talent increases. While many may equate this program solely to the operator level, the industry is experiencing this same challenge for middle- and high-skill jobs as well. The IPC Workforce Development Program seeks to address the various challenges facing the industry at each skill level.

Finding New Talent

The IPC Education Foundation (IPCEF), in conjunction with the IPC Education Team and industry volunteers and partners are actively

working on identifying and driving new talent into the industry. A variety of pipeline programs are in use or in development to increase interest in a career in electronics by providing accurate information to students, parents, teachers, and guidance counselors at the high school level.

We are also collaborating with schools to develop and offer basic skill training classroom curricula and online programs covering basic electronics skills. In the near future, we will be offering these students the opportunity to gain IPC badges and credentials that identify their competencies for potential employers. A key goal of this initiative is to spark an interest in electronics careers and increase the electronics industry's chances for obtaining new talent. IPCEF is already working with member companies that have generously volunteered to offer tours to students and a firsthand look at the high-tech environment in their local facilities. We are looking to expand this program to help connect students who go through these programs with potential jobs in the industry.

We are looking to expand this program to help connect students who go through these programs with potential jobs in the industry.

Onboarding Training Programs

Finding new talent is just the first step. Onboarding training programs are the foundation of a quality training system. IPC is working with industry to develop engaging, effective, and efficient onboarding programs that help bring new workers up to full productivity quickly, while also increasing knowledge retention. But onboarding talent is not the end of the line. To retain talent in the industry we must turn jobs into careers. We can accomplish this by identifying career pathways and growth opportunities and providing programs to help our workers achieve their full potential. In a tight labor market, it can be difficult to sell a candidate on starting as an operator and remaining as an operator. Through the Workforce Development program, IPC seeks to help employers identify the members of their team with the most potential, and guide them into more advanced roles, helping the industry with the jobs they need today while building a base of talent for the future.

The rate of technological innovation is growing at an exponential rate. This requires a new approach to the challenge of continually upskilling our existing talent. Upskilling programs help organizations improve productivity and employee morale, creating an environment where employees continuously learn, grow, and feel valued.

Organizations that have implemented the IPC Workforce Development program report that staff members learn more in less time (some identified a 50% decrease in training times) and retain what they learned. We continue to work closely with industry subject matter experts to ensure that we provide engaging educational experiences that focus on the right information taught to the right depth and in the right context.

One of the first career pathways tackled by IPC was PCB designers. PCB designers were retiring at an alarming rate and the industry didn't have the proper mechanisms in place to train new talent. IPC created a series of programs that lasted six to eight weeks, with entry level, mid-level, and advanced/specialized programs that cover the spectrum of PCB design knowledge and skill. These programs blend the convenience of online distance learning with the practicality of a project-based curriculum. Every week, students are tasked with completing a real-world design project that applies the skills they learned. They then receive feedback and coaching from our expert instructors. This is just one aspect of the IPC Workforce Development program.

Available in self-paced and instructor-led options, IPC's Electronics Workforce Training courses are designed to meet the growing needs of a rapidly changing industry, while providing flexibility and reducing training costs. In addition to PCB design, other courses include:

- Electronics Assembly for Operators
- Electronics Assembly for Engineers
- Wire Harness Assembly for Operators
- IPC-A-610 for Operators
- IPC-J-STD-001 for Operators
- ESD Control for Electronics Assembly
- CFX courses, and many more

With a variety of courses available, we are directly addressing the critical issues of pipeline, onboarding, and staff retention in a way we hope attracts the best and brightest to our industry. For information on how we can help you meet your education and training needs, visit training.ipc.org. DESIGN007

References

1. U.S. Manufacturing Skills Gap Could Leave As Many As 2.1 Million Jobs Unfilled by 2030, Deloitte.com, May 4, 2021.

Dr. John Mitchell is president and CEO of IPC. To read past columns or contact him, click here.

Navigating the Supply Chain Storm With ICAPE

Interview by Barry Matties

Guillaume Chauvet, vice president of sales-Americas East for ICAPE Group, discusses managing the supply chain through transportation issues, raw material shortages, and longer production times. He also details how he helps customers manage different suppliers and often divergent technologies.

Barry Matties: Guillaume, would you give us a better understanding of how your company operates?

Guillaume Chauvet: The company was created by Thierry Ballenghien. He had managed PCB shops in France, and in 1999 he saw the PCB business going to China, but it was complicated to go there. French people didn't have strong ties to Asia, so it was a really good spot for him to open a company, to buy PCBs directly from Taiwan and China with our own qualified supplier, and then sell them in France. That's how we started: by qualifying a PCB shop in Asia and creating more business in France.

Guillaume Chauvet

It was quite successful. At that time, everyone tried to go to China, but it's not easy when you don't know the business, how to qualify PCB shops, etc. PCBs are very technical. Now it's a bit different. It's easy to buy from Asia, people are used to it, the quality is quite good, and there are many PCB experts there.

In 2004, Thierry opened a new company called CIPEM, which was doing more or less the same as ICAPE but for custom parts like harnesses, coils, and metal parts. There are custom parts that were easier to buy in China and were sometimes a better price. The target is to sell more product to our customers. We were selling PCBs to some contract manufacturers, and we said, "You have trouble finding good suppliers for the harnesses, so let us provide them for you. We have solutions for you." It was really to upsell the existing customers.

As we've grown, we have opened a big office and lab in China to control the quality of our Asian suppliers. Now we also have inspectors in the factory who just check the quality of the product directly out of the lines. We send the product to the lab to validate the sample before the customer receives the boards.

To read this entire conversation, which appeared in the November 2021 issue of *SMT007 Magazine*, click here.

Sponsored link: www.icape-group.com

RealTime with... American Standard Circuits: Thermal Management

Article by Pete Starkey
I-CONNECT007

In the third of a series of three *RealTime with... interviews,* I-Connect007 managing editor Nolan Johnson received knowledgeable and informative answers from Anaya Vardya, John Bushie, and Dave Lackey of American Standard Circuits to his questions on the topic of thermal management.

Anaya Vardya began by clarifying the terminology, describing thermal conductivity as a material property defining how quickly heat was transmitted through a piece of that material, whereas thermal management was about analysing the entire system, trying to understand how much heat was being generated, and using appropriate techniques to dissipate that heat as efficiently as possible. On printed circuit boards, straightforward solutions like thermal vias and heavy copper weights could be used for dispersing heat from packages. He invited Dave Lackey and John Bushie to discuss some of the more esoteric techniques.

Johnson asked Lackey to explain the difference between insulated metal substrates and metal-clad PCBs. There was basically none; both design structures involved bonding the PCB to some sort of metal substrate. A simple example was a single-sided circuit board bonded to an aluminium plate with a thermally conductive adhesive material, such as was commonly used to dissipate heat from LEDs. For additional capability, a double-sided PCB with plated through-holes could be used, fabricated on a substrate of FR-4 or thermally conductive laminate, and bonded to an aluminium plate with thermally conductive prepreg.

Another option was to bond circuits on both sides of a metal core, usually aluminium or copper. The two circuits could operate independently or be interconnected by plated-through-holes drilled through insulating plugs in the metal core. Lackey discussed several alternative metal-backed and metal-core alternatives.

Johnson was interested to learn whether multiple functions could be achieved. Bushie picked up this query. In general, with higher levels of integration and more power per unit area, more attention was paid to thermal management using a variety of techniques and materials, combining multiple functions as well as dealing with the heat. Bushie commented on the increasing convergence of RF and digital circuitry in the same design, together with the growing need for thermal management. People were starting to integrate metal cores into these designs, to decrease the overall package size. His example was a multi-function PCB with the control circuitry on one side of the structure and RF circuitry on the other side. Low-loss RF materials, high-speed digital materials, thermally conductive materials, and metal cores were integrated to increase the functional density of the structure. With flexible circuitry being progressively incorporated to minimise connector real-estate and increase reliability by eliminating cables and connectors, all these materials were coming together into a multifunctional circuit board, and what may have been two, three, or four discrete circuit boards could be combined into one smaller package with a higher level of functionality and a variety of circuit structures.

Johnson remarked that with all these dynamics in play—metals, materials, thermal management—designers were faced with difficult choices, maybe for the first time, often with limited experience. He asked Bushie what advice he would offer.

"This starts to sound like an old song...get your fabricator involved!" Bushie responded. "There's a wealth of knowledge and applications engineering experience at your fabricator; we're here to help. At the end of the day, we want to help you design something that we can make—rapidly, reliably, and functioning as you wanted it. Feel free to utilise us."

In 12 minutes, this *RealTime with...* video combined a primer in thermal management, an indication of how complex a topic it could turn out to be, and a comforting assurance that there is an abundance of support and assistance available to guide the designer through the maze of material selection and structural possibilities, provided that the fabricator is consulted at the beginning of the process rather than expected to sort out the deficiencies of an inferior design when it is presented for manufacture. I enjoyed the experience! **DESIGN007**

Pete Starkey is an I-Connect007 technical editor.

Visit I-007eBooks to download your copies of American Standard Circuits' micro eBooks today:

- *The Printed Circuit Designer's Guide to... Fundamentals of RF/ Microwave PCBs*
- *The Printed Circuit Designer's Guide to... Flex and Rigid-Flex Fundamentals*
- *The Printed Circuit Designer's Guide to ... Thermal Management: A Fabricator's Perspective*

APEX EXPO IPC 2022

Revving Up Design

Feature Article by Patrick Crawford
IPC

During IPC APEX EXPO 2022, we are dedicating one portion of the show floor to PCB design. Inspired by the tenets of the IPC-2231A DFX Guidelines document—in short, good design takes all subsequent electronics manufacturing steps into account—we wanted to bring PCB design to the show floor, which is traditionally more focused on exhibiting fabrication and assembly technologies.

Yes, everything starts with design.

There will be an event in the Design@APEX booth every day of the exposition, starting on Tuesday, January 25 with our IPC Design Competition 2022 Finals. The competitors are currently working on their preliminary designs, and the finalists will be invited to compete in a layout competition at APEX. I'll be moderating the event a la a PGA Tour commentator (sports jacket and all) and we'll have some interactive Q&A with the competitors, as well as some special guest judges who will stop by to say hello.

Wednesday will be a Day of DFX with sessions dedicated to that cross-section of manufacturing and design. Starting in the early afternoon, we will host an Ask Me Anything (AMA) with some of IPC's top design committee volunteers, where we plan to spark a dialogue among attendees about how they can design and manufacture their products better by implementing DFX principles. The rest of the afternoon will feature 30-minute talks intended for passersby to drop in and learn more about IPC's involvement in the many facets of DFX—design for fabrication, manufacturing, test, environment, etc.

Both the AMA panel roster and a comprehensive schedule of talks will be published in the first week of January.

Finally, Thursday will be dedicated to STEM, specifically, how PCB design affects the entire electronics manufacturing industry and how students can get involved as they graduate from high school and move onward with their education. Our friends at Altium have put together a great group of individuals who are involved in their Upverter Education and Training departments who will be standing by in the Design@APEX booth to speak with students during their show floor tours. I'm personally very excited for this as I'm passionate about introducing more students to electronics and science in general. DESIGN007

Patrick Crawford is the manager of design programs and related industry programs at IPC, and an I-Connect007 columnist. To read past columns or contact him, click here or email PatrickCrawford@ipc.org.

Shaping Flexible Circuits

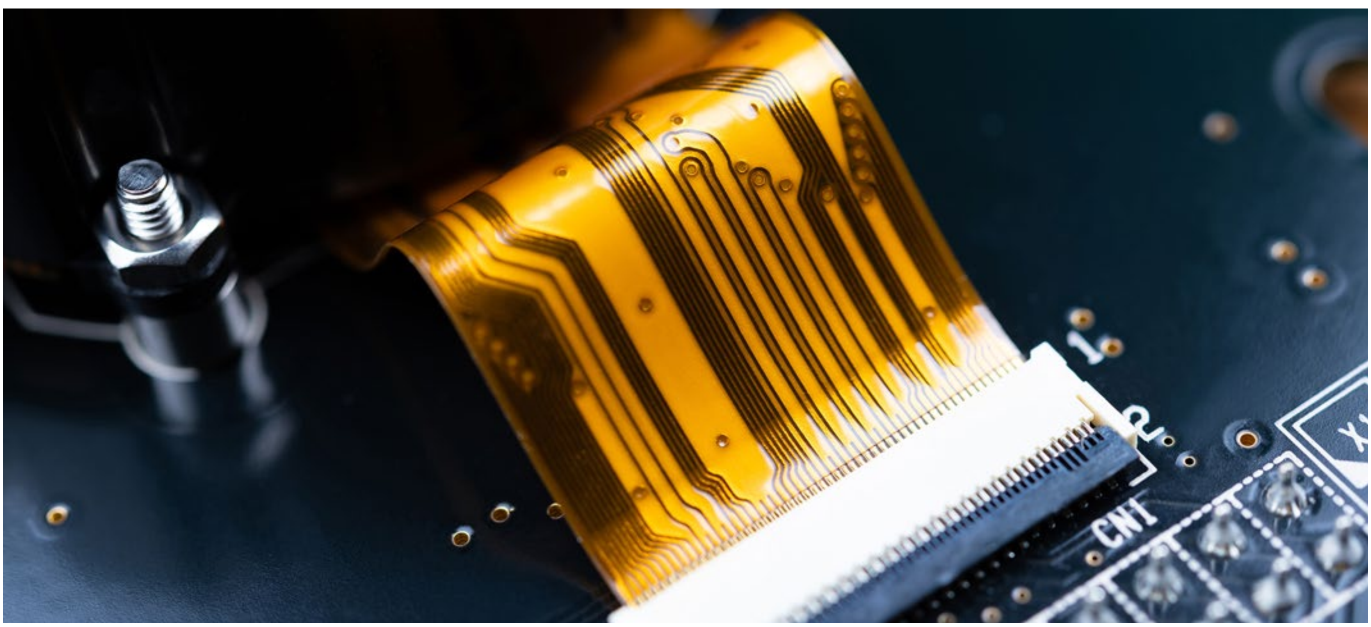
Flexible Thinking

by **Joe Fjelstad**, VERDANT ELECTRONICS

It is axiomatic that bending and folding are fundamental to flexible circuits. The reality is that most flex circuit applications are ones where the circuit is made to conform to the confines of the package that contains it to meet product design objectives. This matter is often glossed over but it is often very important to get the flex circuit to take on a reasonably permanent shape to facilitate its installation into a housing.

Bending flex circuits for static, form-to-fit applications is common; however, getting the flex circuit to hold shape often requires more attention and, in some cases, more processing. This is because flexible circuits, depending on the material used, have varying amounts of elastic memory; that is, the circuit will often try to return to its planar shape. Thus, there is need in such cases to use some established design and process principles for more accurately and permanently shaping flexible circuits to fit permanently in their application.

The first principle of getting a flex circuit to hold shape is to maximize the metal content of the design. This can be accomplished by increased metal thickness, total metal area in the design, or both. Metals, such as copper, have higher moduli of elasticity and tensile



strength than the flexible polymer elements of flex circuit construction but they also are capable of plastic deformation and will permanently deform plastically when bent, provided its elastic limit is not exceeded to the point of fracture.

Many polymers (elastomers are generally excluded, though they can also take a set over time) will also permanently deform if their elastic limit is exceeded. In the case of flex circuit laminates, however, the elastic limit of the flexible carrier film is normally many times greater than the elastic limit of metal. Thus, when the composite structure we call a flex circuit is bent and holds shape to some degree, the metal has plastically deformed while the polymer is still likely to be in its elastic range.

To prevent the polymer from snapping back after bending the flex circuit, it must overwhelm the elasticity of the polymer. Copper is stronger and higher in elastic modulus, however if the circuit traces are thin and small or few relative to the width of the bend area (i.e., the copper is a low percentage of the surface area through the bend), the remnant elastic strain in the polymer may cause the flex circuit to try and return to its original flat shape.

One solution is to uniformly or selectively widen circuit traces through the bending zone to maximize the copper in the bend area as illustrated in Figure 1. Alternatively, if there are electrical/electronic concerns about the trace widths for controlled impedance reasons, fill the spaces between traces with non-functional metal. However, to complete the thought, if the signals are controlled impedance, this method may alter characteristic impedance of the signals and should be checked or modeled. Also, if the design has a weight budget, the impact of the weight increase due to copper should also be considered.

As alluded to earlier, if widening the traces alone is not sufficient, then one of two analogous methods can be employed. One can either use a thicker metal foil or use a thinner flexible base material. The objectives remain the same as they were in the first case: make certain that the metal can overwhelm the polymer in order the hold the final shape. There are advantages and disadvantages to both paths. Making the copper thicker may make etching a bit more difficult; it will also take longer to etch and use more chemistry. On the other hand, making the polymer thinner could make handling a bit more difficult and the strength of the final assembly will not be as great as the alternative method.

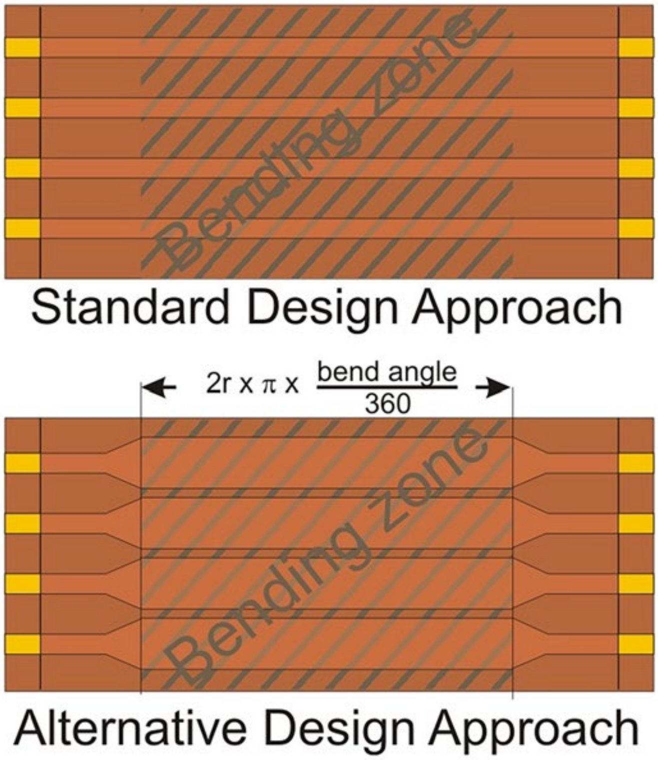
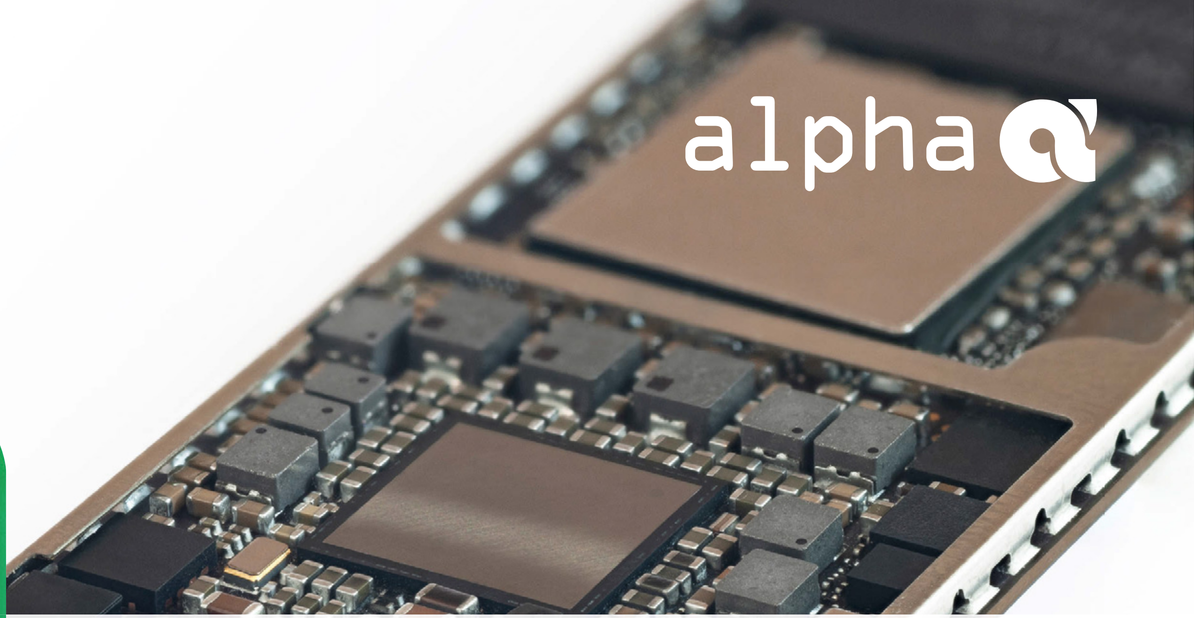


Figure 1: To create a permanent bend, copper traces can be made wider through the bend area. Notice that the circuits are tapered before they enter and after they egress the bend area. An alternative is to use thicker copper, in which case, sculptured flex methods may be best. (Source: *Flexible Circuit Technology 4th Edition*)

In many cases, slight overdriving of the circuit in the bend area can help to create a stable and more permanent shape but be careful that the guidelines for the minimum bend radius for the flex construction are not violated. Remember that when permanently bent and plastically deformed, the copper is normally thinned in



alpha

ALPHA® OM-372

High Electrochemical Reliability
Ultra-Fine Feature No-Clean Solder Paste

ALPHA OM-372 is designed for superior performance on assemblies with ultra-fine pitch components requiring excellent stencil transfer efficiency and high electrical reliability, such as those found in mobile and wearables, computing, and medical assemblies.

ALPHA OM-372 Key Features:

- Best-in-class electrochemical reliability on fine pitched low standoff packages.
- Ultra-fine feature printing and reflow capability down to 008004 components.
- Minimum post reflow residue provides high reliability performance for fine pitch, high density designs when flux is entrapped under devices
- Excellent HiP/NWO Performance on high I/O count packages.
- No-Clean, Zero-Halogen

MacDermid Alpha
ELECTRONICS SOLUTIONS

www.macdermidalpha.com

the bend and is thus weakened. If a truly accurate predictive solution is desired, one can use finite element modeling methods.

Another way to get the flex circuit to hold shape for the application is to form it, commonly done by using a mandrel designed to bend the circuit into shape. The manufacturer can then heat the fixture to a temperature near the glass temperature of the adhesive used in the flex circuit and allow the circuit and fixture to cool in place before removing it. This approach works easiest with polymers and/or adhesives that have relatively low melting points.

Another option is to use less common base materials. These would be materials of low strength and having little, if any, elastic strength. Unreinforced FEP or PTFE (i.e., Teflon), for example, fall into this category. This combination will allow the user to permanently deform the circuit into the desired shape.

To summarize, getting flex circuits to hold shape in an application is not that difficult, but it does take some attention. Several options have been discussed but the method of choice relative to those mentioned here will obviously depend on the demands of the design and its application. FLEX007

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

Elementary, Mr. Watson

We've Never Done It That Way Before

John Watson

The September edition of *Design007 Magazine* discussed the theme of collaborating and working with a team. In that issue, I wrote a feature article called "PCB Design Is a Team Sport." After that edition was published, I had several follow-up questions and conversations with individuals; they agreed on the importance of teamwork but felt that it's easier said than done. It's challenging because of the inherent problem of team members accepting or handling change very well. Change is a word that sends shivers down the spine of some. You know those sort of individuals. They're easy to identify. The ones that constantly remind everyone, "We never did it that way before." As if how we did things in the past was so much better.

Why don't you hear much reminiscing about the good old days of PCB design? Maybe because it meant long hours at a light table with a sheet of mylar and endless rolls of tape, going home at the end of the day with more scraps of tape on you than finally got into your design, it took hours to get rid of the spots in front of your eyes. I know some readers are scratching their heads, wondering what I'm talking about.

The truth is, we are a part of an industry that significantly influences how people live. There is not an area of our lives where electronic devices don't have an impact. With that said, the desire for better, innovative, faster, and smaller devices are constantly causing our industry to change and push forward. I would say our industry is reinventing itself constantly. The Greek philosopher Heraclitus had it right when he said that "The only constant in life is change."

To read this entire column, click here.

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

Flex007 Highlights

AT&S Continues Strong Growth Course; Revenue Up 30% in 1H of 2021/22 ▸

Consolidated revenue rose by 29.7% to € 697.6 million in the first half of 2021/22 (PY: € 537.8 million). Adjusted for currency effects, the increase in consolidated revenue even amounted to 34.8%.

Rogers Reports Q3 2021 Results ▸

"Rogers' strong position in the burgeoning EV/HEV markets was again evident in our third quarter results, despite some near-term supply chain challenges," stated Bruce D. Hoechner, Rogers' president and CEO.

Taiyo Discusses New Inkjet Technology and Solder Mask for Flex ▸

At PCB West, Nolan Johnson spoke with Brian Wojtkiewicz of Taiyo about some of the company's new inkjet technology, which will be manufactured in the U.S. Brian also discusses a new solder mask that has been developed especially for the flexible circuit segment of the industry.

All Flex Discusses Merger and New Medical Applications ▸

Nolan Johnson recently spoke with Jamin Taylor, vice president of sales and marketing at All Flex Flexible Circuits in Minnesota. The discussion revolved around the company's recent merger with Printed Circuits Inc., and some creative new flex applications, including flexible heaters and catheters for the medical field. We're working on a few new applications and products, like being able to do some fine lines and flex materials, but with large format, a longer format than usual."

Flexible Thinking: Flexible Circuits—A Catalyst for Technological Evolution ▸

With only a wee bit of prejudice, columnist Joe Fjelstad would argue that flexible circuits are among the most adaptive and adaptable of all electronic interconnection technologies and perhaps the most catalytic as well.

Royal Circuits Hires Herb Snogren to Lead Entry into Ultra-HDI Technology ▸

Royal Circuits announced Herb Snogren has joined the company's management team as an industry consultant. Snogren will be leading Royal Circuits' latest initiative to add ultra-HDI technology to its existing PCB manufacturing services.

FLEX Conference to Co-locate with SEMICON West in 2022 ▸

SEMICON West, North America's premier microelectronics exhibition and conference that unites players across the entire electronics manufacturing and design supply chain, and FLEX Conference, the annual event focused on flexible hybrid and printed electronics innovations, will co-locate at the Moscone Center in San Francisco, July 11-14, 2022.

Trackwise Delivers Record-breaking 72m Multi-layer Flexible PCB ▸

Trackwise Designs plc, the UK-based manufacturer of specialist products using printed circuit technology, has broken its own record for the world's longest multi-layer flexible printed circuit (FPC), delivering a 72m FPC for in industrial application.

Challenges in Simulating a Wirebonded CoB on a Rigid-Flex PCB

All Systems Go!

by Brad Griffin, CADENCE DESIGN SYSTEMS, INC.

There are many good reasons to use a chip-on-board (CoB) implementation. When this is combined with wirebonding and the use of a rigid-flex PCB, challenges mount. An application that demands all three—CoB, wirebonding, and rigid-flex PCB—is a camera module that goes into a mobile application, the sample design used to illustrate the design and analysis challenges in this article. If you are not aware of and prepared for the potential pitfalls, it is highly likely that your project could fall short or even fail.

If the camera module goes into a mobile application, cost, performance, footprint (in all three axes), and time to market become requirements that must be met. Cost and footprint justify the use of CoB implementation due to the elimination of a package; performance and cost justify the wirebonding of the image sensor die directly on to a rigid-flex PCB, since the short wires with the eliminated package improve performance and reduce cost; the restricted footprint within a mobile device calls for the use of a rigid-flex PCB. Figure 1 shows a rigid-flex PCB with an image sensor implemented using CoB and the MIPI signals driven to the connector on the other end of the rigid-flex PCB. The impedance of the MIPI signals traversing from the wirebonded chip to the connector must be a consistent 100-Ohm path.

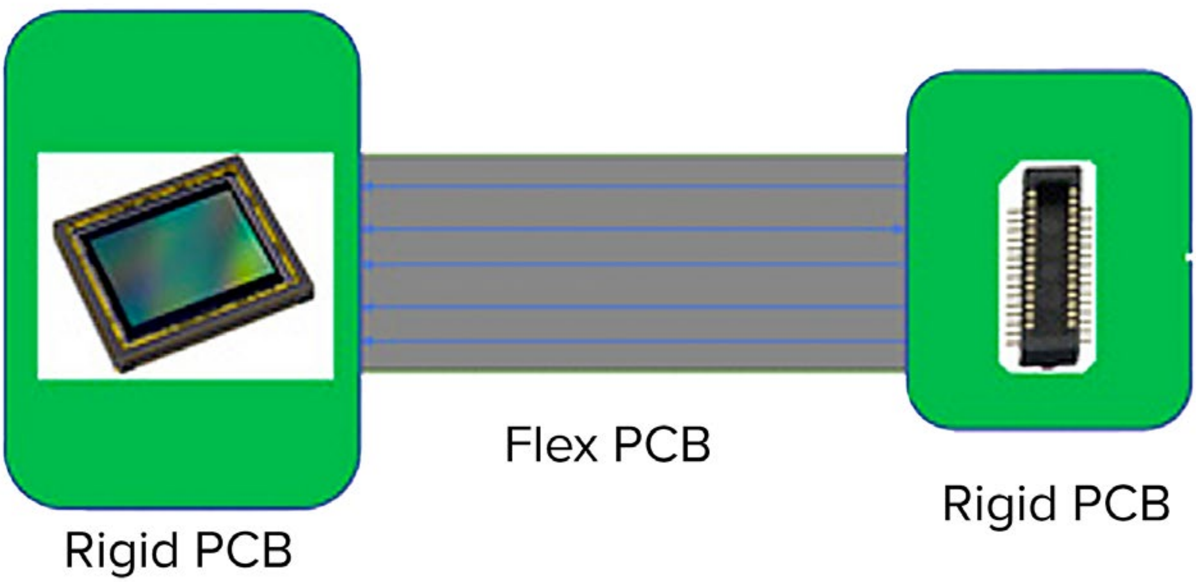


Figure 1: Rigid-flex PCB with CoB and connector.

The use of rigid-flex PCB, CoB implementation, and wirebonding alone could lead to a slew of potential pitfalls or challenges:

- Designing the rigid-flex with different stack-up between the rigid and flex sections
- Wirebond routing in the miniature board area
- Use of stacked vias for routing
- Controlling the impedance of 100 Ohm
- Extraction of the 3D wirebond in 2D (with all legacy tools) does not help with accuracy
- Simulation time for extracting the wirebond profile with respective signals

The CoB assembly process, commonly referred to as direct chip attachment (DCA) technology, refers to the assembly/technology process in which a die is directly mounted, electrically connected, and typically encapsulated with a silicone- or epoxy-based material in the final PCB. Figure 2 shows a typical implementation of the CoB and the wirebonding.

The rigid-flex design is best created using a PCB design tool with the ability to define and add multi-stackup and constraints for rigid-flex, including inter-layer and comprehensive spacing rules. When the CoB is a large chip with a greater number of pins than a typical PCB tool can handle, you will need to use the SiP capability of a package design tool.

Figure 2 and Figure 3 show the final fabrication layout images of the product and 3D view with wirebond profile.

After the layout of the CoB in rigid-flex, create the MIPI/xTalk channel topology and perform signal extraction and simulation of the channel. If a custom compliance kit/MIPI is available, perform compliance checks and generate the compliance signoff pass/fail report. All of these are best performed using a fast extraction and simulation solution.

Next, it is important to simulate the entire camera module including the rigid-flex, wirebonding, channel, and the connector. This is typically done using the divide and conquer approach by extracting each component as a whole or in parts and merging them together manually in a SPICE file for simulation. This is no longer true with new extraction tools, which can extract the rigid-flex board, wire-

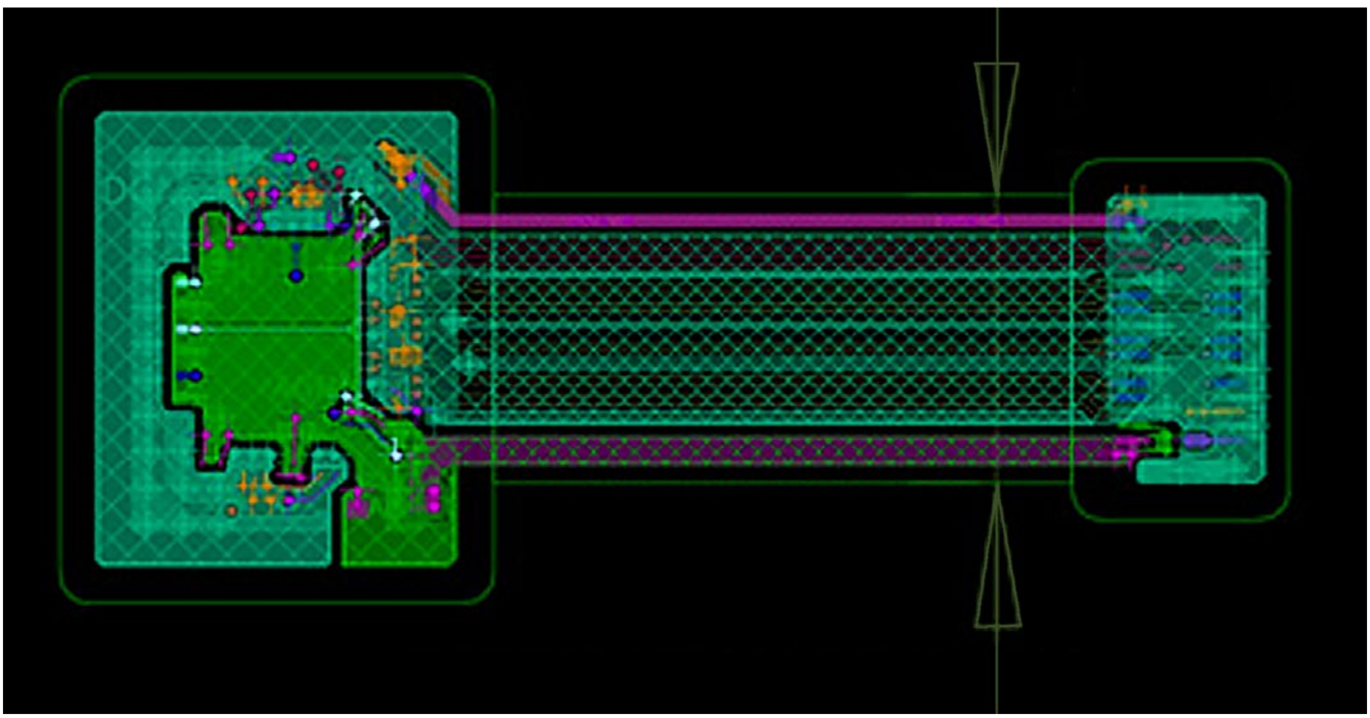


Figure 2: Fabrication drawing.

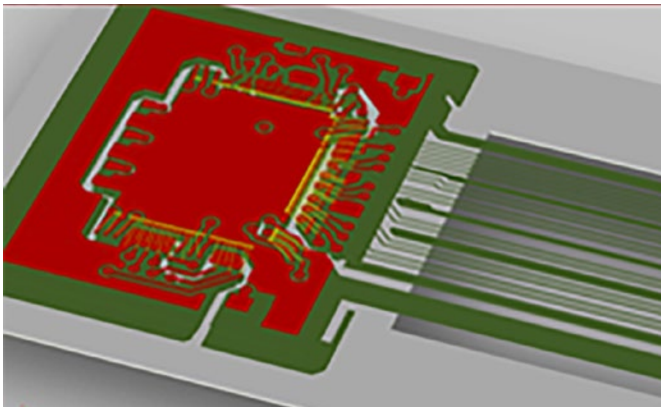


Figure 3: 3D view with wirebond profile.

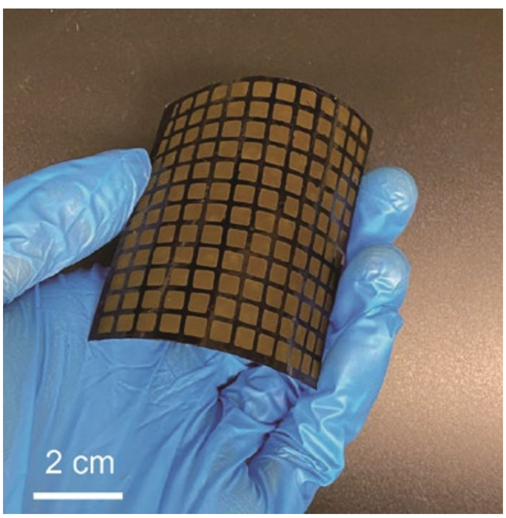
bond, and the connector together, and perform that in 3D. The extracted 3D wirebond profile needs to be brought into the simulator, which is typically a manual process with legacy tools. This also has changed with new tools, which perform faster and yet accurate 3D extraction of the design, and directly read the 3D wirebond profile from a PCB file. The new tools perform much faster simulation of the design, including return and insertion losses, and power-aware analysis with virtually unlimited scalability. This improves the accuracy of extraction, saves time, and eliminates the risk of introducing manual errors in the process.

Learn how a customer applied this methodology to solve their camera module design that went into a mobile device. **FLEX007**

Brad Griffin is a product management group director for the Multiphysics System Analysis Group at Cadence Design Systems, Inc., and the author of *The System Designer's Guide to... System Analysis* (a free eBook available for download). To read past columns or contact Griffin, click here.

Flexible, Wearable X-ray Detector Doesn't Require Heavy Metals

X-ray imaging is a fast, painless way for doctors to see inside a patient, but radiation detectors, which go under the body part being imaged, are rigid panels that contain harmful heavy metals. Now, researchers in ACS' Nano Letters report a proof-of-concept wearable X-ray detector prepared from nontoxic metal-organic frameworks (MOFs) layered between flexible plastic and gold electrodes for high-sensitivity sensing and imaging. Detectors that could conform to rounded body parts or mold to the inside of confined spaces could be beneficial in some applications. The researchers mixed a solution of nickel chloride salt and 2,5-diaminobenzene-1,4-dithiol (DABDT) for several hours, creating a MOF in which nickel linked the DABDT molecules. In initial tests, the nickel-containing MOF was more sensitive than recently reported detectors when irradiated with 20 keV X-rays, equivalent to the energy released during medical diagnostic imaging. Then, to make a flexible X-ray radiation detector, the team sandwiched the nickel-containing MOF between gold film electrodes, one of which was on a flexible plastic surface. They used copper wires to transmit current from each pixel of a 12x12 array and covered the whole device with a silicone-based flexible polymer. Finally, they placed an aluminum letter "H" on the detector and irradiated it with X-rays, measuring a much lower current output underneath the H than under the unimpeded material. This device is promising for the next generation of radiology imaging equipment and radiation detection when wearable or flexible devices are needed.



(Source: ACS—American Chemical Society)

TOP TEN

EDITOR'S PICKS

Fresh PCB Concepts: High-tech PCBs from Design to Volume Production


Ruben Contreras

Columnist Ruben Contreras talks about producing high-tech PCBs from design to volume production, since it is within this area that customers often do not reach their initial targets regarding time, cost, and performance.

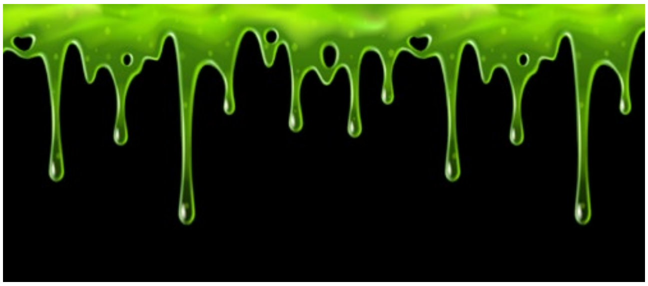
Averatek Moves Ahead With A-SAP™, ASAP


Vinton & Denning

During DesignCon, the I-Connect007 Editorial Team spoke with Mike Vinson and Paul Dennig of Averatek. They discussed the company's latest advances in air cavities and semi-additive processes, and what this new technology will mean to the industry, especially PCB designers. Vinson said, "Over the next 12 months, I think you'll see more involvement from a number of areas—everything from sensors to the actual printed circuit boards themselves and miniaturization."

Beyond Design: The Impact of No-Clean Flux Residue on Signal Integrity

Most electronic products today are assembled using the no-clean soldering process. The need for no-clean solder pastes emerged in response to legislation against the use of ozone-depleting chemicals, and the appeal of removing the costly flux cleaning operations in the assembly of PCBs.




Joe Clark

DownStream Adding More Flex, Rigid-Flex Functionality

Nolan Johnson recently spoke with Joe Clark, one of the co-founders of DownStream Technologies. We discussed the company's latest software release, which adds more flex and rigid-flex capabilities, as well as updates to the scripting in DownStream's tools. As Joe explained, "With our current release of software, we focused on rigid-flex designs, which is becoming a common design challenge for our customers."

PCB Talk: Additive Electronics—Are You One of the Curious?

The term "additive electronics" is a broad term in our industry. To many this suggests 3D printing and the processes used to form circuit patterns with these additive methods. To others, this term conjures the image of newer PCB fabrication techniques that use semi-additive PCB fabrication processes to realize line width and space below the traditional 75-micron (3 mil) capabilities that are typically seen with subtractive etch processing.

Polar Instruments: Pandemic and Parts Shortage Lead to More R&D Time


Geoffrey Hazelett

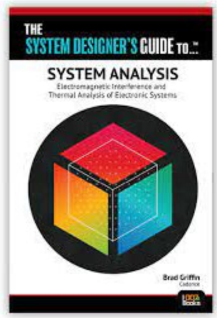
During PCB West, Nolan Johnson met with Geoffrey Hazelett, VP of sales for Polar Instruments. He explained how the pandemic actually helped Polar's R&D in the long run, and why the current parts shortages may offer a similar benefit for OEMs, who can now spend more time in what he calls "forced R&D."

Manage Your Data and Document Everything

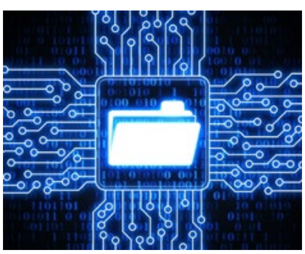
At a small company, it's important to have rules and guidelines about managing documentation. In a startup environment, this type of work usually falls on engineers and the technologists doing the actual testing.

Excerpt: *The System Designer's Guide to… System Analysis,* Chapter 3

The third chapter of this book, "Limitations of Today's Electronic System Design," focuses on the variety of limitations that today's electronics system designers face as data becomes ever more complex and the industry seeks alternatives past Moore's law.



Connect the Dots: Finding Value in Gerber Files



Converting to Gerber is one way to perform a double check of your PCB design that can pre-answer questions from your manufacturing partner and pre-solve problems with the boards themselves.

DFM 101: PCB Via Structures


Anaya Vardya

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

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

Career Opportunities

WHELEN®
ENGINEERING COMPANY, INC.

Galvanic Systems Director

Whelen Engineering Co. seeks FT Galvanic Systems Dir. in Charlestown, NH to lead technical team to optimize GreenSource Fabrication, LLC Division's first-gen equip. by applying PCB mfg. concepts per cust reqs. Ensure process engg. meets co.'s needs; develop and validate process changes; plans to improve process capability using statistical & root cause analysis & eval'ing equip, including Atotech equip, thru design of exper & testing; travel int'lly 15-25% to eval biz plan & strategy to markets. Min reqs: U.S. Bach degree or foreign equiv. in chem sci or chem engg; knwl of entire PCB mfg. process, including process flows, indiv. processing steps, & tooling, w. knowledge of PCB pattern plating, including subtractive etching processes, additive processes, and printable techs as demo'd by 12 yrs' exp. in PCB industry; Theoretical knwl of PCB Plating Processes, including MLB, HDI, and SLP-type PCB fab processes, as demo'd by 10 yrs' exp w. PCB plating processes; 5 yrs' exp working w. Atotech Equipment prod lines & their specialty chems; Prior work exp in R&D enviro. including app of lab analysis concepts and knowledge of cross section and wave form patterns.

Apply to: Corinne Tuthill,
ctuthill@greensourcefab.com or at
Greensource Fabrication, LLC,
99 Ceda Rd, Charlestown, NH 03603

apply now

schmoll maschinen
... one step ahead.

Service Engineer

Schmoll Laser and Direct Imaging

Reports to: Field Service Manager
Location: North America

SUMMARY:
Provide expert-level service on multiple laser drilling and direct imaging product lines. Maintain high customer satisfaction, timeliness, accuracy, efficiency, cost effectiveness, and safety.

DUTIES AND RESPONSIBILITIES:
- Install, commission, and maintain Schmoll products at customer sites. Perform modifications and retrofits as needed.
- Troubleshoot, diagnose, and calibrate products via telephone or at customer sites.
- Handle a wide variety of problems, issues, and inquiries.
- Provide training for customers and others in the effective operation, calibration, and maintenance of all products.
- Lead the project management team for retrofit/upgrade requests and recommendations for Schmoll equipment until the end of commissioning and final payment.
- Assist customers with potential optimization of their machine operations and work with clients on application improvements.

QUALIFICATIONS:
- Must possess a valid driver's license, clean driving record, major credit card (for business travel), and passport.
- Ability to read and interpret technical documentation, compile reports, and compose routine correspondence, define problems, collect data, and draw a valid solution.
- Must be able to travel extensively, partly international, to support customer needs. While Burkle makes every attempt to avoid Sunday and Friday evening travel, sometimes it is required.

apply now

Career Opportunities

MacDermid Alpha
ELECTRONICS SOLUTIONS

Technical Marketing Specialist

Waterbury, CT

JOB DESCRIPTION:
Responsible for providing technical knowledge and support to marketing communications professionals. Cross training and acting as liaison between the Innovation and the Marketing Communications teams for both Circuitry Solutions and Semiconductor Solutions.

Chemist 1

Waterbury, CT

JOB DESCRIPTION:
Perform analysis—both chemical and mechanical—of customer-supplied samples. These include both structural and chemical testing using various instruments such as SEM, Instron, ICP, and titration methods. Perform various failure analysis functions, including, but not limited to, chemical analysis, SEM analysis of customer parts, and cross-section evaluation.

Applications Manager

Waterbury, CT/New England Region

JOB DESCRIPTION:
Applications Manager in the Electronics Specialties/Circuitry Solutions group to provide applications process knowledge, training and technical support of new products leading to sales revenue growth. Requires working through the existing sales and technical service organizations to leverage this knowledge globally. Experience in multilayer bonding along with dry film and solder mask adhesion processes a plus.

apply now

CML

Sales Manager

New Business Development (m/w/d) – Sales Territory: Germany

CML Group is a global leader in the Printed Circuit Board industry, specialized in PCB manufacturing and sustainable PCB supply solutions. Our products are made to the highest quality and reliability standards, including automotive requirements.

For the expansion of our target markets, we need you to generate new business, drive new projects from RFQ stage and manage the customer relationship.

Your responsibilities:

- Develop new customers and build long-term customer relationships
- Understand the customer requirements and acquire new contract enquiries from all market sectors
- Proactive market and customer research
- Identify new potential electronic industry sectors
- Result-oriented sales management including support and consulting on new projects
- Independent management and organization of your accounts
- Price and contract negotiations with customers and contractual partners

Your profile:

- Several years of professional experience in sales and key account management
- Knowledge of printed circuit board production/ industry would be an advantage
- Fluent in Business English and willingness to travel internationally
- Flexible and an open-minded mentality
- Strong communication skills, team player
- Self-motivated, well-organized, professional
- Your home base is in Germany

Interested? Looking forward to your application!
Please send your application to hr@cmit.support.
For more information visit
www.cml-globalsolutions.com

apply now

Career Opportunities

Career Opportunities

Fuji America Corporation is a rapidly growing electronics assembly equipment distributor. We support the factories of the future and smart factories globally. We offer an exciting and challenging career for a software support engineer and an applications engineer who want to join our growing company.

Software Support Engineer

As a software support engineer for Fuji America Corporation, you will be a customer-facing technical advisor with the opportunity to solve technically complex problems for our proprietary software. As a trusted advisor to our customers, you will have influence over a broad range of solutions that create business value. As a valued member on our team, the software support engineer will use advanced troubleshooting methods and tools to solve technically complex problems. These highly complex, escalated problems require broad and in-depth product knowledge, as well as exceptional troubleshooting skills.

- Field installation of proprietary software/automation equipment throughout North America
- Field troubleshoot, repair, training, and process support of proprietary software
- Provide remote and on-site technical support
- Troubleshoot Windows 10/Windows server installing, configuration, and support
- Networking experience—setting up and supporting networks.
- Exposure and/or experience with Oracle or Microsoft SQL server databases
- Strong verbal communication skills with both customer and other technical depts.
- Flexibility to travel and perform job assignments on short notice
- Strong aptitude with current computing applications and networking processes

Experience

- Bachelor of Science in computer science or related field preferred

Applications Engineer

As an applications engineer, you will be responsible for doing cycle time and studies in preparation to make recommendations of Fuji products for customers' applications. Support implementation of activities within the technical center such as customer visits, demonstrations, evaluations, testing, inspection of Fuji products, including peripheral equipment from other vendors.

- Assist sales representatives in technical aspects relating to machine and software functions and utilization.
- Assist sales representatives and customers with providing CTA (Cycle Time Analysis) to them for recommending Fuji products to customers' specific applications. This includes the sFAB machine as well as all other SMT machines.
- Schedule and perform product demonstrations on all available types of equipment and software to potential and existing customers.
- Test and evaluate existing as well as new technologies on equipment and software performance and reliability.
- Assist in the coordination of any new FAC projects by utilizing your full potential.
- Responsible for the setup of the equipment and its demonstration for various trade shows.
- Assist FAC staff in any technical issues which may require attention.
- Assist in the coordination of design and manufacture of customs tooling for placement equipment.
- Perform inventory checks every six months according to the schedule and manner regulated by the company, if applicable.

Experience

- Minimum five years programming/computer experience
- Bachelor's degree preferred

apply now

Career Opportunities

ICAPE GROUP

PCB Field Engineer– North America Operations

ICAPE Group is a European leader for printed circuits boards and custom-made electro-mechanical parts. Headquartered in Paris, France, we have over 500 employees located in more than 70 countries serving our +2500 customers.

To support our growth in the American market, we are looking for a PCB Field Engineer.

You will work in our North America technical center, including our U.S. technical laboratory, and will be responsible for providing technical and quality support to our American sales team.

You will have direct customer contact during all phases of the sales process and provide follow-on support as required.

RESPONSIBILITIES INCLUDE

- Feasibility recommendations
- Fabricator questions and liaison
- Quality resolutions
- Technical explanation (for the customer) of proposals, laboratory analysis or technology challenges

REQUIREMENTS

- Engineering degree or equivalent industry experience
- 5 years' experience with PCB manufacturing (including CAM)
- Excellent technical understanding of PCBs
- Experience with quality tools (FAI, PPAP and 8-D)
- Good communication skills (written and oral)

Communication skills are essential to assist the customer with navigation of the complex process of matching the PCB to the application.

SALARY

Competitive, based on profile and experience. Position is full time in Indianapolis, Ind.

apply now

ventec
INTERNATIONAL GROUP
騰輝電子

Customer Service Representative, UK

We are looking to expand our UK Customer Service/Internal Sales team. As Customer Service Representative you will provide great sales and customer service support and respond to the needs of clients from industries including Aerospace, Defence, Automotive and Pharmaceutical. Duties include:

- Maintain & develop relationships with new and existing customers
- Make rapid, accurate cost calculations and provide quotations
- Accurately input customer orders through bespoke MRP System
- Liaise with colleagues at Chinese HQ and other Overseas Business Units to manage domestic and international requirements
- Assist sales team with reporting, sales analysis and other items at their request

Skills and abilities required for the role:

The ideal candidate is a proactive self-starter with a strong customer service background. Friendly, approachable, and confident, you should have a good phone mannerism and be computer literate.

- Previous experience in a Customer Service background, ideally management or supervisor role
- Experience with MRP Systems
- Good working knowledge of Microsoft Office Tools such as Outlook, Excel etc.

What's on Offer:

- Excellent salary & benefits commensurate with experience

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

Please forward your resume to HR@ventec-europe.com

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

Test Engineer (TE-MD)

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

- Candidates must have at least three years of experience with in-circuit test equipment. A candidate would develop and debug our test systems and install in-circuit test sets remotely online or at customer's manufacturing locations nationwide.
- Candidates would also help support production testing and implement Engineering Change Orders and program enhancements, library model generation, perform testing and failure analysis of assembled boards, and other related tasks.
- Some travel required and these positions are available in the Hunt Valley, Md., office.

apply now

Sr. Test Engineer (STE-MD)

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

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

We proudly serve customers nationwide and around the world.

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

apply now

Career Opportunities

Career Opportunities

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

Prototron Circuits

Sales Representatives

Prototron Circuits, a market-leading, quick-turn PCB shop, is looking for sales representatives for all territories.

Reasons you should work with Prototron:

- Serving the PCB industry for over 30 years
- Solid reputation for on-time delivery (99% on-time)
- Excellent quality
- Production quality quick-turn services in as little as 24 hours
- AS9100
- MIL-PRF- 31032
- ITAR
- Global sourcing
- Engineering consultation
- Completely customer focused team

Interested? Let's have a talk.
Call Dan Beaulieu at
207-649-0879
or email to
danbbeaulieu@aol.com

apply now

Career Opportunities

SMT Operator
Hatboro, PA

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

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

Duties and Responsibilities:

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

Requirements and Qualifications:

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

We Offer:

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

apply now

SMT Field Technician
Hatboro, PA

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

Duties and Responsibilities:

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

Requirements and Qualifications:

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

We Offer:

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

apply now

Career Opportunities

SIEMENS

Siemens EDA
Sr. Applications Engineer

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

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

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

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

apply now

Koh Young Technology

Logistics Assistant

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

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

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

apply now

Career Opportunities

BLACKFOX
Premier Training & Certification

IPC Instructor

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

Independent contractor, possible full-time employment

Job Description

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

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

Qualifications

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

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

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

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

apply now

U.S.C.
U.S. CIRCUIT

Plating Supervisor

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

Competitive benefits package. Pay will be commensurate with experience.

Mail to:
mfariba@uscircuit.com

apply now

Career Opportunities

Career Opportunities

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

CAD/CAM Engineer

Summary of Functions

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

Essential Duties and Responsibilities

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

Organizational Relationship

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

Qualifications

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

Physical Demands

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

apply now

INSULECTRO

Are You Our Next Superstar?!

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

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

View our opportunities at
Insulectro Careers (jobvite.com)

apply now

Career Opportunities

Career Opportunities

MANUFACTURERS OF QUALITY PRINTED CIRCUIT BOARDS

Pre-CAM Engineer

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

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

apply now

Process Engineer

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

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

apply now

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

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

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

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

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

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

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

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

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

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

TECHNOLOGY SPECIALIST: **BRYSON MATTIES**

COVER: **SHELLY STEIN**

COVER IMAGE: **ADOBE STOCK © PIXEL-SHOT**

DESIGN007 MAGAZINE

DESIGN007 MAGAZINE®
is published by BR Publishing, Inc.,
942 Windemere Dr. NW, Salem, OR 97304

December 2021, Volume 10, Number 12
DESIGN007 MAGAZINE is published monthly,
by BR Publishing, Inc.

ADVERTISER INDEX


Problems solved!
PCB007 MAGAZINE
SEPTEMBER 2021
PCB007 MAGAZINE
OCTOBER 2021
PCB007 MAGAZINE
DESIGN007 MAGAZINE
OCTOBER 2021
DESIGN007 MAGAZINE
NOVEMBER 2021
DESIGN007 MAGAZINE
DECEMBER 2021
ART & SCIENCE OF PCB DESIGN
SMT007 MAGAZINE
NOVEMBER 2021
OR
SMT007 MAGAZINE
DECEMBER 2021
COMING SOON
01 22 22
START PLANNING NOW WITH OUR IPC APEX EXPO PRESHOW COVERAGE
SUBSCRIBE NOW