AUGUST 2021

I-Connect007

DESIGN007 MAGAZINE

DESIGN EDUCATION:

Not Just a Roll of the Dice

DESIGN007 MAGAZINE

Design Education: Not Just a Roll of the Dice

This month, we asked a variety of expert contributors, seasoned and not-so-seasoned, to describe their path to becoming successful PCB designers and their advice to new designers seeking to continue their education.

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

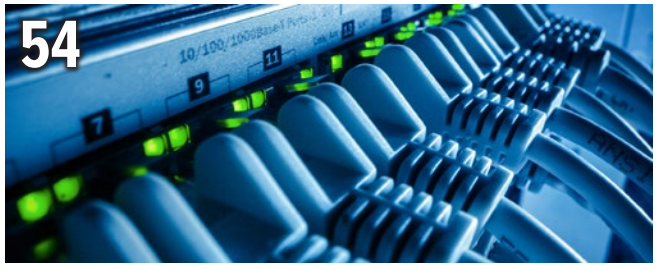
Every day we get questions like those. And we love it. We have more than 490 PCB experts on 3 continents speaking 19 languages at your service. **Regardless where you are or whenever you have a question**, contact us!

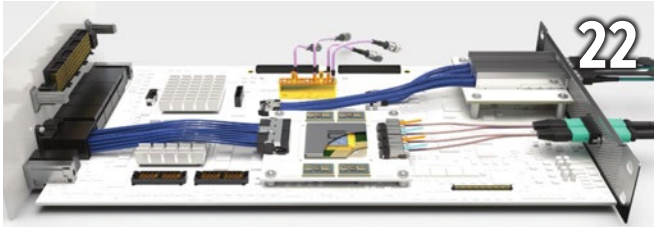
What's your PCB question?
www.ncabgroup.com/PCBquestion

Reliable answers. Reliable PCBs.

NCAB GROUP

DESIGN007 MAGAZINE





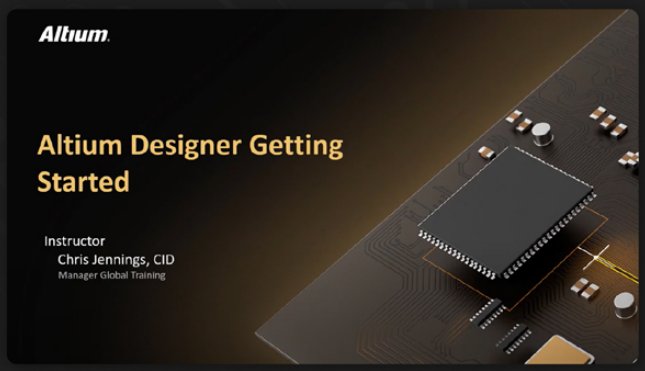
ALTIUM ACADEMY

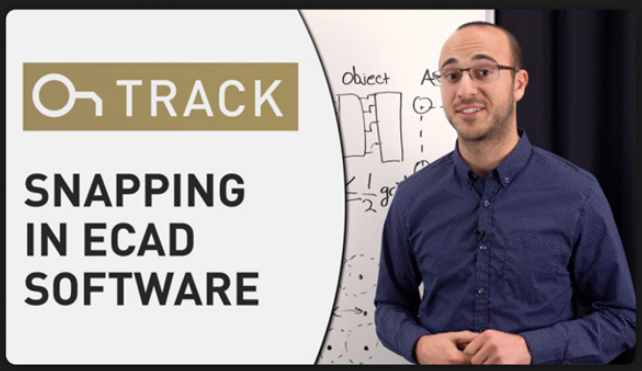
Online Education for PCB Designers and Electrical Engineers

Access over 350 tutorial videos

- On-demand Webinars
- What's New with Altium Designer
- Getting Started with Altium 365
- Whiteboard Series

Watch Now



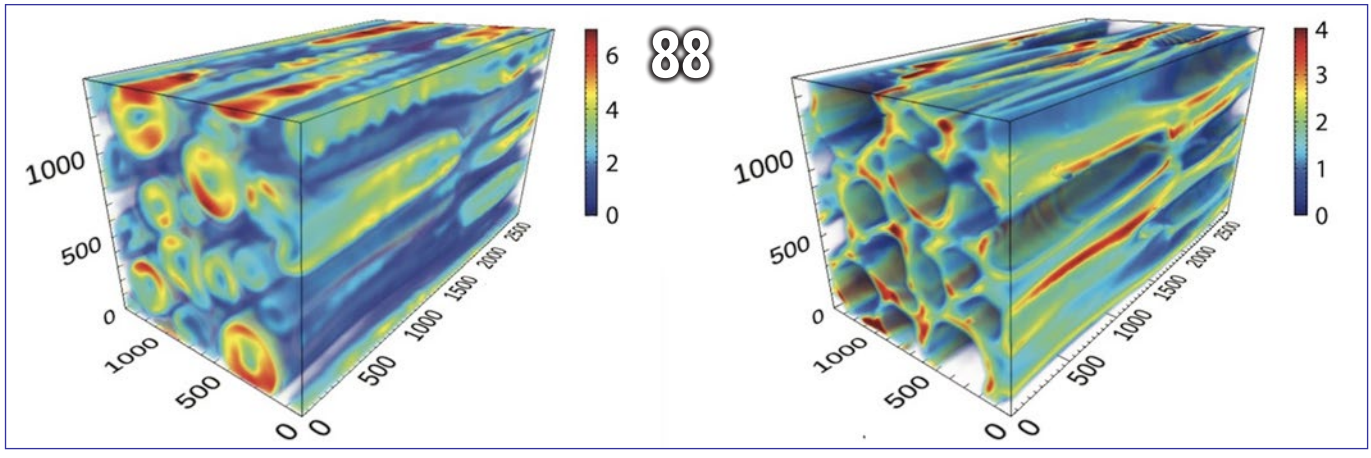


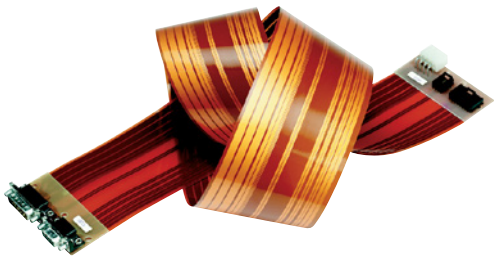
Altium | YouTube

FLEX007

This Month in Flex

Flexible and rigid-flex circuits offer an array of advantages for technologists and inventors who are seeking to push the proverbial envelope. This month, Flex007 looks into the modus operandi of today's flexible circuit innovators, and the need to ask both "Why?" and "Why not?" when developing new electronic interconnect technologies.

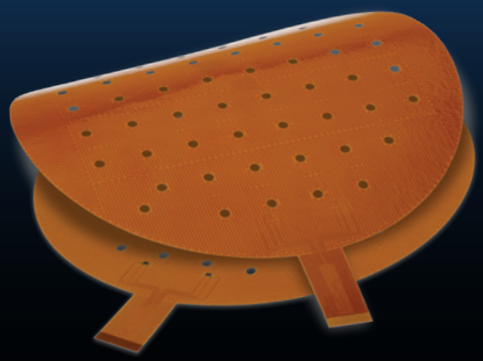




all flex

Flexible Circuits • Heaters • Assemblies

- Design application assistance
- Component assembly
- Fast response
- Quick delivery



All Flex Flexible Circuits & Heaters
1705 Cannon Lane
Northfield, MN 55057
(800) 959-0865
(507) 663-7162

allflexinc.com

Design Education: Not a Roll of the Dice

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

If you're a new PCB designer today, you may feel like a first-level fighter in "Dungeons and Dragons." You thrive on the variability and complexity of this career but moving up to the next level is often the result of a series of choices that you have to make—often without knowing what's going on.

But there is one thing that you can control: your education. And the more you know, the more control you have over your career.

Fortunately, new designers have access to more educational and training resources than ever before. There are live and on-demand webinars, YouTube videos, blogs, podcasts, and now live, in-person conferences and trade shows are back.

You can join a design organization like PCEA or IPC Design and attend chapter meetings online, even if you live across the country from that chapter. You can read industry magazines like this, technical papers from conferences, books written by the top design experts in the world, and the free eBooks published by I-Connect007, including *The Printed Circuit Designer's Guide to…* series, which have proven wildly popular. (Now that I'm a book editor, maybe I need a blazer with elbow patches and a meerschaum pipe.)

But you must do your due diligence, like you were buying a company. I'm sure you won't be surprised to learn that not every PCB design video on YouTube contains design techniques that would make Rick Hartley say, "Damn, why didn't I think of that?" It's up to you to separate the good training from the bad, the wheat from the chaff, to get slightly biblical.

"It's up to you" is the operative phrase here. It's up to you to continue your design educa-

tion—whether or not your boss pays for it—and that requires filtering through all of this content, whether it's online or in a hardback book. There's a lot of noise, marketing content, and bad science masquerading as real design information.

It's up to you to sift through all that raw data. Pick your favorite design instructors and read everything they've ever written. If you're not sure who the design experts are, look to see who's speaking at DesignCon, PCB West, and AltiumLive. There's a reason that the same group of speakers has been teaching PCB design at conferences around the world for years: they get good evaluations, so they get invited back. They're like good bands; they pack the venue every time. Follow them.

This month, we asked a variety of expert contributors, seasoned and not-so-seasoned, to describe their path to becoming successful PCB designers and their advice to new designers seeking to continue their education. Instructor Susy Webb explains how much PCB design has changed since she started her career, and how her class curriculum has evolved since she began teaching in 2003. Tomas Chester discusses his entry into PCB design during the 2008 downturn and what college did and didn't teach him. Tamara Jovanovic shares a story about the day she went from assisting on design projects to designing her first PCB from start to finish.

Patrick Davis explains why new PCB designers usually cost their employer money in their first year, and don't become profitable until their third. Tom Hausherr drives home a great point in his feature article: You can't learn PCB design on your own. It's just not going to happen. Michael Steffen stresses the importance of being a problem-solver, and why all the design curriculum in the world can't help you if you don't know how to apply this information. Kelly Dack discusses his work educating the "Ians, Ashleys and Zachs of the world" in proper DFM strategies. Martyn Gaudion delves into the many "peripheral influencers" that young designers need to be aware of, many of which seem to have no connection to PCB design. Judy Warner walks us through Altium's educational formats, from YouTube videos to trade shows. And John Coonrod points out the many benefits of having a wide-ranging knowledge of engineering and fabrication processes.

Speaking of DesignCon, we'll be bringing you coverage of this venerable show, which runs from August 16-18. Editor Nolan Johnson and I are looking forward to getting back to covering trade shows and conferences! I hope to see you there. **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.

Design Education and Training for Today and Tomorrow

Feature Interview by the I-Connect007 Editorial Team

PCB design and design engineering are evolving constantly, and design education and training must also evolve to meet the needs of tomorrow's young designers—yes, there are young designers again.

The I-Connect007 Editorial Team recently spoke about this issue with design instructor Susy Webb, CID. Susy meets many of the new designers through her various PCB design classes at PCB West and other PCB design conferences.

We asked Susy to discuss how design and training have changed in the last few decades, where design education is headed, and how today's crop of new designers differs from the "graybeards" who are planning their retirements. She also recommends design education resources from a variety of media formats.

Andy Shaughnessy: Nice to talk to you again, Susy. Would you share your training background?

Susy Webb: I was brought into this field by a friend of mine. I didn't have any engineering background at that time, but you really didn't need it then. What you needed was the manual dexterity, a lot of patience, and the ability to basically lay down parts and connect the dots; that's pretty much what we did 35–40 years ago.

But things have changed so dramatically over the years to where currently you need to know a lot of physics, signal integrity, power integrity, EMI and DFM, organizational skills, and good communication, you name it. And now, since we who design boards are like the cog in the wheel, we need to know basically everything. Not only do we need to know it, but we need to know why it is important, so that we can choose what priority or what focus we are giving the particular board that we're designing. Because sometimes things compete, like power integrity and signal integrity, or the SI and the EMI, you need to know which one to give the highest priority to as you're designing each board, because each one is different.

EMC

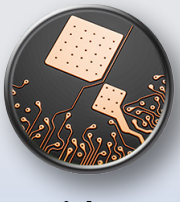
STRIVE FOR **EXCELLENCE**

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

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

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

World's largest producer of environmentally-friendly advanced laminate materials



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

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

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

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

TECHNICA, U.S.A.

North American Master Distributor
1-800-909-8697 • www.technica.com

Susy Webb

I started doing basics design classes in 2004. I've been doing this for 17 years. My basics class has changed dramatically over those years; in the beginning, it was, "This is a component symbol, and this is a PCB symbol." It was to teach people who were green to the business about how to do it. But now, so many of the people coming in not only want to know the basics, but they also want to know how everything else works in the design, and so my classes have changed to cover that kind of information as well.

Shaughnessy: Right. It does seem like we're asking designers to do a lot more of what used to be in the electrical engineers' domain.

Webb: And vice versa. I was brought in by the friend who just recognized that I had some ability to do something like this, and we did it on Mylar with black tape. But then EDA computer software came along, and it required a whole new set of skills. Time has also brought in the need to understand and implement many more engineering needs like how to work with impedance control, signal and power integrity, and EMI control.

Shaughnessy: What are some of the other changes that you've noticed in your classes? When the attendees come in, they're dealing with a whole different skill set than 15 years ago, right?

Webb: Fifteen years ago, the classes were filled with technicians or others who wanted to try their hand at design but didn't really understand the ins and outs of the profession. Or it was somebody who had an idea that they would like to learn about it all but didn't really know where to start. But now, I stand in front of the class and ask, "Who is an engineer?" And most of them are. A few are mechanical, assembly, or some other discipline along those lines, but most are electrical engineers, and their companies are asking them to design boards.

When that started being a trend, the engineers really didn't have any place to learn about the ins and outs of the job and who you had to please, which is a really important part of the job. You have to please the DFM people, the testing people, the people who may use the board in the field, and you have to please the other electrical engineers on your team. It's very important that as many of their issues as possible are addressed for design so that the design is easy to build, test, and use. We can't always please all of them, because sometimes things conflict, but the more you can design to good standards and meet as many of the other people's needs as possible, the better the board or project will turn out. Unfortunately, it is very easy to design a board that is not manufacturable, with an exorbitant cost, and that does not work properly. Since we don't want to waste our time, it is important to understand the physics and electrical needs of the project and implement them into the board as we design it.

Shaughnessy: Years ago, speeds were slower, so you could be close and it would still work.

Webb: Right. Things were so much slower back then. You could wander a signal all over the

board and it really wouldn't make a difference. But that hasn't been the case for more than 20 years. It's increasingly more of an issue as time goes on, because electronics are changing dramatically, and if you're not keeping up, you're falling behind. You just can't rest on your laurels; you have to continue to learn.

Shaughnessy: Today's designers also need to know some of these so-called soft skills, like communication and problem-solving. Happy Holden has written about the need for engineers to develop their problem-solving skills. Does anybody teach communication for designers?

Happy Holden: Yes. HP sent me to a three-day class on communication that's still taught today. The class gave me tools, techniques, and methodologies that were much more sophisticated and complete than I had thought about.

Webb: You absolutely must be able to problem-solve and make decisions in the engineering field, and there's not always somebody there to help answer questions. There are thousands of questions and decisions to be made throughout the process, so it is very important to keep learning and understanding the information that you're working with. Hopefully, there are still some mentors out there, people who are ready to help new designers. People contact me all the time on LinkedIn and say, "I have a question..." and I am happy to help as I know how hard it can be to put all the pieces together. Hopefully, young designers can still find someone who understands these concepts and be able to trust them about the pros and cons of your design.

I not only talk about the engineering concepts when I'm teaching, but I like to show tons of pictures in my classes because it's important to have the attendees see what I'm talking about, since I can't design a board live for them. I try to put in examples; if you orient the part this way, you'll get this result, but if you orient the part that way, you might get much better results, because of this or that. It's important to explain "why." I cover as much as possible, from the engineering and manufacturing concepts to the practical implementation of them.

Shaughnessy: Some designers have predicted that in the future, each designer is going to have to be a specialist instead of a generalist. What do you think?

Webb: I am not sure what they are specifically talking about. People are learning more about things like physics, how to do simulation, the concerns that are needed for design-for-manufacturability, or how to create good library parts and maintain a good library management system. But in the jobs that I have seen on LinkedIn or others, the listings require more of a generalist. They tend to want everything from the person they are trying to hire. They want him or her to be an engineer, a scientist, and a designer, while understanding DFM.

They tend to want everything from the person they are trying to hire.

So, even if a person is going to focus on a specific area of expertise, I think they really must understand all the areas, because as a board designer, you have to understand how everything affects everything else.

Holden: Yes. There are a lot of common skills, but people who become experts in HDI might not be experts on flex or rigid-flex.

Webb: Exactly. As a general designer, you must understand HDI and materials—not to the same level that somebody who specializes in

that, but they certainly must understand all the different areas that are affected by the board design. There are so many people who don't understand HDI. Happy taught classes on HDI for years, and yet I still get questions and comments from people who are doing things that make the board either unmanufacturable, or so expensive it's crazy. We have to keep up with technology.

The problem is that many new designers—engineers, for example—still look at designing as connecting the dots. They don't understand all the other people who are involved, and so they don't know what they don't know. Somebody's got to teach them about the fast rise time signals, the physics, the processes, and the ways to orient things that are placed and then laid out. All those kinds of things are so important to make the board work right, to not have noise, and to make it manufacturable, with good documentation when they send it out.

The problem is that many new designers—engineers, for example—still look at designing as connecting the dots.

Shaughnessy: You've watched the growth of virtual classes and conferences over the past year during the pandemic. Do you think the virtual format is here to stay?

Webb: I think the people whose companies can't afford to send them to the live conferences like the virtual format, but I think everybody should go to live conferences, if they can.

In person, you can be wowed by the interactions with people that have the same goals in mind, you can talk to people about a specific issue, and you can share thoughts and encouragement to people. You can find people that you can ask questions of after you go home in a mentor-type situation. And so, the interaction between people is one of the great things about going to the conferences and meetings.

Even in my classes, somebody will ask me a question and I will try to answer the question to the best of my ability. Somebody else will chime in with a possibility, and then somebody else will do the same. There's a lot of give and take at the conferences that you just can't get in virtual format.

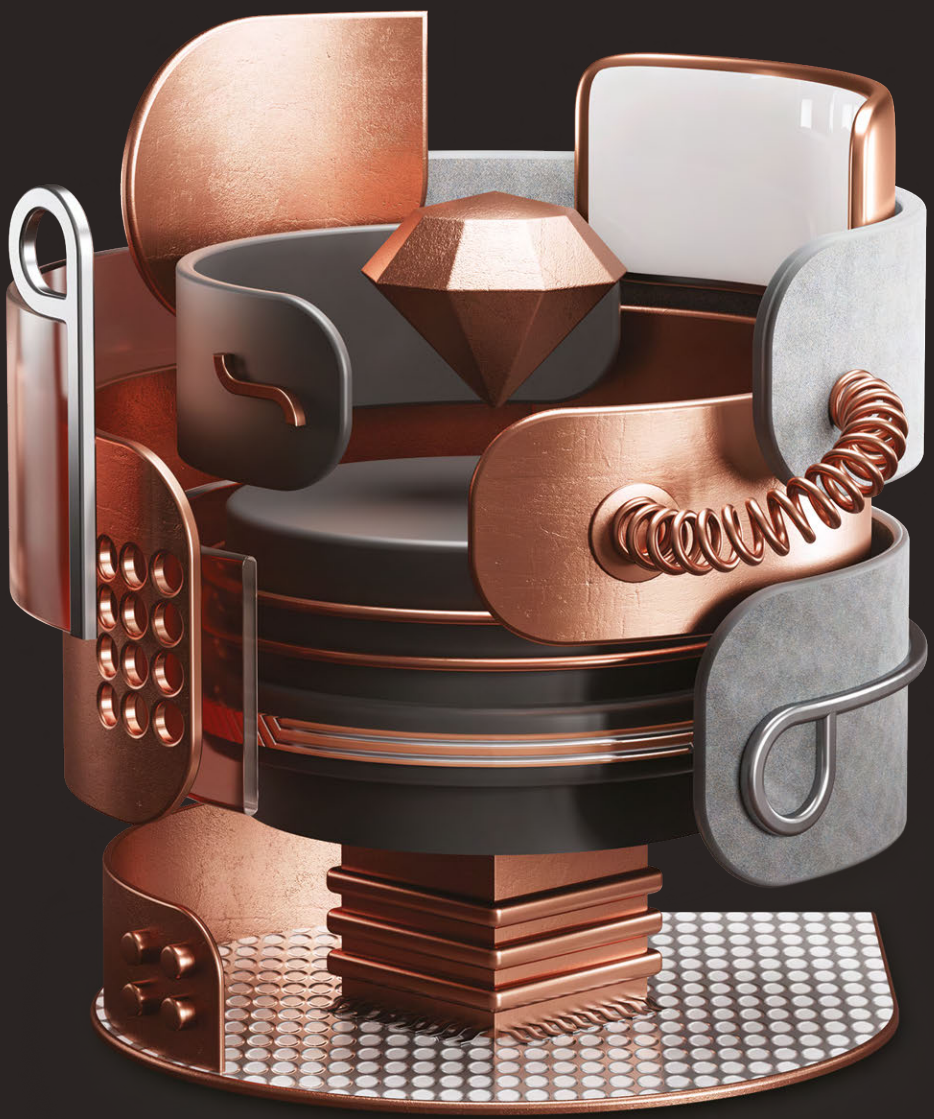
Nolan Johnson: If this continues and we use an in-person format as well as the ability to join in virtually, it creates two tiers of involvement, doesn't it?

Webb: Yes. I think that live, you just get a better interaction as a teacher. As a presenter, I like it, because I can look out at their faces, see if they're understanding what I'm talking about, and we have interaction through questions on the presentations. But when you are dealing with people virtually, they don't necessarily step up and ask questions like they do in a classroom setting. There's just not as much interaction with online classes.

Shaughnessy: Sure. When your class is over, what educational resources do you recommend for your attendees?

Webb: I am a big proponent of reading books. I used to be concerned about that, because the industry is changing so much, but it really depends on the topic. When a book covers physics, it's not the physics that are changing. It might be the speed of the signal that's changing, but the principles discussed in the book are still very much adequate for what we need to do today. So, I ask attendees to list the design books and articles that they read. Then I suggest that they read books and articles by authors who are instructors at the conferences,

WE DO IT ALL ...

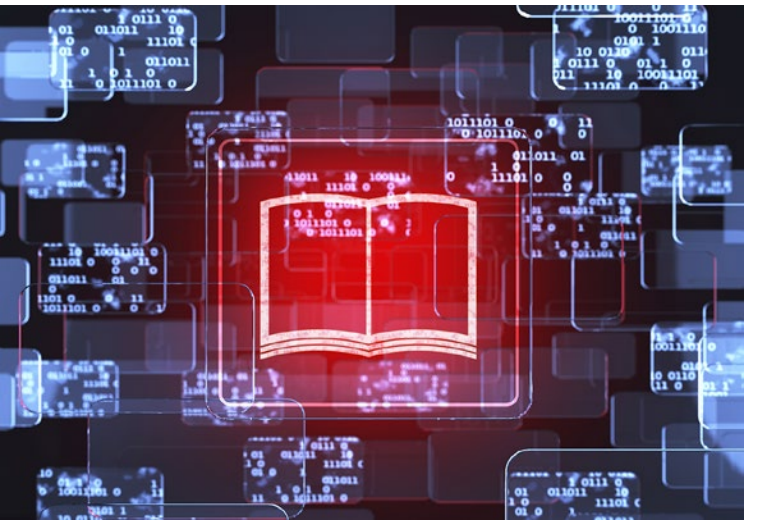


We offer an all-in-one solution through our experience, integrity, and clear communication. Get on board for our joint journey into the future.

www.PCB-technologies.com

ALL IN ONE

PCB TECHNOLOGIES™
leading the way



like Eric Bogatin, Lee Ritchey, Rick Hartley, or Happy Holden.

Anyone can write an article, but that doesn't mean they are correct. You need authors who you trust and ask questions when you can. But there are many educational opportunities. The PCEA (Printed Circuit Engineering Association) has technical meetings, and they're online too. There's a lot of information posted on LinkedIn. And I take a special interest in online articles written by those I know and trust, such as the ones I just mentioned.

Shaughnessy: What do you think about YouTube? I've heard designers say, "I've learned some tips on YouTube, but you have to really look for it."

Webb: I actually have as well. Sometimes I will be figuring out a software issue and I'll go to Google, which leads me to a YouTube video about how to do what I need to do. There's so much available now for designers that wasn't available when I first started. We needed mentors in the past so we could turn to them and say, "I can't figure this out, can you help?

Holden: You know, Susy, if you go to Google, you'll only get 10% of what's available about circuits.

Webb: Yes, of course.

Holden: You're missing 90% if you don't use specialized software or techniques. That's why I make it an entire chapter in my book. Everybody thinks Yahoo and Google have access to everything. No. Yahoo and Google have access to everything about Hollywood stars, but for circuit board design, they only have 10% of the information, and it's a very elaborate procedure to get to the other 90%.

Webb: That's right. And there are some things on YouTube, as I said. But I definitely think you need to read all kinds of articles and books. The PCEA has a wonderful listing of books and articles that we have vetted as correct information. And there are classes that are sponsored by software companies, PCEA, and IPC.

At PCEA, we're developing a weeklong class. The textbook is written by the PCEA EDU group, which consists of myself, Rick Hartley, Gary Ferrari, Mike Creeden, and Steph Chavez. This book is being used as training material through EPTEC, a company that does training in different areas around PCB design and manufacturing. We're trying to fill that mentor void by having reference document information, plus the in-person teachings that would help a person understand it all.

Shaughnessy: A few days ago, Happy summed it up: Good PCB design information is very specific, but very hard to find. Is that the case in your view?

Webb: Yes, it is. That's the main reason you should go to somebody you trust; if you don't trust anyone, you need to start attending conferences. I know that's difficult, because some companies will not send people to conferences. I paid my own way to conferences for years, because my company wouldn't pay, and I wanted to learn more. But it does pay off in the long run, because learning all that information paid off for my job satisfaction, my job titles, and my paycheck.

I believe the best thing that a person can do is go to every bit of training that they can get. Everybody who teaches a class about HDI or DFM, for example, is going to approach it from a different angle, so don't just take one class on a certain topic.

Shaughnessy: Speaking of DFM, we've had this ongoing discussion about whether designers need to be experts in fabrication. What do you think?

Webb: I think designers must understand manufacturing. You don't have to be an expert at it, but you have to understand what they need, how they need it, and why they need it, because if you understand the why, then you can understand the reasoning better, especially when things conflict. A person needs to know a lot about DFM and have experts they can call on when they need help.

I teach a class called "Building a Bridge from Design to Manufacturing" because designers don't necessarily understand what the manufacturers need, nor do the manufacturers understand that what they're asking for is difficult from the design perspective. We have to understand each other, and that manufacturers are not just asking for something because it makes their job easy, but rather because it really is a requirement for them to produce the board. By the same token, they must understand why it can be difficult for us to accomplish what they want.

Shaughnessy: What would you say are some of the must-haves as far as training, education, and certifications? What do you think every designer should have under their belt?

Webb: The CID is changing somewhat, and I am not clear what all those changes are. There is a new training class and certification from the PCEA. The great thing about the PCEA class is that the information was written by longtime designers and speakers who really understand the information. To me, the most important thing is just to go for training, and that means software training, conferences, online classes, etc. LinkedIn, as I mentioned earlier, posts a lot of good articles.

One must be hungry to understand where the business is and where it's going, in my opinion. If that means spending time, effort, or money for the designer to find the path for more knowledge, then that's what they need to do. It does not have to cost personal money. If they approach management with a written "sales pitch" about how more training will benefit the entire company because the boards will be designed correctly the first time, many times their company will agree to pay.

One must be hungry to understand where the business is and where it's going, in my opinion.

Engineers understand the electronics involved in designing a circuit on a schematic, but they may not understand fully what's happening inside the board. For example, I can't tell you how many times I've explained energy fields to an engineer, and they say, "No, no, the signal is in the trace." Actually, the signal energy is in the energy fields that surround the trace and if you don't understand how to design and control those energy fields as the signal travels through the layers of the board, the signal and the board may not work properly.

Another example of misunderstanding is EMI. I can't tell you how often I've heard, "We don't have to pass EMI testing and regulations, so we don't need to know about EMI." Well, yes, they do, because EMI is also crosstalk, interference, or inadvertent antennas which are caused by not controlling the higher-fre-

quency signals we are working with just by using today's parts.

To me, it's very important that you learn to discern which information is right for the particular board the designer is working on. For example, datasheets will tell you you're supposed to do A, B, and C to design the part onto the board. Many engineers have told me, "The datasheets say A, B, and C, so it must be right." In reality, the information might not be right on this board. When the designer is just relying on the datasheet or the application note they are not getting the whole picture. They're getting the part of the picture that the manufacturer made work in one particular setting, on one particular board, with one particular stackup. The designer's board, stackup, and everything else may be very different.

To me, it's very important that you learn to discern which information is right for the particular board the designer is working on.

It reminds me of when you see a software demonstration; the software always works beautifully when they demonstrate, because they have it set up perfectly. But when you sit down and try to do the same thing, it doesn't always work well.

Shaughnessy: Right. What do you think about specific CAD tool training from the EDA companies?

Webb: Some of them have conferences and classes, and they try to teach things above and beyond the basics of their tool, and that's a very good thing. Everybody needs to understand the basic information. But if you can go to training where they teach about their tool and some more advanced information as well, then you've gotten two good things in one because you understand why to do things a certain way.

As an example, we all know that software companies push their autorouters. I have a friend who has worked with four or five different companies as part of their autorouter development teams. I recently asked him what he thought about autorouters in general, and he said, "They're probably not a good way to design a board." He said they are not able to keep up with current high-speed design technology, so they are always doing a catch-up kind of work.

I have always felt that way, but I was surprised someone who was in the development of those tools said that as well. I put this information into one of my classes, because I know that there are people who want to plug the board into an autorouter and come back the next day and it's finished. Routers don't understand about return current and they don't know about where the energy of the signal really flows from layer to layer in the stackup. Those things need to be controlled by the designer. In my opinion, autorouters can't do nearly as good a job on complicated boards as the experienced designers can.

Shaughnessy: Right.

Webb: You asked about four-year colleges, and their education curriculum for design. There is currently very little about PCB design in most four-year engineering degree programs. The PCEA is working with some colleges to develop a semester-long course about PCB design to help engineers understand what is needed after they graduate. It is a work in progress, so definitely watch for that. In the meantime, there is the one-week class the PCEA group of designers is doing with EPTAC. There are also people who teach one-, two-, and three-day classes about PCB design as well. We are not trying to be all things to all people, certainly, but we're trying to be the mentor that isn't there anymore.

Holden: I give free online lectures on PCB design at Michigan Tech. It's a semester course. You're welcome to attend. Just email Marc Carter. He's been covering the development of PCB courses at MTU and elsewhere in his PCB007 column.

Webb: I would be interested in attending those classes. I'm still working full-time, so I don't know how often I can attend them, but thank you.

Shaughnessy: Susy, do you have any advice for a new designer starting out? As they're leaving your class, what do you tell them?

Webb: For the seasoned designer, be sure that you keep your mind open to many possibilities, because you may not always be with your current company, and other companies may do things very differently. When/if you go someplace else, you may be required to do other things, and it's important to be well-rounded. It helps a person be more valuable and land that new job. The reason I bring that up is because I've interviewed designers who had been laid off, but they felt that the way things were done at their previous company was the only way to do things, and they did not want to change that. I believe an open mind is a good tool to have.

For the new designer, or the engineer designing boards, I would say to develop a passion for the profession so that their natural inquisitiveness leads them to always want to understand more.

Shaughnessy: Beware tribal knowledge, right?

Webb: Yes. Many companies have rigid processes that work for them, but other skills may be needed at another company. There are thousands of ways to do these things, so a designer needs to be flexible and be able to think for themselves and make decisions within the confines of the company processes.

Shaughnessy: Great. This has been really good. Thank you for talking with us. I hope we can get together in person this year.

Webb: Thank you, Andy. I enjoyed it. **DESIGN007**

To contact **Marc Carter** about his work with PCB curriculum at Michigan Tech, click here.

Susy Webb is a senior PCB designer at Design Science PCB in the greater Houston area.

Fly-over Technology—When It All Gets Too Fast

Beyond Design

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

As frequency and rise times increase, SerDes design is becoming more critical. Losses caused by the skin effect, fiber weave effect, surface roughness, vias, and connectors are often too excessive for reliable communications. PCB dielectric materials such as AGC (Nelco) N4000-13SI, Isola I-Speed and Panasonic Megtron 6 are reasonably priced, low loss, temperature stable, moderately flat weave laminates applicable to the 2, 4 and 10 Gbps channels. However, for such applications as PCIe-Gen 4 (16Gbps) and higher, one may need to consider alternatives for stackup construction to meet the higher performance requirements.

So far, we have been able to deal with fiber weave skew, using a variety of physical layout and laminate construction techniques, while still using the more common existing PCB materials. Isola GigaSync material, for instance, was developed in 2013 to specifically reduce fiber skew, but unfortunately its dissipation factor (Df) and dielectric constant (Dk) are too high for today's serial channels. These homogenous materials try to match the Dk of the polymer to that of the glass, with some boundary modification also being performed where the polymer and fibers contact. This effectively eliminates glass weave skew. Unfortunately, glass is still fairly high Dk when compared to the newer polymers that come in the 2.5-2.8 range. A better alternative is to either use hybrid stackups with mixed woven and non-woven materials to lower the cost or use spread-glass material. Isola I-Speed MS, for instance, is mechanically spread glass, expanded in both the warp and fill direction, which is an improvement especially if two-ply construction is employed.

But as frequency continues to soar, cutting edge interconnects running signals at 28 Gbps

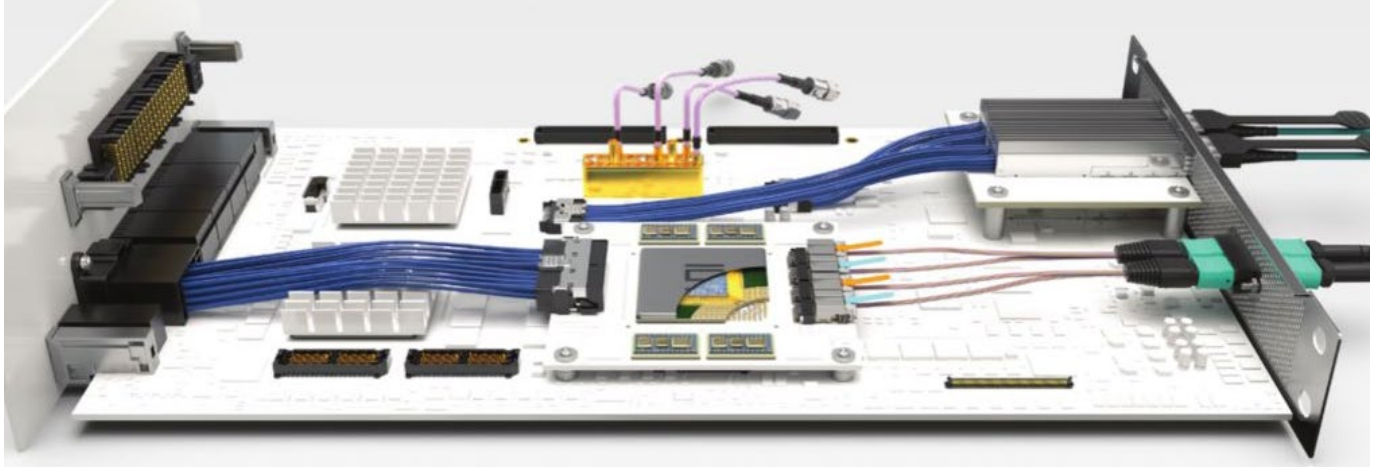
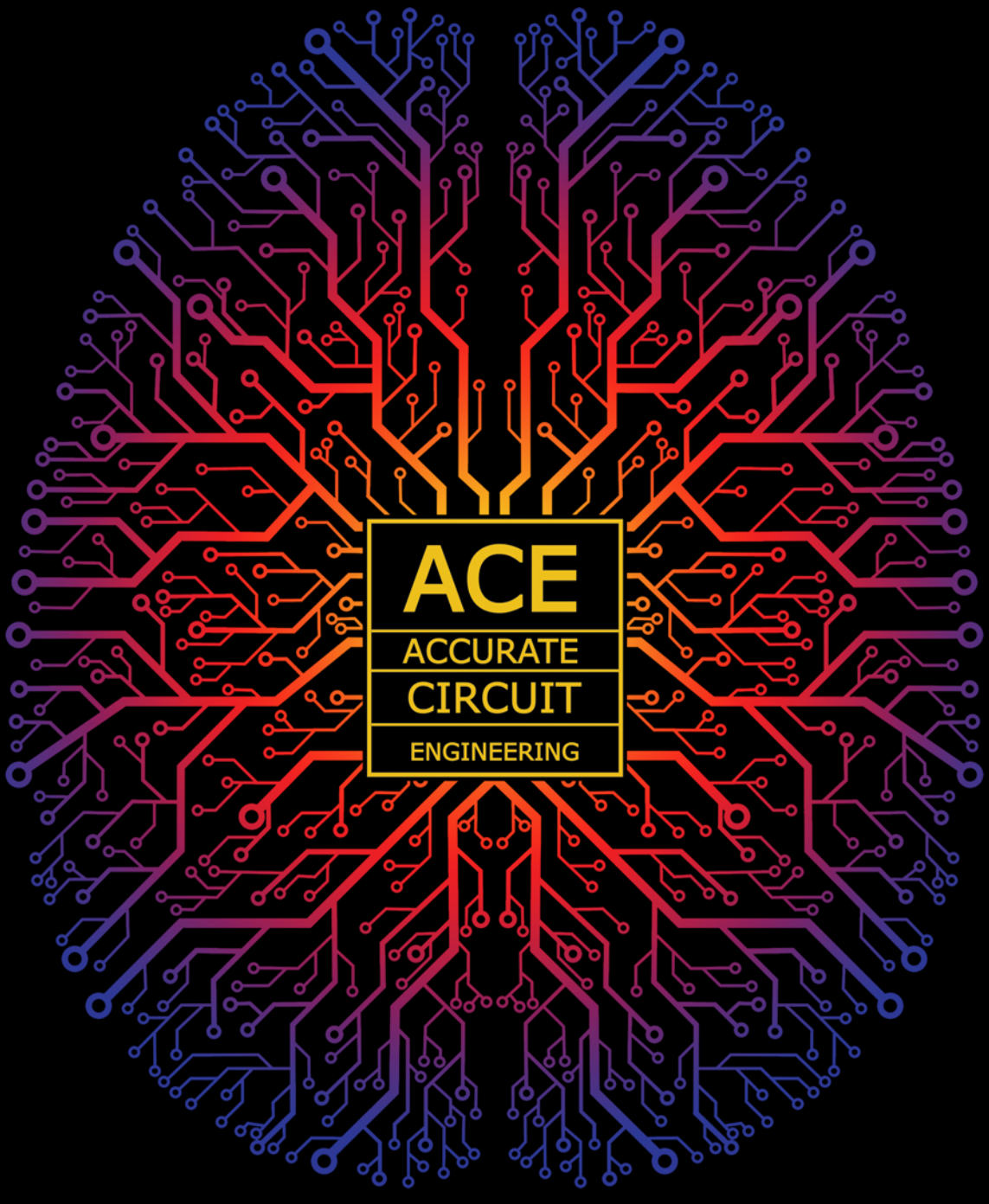


Figure 1: Twinax Cable Flyover™ Technology. (Source: Samtec Inc.)

Don't Overthink Your Prototype Builds



Go with the team that can make your ideas a reality...

RF • MICROWAVE • HIGH-SPEED • QUICK TURN EXPERTS

ACE
ACCURATE
CIRCUIT
ENGINEERING

Accurate Circuit Engineering
3019 Kilson Dr
Santa Ana, CA 92707

(714) 546-2162 • sales@ace-pcb.com

For technical assistance, call:
James Hofer
(714) 227-9540

www.ace-pcb.com

	FR-4	Megtron 6	Twinax	Optics
10 Gbps	< 250	> 250	> 250	> 250
14 Gbps	< 125	< 250	> 250	> 250
28 Gbps	< 50	< 125	> 250	> 250
56 Gbps		< 50	> 250	> 250
112 Gbps			< 250	> 250

Table 1: Maximum trace length (mm) vs. bandwidth (Gbps).

and beyond require a rethink (Table 1). Or, in the case of Samtec's solution (Figure 1), a complete rerouting of the signals using Flyover™ cables that remove the signals off the backplane entirely, from chip to off-system connector. They have developed cables and connectors specifically to handle high-speed signals that can pass above and across the board. Using cables for 56 Gbps signals cuts losses by about half compared to PCB traces. This saves PCB cost by reducing the layer count, the material and specification requirements while adding flexibility to high-speed interconnects.

The role of fly-over cables is to isolate signals from the limitations of the PCB materials. As signal speeds increase, the dielectric material's Df and Dk become an issue, and traces need to be shaped and routed perfectly, without skew, to avoid signal coupling, crosstalk, and electromagnetic compliance (EMC) issues. Fly-over cables are ideal for 28, 56 and 112 Gbps data rate serial links.

Recently, the shift from non-return-to-zero (NRZ) to pulse amplitude modulation 4-level (PAM4) encoding for leading-edge server backplanes has made it extremely difficult to meet jitter and noise requirements over any useful length of board, despite enormous advances in channel characterization and equalization. NRZ is a modulation technique that has two voltage levels to represent logic 0 and logic 1. While PAM4 uses four voltage levels to represent four combinations of 2-bit logic–11, 10, 01, and 00 (Figures 2 and 3).

The goal of these protocols is to transmit data efficiently over co-ax, fiber, or PCB interconnects, but each uses a different method and has its benefits and drawbacks. The well-established NRZ is good for short distance runs, has a throughput of 1 bit per unit interval (UI), minimizes current change, and has a signal and noise ratio (SNR) of 0 dB. On the other hand, PAM4 can transport twice the signal of NRZ (throughput of 2 bits per UI) because it operates on four levels (Figure 3). But this makes reflections three times worse than NRZ, resulting in an SNR of -9.54 dB. Unfortunately, lower-loss cables do not dampen reflections as well as those with higher loss. The increased reflections raise the noise floor, which is critical to PAM4 encoding, especially on shorter cable lengths. However,

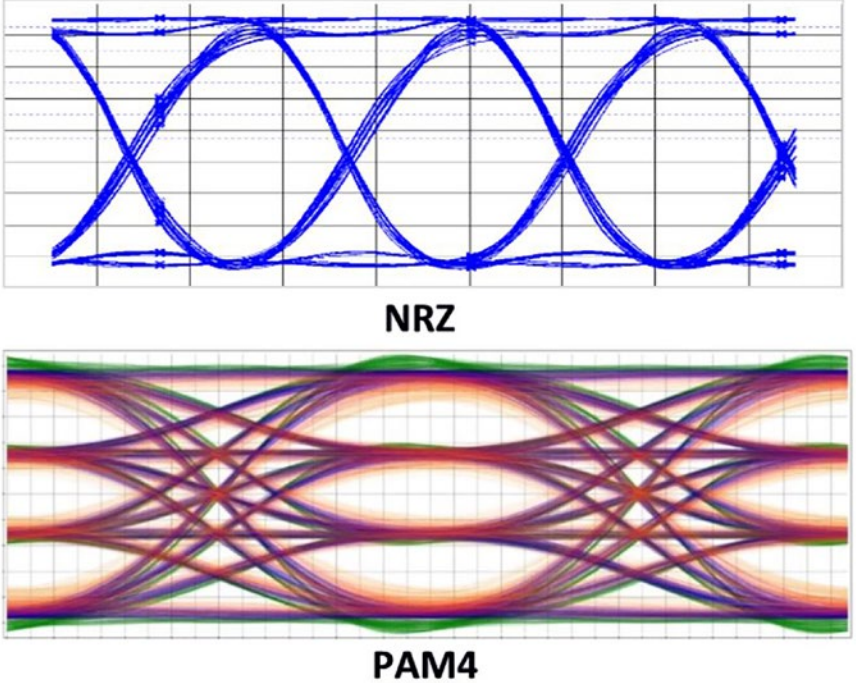


Figure 2: NRZ eye vs. PAM4 eye (Source: Xilinx).

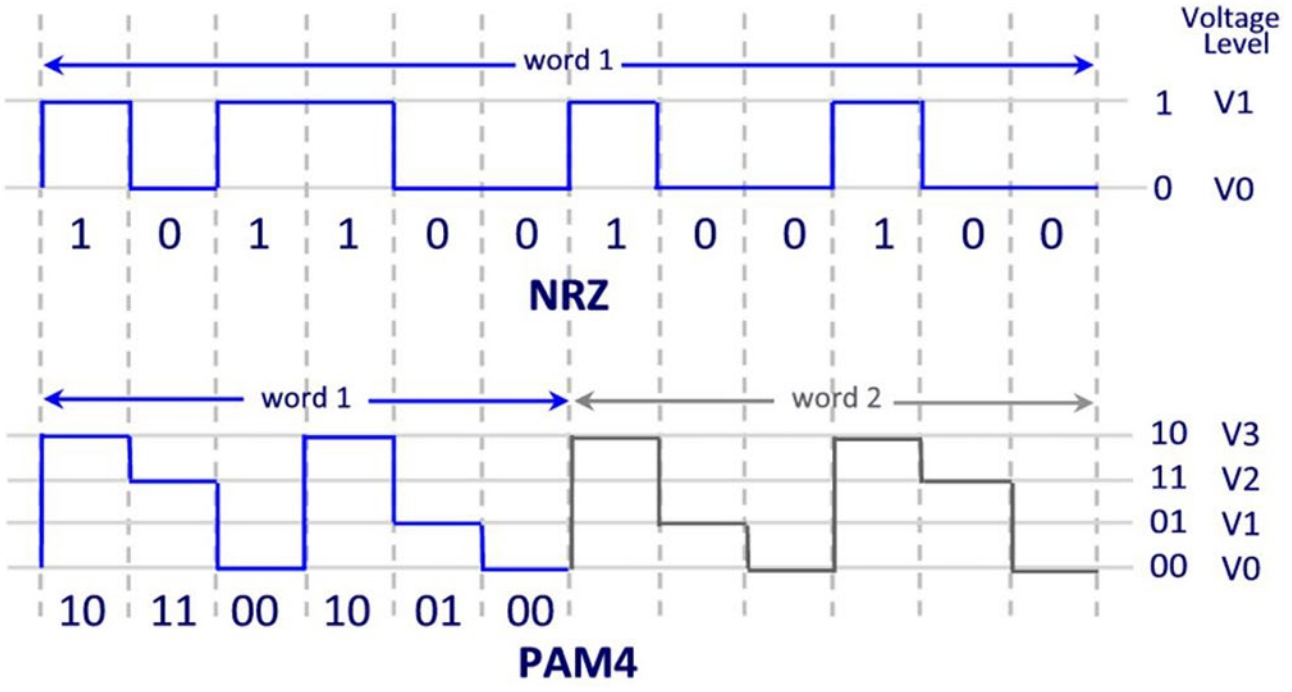


Figure 3: NRZ vs. PAM4 encoding.

NRZ's higher Nyquist frequency (the highest frequency that can be coded at a given sampling rate in order to be able to fully reconstruct the signal) results in higher channel-dependent loss, making PAM4 a more viable solution for high-frequency serial communications.

Using ultra-low skew twin-ax cable to route signals over the PCB is a key performance enabler as signal range and integrity requirements continue to become ever more important in high-speed applications. Fly-over technology provides performance and cost advantages compared to lossy PCBs, with up to 112 Gbps performance at 150 mm cable length. Fly-over also allows designers to go from one board to another as a flexible backplane architecture within a rack, as well as being used as a rack-to-rack interconnect. The traditional backplane architecture is retained, but the system offers lower PCB complexity while achieving higher performance targets.

However, with the growth of 5G data traffic and AI computing, there is a need for faster connectivity to meet the increasing bandwidth. Consequently, serial speed beyond 112 Gbps per lane is now required. If we follow the SerDes technology progression, by doubling the data rate per lane every two years, the next generation I/O data rate will be 224 Gbps. With the signaling rate increasing, electrical channels like PCB traces or copper cable both have bandwidth limitations over certain reach distances. Alternatively, optical fiber cables can be used to transmit high bandwidth data over both short and long distances. Also, there are new emerging technologies such as chiplet and co-packaging optics (CPO) where most of the channel operates in the optical domain.

Key Points

- Applications such as PCIe-Gen 4 (16 Gbps) and higher may need high-performance materials for stackup construction.
- Fiber weave skew has been dealt with using a variety of physical layout and laminate construction techniques while still using the more common existing PCB materials.

- Isola I-Speed MS, for instance, is mechanical spread glass that is an improvement especially if 2-ply construction is employed.
- Fly-over cables remove the signals off the backplane entirely, from chip to off-system connector.
- Using cables for 56 Gbps signals cuts losses by about half compared to PCB traces.
- The role of fly-over cables is to isolate signals from the limitations of the PCB materials.
- Fly-over cables are ideal for 28, 56 and 112 Gbps data rate serial links.
- The shift from NRZ to PAM4 modulation encoding has made it extremely difficult to meet jitter and noise requirements over any useful length of board.
- PAM4 is a more viable solution for high-frequency serial communications.
- Fly-over technology provides performance and cost advantages compared to lossy PCBs, with up to 112 Gbps performance at 150 mm cable length.
- With the signaling rate increasing, electrical channels like PCB traces or copper cable both have bandwidth limitations.
- Optical fiber cables can be used to transmit high bandwidth data over both short and long distances.

Resources

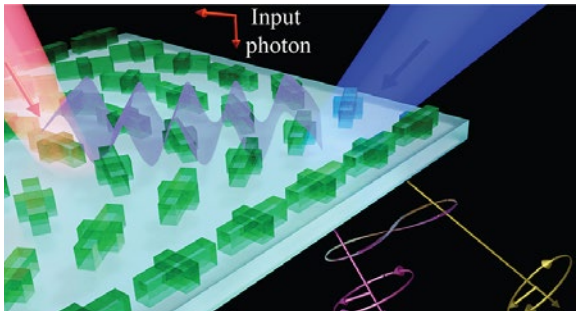
1. Twinax Flyover Cable Systems | System Optimization | Samtec
2. "Flyover Cables: Inevitable, but Not Easy," by Patrick Mannion
3. "Understanding NRZ and PAM4 Signaling," by Brian Niehoff
4. SI List, by Scott McMorrow, Samtec

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

New Quantum Research Gives Insights Into Mastering Quantum Light

A team of scientists at Los Alamos National Laboratory propose that modulated quantum metasurfaces can control all properties of photonic qubits, a breakthrough that could impact the fields of quantum information, communications, sensing and imaging, as well as energy and momentum harvesting. The results of their study were released in the journal *Physical Review Letters,* published by the American Physical Society.

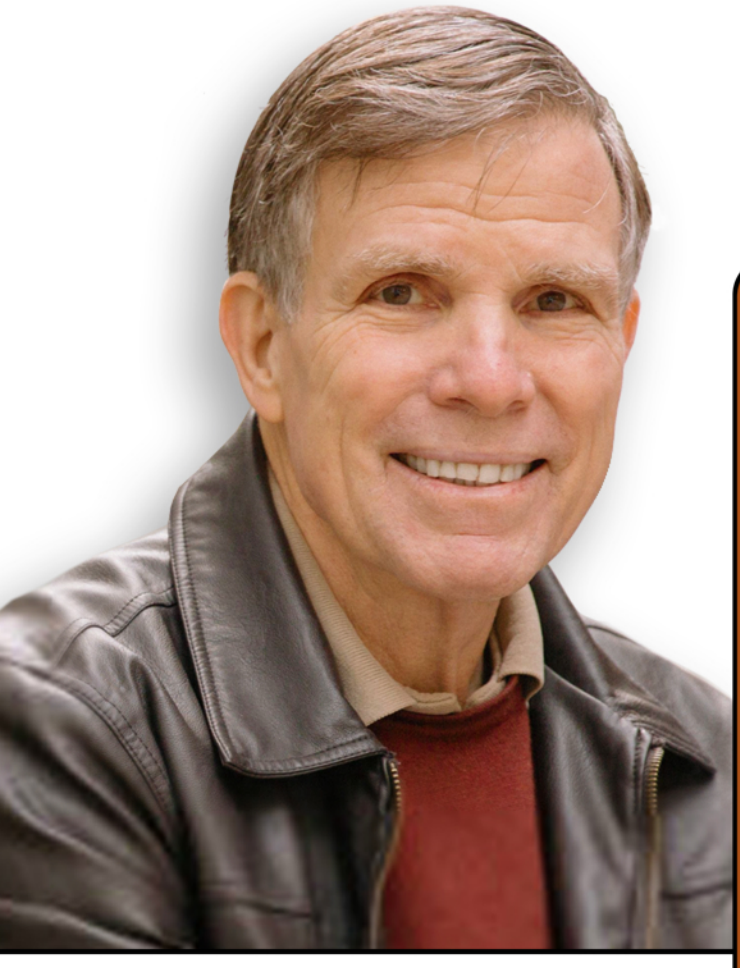
Metasurfaces are ultrathin structures that can manipulate light in ways not usually seen in nature. In this case, the team developed a metasurface that looked like an array of rotated crosses, which they can then manipulate with lasers or electrical pulses. They then proposed to shoot a single photon through the metasurface, where the photon splits into a superposition of many colors, paths, and spinning states that are all intertwined, generating so-called quantum entanglement—meaning the single photon is capable of inheriting all these different properties at once.



Harnessing photons that exist in the vacuum and shooting them in one direction should create propulsion in the opposite direction. Similarly, stirring the vacuum should create rotational motion from the twisted photons. Structured quantum light could then one day be used to generate mechanical thrust, using only tiny amounts of energy to drive the metasurface.

(Source: Los Alamos National Laboratory)

I-007e WORKSHOP



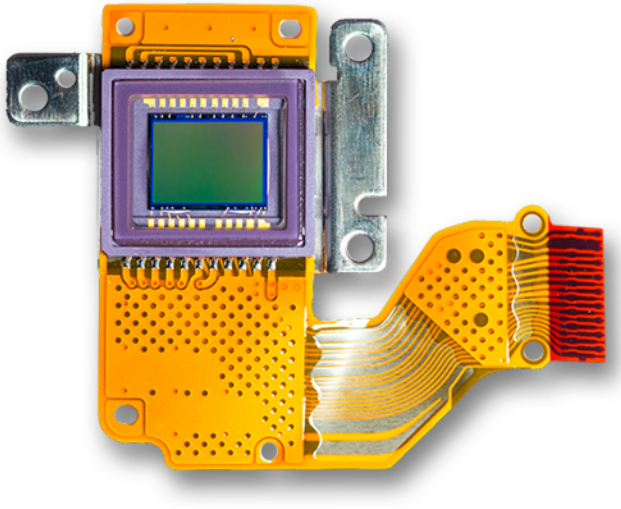
Online Training Workshop Series:

Flexible Circuit Technology

with Joe Fjelstad

This free on-demand workshop series is a comprehensive look into the structures, applications, materials, and manufacturing processes of flexible printed circuits.

START WATCHING



PRESENTED BY

I-Connect007
GOOD FOR THE INDUSTRY

SPONSORED BY

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

Not Just a Roll of the Dice

My Road to Becoming a Great Designer


Tomas Chester

Feature Article by Tomas Chester
CHESTER ELECTRONIC DESIGN

It was halfway into 2009, I had recently graduated from the University of Guelph with a bachelor of engineering in systems and computing, and the options for employment were slim. Unfortunately, I had entered the workforce during the very tail end of the 2008 financial crisis and was having difficulty finding employment in any field. It would be the better part of a year before I was able to start in the first position in my field as a junior electronics engineer. My goal had been to simply get in the door. I knew that experience and simply getting in would allow me to start my career with direction and purpose.

University had taught me the basics of electrical circuits, and more importantly it had provided me with skills to work within a team and clearly identify issues requiring attention. With all the prior electronics engineers and designers having left the company, it fell to me to decipher the state of various projects and forge ahead with completing the design and verifying the hardware result.

With limited knowledge of how to create a PCB, I had to seek out opportunities to develop important skills such as layer stackups, trace widths and spacing, and design assembly, while at the same time trying to navigate a design tool I had only just learned existed. My co-workers were in similar situations, learning software, understanding complex data conversions, and trying to decipher their own tangled webs. However, working as a team allowed us to assist one another in finding innovative solutions. It was here I learned one of the most important lessons of my career: you need to support and be supported by a great team.

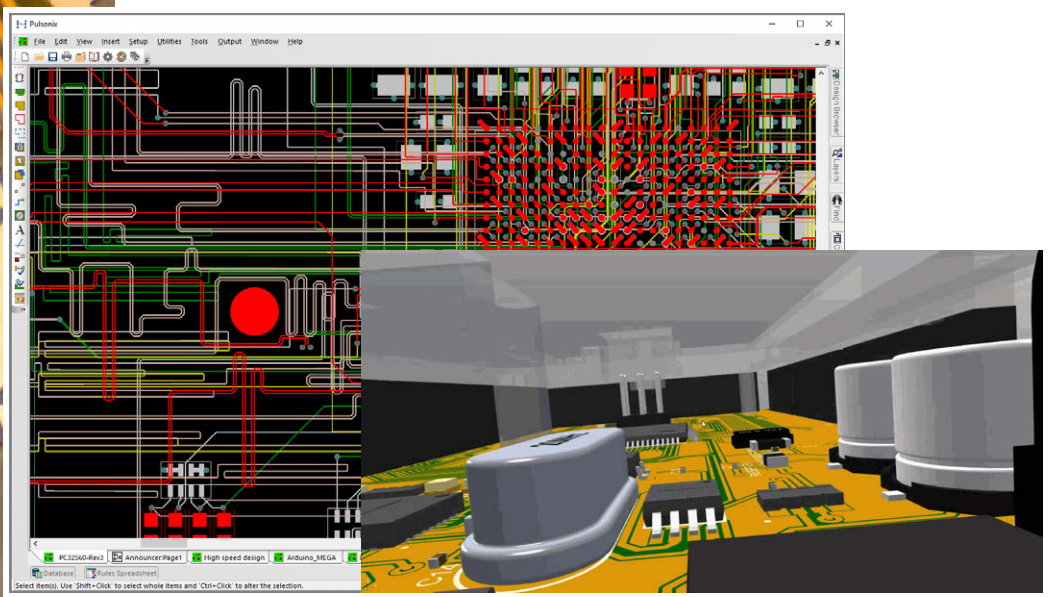
As the small successes mounted, a major project loomed on the horizon. The existing system architecture was not designed for the anticipated data throughput that was going to be required. With a young and relatively inexperienced team, we considered consulting contractors and external design bureaus. It was at this point that I expressed interest in running the project internally. This would be a big task with significant risks; however the benefit would be full control of the project, an increased understanding of our architecture,

Pulsonix PCB Design Software

PCB design done right

New Version 11.0 Increases Speed up to 80%

Multi-threading and GPU processing in this new release create a gigantic leap in speed, dramatically reducing the time it takes to perform many PCB design operations, such as Design Rule Checking (DRC), Copper Pouring, and Gerber generation. That's PCB done right!



Advanced PCB Design Features

- Intuitive placement and routing
- Dynamic copper pour
- Full rules engine for automatic DRC
- High-speed design rules
- Heads-up display length matching
- Rigid-Flex, embedded components, and chip-on-board
- Revision control and PLM integration

Try Pulsonix yourself at pulsonix.com/trial

PULSONIX

and a growth in design experience for a young team.

The decision was made to keep the project internal, and I was given the opportunity to increase my knowledge of PCB design, as well as the ECAD tool that would make it possible. Slowly, the team grew with new members bringing varied experiences and innovative ideas to the table. This provided fresh perspectives on the use of our software tool, as well as general and advanced design concepts. This was vital to my professional development, as each designer has a unique method of routing and solving design problems. Spending a morning or afternoon watching how someone else uses the same design tool you do can show you new shortcuts or design methods for resolving obstacles.

Spending a morning or afternoon watching how someone else uses the same design tool you do can show you new shortcuts or design methods for resolving obstacles.

Sadly, all great things come to an end, and decisions made from on high meant that our Canadian research and development team was restructured, which meant that my position was no longer required. Thankfully, I now had nearly three years of experience to assist me with finding my next position. At this point, I reached out to the network of industry contacts I had developed through the various projects from my career thus far. This included looking into the companies we had considered for the system architecture project. I reached out to one of those companies and was invited for an interview.

When they offered me a position, I decided to accept. I took this opportunity to reflect on my experience, professional learning, and the industry to discern a path forward in my career. As a hardware designer/engineer, I would have options for moving into management positions or I could stay in a technical role and start to specialize. While I knew it was too early in my career to be making these decisions, I knew that I wanted to have a clear goal to be working toward.

Having a clear goal of becoming a technical specialist in PCB design allowed me to focus my energy on professional learning and development. Having co-workers with skills in other design areas allowed for a sharing of knowledge and a sounding board for complex situations. Learning all I could from co-workers, as well as watching any teaching information I could find online, helped to increase my design knowledge. This thirst for knowledge brought me to collaborate with, and learn from, colleagues from other departments. This gave me the chance to understand integration from both the mechanical and software perspective, bringing new insights to my hardware designs.

Today, I am now a full-time self-employed entrepreneur running my own business. While my current responsibilities are greater and more varied than in previous positions, the one that has always been my highest priority is my commitment to professional growth and learning. When working for a company, you need to seek out challenges and push for training in whatever capacity you can find. You should never stop learning and trying to improve yourself. Just because you don't need it now, does not mean you won't need it in the future. Every step forward in that direction will enable you to become a great designer. **DESIGN007**

Tomas Chester is founder and hardware designer with Chester Electronic Design in Guelph, Ontario, Canada.

Not Just a Roll of the Dice

My PCB Design Education: A Continuous Process


Tamara Jovanovic

Feature Article by Tamara Jovanovic
HAPPIEST BABY, INC.

I remember the first time my boss asked me to design a PCB for some in-house testing. It was wintertime and we were in a meeting room with a few other co-workers. Someone suggested we design our own PCBs for testing to mitigate issues from the set-up with development kits. My boss agreed, and everyone turned to me, an electrical engineer.

Immediately, anxiety started spreading through my entire body. I could feel my palms sweating and my heart rate increase as reality hit me; I had never designed an *entire* board by myself before. Until then, I had read books, blogs, magazine articles, and Altium support pages on how to use their tools. I had watched countless YouTube videos, done some of the sample projects on my own, and retouched some of the existing designs. However, I had never designed an entire PCB on my own from start to finish.

I felt like there was still so much I needed to learn, and I was terrified I wouldn't be able to deliver. Despite the panic setting in and my legs shaking from anxiety, I blurted out, "No problem. I can do it." Everyone in the room nodded at me and we moved on to the next topic.

I barely remember what was discussed during the remainder of that meeting. All I could think was, "How am I going to pull this off? What in the world am I going to do?" I was trying to calculate in my head how many hours it would take for me to thoroughly read the 1,500-page *Printed Circuits Handbook* by Happy Holden and Clyde F. Coombs. In that moment, it didn't matter how small or straightforward the PCB was supposed to be, I felt like I wasn't prepared enough to do this on my own. But there is a reason why some parents effectively teach their kids to swim by throwing them into the deep end of the pool and letting them figure it out on their own.

When I calmed down a little bit from the initial shock (of being asked to do my job), I remembered that I had designed circuits and complex electronic systems before. I realized that I had the knowledge and the resources to complete this task, and all I needed to do was implement it into a new application. I

started by reading the main IC's datasheet and I found a reference design for the part, as well as design suggestions and additional part recommendations. Most (if not all) ICs have reference designs in their datasheets, and they are a great starting point for designers. I created a schematic, ran my design rule check and surprisingly, everything passed.

I moved on to layout. Once I got started, the whole process was flowing. At that point, I couldn't remember why I was initially so stressed out about it. I had some doubts and worries along the way, but I was hoping if I had followed all the rules and design specifications that the board would work. After a nerve-wracking two weeks for fabrication and assembly, the boards came back fully functional, and we still use them to this day.

After a nerve-wracking two weeks for fabrication and assembly, the boards came back fully functional, and we still use them to this day.

Looking back, I realize that experience played a big part in my PCB design education; everyone has to start somewhere. As I was designing more boards and working on more complex designs, I got comfortable using the tool and more confident about my skill. Even as I was gaining more experience, I still had ups and downs along the way.

There was a time I sent out boards for fabrication and heard back from the manufacturer that my design was not manufacturable. I've had to modify designs quickly in order to meet deadlines and schedules, and I've made hand alterations when I have accidentally missed something in the design. All of it—the good and bad—was the best thing that could have ever happened to me. Being under pressure to deliver functioning boards, on time, made me aware of what could go wrong, and it forced me to employ a systematic approach to every design I work on. I started keeping notes of the tricks I learned along the way and because of that, I ensured that I wouldn't make the same mistakes again.

Here are some of the design practices that I've been following while designing boards.

1. Understand your manufacturer's capabilities.

Altium Designer allows designers to set rules to anything you want: trace width, spacing, impedance control, via size, etc. But it is important to be aware that once you are done with your design, you need to send it to a fabricator to bring it to life. Unfortunately, designers sometimes create designs that are not feasible in real life. I've found that it is best to speak to your fabricators *first* and understand their design limitations and capabilities. There are industry standards that can be followed, but for more complicated designs that are constrained with space and rules, it is always best to start with getting all the information you need to deliver the best design.

2. Understand your design requirements.

I always spend a significant amount of time understanding what needs to be done and familiarizing myself with the parts. Reading the datasheet and application notes thoroughly can considerably decrease the amount of time you spend on the design. In addition, I like to validate my designs before I send them out for fabrication. By simulating and testing the design beforehand with software, development kits, and breadboard components, I ensure design functionality and have the freedom to test multiple configurations of the design before I hand off the data package. It also makes me more confident about my design.

3. Set the rules.

Make sure to set all your rules before you start working on layout. This can make your life exponentially easier, and it helps prevent mistakes along the way.

4. Note the lessons learned.

After you are done with a design and you are ready to work on the next revision, always review what you did, what went wrong (if anything), and identify areas for improvement. Mistakes happen, but if you hold yourself accountable for what you created, you're going to see that there is always something you can do to improve your designs.

5. Never stop learning.

Finally, always continue your design education. You can't rest for too long, because technology is constantly evolving. When I am working on something new, I always try to get my hands on as many resources as I can. Maybe I haven't thought of an idea that someone else already encountered and I can learn something new. It's always good to try to expand your knowledge; you never know when a tip or technique will come in handy.

As with most things, I believe experience plays a big part in PCB design, so I try to practice as much as I can on my own. I feel like I still have so much to learn, but I am excited to try out some new techniques I've read about, and I hope that one day, if I keep this up, I can become a great designer. **DESIGN007**

Tamara Jovanovic is an electrical engineer at Happiest Baby, Inc., a developer of smart baby beds, based in Los Angeles, California.

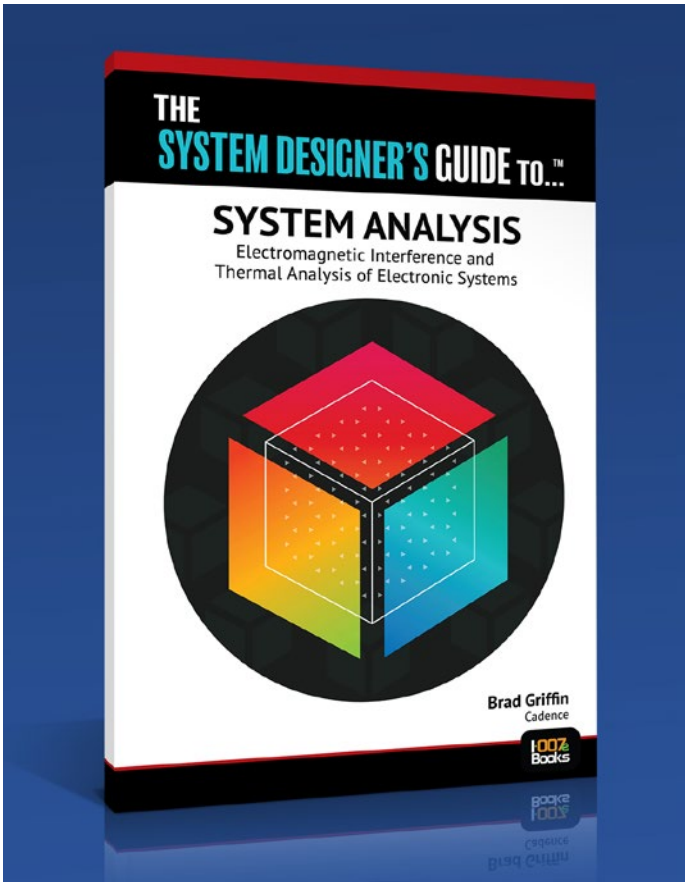
New I-007 eBook Highlights System Analysis

In this latest title from I-007eBooks, readers will learn how system-level analysis of complex and high-speed electronic designs is critical to solve electromagnetic, electrothermal, and electromechanical simulation challenges and to ensure that the system works under wide-ranging operating conditions.

According to signal integrity expert Lee Ritchey, founder and president of Speeding Edge and author of *Right the First Time: A Practical Handbook on High-Speed PCB and System Design:* "Author Brad Griffin offers readers a very good explanation of system-level analysis of complex and high-speed electronic designs. I highly recommend this book for those who are getting started with the design of PCBs and systems that are used in data centers."

Download your free copy today!

You can also view other titles in our *full library. The System Designer's Guide to...* series is specifically dedicated to educating the printed circuit and systems design sectors and serves as a valuable resource for people seeking the most relevant information available.



Not Just a Roll of the Dice

Hiring a Young PCB Designer: What's the Break-even Point?


Patrick Davis

Feature Interview by the I-Connect007 Editorial Team

How long does it take for an inexperienced PCB designer to get up to speed and become a regular member of the design team? We recently spoke with Patrick Davis, product management director for Cadence Design Systems, about his company's hiring and training processes.

Patrick explains his theory that PCB designers typically cost their companies money during the first year of their career, though they hit the break-even point in their second year and begin being profitable in the third. Realistically, how long did it take you to become a solid PCB designer?

Barry Matties: Patrick, what is Cadence doing to find talent? Are you reaching out to universities and vocational schools to "sell" the design career, if you will?

Patrick Davis: Yes, we are. We have a very good relationship with a university. I am working with a few people to develop what is basically a PCB Design 101 class. We'd like it to be like Mary Sugden's school, the Copper Connection, from years ago in San Jose.

I'm working with a couple of people who are putting together a design curriculum for those who want to learn to be designers. We're helping them with the tool set and understanding the process; they are going through six months of school to become a beginning designer. It takes about six months before you can go ahead and make your own mistakes. You get those people who want to do this, and then you teach them.

It just takes time to learn how to design, and I figure I need three years to break even on a new employee with zero experience. When I bring new designers in, I usually say, "Give me three years. There will be one year of training to actually become useful, and I'm going to lose money on you because you're going to make so many mistakes. In the second year, we actually break even with you. This is your transition year where you start to figure out what your style is, what you're good at, and this is when you really start to learn. In your third year, you're now making money for us."

Gain knowledge, save time and improve your bottom line with ASC's

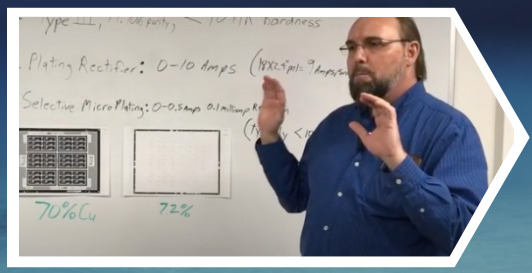
FREE RESOURCES

WEBINARS

Attend weekly free webinars focused on different PCB topics

eBOOKS

Download concise and relevant books written by the ASC Experts



EXPERTS BLOG

The ASC Experts help you learn more about advanced PCB technologies

ONLINE DFM

Upload Gerber files and get results back in just 60 minutes

American Standard Circuits

Creative Innovations In Flex, Digital & Microwave Circuits

www.asc-i.com | (630) 639-5444

I'd almost prefer the inexperienced types over the people who have half-baked design experiences or bad design habits. There are a lot of new schools out there that are starting to come up and I think it's great; a school can give you that first six months into PCB design so you're useful. So, I'm not losing as much money on you when you come in, and hopefully you can pick it up quicker.

Andy Shaughnessy: Susy Webb teaches a PCB design basics class at PCB West, and sometimes the class is full of engineers. The EEs are becoming designers, but designers say that EEs don't know how to look at it artistically, like a designer. On the other hand, designers are now doing more of what was in the realm of the engineer not long ago; they're all switching hats.

Davis: Sure. What always ends up happening is that you get two kinds of engineers. You get the engineer who sits there with bad garlic breath, reading over your shoulder, telling you what's going on, and you want to say, "Please don't eat garlic next time."

And then you've got the engineer who throws it over the wall and says, "Go ahead and I'll give you something in about five or six weeks." Between those two worlds, we end up needing the engineer to become more involved, and they need to have a tool set that they're familiar with, and can actually play with and make things.

But there is a definite gap among the designers. At my previous position, I had some old-timers who had been designing for 25 years, but I was also picking people right out of San Jose State. These young engineers just wanted to design. And then you have the crazy people. We had a PCB designer once who said, "You've got to have a special kind of stupid to be a designer, and I think I've got it."

The young guys I was teaching understand signal integrity and power integrity, but I said, "Go on the floor. I'll pay you to spend a week on the shop floor to understand the manufacturing process." Manufacturing is just as important as signal integrity and power integrity to a designer.

Matties: If an OEM is looking to hire a designer, what should they consider first?

Davis: Aptitude. As I said, it takes a special kind of stupid to be a designer. You've got to want to sit in the cave, get lost into the puzzle, and you have to love the puzzle. You have a three-dimensional puzzle that is bound by the laws of physics, manufacturing capabilities, time, and that's it. Everything else is within your world. If you're good at doing things like Sudoku puzzles, and you love the challenge of it, then that's great. That is one thing that very logical people who come into this career just can't handle.

To be successful at design, you must have a thirst for learning, because the learning is always going to be there. You can't stop learning. The business is moving too fast. Especially if you're at an OEM, you can get locked in, thinking, "I'm designing this one thing for the 747th time in my career." You have to keep learning all of the time.

Designers need to have a thirst to learn, and they need that curiosity to sit around and work through puzzles. I don't care what your degree is in. Degrees help; I won't have to explain physics and inductance, capacitance, and resistance. I can talk at a higher level to someone with a degree. But I just want people who are intelligent and willing to learn.

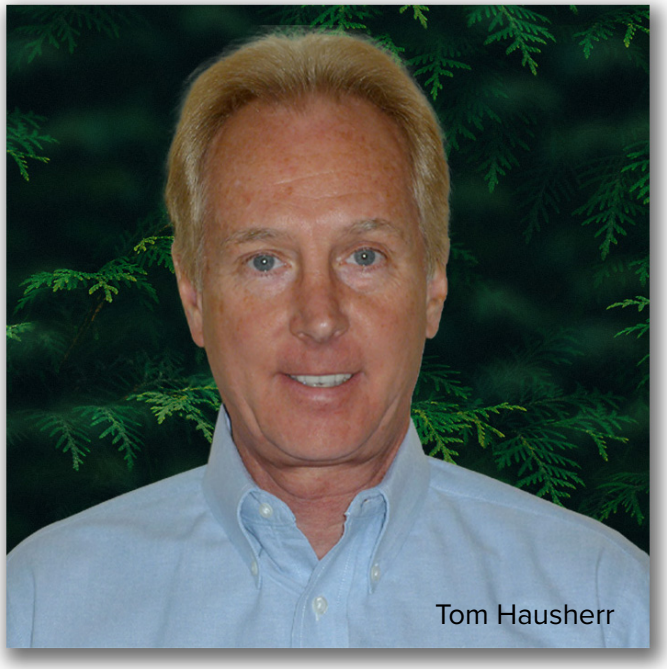
Once you understand the physics behind it, everything else is based on that. Your return path, your power, how things are placed, sine waves, and timing are all based on simple physics. You don't have to have a Ph.D. just to understand the few pieces of it that you need to know.

Matties: Thanks for your time, Patrick.

Davis: Thank you. I appreciate it. DESIGN007

Not Just a Roll of the Dice

You Can't Learn PCB Design on Your Own


Tom Hausherr

Feature Article by Tom Hausherr, CID+, CIT
PCB LIBRARIES

When Editor Andy Shaughnessy asked me to comment on the best path for new PCB designers to learn the trade, I had one immediate thought: I don't think anyone can learn PCB design on their own. It's just not possible today.

This job takes a lot of hard work. I got where I am by working 60- or 70-hour weeks for decades. I would spend 40 hours a week learning PCB design and 30 hours a week reading the CAD tool manual.

Even back in the 1970s, we had mentors to answer our questions and teach us the ins and outs of design. Unfortunately, new designers don't always have the benefit of mentors. Smaller companies may not have the time or resources to assign mentors to new hires. Your boss isn't responsible for your design education; you are.

There's always more to learn in PCB design, and more than one way to learn. Even decades after I began designing PCBs, I'm still learning. In 2011, I learned Altium and SolidWorks, and even though I knew PCB design inside and out, I had a 15-month learning curve to master these tools. I was surrounded by engineers and designers who knew the tools, and I was not shy about asking questions every day. Learning PCB design includes learning and mastering PCB library creation—that alone is a tough project.

My best advice for new designers is the same that I gave my son Luke back in 2003. I told him to go to work for Mike Creeden in his service bureau, San Diego PCB Design, and make minimum wage until he became productive enough for a pay raise. So that's what he did. Of course, Luke drove Mike crazy, asking him 1,000 questions along the way, but that's how you learn PCB design. Eventually it paid off. Now, 18 years later, Luke is one of the top PCB designers in San Diego.

There's no magic wand for this. If I wanted to learn PCB design from scratch today, I would do exactly what my son did; I would apply for a starting position working for peanuts, provided that the company taught me how to design boards.

A Few Pointers

- Visit multiple assembly and fabrication shops for both prototypes and production. Many designers haven't been inside a board shop in decades, if ever. Ask questions and try to understand these processes, because many design mistakes are related to improper DFM and DFA practices.
- Take an IPC-J-STD-001 training course and learn how to solder parts onto PCBs. Learn all the terms, definitions, and acronyms. When I first started in CAD design, I felt almost like I was learning a new language.
- Learn every component package. There are only 50 types of standard packages. What functions do they perform? Resistors, capacitors, diodes, and inductors comprise 80% of the parts on a typical PCB design. Learn the reference designator chart, as well as the basic component lead forms: gull wing, rectangular end cap, J-lead, castellation, bottom terminals, BGA, CGA, LGA, through-hole, etc.
- Familiarize yourself with the major CAD vendors—Cadence, Siemens, Zuken, and Altium. When you pick a CAD tool, learn every feature that it has. Read the user guide.
- Read books by high-speed design experts such as Lee Ritchey and Eric Bogatin. When reading content that focuses on signal integrity, be sure to understand the importance of a well-designed reference plane for your return current. Again, ask questions.

I would also advise young people who may be interested in PCB design to get familiar with math, especially geometry. High school and college geometry courses are very important and knowing Cartesian coordinates and the Pythagorean theorem are a must.

In the early 2000s, IPC predicted that the PCB designers of the future would be electrical engineers, and we're seeing evidence of that trend today. So, go to college, at least get a bachelor's degree in electrical engineering, and take drafting and geometry classes. You can take online classes today, and all you need is a computer with an internet connection.

An electrical engineering degree will open the door to employment at the entry level. Then you'll be ready to really start learning PCB design. **DESIGN007**

Tom Hausherr is president of PCB Libraries.

Integrated Tools to Process PCB Designs into Physical PCBs

Visualize

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

Verify

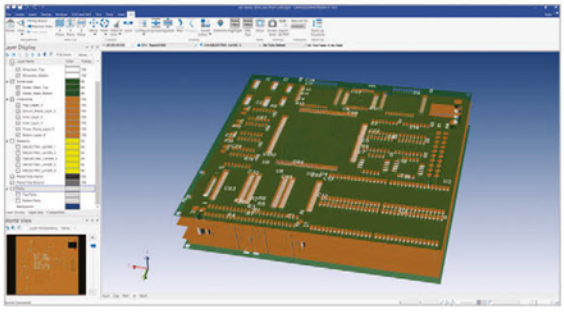
Ensure that manufacturing data is accurate for PCB construciton.

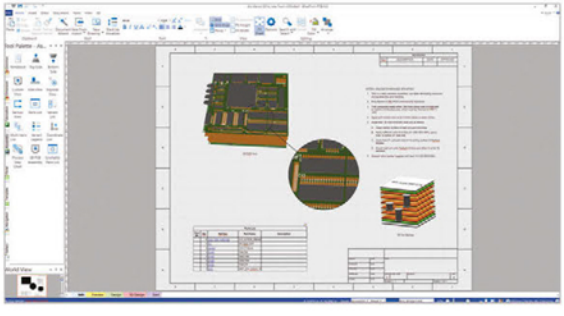
Panelize

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

Document

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





CAM350®

Verify and Optimize PCB Designs for Successful Manufacturing

BluePrint-PCB®

Create Comprehensive Documentation to Drive Fabrication, Assembly & Inspection

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

DownStream Technologies

800-535-3226 | downstreamtech.com

NOW WITH 3D

Not Just a Roll of the Dice

Seeking Out Design Education Resources


Michael Steffen

Feature Article by Michael Steffen, CID
CRYSTAL GROUP

Over the last few decades, electronics have become an exponentially more important part of our daily lives, and in turn electronics designers have become highly sought after. The field is massively diverse in focus to cater to many different markets. This has led in some ways to much broader availability of resources to the junior designer, but on the flipside, the availability of "good" resources can still be lacking. Let's take a look at some of the resources available, and how to make the best of them.

I've been doing PCB design for about 15 years. I started in high school, drawing out simple circuits in an image editor and raiding the local Radio Shack for ferric chloride and copper-clad boards for toner transfer etching. The results were crude, and often didn't work, but the process was fascinating—and truthfully still is. For as empowering as it can be, PCB design has a shockingly low barrier to entry. Radio Shacks are harder to find today, but the materials are all still readily available from online shops. In contrast to my early days of hand etching, there are now dozens of PCB fabricators willing to make full-featured prototype boards for just a few dollars.

Furthermore, over the last decade the number and quality of free-to-use EDA software options has exploded, and features that were once only available in software suites costing many thousands of dollars are now making their way into open-source offerings. There has practically never been a less expensive time to get into the field.

But while access to tools and supplies has improved dramatically, access to quality training has not kept pace. Sure, there are thousands of videos on YouTube, but they all convey slightly different information in a slightly different way, and not always accurately. And higher education is not always the solution either; my own alma mater did not offer a PCB-specific course at the time I was working on my degree. So, it can be difficult for a junior designer or engineer to find the practical information they need to realize their design.

There certainly are quality resources out there. Many of these are in the form of topic-specific books; there is great knowledge in the

While others struggle just to keep pace, we stay ahead of the curve by constantly investing in the equipment, people and resources needed to be the best total PCB solutions provider in the industry.

VIEW OUR CAPABILITIES

EAGLE electronics Inc.

www.eagle-elec.com

texts of Howard Johnson, Eric Bogatin, and Bruce Archambeault; for someone with a design or two under his belt these are quite valuable reads. But particularly for very new designers, the terminology in these books can be intimidating, especially if the designer does not have a deep engineering background.

Of course, if senior staff are available as mentors, this can be one of the most valuable resources of all. Given their knowledge of your organization's exact processes and preferred design methodology, mentoring by a more senior designer is probably the most direct and focused way to get the information needed to begin as a junior designer. Unfortunately, in many cases a mentor is not available or practical for the organization to provide, and designers are left to fend for themselves.

Of course, if senior staff are available as mentors, this can be one of the most valuable resources of all.

For a junior designer, I think a good compromise can be to spend time studying commercial products. Millions of designer-hours have been invested in optimizing the PCBs in the products we use every day, and it is possible to glean some of that knowledge with a keen eye. I always keep a drawer full of scrap boards somewhere around my workstation, and when I hit a wall on a design I like to pull out a relevant one and spend a few minutes looking it over. It doesn't always result in a critical insight, but it helps to break my mind out of its block and suggest different ways that things can be done. This can be particularly useful for a junior designer without a lot of personal experience to pull from. In any case, it can prompt further research.

But for all the concrete resources available, there's one more nebulous concept that deserves mention, and which can be used to make the best of the resources you do have. There's a phrase I like to use, perhaps to the detriment of my own career prospects: I'm not a great engineer, but I'm a pretty good problem solver. I don't easily retain book knowledge, and to this day I'd have to pick up a textbook to quote anything but the most basic engineering equations. But I have always understood the importance of being able to analyze a problem or task and break it into smaller parts.

The ability to think critically about a design, determine its inputs and outputs and the functional blocks, and separate them into workable pieces is to me the most important skill an engineer or designer can have, because with it an arbitrarily complex design can be made manageable. This can be applied to design, breaking down the structure of a widget to its elemental functions and handling the important features of each one individually. Or it can be applied to testing and troubleshooting, identifying the signals going in and coming out, and what operations should be happening in the middle to narrow down a problem area. Anyone can pick up a textbook when they need to, but problem-solving skills like this are more abstract and usually not taught outright.

Ultimately, by that logic, your success as a designer depends not only on what resources you have available, but also on how you apply them. In my experience, the best designers are ones that are flexible, that can adapt to changing requirements or unfamiliar scenarios. Because as the market for electronics continues to evolve, there will continue to be new and unique problems to tackle. **DESIGN007**

Michael Steffen, CID, is a senior electrical engineer with Crystal Group.

SIEMENS

Making DFM Accessible For Any Business Size

From home garage to enterprise - PCBflow makes it a breeze to certify designs for manufacturing

Siemens solution enable companies to optimize the entire Electronics Manufacturing process in a smart and efficient way. Our lean manufacturing experts will walk you through the design-to-manufacturing flow and show you how to leverage the digital twin to improve your processes.

www.pcbflow.com

Learn More

Not Just a Roll of the Dice

Paying it Forward: DFM Education for Engineering Grads


Kelly Dack

Feature Article by Kelly Dack
EPTAC

A few years ago, I had the opportunity to work for a prototype PCB supplier in Seattle. My role at this manufacturer was to represent the company's capabilities to the many PCB designers and engineers in the area and to reach out to help them understand important DFM considerations in their designs.

Earlier, I'd had a life-changing realization regarding the important requirements of our low-volume, in-house PCB assembly stakeholders. Could it be that I was being given an opportunity to see more—this time from the perspective of the PCB fabricator? They too face challenges every day from designers who should know more about DFM for PCB fabrication. You might say that over the past six years I have become "woke" to the daily challenges our PCB manufacturing stakeholders face when designers and engineers show disregard for solid DFM practices.

You may remember three years ago that this magazine was asking, "Who will fill your shoes?" While working in Seattle, it did not take me long to answer that question. Every day for my job, I had the opportunity to drive to several Seattle-based design companies and meet with their engineering staffs, usually full of young, recently graduated electrical engineers. Almost every company I visited had employed several recent EE graduates (all named Zach, Ian, or Ashley, it seemed) who were tasked with laying out PCBs.

I could tell in a heartbeat that these kids were smart. Some of them were involved in the Maker subculture and had already been exposed to etching printed circuits and finding creative ways to manually solder their own PCB assemblies. This exploding population of creative brilliance was dangerous, though. It seemed they could learn new layout and signal integrity software in a matter of days. Educated in the universities, with a two-week survey course on PCB design if they were lucky, these electronics engineering graduates were turned loose into the industry.

Their ability to learn the layout tools quickly without being educated in DFM enabled them to pump out unmanufacturable designs just as quickly. We saw PCBs with odd numbers of

APCT

Passion I Commitment I Trust

Flex & Rigid Flex Manufactured At Lightning Speed!

APCT Anaheim Offering Quick Turn Capability For Flex & Rigid Flex Product!

APCT Anaheim Offers

- 1 – 2 Layer Flex With Stiffener: 5, 7, 10, 15 Day Options
- 4 – 6 Layer Flex With Stiffener: 7, 10, 15 Day Options
- 4 – 6 Layer Rigid Flex: 7, 10, 15 Day Options
- 8 – 12 Layer Rigid Flex: 7, 10, 15 Day Options
- 14 – 16 Layer Rigid Flex: 10, 15 Day Options
- 18 – 22 Layer Rigid Flex: 15 Day Options
- 2X Lam Cycle Rigid Flex HDI: 20+ Day Options

"Bookbinder Know-How"

APCT.com | 512.548.7435

Printed Circuit Board Solutions

APCT.com	APCT Santa Clara HQ	APCT Anaheim	APCT Orange County	APCT Wallingford	APCT Global
	408.727.6442	714.921.0860	714.993.0270	203.269.3311	203.284.1215

Figure 1: Kelly and Mark Thompson fighting the "War on Failure."

layers, unbalanced copper, quarter-inch-thick PCBs with tiny vias which would not plate through—you name it. Suddenly, with little time to react, our fabricators were being deluged with overly creative designs which could not be manufactured.

These smart young engineers were sorely in need of an education on DFM. I'd found my calling. I teamed up with CAM expert and manufacturing industry icon Mark Thompson, and we were off to the races. I'd known Mark for years and he was always a helpful DFM mentor while I had worked at the gaming company years earlier. It was an honor to work side-by-side with him and help educate these new PCB designers.

We invited these young engineering customers to tour our fabrication facilities and provided "lunch and learn" sessions. Together we recorded a variety of video interviews and wrote articles to help fight the war on DFM failure.

We were positioned to make a difference and we were. Slowly, the PCB designers we had schooled were sending balanced, even-layered designs with nicely spaced traces and appropriate widths with consideration for copper weight and outerlayer/innerlayer processing.

This was a DFM win for both design and manufacturing to be sure. These recent grads didn't know what they didn't know, and it wasn't their fault. Most EE curriculum doesn't include much manufacturing content, though that is changing, as Marc Carter chronicles in his column.

I really enjoyed my time teaching these grads. These smart young people just needed someone to explain the best practices for DFM, and they were off to the races. I'm sure that the Zachs, Ians, and Ashleys are on their way to becoming solid PCB designers who will in turn share their knowledge with the next generation. **DESIGN007**

Kelly Dack, CIT, CID+, is an instructor with EPTAC and communications officer with PCEA.

Design Education Organizations, Conferences, and Webinars

As the world returns to some semblance of normalcy, PCB designers and design engineers now have access to a variety of educational resources, both online and in-person. You can also join organizations like PCEA and IPC Design and attend online chapter meetings—free of charge.

Never before have there been this many opportunities to expand your design skill set. Take advantage of these conferences and webinars. Your design education is your responsibility!

Upcoming Live Conferences

DesignCon 2021
San Jose, California, USA
August 16–18, 2021

PCB West 2021
October 5–8, 2021
Santa Clara, California, USA

PCB Carolina 2021
November 10, 2021
Raleigh, North Carolina, USA

IPC APEX EXPO 2022
January 25–27, 2022
San Diego, California, USA

PCB East 2022
April 11–13, 2022
Marlborough, Massachusetts, USA

IPC Online Classes

PCB Design for Military & Aerospace Applications
Every Monday and Wednesday
August 30–October 6, 2021

PCB Design for Rigid-Flex Boards
Every Monday and Wednesday
August 30–October 6, 2021

Introduction to PCB Design II
Every Tuesday and Thursday
August 31–October 21, 2021

EPTAC Online IPC Design Certification Classes

IPC Designer Certification (CID)
August 9
September 7
October 4
November 15

IPC Advanced Designer (CID+)
September 13
November 15

PCB007 Highlights

Foundations of the Future: IPC Education Foundation Lessons Learned ▸

Looking back on our second year, the IPC Education Foundation takes pride in how we adjusted our engagement initiatives during the pandemic. We couldn't rely on our original strategy of hosting and attending a range of in-person activities and events.

Punching Out! Keep Your Saw Sharp ▸

It sounds very simple, but work is much easier with a sharp saw. Abraham Lincoln may have said, "Give me six hours to cut down a tree and I'll use four hours to sharpen the ax." Chainsaws are much, much quicker, but the concept still applies. Tom Kastner explains how this relates to your business.

EIPC Technical Snapshot: Thinking Differently in a Post-Pandemic World ▸

President Alun Morgan introduced the 10th in the series of EIPC's monthly Technical Snapshots. He said that although previous webinars had been directed at specific technical issues, the focus of the present session had been expanded to include a good measure of business content to provide better understanding of how it had developed during the pandemic, and also some pointers to how it would look and feel as we began to emerge from such unusual times.

IBIDEN Selected for FTSE4Good Index Series, FTSE Blossom Japan Index ▸

IBIDEN Co, Ltd. is pleased to announce that it has been selected for FTSE4Good Index Series for the sixth consecutive year, and FTSE Blossom Japan Index for the fifth consecutive year.

MKS Instruments to Acquire Atotech ▸

MKS Instruments, Inc. announced that they have entered into a definitive agreement pursuant to which MKS will acquire Atotech for $16.20 in cash and 0.0552 of a share of MKS common stock for each Atotech common share.

Hire or Be Hired at jobConnect007 ▸

On July 2, 2021, I-Connect007 released the inaugural issue of jobConnect007, a monthly guide to job openings in the printed circuit board/electronics manufacturing industry.

Meet Luca Gautero, New PCB007 Columnist ▸

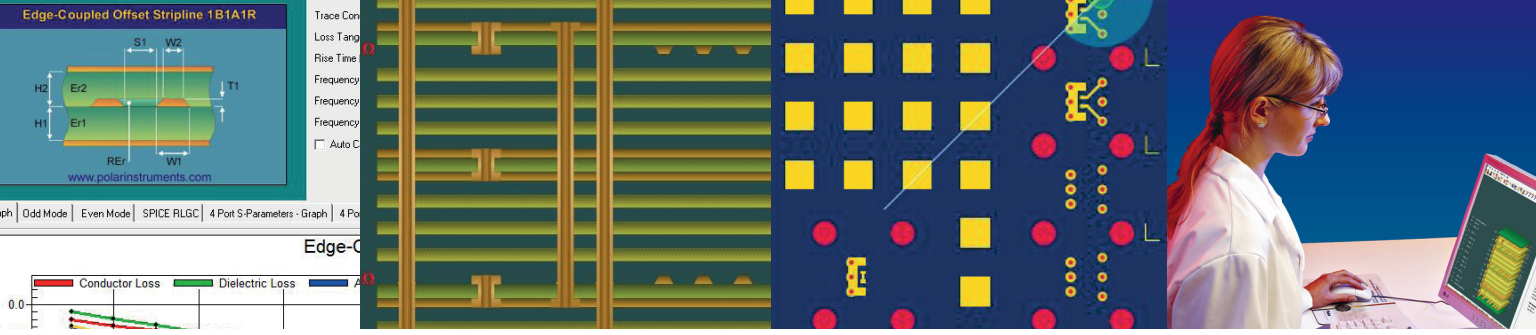
We are pleased to announce the latest addition to our team of expert columnists, Dr. Luca Gautero. In his column each month, Dr. Gautero will cover inkjet application for solder mask from a variety of angles. His technical perspective will address maintenance, material challenges, and patterning capabilities. Social responsibility and cost effectiveness will also be illustrated.

American Standard Circuits Acquires Orbotech Precise 800 ▸

American Standard Circuits has recently announced that they have purchased Orbotech's Precise 800.

Dan's Biz Bookshelf: Go Live! Turn Virtual Connections Into Paying Customers ▸

If one thing has become clear during the pandemic, it's that more people are using social media to communicate. Use that to your advantage in your business. Dan reviews helpful tips from this book by Jeffrey Gitomer.

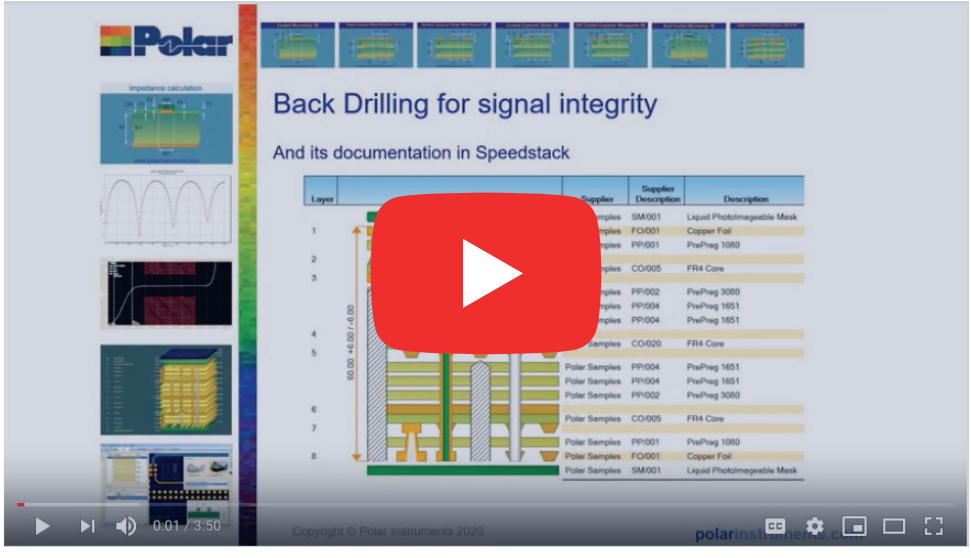


SUBSCRIBE

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

PCB Signal integrity tools for design & fabrication

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



polarinstruments.com

PCB Design Education—What 'They' Don't Tell You

The Pulse

Feature Column by Martyn Gaudion, POLAR INSTRUMENTS

Lots of people are far more qualified than I am to talk about PCB design, but what fascinates me about both PCB and electronic design is the range of disciplines you need to be aware of, especially when moving into high speed/high power/high reliability arenas. For a new designer entering this space for the first time, it can be quite an eye-opener (no wordplay intended) to discover just how many different disciplines are involved in turning a good design into a fit-for-purpose PCB.

Specialist in How Many Disciplines?

One of the changes that has engulfed all industries in the past 50 years is the increasing level of specialisation required. Gone are the days when an engineer could have an idea, design the electronics, lay the board out, and maybe even build and etch the PCB themselves. When Polar was founded in the mid-1970s, that's just what could happen. And while this is still possible for the most basic of designs, a PCB designer now needs to at least be aware of a whole spectrum of specialist disciplines which will entail dealing with other companies and industry professionals, to determine where, and where not, external advice is needed for a specific design.

Material Choices

The proliferation of materials in the past 20 years means that a new designer is faced with a bewildering choice of materials for any design; the temptation to "mix and match" to ensure best performance/value for money trade-off means a designer will often need to speak to a base materials specialist for OEM work, or their PCB supplier, or a PCB value-added reseller. Many of these companies now employ former fabrication staff with extensive materials knowledge. Stretched supply chains mean these specialist intermediaries


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

can help especially on low volume or prototype orders.

What to Ask?

I often encounter designers who are reluctant to ask about fabrication and materials as they only feel comfortable asking "when they know what to ask." I also see some new designers who try to infer everything off a material datasheet, and whilst this gets part of the way to the answer, the datasheet is just that—information about a raw material that the fabricator "processes" into a finished product. Just like baking a loaf of bread, the raw ingredients for a PCB are not the same as the finished product. Pretreatment for adhesion and pressing impact the raw material in a way which will be fabricator dependant and will differ from the raw material specs. Not quite as extreme as trying to make a sandwich out of a bag of raw flour, but you get the picture. If you don't know what to ask, there are plenty of YouTube resources that cover PCB topics, along with the networks related to industry bodies like IPC and EIPC. These organizations aim to bring together networks of specialists with disciplines from chemistry through metallurgy, drilling, high-speed signalling, reliability, and thermal capability.

More and More

As technology pushes higher speed, more density and more durability, and legislation restricts the use of some materials, there seems to be ever more to contend with from a designer's perspective. As I mentioned, the only way to deal with this is to discuss requirements for each design with the appropriate source of knowledge from each discipline. The key for a new designer is to be aware that, depending on the design, non-electronic parameters interplay with the design outcome. It is quite fascinating to me that so many disparate disciplines need to work in harmony to produce the most fit-for-purpose design at the most economical cost.

Sometimes the Laws of Physics Work in Your Favour

Fortunately, sometimes technology drives in a direction to make life easier; the drive to ever-shrinking designs means that often high-speed traces can be kept quite short—so short that some parameters which need careful treatment on a larger scale can be of low enough impact to be ignored. Insertion loss is one case where loss is directly proportional to trace length, so keeping traces short can mean you can perhaps build on a lower grade base material. If the traces are very short compared with the highest frequency content, maybe characteristic impedance is not an issue there either.

A Sticky Problem

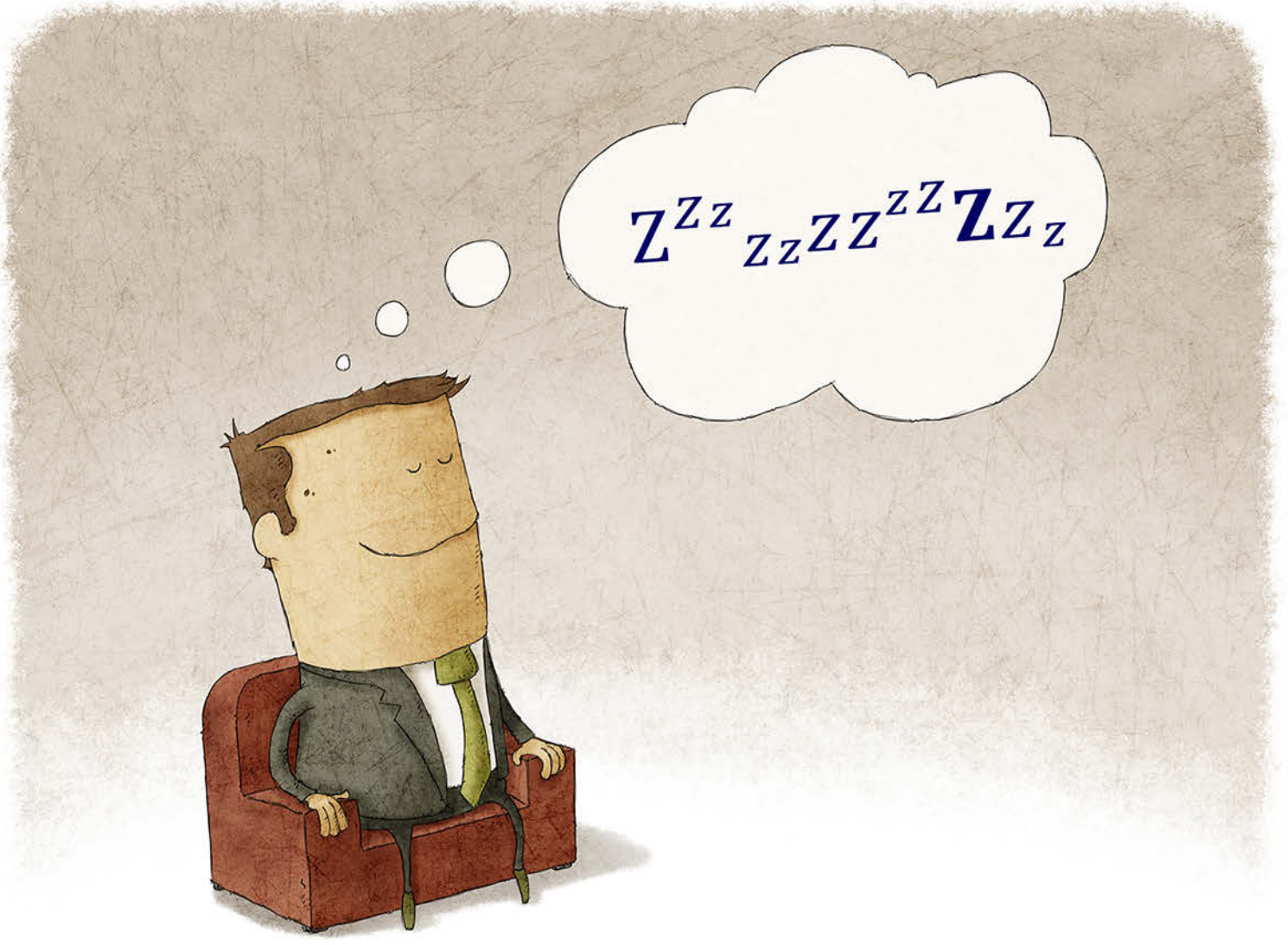
On the highest-speed interconnects, it is well known that copper roughness impacts the signal losses, and complex modelling is needed to calculate the impact, but the roughness is a "necessary evil" to keep the board from delaminating in the assembly process. Here, the chemists are working their magic to provide treatments and processes that chemically bond the copper to the dielectric, and emerging processes for bonding smooth copper could ease one task from the list of design considerations over the coming years.

Conclusion

A new PCB designer, apart from knowledge of layout—which is after all the job description—needs to be aware and ready to discuss a range of peripheral influencers on the PCB's characteristics, many of which would seem to the casual observer to have no connection whatsoever to electronic interconnect. However, from an industry which measures time in inches, and thickness in ounces, that surely should come as no surprise. **DESIGN007**

Martyn Gaudion is managing director of Polar Instruments Ltd. To read past columns or contact Gaudion, click here.

We DREAM Impedance!



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

- Transmission line impedance and
- Power Distribution Network impedance

DISCOVER MORE

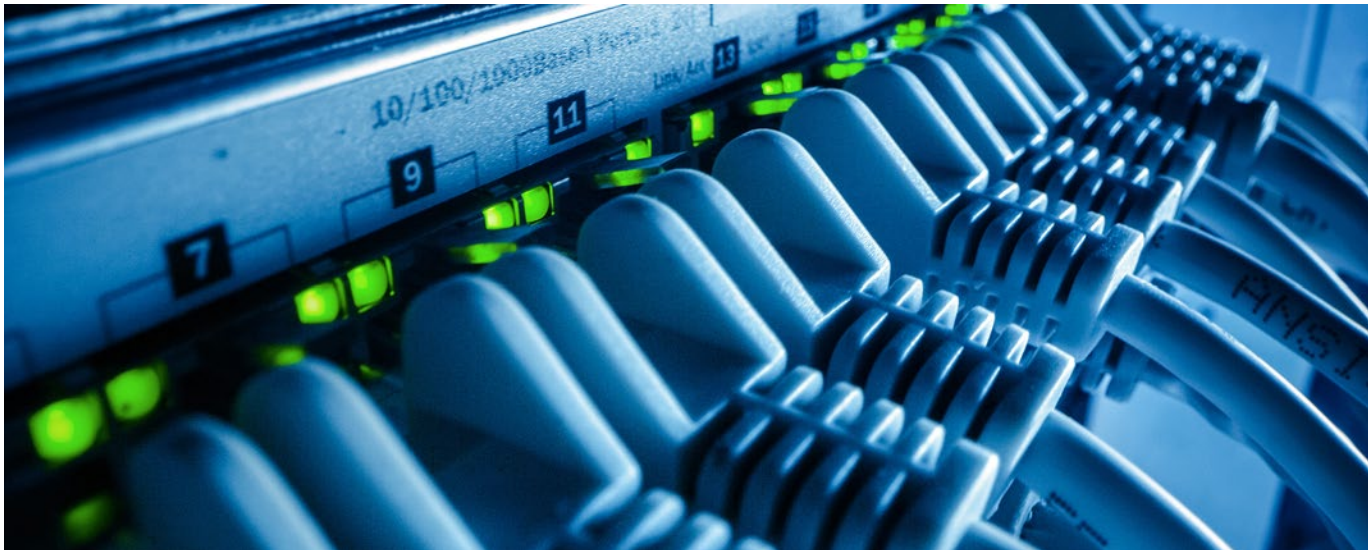
iCD software quickly and accurately analyzes impedance so you can sleep at night.

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

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

iCD
In-Circuit Design Pty Ltd

www.icd.com.au



The Impact of Via Structures on Multi-Gigabit Signal Transmission

Article by Chang Fei Yee
KEYSIGHT TECHNOLOGIES

Abstract

This article briefly introduces various via structures on the printed circuit board (PCB) for layer transition purposes. It also investigates the impact of these via structures on multi-gigabit transmission by analysing time domain reflectometry (TDR), differential insertion loss (Sdd21), and eye diagrams.

Introduction

The via is an essential structure used to make an electrical connection between the layers in multilayered PCBs. Connecting multiple layers of the board makes it possible to reduce the dimensions of the PCB. The via structures available in the PCB industry include through-hole vias and microvias, as depicted in Figure 1.

A through-hole via goes straight through the PCB from top to the bottom and can be applied to connect all the layers of a PCB. It is the most common via and is easiest to construct. A microvia, on the other hand, connects the outermost layer of a PCB to an inner layer, but is not visible on the other side of the board. Microvias are classified as staggered or stacked. With stacked microvias, the via barrels are constructed on top of each other in distinct layers. Staggered microvias, however, are scattered in various layers, and each via barrel is linked with a short horizontal conductive bar.

There are two major advantages of selecting microvias over through-hole vias. First, the microvia has a smaller dimension and aspect ratio, which facilitates the miniaturization of

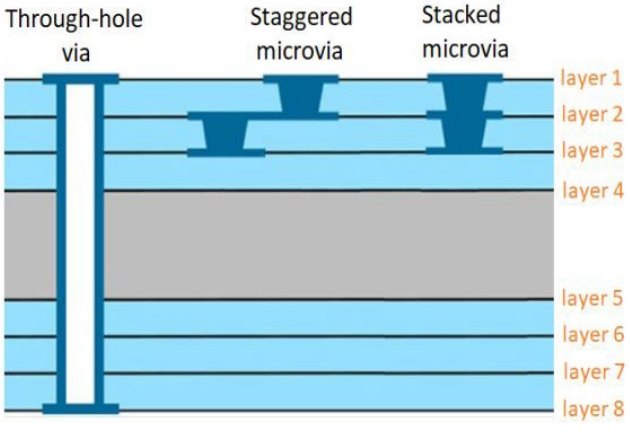
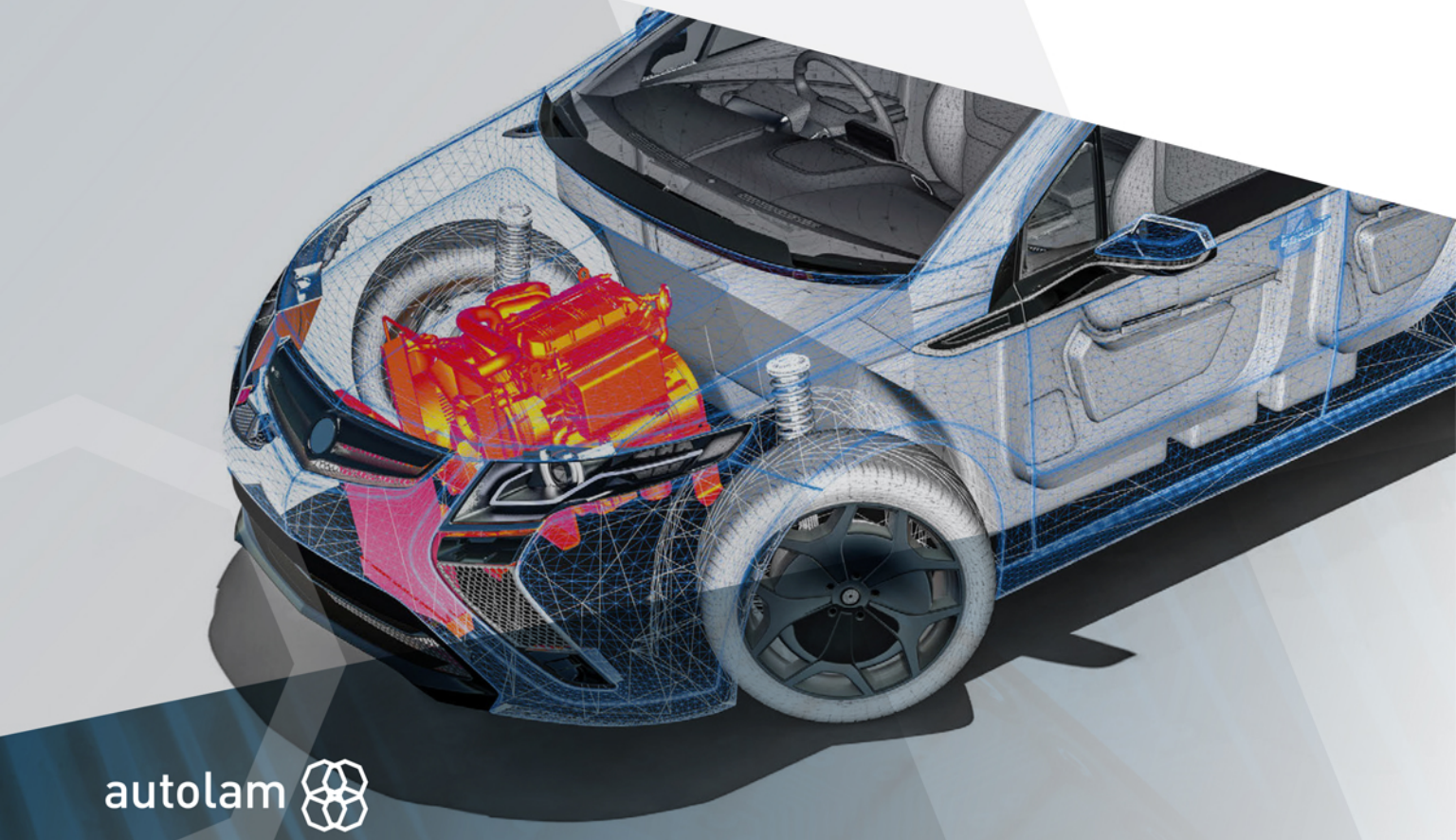


Figure 1: Various via structures on PCBs. (Source: ourpcb.com)

ventec
INTERNATIONAL GROUP
騰輝電子

autolam: Base-Material Solutions for Automotive Electronics



autolam

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

IATF 16949:2016 CERTIFICATION

venteclaminates.com

the PCB[1]. Secondly, the dangling stub in a through-hole via's layer transition causes degradation to the signal transmission[2] due to the inverse proportional relationship between the quarter wave resonance frequency and the via stub length[3].

This article outlines our two-stage study of the impact of through-hole vias, staggered microvias, and stacked microvias on multi-gigabit transmission. In the first stage, the three-dimensional models of these via structures were constructed, followed by the extraction of the s-parameter using Keysight EMPro[4]. In the second stage, the s-parameter files of these via structures are imported to Keysight ADS[5] to analyse the time domain reflectometry (TDR), differential insertion loss (Sdd21), and eye diagram. The analysis and results are explained in the following section of this article.

Analysis and Results

Five transmission line (TL) models were constructed with six-layer PCB stack-up listed in Table 1. Each 1-ounce copper layer is insulated by low-loss dielectric material.

For TL model A (Figure 2a), a 6-inch differential pair with 100-ohm impedance (i.e., trace width 6 mils, spacing of 4.5 mils) on layer 1 is transitioned to a 6-inch pair on layer 3 using a through-hole via. The through-hole via has a drill diameter of 8 mils, pad diameter of 20 mils, a clearance of 28 mils, and a center-to-center separation of 27 mils. This via also comprises a 60-mil dangling stub between layer 3 and 6, highlighted in pink.

PCB layer	Layer assignment
1	Signal
2	Plane
3	Signal
4	Signal
5	Plane
6	Signal

Table 1: A six-layer stack-up.

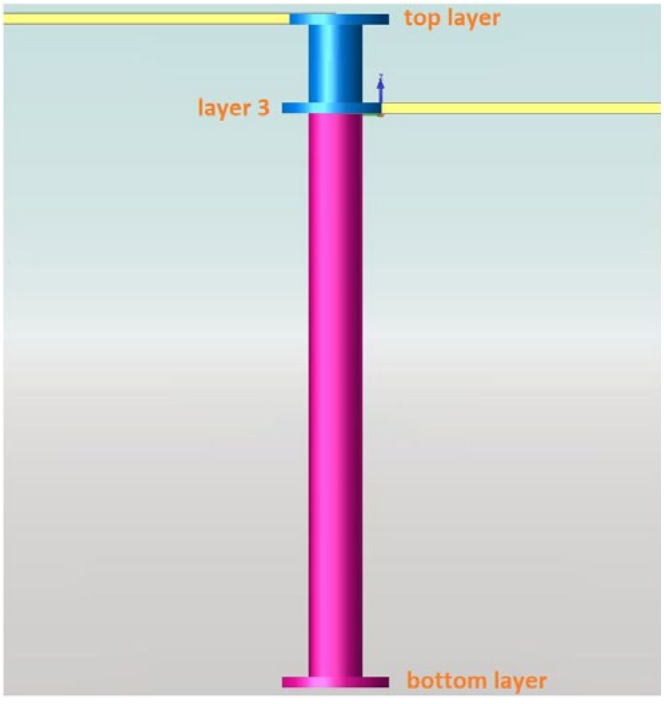


Figure 2a: Cross-sectional view of through-hole via.

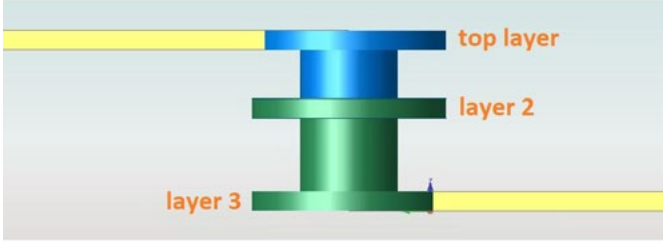


Figure 2b: Cross-sectional view of stacked microvia.

Model B (Figure 2b) is much like model A, except the via is replaced by a stacked microvia. This microvia has a drill diameter of 6 mils, pad diameter of 12 mils, clearance of 22 mils, and a center-to-center separation of 20 mils.

TL model C (Figure 2c) is similar to model B, except the stacked microvia is replaced by a staggered microvia. This microvia has a drill diameter of 6 mils, pad diameter 12 mils, clearance of 22 mils, and center-to-center separation of 20 mils. The short bar connecting the two via barrels has 6-mil width and 10-mil length, highlighted in red.

Meanwhile, the TL model D resembles model C, except the short bar linking the two

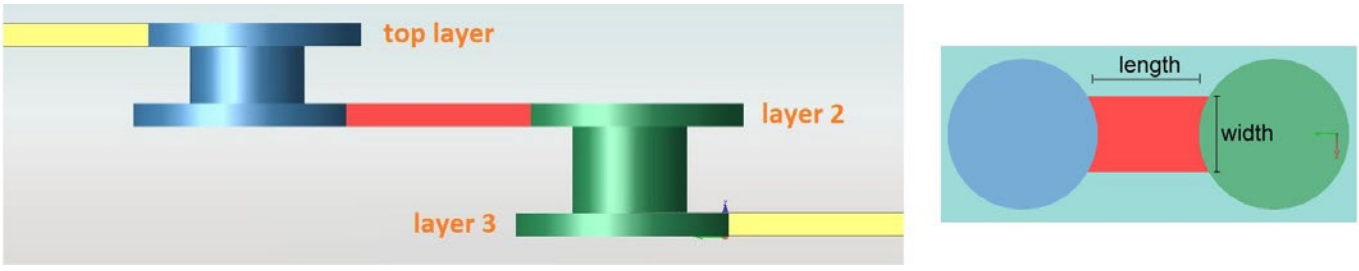


Figure 2c: Cross-sectional (left) and top (right) view of staggered microvia.

via barrels is decreased to 5-mil in length. On the other hand, the TL model E is like model D, except the short bar connecting the two via barrels is widened to 9-mil width.

As shown in the TDR plot depicted in Figure 3, the through-hole via stub in TL model A causes a capacitive impedance discontinuity of 22 ohms. With stacked microvias, the capacitive impedance mismatch experienced by TL model B is reduced to 3 ohms. Meanwhile, with a short bar in the staggered microvia, the impedance mismatch encountered by TL model C becomes inductive and rises to 6 ohms. The inductive discontinuity in staggered microvias can be diminished by shortening and widening the short bar, as experienced by model D (5-ohm discontinuity) and E (4-ohm discontinuity) respectively.

Hence, the stacked microvia experiences the minimal impedance mismatch vs. the other via structures. But if staggered microvias are selected for signal routing, the short bar shall be kept at the minimal length to minimize impedance mismatch to comply with DFM standards.

In the Sdd21 plot depicted in Figure 4, the through-hole via stub in TL model A causes a resonant dip of -65 dB at 22 GHz. With stacked microvias, TL model B experiences Sdd21 of -3.6 dB at 10 GHz and -9.5 dB at 50 GHz respectively. Meanwhile, the impact of inductive impedance discontinuity due to the short bar in the staggered microvia, becomes significant for frequency range beyond 20 GHz. The Sdd21 encountered by TL model C is worsened by 0.1 dB at 20 GHz and 1.1 dB at 50 GHz respectively, versus TL model B. By shortening and widening the short bar in the staggered microvia, the Sdd21 at 50 GHz encountered by model D and E is reduced by 0.4 dB and 0.6 dB respectively, versus C. Therefore, stacked microvias experience the minimal insertion loss.

The impact of via structure is further studied by analysing an eye diagram at the receiving (Rx) end, with the injection of non-return-to-

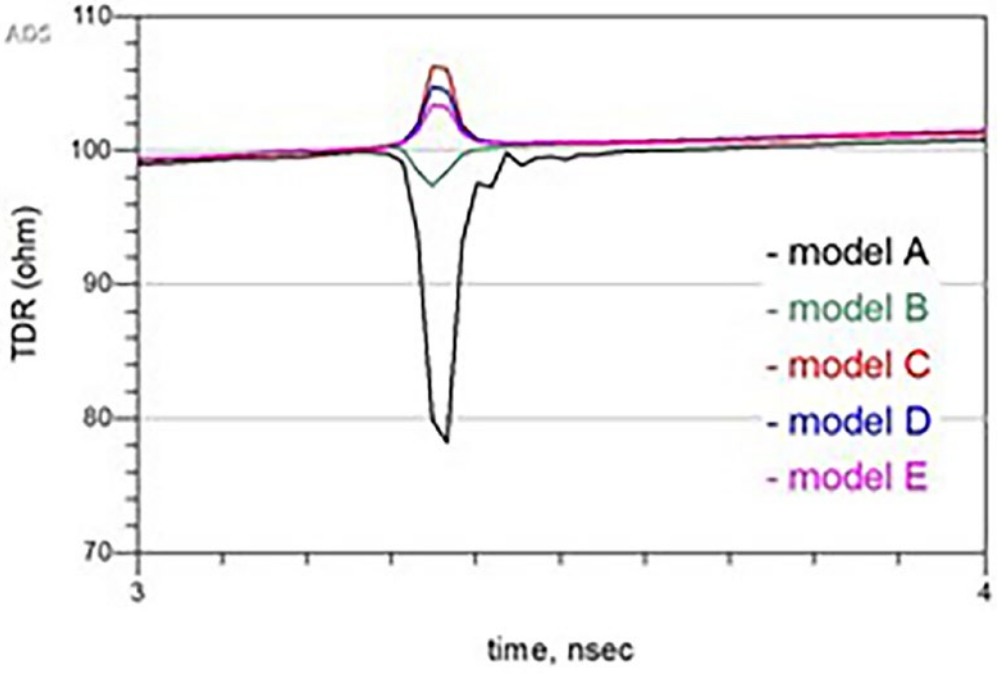


Figure 3: TDR plots for TL model A, B, C, D and E.

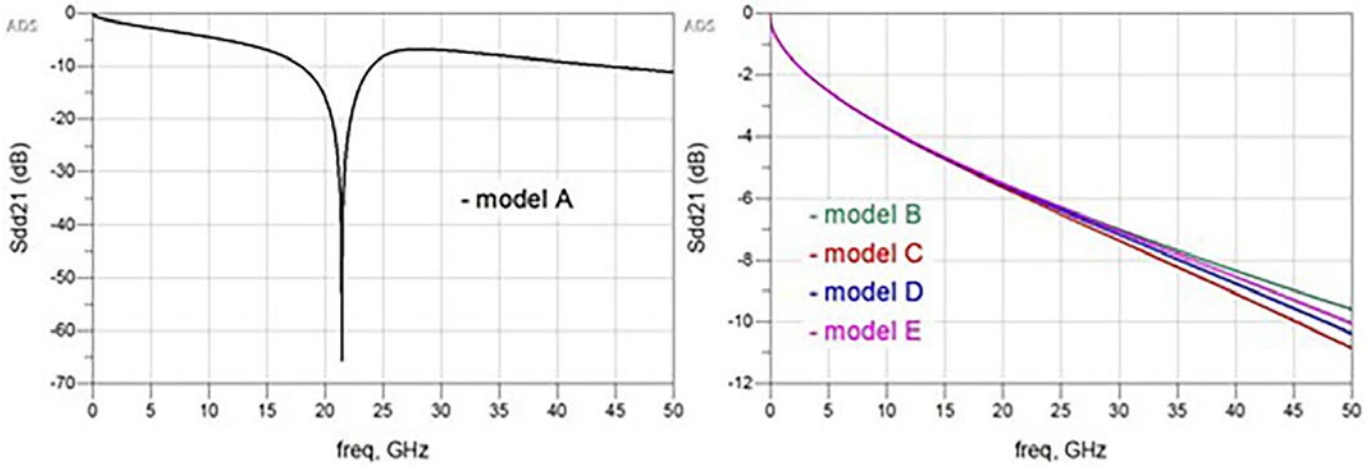


Figure 4: Sdd21 plots for TL model A, B, C, D and E.

zero (NRZ) signal at PCIe 4.0 16 Gbps (i.e., fundamental frequency 8 GHz, rise/fall time 25 ps) and PCIe 5.0 32 Gbps (i.e., fundamental frequency 16 GHz, rise/fall time 12.5 ps) on transmitting (Tx) end of TL model A, B, C, D and E respectively. The NRZ signal has 1 Vpp amplitude.

As shown in Figure 5, at 16 Gbps transmission, TL model B with a stacked microvia experiences an eye height of 657 mVpp. The eye height encountered by TL model C with a staggered microvia is reduced by 3%, to 638 mVpp. Model D and E with shortened and widened short bars in staggered microvias experi-

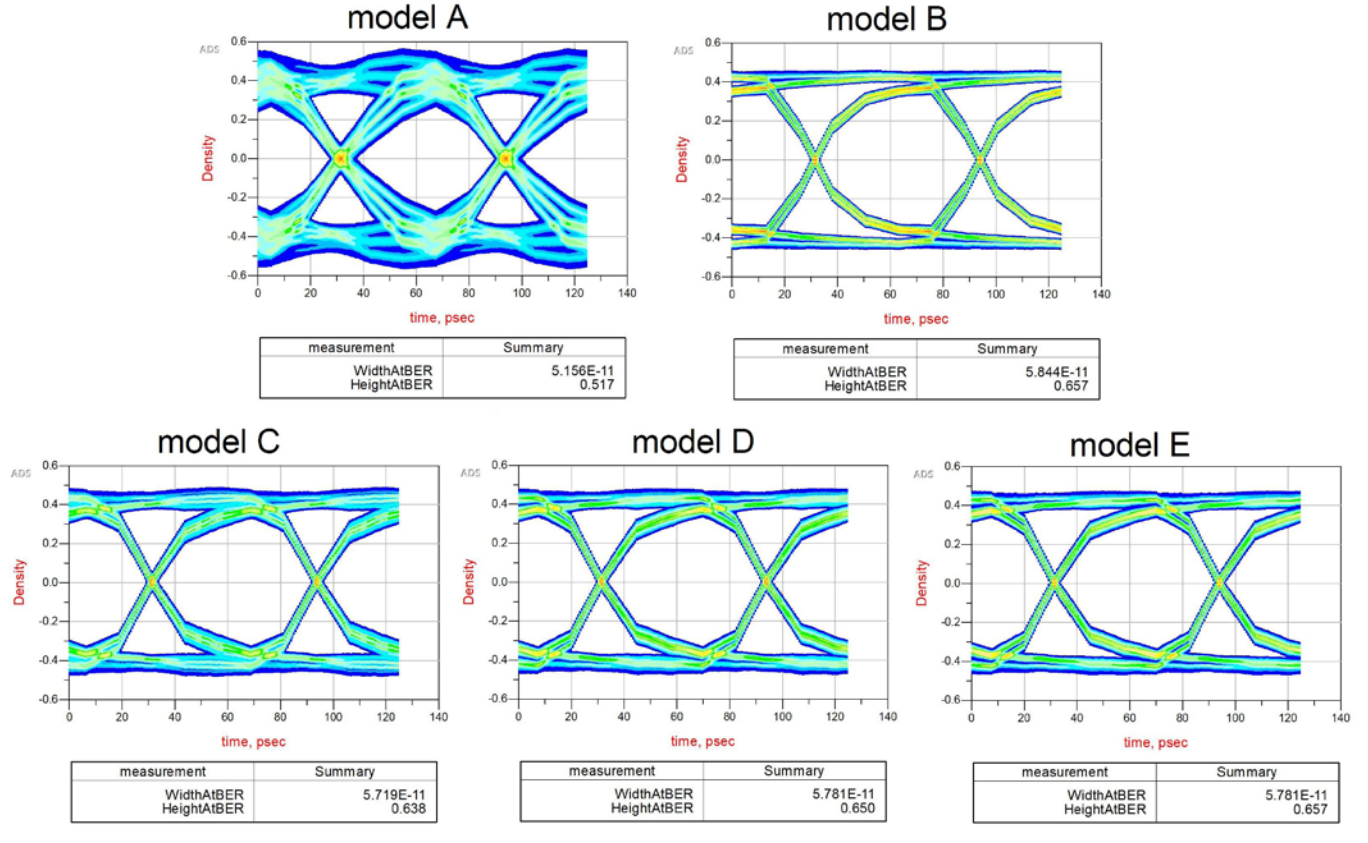


Figure 5: Eye diagram for TL model A, B, C. D and E at 16 Gbps transmission.

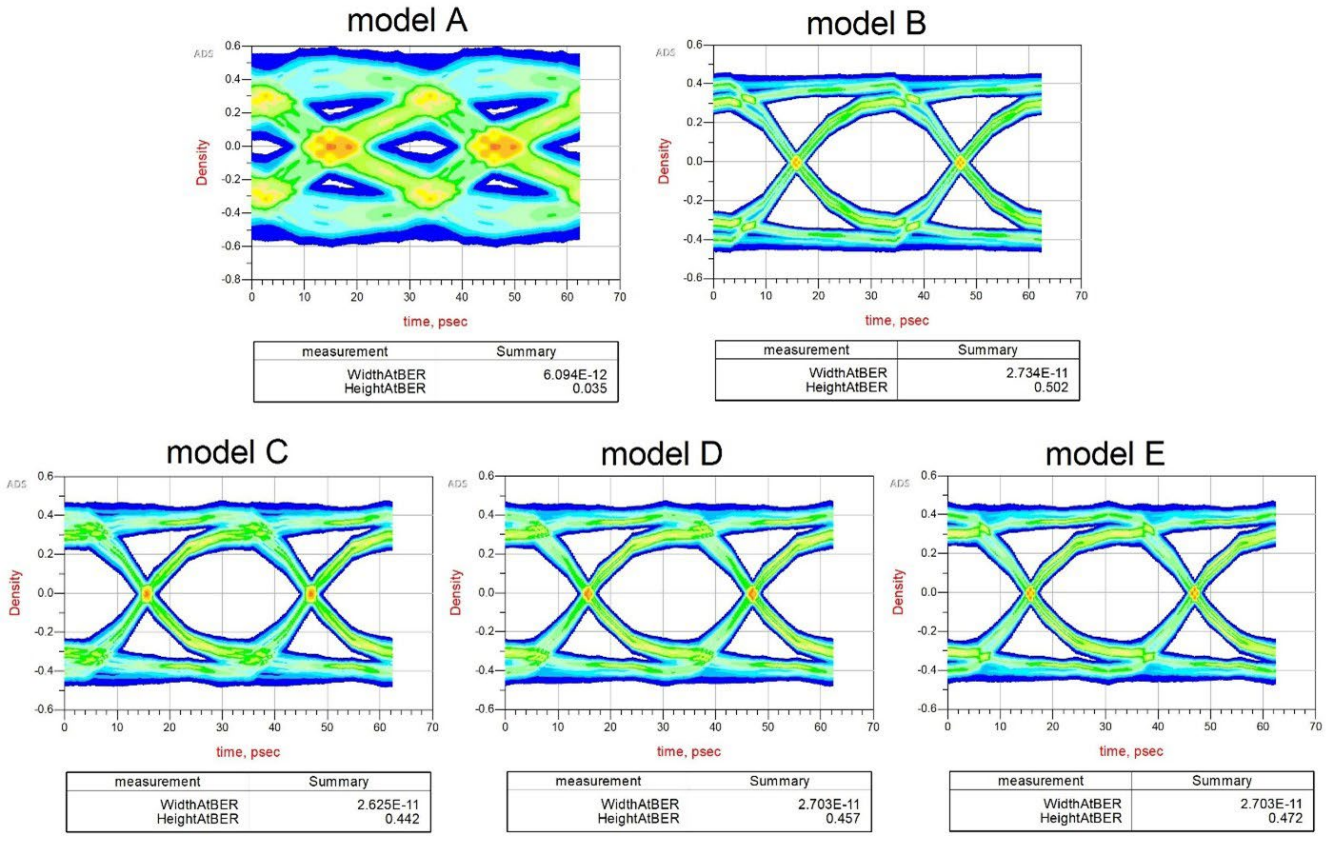


Figure 6: Eye diagram for TL model A, B, C, D and E at 32 Gbps transmission.

ence an eye height reduction of less than 1.1 % vs. B. The eye height faced by TL model A with through-hole via stub is diminished by 21 % vs. B, to 517 mVpp.

Meanwhile, regarding Figure 6, at 32 Gbps, TL model B with a stacked microvia experiences an eye height of 502 mVpp. The eye height encountered by TL model C with a staggered microvia drops by 12% to 442 mVpp. Model D and E with shortened and widened short bars in a staggered microvia experience an eye height reduction of less than 9% versus B. The eye faced by TL model A with a through-hole via stub is almost closed.

Summary

In this article, the performance of various via structures are compared relative to impedance discontinuity, insertion loss and eye diagrams. The analysis results indicate that stacked microvias encounter the least impedance discontinuity and insertion loss versus the other via structures. However, if staggered microvias are selected for signal routing, the short bar shall be kept at the minimal length to minimize impedance mismatch to comply with DFM standards. **DESIGN007**

References

1. Image courtesy of Sierra Circuits.
2. T. Kushta et al, "Resonance stub effect in a transition from a through-via hole to a stripline in multilayer PCBs," 2003.
3. E. Bogatin, "The Quarter-Wave Stub Frequency: Rule of Thumb #17," EDN, 2014.
4. Keysight EMPro.
5. Keysight ADS.

Chang Fei Yee is technical lead in hardware and signal/power integrity for Keysight Technologies.

Unfolding Altium's Design Education Platforms

Feature Interview by the I-Connect007 Editorial Team

For this issue on PCB design education, we knew we had to speak with Judy Warner, director of community and industry engagement for Altium. Judy has been heavily involved with Altium's educational programs, including the technical presentations for their annual conference, AltiumLive.

The I-Connect007 Editorial Team asked Judy to discuss Altium's educational plans going forward, specifically AltiumLive's transition back to live, in-person shows soon to be offered in a new hybrid format.

Andy Shaughnessy: Great to talk to you, Judy. We're discussing PCB design education and training. Would you give us an update on the next AltiumLive show?

Judy Warner: For AltiumLive, we were planning to go 100% virtual again this October, but for a variety of reasons, we did not do that. We've pushed AltiumLive out to January 2022 and we'll be co-locating the show with IPC APEX EXPO in the San Diego Convention Center.

Shaughnessy: That's a smart idea. It makes it easier for media to cover both shows.

Nolan Johnson: Definitely. I can't help noticing that IPC has been looking for a way to include designers in this whole conversation. You bring a critical mass of designers, and this addresses one of the larger issues we often discuss of getting designers to really understand what happens on the manufacturing floor.

Shaughnessy: Judy, how would you describe Altium's philosophy toward design education? How does this tie in with AltiumLive?

Warner: Overall, design engineers are at the heart of this work. Obviously, our end goal is to create and sell powerful software solutions that help design engineers to succeed. If they're not fully succeeding, then we need to step up as industry leaders to fill those gaps. We can create the best tools in the world, but if

Candor
Circuit Boards

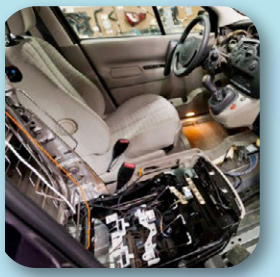
Innovative solutions for complex circuit board manufacturing

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

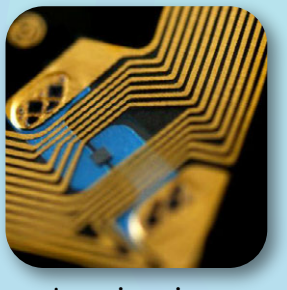
RF/Microwave PCBs

Power Electronics

High-Speed Data



Automotive Electronics



Lamination

Startups

Our technology produces what our customers dream!

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

we don't empower designers with educational resources to effectively leverage those tools and offer training in solid design principles in an intelligent and effective way, then it's an incomplete solution.

The more that we can be an empowering solution and resource provider, the more we allow them to not just succeed with our tools but help them successfully navigate the requirements of all their stakeholders and partners. This all happens within a connected industry ecosystem that provides both tools and complementary educational content. By co-locating AltiumLive at IPC APEX EXPO, we will be able to help connect designers to that event's manufacturing ecosystem. This is designed to be a physical manifestation of the digital ecosystem we are building on the heels of Altium's introduction of our cloud-based platforms Altium 365 and Nexar. We've had a tagline for AltiumLive for four years now: Learn, Connect, Get Inspired.

Obviously, our end goal is to create and sell powerful software solutions that help design engineers to succeed.

Johnson: The results are better yields and more affordable costs for the fabrication of boards because they made good choices. Plus, more resilience and the ability to fabricate at multiple fabricators because they were designed in a way that was interchangeable, suddenly that makes the OEM more profitable.

Warner: Exactly. In this way, we're correlating our customers' success to our own. Of course like every other company, we promote and sell software, we do all the typical activities that a business would do, but we believe at the heart of it, we must have a fundamental user and ecosystem-centric focus. Education is one way to embody this philosophy. Enabling them to have visibility, to be successful, to save money, to look good with their bosses, improve time to market, and all the things that indicate success for them, well then, we've been more than just a tool provider. We have entered a partnership with them where we're committed to a belief that when our customers and partners succeed, we succeed.

Johnson: Are there portions of all your multiple approaches that seem to be working better? Is there a front runner with respect to how your audience is connecting with your content?

Warner: First, there are webinars, because people can engage, ask questions, follow up, and watch it again. On-demand education does well, particularly technical on-demand education, where perhaps we reveal designers' blind spots. One example is a livestream Rick Hartley did for us on proper grounding that has almost 80,000 views. Design engineers do not get this kind of field theory at university, but it causes them problems regularly, and Rick shows them how to solve this pervasive problem. Another example of ecosystem-centric content that performed well was a podcast I recorded with Kelly Dack talking about laying out rails for assemblies. Honestly, I didn't think it would be that popular or engaging, but it is one of the most popular podcasts to date. I suspect that was just because it's a fundamental and practical thing they needed to know and could apply immediately. Kelly did a good job explaining the different ways to cut out rails, why they need to be a minimum width, and he brought on show-and-tell boards. It ended up being quite valuable.

We have two types of content: the tool content and the technical design education content. For most of our customers, they value having access to both. The podcast has really expanded, and we are quickly approaching

100,000 downloads. We have a monthly newsletter and that performs well. It's opened by tens of thousands of people, and it is translated into six languages, so it has a global audience.

We have our main Altium YouTube channel, which has many videos and playlists. The AltiumLive content lives there and is always accessible. The other channel that's really taken off is the Altium Academy YouTube Channel. That's where we put a lot of our educational resources. There are video versions of the podcast on our main YouTube. Our on-demand webinars are on both channels, so it's a good aggregator of our educational content.

Shaughnessy: Now, at your live events, you drew more designers than I've ever seen at a show. Didn't you have 300 designers at the last AltiumLive?

Warner: Yes. Actually, we had 500 designers at our last live conferences, both in San Diego and Frankfurt. However, we had over 2,000 per location when we went virtual and had over 8,000 people register. So, the appetite for education is there. We now have a full-time head of education, Rea Callender, and he's working with his team to develop Altium Academy, which will be more of a formal process-driven initiative. We're going to double down on educating designers.

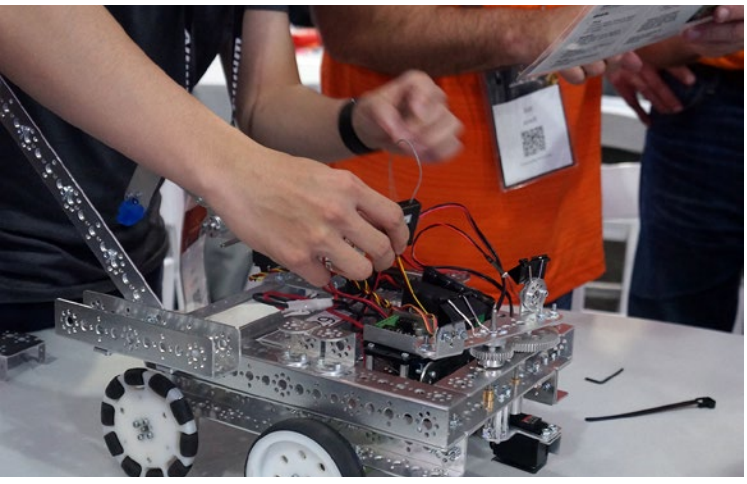
I've been handing off all I can to Rea and am very excited about what he and his team are doing as he gets the Academy off the ground. I think that's due to launch in the October/November timeframe. Rea put together our Upverter student education for high schools and universities, but then he will also be working on tightening up the quality of education. Eventually we're going to have a certification program and do all kinds of exciting things, so stay tuned for that. Like many, our education team believes that training needs to be in bite-size pieces, modular, chaptered, and digestible, and then you have something meaningful at the end. It's immediate ROI that shows up for design engineers in their next day at work and that's why we think credentialing eventually is going to be somewhere we absolutely need to go.

Judy Warner

Shaughnessy: You mentioned the Academy about a year ago; it was just about to get started.

Warner: Yes, then COVID hit, and it was sidelined temporarily. But now it's fully underway, and it's got traction. Rea will also be working with the IPC Education Foundation during AltiumLive. I think they might be doing some student design competitions with high school students; we really believe that we must expose, engage, and inspire students at the high school level so they know what to major in at university. The Upverter education program is growing like a weed, and we're hoping Rea will do the same thing for university students and then for professionals, and eventually we will offer certification. So when you get this training, you'll be able to test and certify against it with proctored exams and put it on your resume.

Shaughnessy: As we see with these classes, even online during COVID, people are thirsty for design knowledge.



An example of AltiumLive 'magic' is the robot team challenge.

Warner: It's so true. It's interesting to engage with universities that are trying to prepare students for industry, but there's a gap between industry and the theoretical and principle-driven kind of electrical engineering courses. Hands-on time in the labs or involvement in engineering student competitions go a long way to help prepare students for industry.

Often they do a capstone project for college that involves PCB layout. They design a simple two-sided board, turn a tassel, and they get thrown into a big OEM job where they're told, "You're going to lay out circuit boards now." They're thinking, "Well, I laid out a 2-inch by 3-inch board for a robot I made in college and now I'm being asked to do HDI and stacked microvias." They're like a deer in the headlights. We want to be part of preparing students for industry in a meaningful way that's recognized by industry.

As an industry, we're always trying to backfill the loss of expertise as our old friends are retiring and we're losing this amazing expertise, wisdom, and tribal knowledge that's retiring with them. Altium has tried to capture some of their knowledge and wisdom, then make that accessible to the entire industry. One of my favorite things to facilitate at AltiumLive is connecting the Rick Hartleys, Happy Holdens, and Eric Bogatins of the world with students, and they sometimes end up doing design reviews or mentoring teams. The same goes for younger professional engineers; it's a magical thing to watch the sparks fly in what we like to call the AltiumLive 'magic'.

There's enough room for everybody. I'm a huge fan of PCEA. I think these education platforms, whether it's PCEA, IPC, Altium, Cadence, Mentor (Siemens), or wherever people can get training. We can't get enough good educational content to these folks fast enough.

Shaughnessy: I know you guys do a lot of work at the college level. You have a student version of Altium Designer, right?

Warner: It's not a student version; it's a time-based license of the full Altium Designer tool. But yes, we do have an academic website and we give away software to engineering teams at universities or to individual students for free on a limited-time basis until their graduation. We've seen a huge uptick in the use of those tools. They may have been using free or open-source tools before, but that just doesn't give enough weight on their resume when they're going to work for SpaceX, Tesla, or Apple. These companies want to see that they've used a professional tool and so it really helps them get jobs.

Shaughnessy: It may be their first real EDA tool.

Warner: Right. Another fun channel we have on YouTube is called Altium Stories. One story shows the kids from Delft University who made an exoskeleton and entered it in a commercial competition against commercially funded companies—and they won. These kids just rock the world, and these are the kids who are getting pulled right out of college and getting the best jobs.

Shaughnessy: It's funny that we keep hearing about how the kids of tomorrow are slackers, but the kids that we meet in this industry are really dynamic, hard-working people.

Warner: They're brilliant.

Shaughnessy: I wonder: Do they have different expectations when they go to a conference or trade show? Is it any different teaching to them? I wonder if it's like they say about EDA software: When millennials buy software, they want it to work right out of the box.

Warner: Well, when you think about it, these students are digital natives. Unlike us, they've grown up on the internet using software. Using technology is like breathing for them, so we're just trying to stay a step ahead of that with tools that have an intuitive and appealing user interface that works like the software and games they grew up on. I think education is going to be the same way. It's an Amazon world, and it's a Google world.

At AltiumLive, we brought all the experts and sound technical training. But then we also brought robot battles and connected the old pros to the young engineers. It was a hoot to watch two designers in their 20s—one from Uber, one from Lyft—talking to Carl Schattke from the largest EV company in the world. He brought out his Mylar tape-ups to a cocktail reception; it just blew their minds, and they loved it. They loved it because they had no idea where we came from, because they grew up with the internet and screens.

Shaughnessy: Is there anything else you want to mention that maybe we haven't talked about?

Warner: One thing I didn't mention, as far as resources go, is our massive collection of technical content available on our resource page—resources.altium.com—which has thousands of blogs, including some by industry thought-leaders, designers, and some of our own video content.

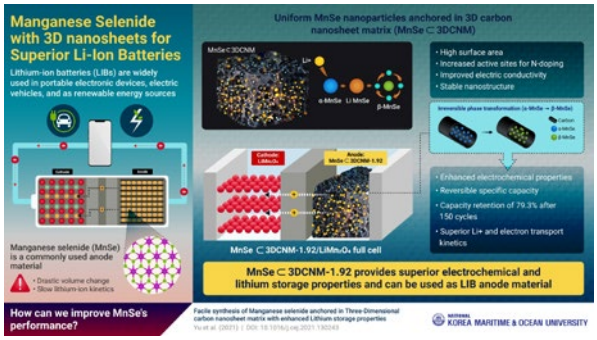
Shaughnessy: It's been great talking with you, Judy. I'm looking forward to some live trade shows.

Warner: Great talking to you, Andy. **DESIGN007**

'Wrapping' Anodes in 3D Carbon Nanosheets: The Next Big Thing

Recently, researchers from the Republic of Korea have found that embedding manganese selenide anodes in a 3D carbon nanosheet matrix is an innovative, simple, and low-cost means of reducing drastic volume expansion while improving the energy density of these batteries.

Lithium-ion batteries (LIBs), which are a renewable source of energy for electrical devices or electric vehicles, have attracted much attention as the next-generation energy solution. However, the anodes of LIBs in use today have multiple inadequacies, ranging from low ionic electronic conductivity and structural changes during the charge/discharge cycle to low specific capacity, which limits the battery's performance.



In search of a better anode material, Dr. Jun Kang of Korea Maritime and Ocean University, along with his colleagues from Pusan National University, Republic of Korea, has designed an anode that, owing to its unique structural features, overcomes many of the existing barriers of anodic efficiency.

The team is excited about the potential implications of their accomplishment. As Kang explains, "Using a conducive filler scaffold, we have developed an anode that boosts the battery performance while simultaneously allowing reversible energy storage. This strategy can serve as a guide for other transition metal selenides with high surface areas and stable nanostructures, with applications in storage systems, electrocatalysis, and semiconductors."

(Source: Korea Maritime and Ocean University)

The Split Planes Challenge

Connect the Dots

by Matt Stevenson, SUNSTONE CIRCUITS

PCBs don't explode every day, but if your design calls for more than one voltage on a power plane layer, spontaneous combustion is just one of the many problems this tricky challenge for PCB designers can cause.

It may seem counterintuitive, but when a PCB fails spectacularly, the cost is less because it's both immediate and easier to measure. If you make a small crater the first time you turn on the first production device, you lose an entire lot of boards and the time spent on designing and prototyping. When less noticeable or intermittent issues occur, they can eventually cost far more depending on how long they remain undiscovered.

The severity of the PCB failure seems to have an inverse relationship to the amount of effort required to fix it. When there's smoke coming out of your PCB, it's relatively easy to find out why. Digital glitches and signal anomalies are more subtle issues that can take many hours of tedious detective work to solve.

Split planes are enclosed regions on an internal plane layer, and problems with them can be among the most difficult design flaws to ferret out. The further a board makes it through the development cycle without someone noticing, the more it costs. Production delays and resources directed at solving the problem can be quite costly, but the worst-case scenario involves malfunctioning boards making it out into the field. Those events create warranty claims, product recalls, damaged reputations, and lost jobs.

Here we will explain why managing split planes can be so challenging and examine some best practices for avoiding this common issue.

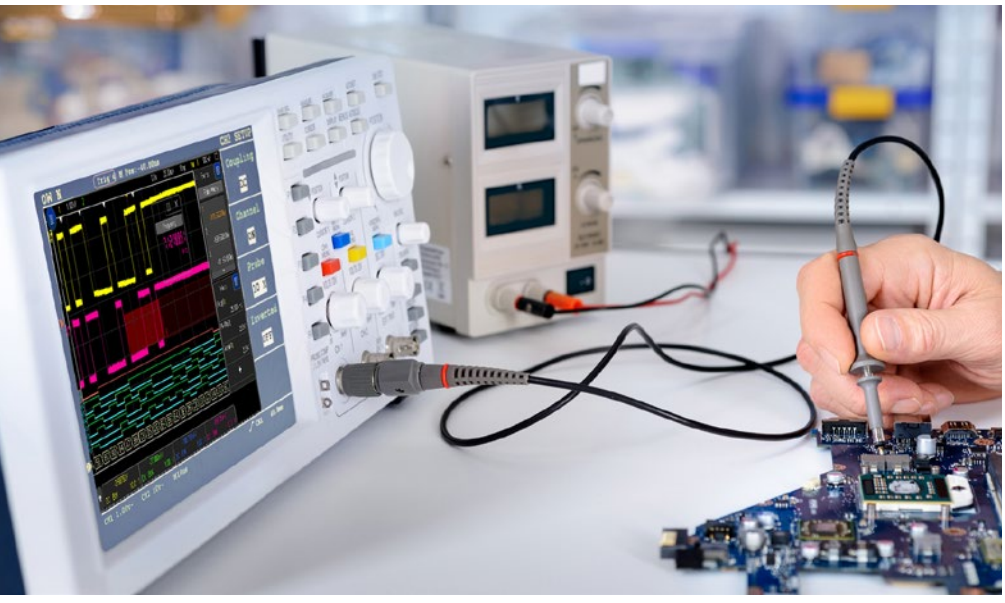
Why Does This Happen?

There often exists a gap between what a design tool allows the designer to do with split planes and how difficult it can be to recognize issues with the design.

Your PCB design tool will offer the ability to assign voltage amounts to the entire plane, but that can lead to problems if your design assigns multiple voltages to the same plane. Problems are avoided

cadence®
CHANNEL PARTNER

EMA Design Automation®



if you divide the plane into separate, electrically isolated areas using a split plane.

Just because different voltage amounts are "isolated" doesn't mean you're out of the woods. When more than one voltage is assigned to a power plane layer, your CAM tool does not check whether the voltage of your via matches the section of the board it's going into. If you have a 12V via going into a 3V section of the board, it will fail.

Mixing digital and analog signals can also create headaches. An analog ground is noisy, and it fluctuates, while digital grounds are quieter and steadier. Connecting digital and analog planes together probably won't result in a board failure that trips the smoke detector, but there will be performance anomalies from the noisy analog ground interfering with signals on digital pins.

If you are relying on manual oversight to discover such issues before production, it is easy to lose track of individual connections—especially on a complex board design with hundreds of vias.

How Do You Best Manage Mixed-Signal Output?

When you need to assign one or more power nets to a layer on a simpler design, applying plane splits that segregate areas containing each voltage is a simple and efficient way to distribute power.

Warning! This method relies on the human eye to ensure accuracy and gets risky for more complex board designs.

We think a better method is to use polygon area fills to make connections. With polygon area fills, when you name every net connection on each layer, they become visible. Vias of different nets will not connect. The enhanced visibility makes it so your CAM tool can perform error checking on your separate area fills and automatically recognize mismatched voltages.

How Do You Prevent Mixed-Signal Problems?

Digital and analog signals don't play well together. Here are some actions you can take to prevent problems with mixed-signal output.

- Keep the power ground and control ground separate for each power supply stage.
- To keep your digital and analog grounds separate, build in a small impedance path. This will limit power circuit interference and help protect your control signals.
- Partition your PCB with separate digital and analog areas.
  - Make sure digital and analog components are assigned to their respective areas
  - Never route digital signals through analog territory and vice versa
  - Straddle analog-to-digital converters along the border of the two territories
  - Use a single, solid ground plane
  - If you must route a signal trace from one area to the other, place it entirely over the PCB's ground plane

- Pay attention to routing. For example, you want your analog grounds crossed only by analog lines. This will reduce capacitive coupling on a large ground plane with lines routed above and under it.

A Designer's Best Friends: Vigilance and Awareness

Designing with voltage top of mind is vital, as is knowing what the design tool's error checker will and will not catch regarding connections to split planes. This is how you avoid burning boards and doing costly rework. If you build your designs such that you can rely on automated error checking, we believe that is the best method for dealing with complex power plane schemes. **DESIGN007**

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

ALL SYSTEMS GO!

Challenges in Analyzing Today's Hyperconnected Systems

Column Excerpt by Suketu Desai
CADENCE

Today's data-thirsty world is looking forward to the next-generation communication systems beyond 5G, the promise of massive connectivity to the internet with extreme capacity, coverage, reliability, and ultra-low latency, enabling a wide range of new services made possible through innovative and resilient technologies. The exponential growth in data speed and networking has introduced numerous design and analysis challenges across a system design. Design teams are challenged to deliver new, differentiated products faster and more efficiently, despite the ever-growing complexity of silicon, package, board, and software for many complex applications in the hyperscale computing, automotive, mobile, aerospace, and defense markets.

Fragmentation Challenges

Design flows are fragmented across chip, package, board, and system levels, making end-to-end simulation harder. An accurate 3D model is the most accurate and reliable method to achieve structure optimization and high-speed compliance of the complex structures found in silicon interposers, rigid-flex PCBs, stacked-die IC packages, connectors, and cables. A high-fidelity interconnect design is a critical factor for high-speed signaling, such as 112G SerDes interfaces that are highly susceptible to any change in impedance that negatively impacts the bit error rate. Optimization entails extensive what-if analyses, including dozens of complex extractions and simulations. The alternative method using legacy tools requires users to partition a design, do piecemeal modeling and analyses, and stitch the results together. This method introduces accuracy loss, user errors, and risk. In addition, the need to merge mechanical structures such as cables and connectors with the system design to create one comprehensive model and simulate as a single piece cannot be accomplished.

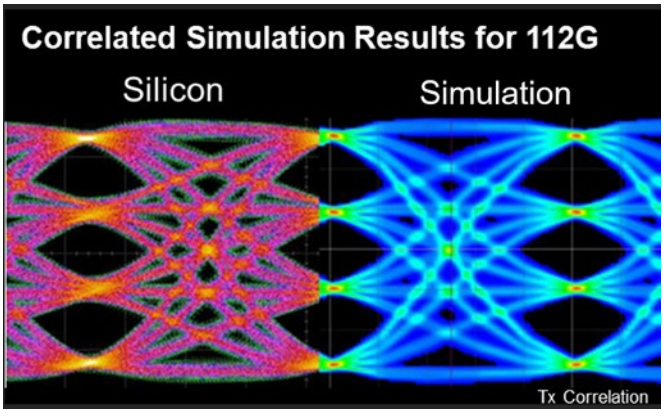


Figure 1: Real measurement from PAM4 silicon (left) compared with modeled output (right).

Click here to read the rest of this column.

Suketu Desai is senior director of product engineering, Multiphysics System Analysis Group, Cadence Design Systems, Inc.

MilAero007 Highlights

Defense Speak Interpreted: POM—Explaining the Process for Defense Budgeting ▸

Anyone hanging around Defense programs will have surely heard of the term "POM." Most of the connotations I have heard say that if you have a POM or will get "POM'd," your program is "skating on solid ice." That led me to infer that if you were in the POM, your program was established. But why and how?

Catching Up With Prototron's Kim O'Neil ▸

I recently sat down with industry veteran and friend Kim O'Neil to get to know more about his story. From his time his two tours in Vietnam, to his time in our industry, Kim has always been a solid contributor wherever he has served. Having known and worked with Kim for a number of years I knew that his was an interesting story that should be told.

Ducommun, an Airbus Detail Parts Partner, Awarded Multi-Year Contract ▸

Ducommun Inc., a global supplier of innovative electronic and structural solutions for the aerospace and defense industry, announced that it has been recognized as an Airbus Detail Parts Partner (D2P) and awarded a long-term, five-year contract to provide a titanium work package for key products on the A320 and A330 programs.

U.S. Navy Awards Sikorsky Contract to Build CH-53K Heavy Lift Helicopters ▸

Utilizing proven advanced manufacturing techniques and continuing to drive down unit price, Sikorsky will build nine more CH-53K™ heavy lift helicopters under a new contract for the U.S. Navy.

International Space Station Astronauts Install Two Boeing Advanced Solar Arrays ▸

Two new, advanced solar arrays provided by Boeing have increased the International Space Station (ISS) power supply after European astronaut Thomas Pesquet and NASA astronaut Shane Kimbrough installed them during three spacewalks in June.

Micross Expands Capabilities; Acquires Microelectronics Business Assets of Ultra CEMS ▸

Micross Components, Inc., a leading global provider of mission-critical microelectronic components and services for high-reliability aerospace, defense, space, and industrial applications announced the expansion of its hybrid and assembly business with the acquisition of assets of the microelectronics business from Ultra CEMS, a leading provider of high-reliability hybrid electronics engineered for the harshest environments.

NASA's Self-Driving Perseverance Mars Rover 'Takes the Wheel' ▸

NASA's newest six-wheeled robot on Mars, the Perseverance rover, is beginning an epic journey across a crater floor seeking signs of ancient life. That means the rover team is deeply engaged with planning navigation routes, drafting instructions to be beamed up, even donning special 3D glasses to help map their course.

www.prototron.com

Serious About Speed

For over 30 years Prototron Circuits has led the pack when it comes to providing quality circuit boards FAST.

Be it Class 3, AS9100, ITAR or MIL-PRF-31032, Prototron has the speed you need.

Prototron Circuits

America's **Board Source**

Educational Accolades for Our Chapter Liaison

The Digital Layout

Feature Column by Kelly Dack, CIT, CID+, PCEA

Introduction

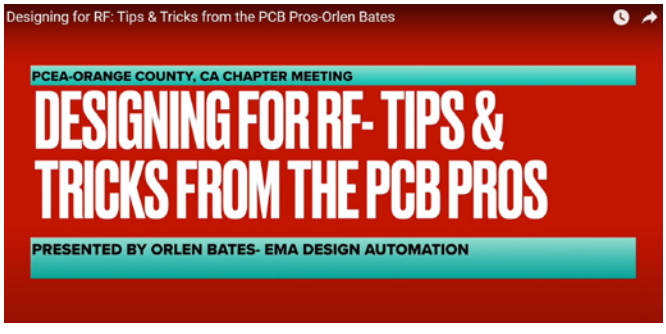
In this month's column, I give kudos to PCEA Chapter Liaison Scott McCurdy, who has been capturing the efforts of the PCEA to educate our members and our industry as a whole; then, I pass the mic to PCEA Chairman Steph Chavez to provide some thoughts on who's responsible for continuing education—you or your employer?

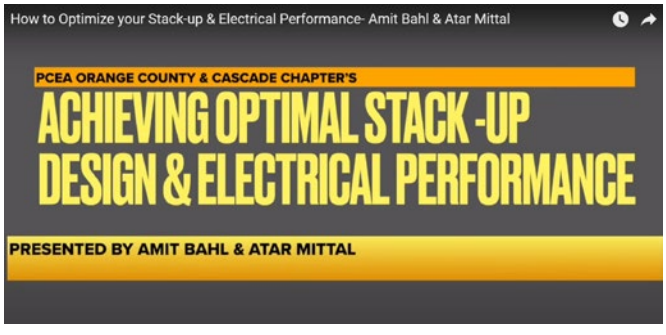
Finally, I provide our readers with a growing list of events for 2021.

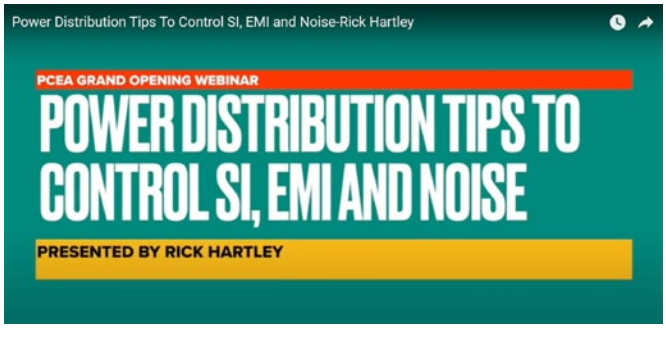
PCEA Updates

Scott McCurdy has been a tireless force in collaborating within the printed circuit engineering industry to bring people together. Not only has Scott worked hard in the industry as director of sales and marketing for Freedom CAD Services in Orange County, California, but for years he has worked outside of the box, presiding as a leader in trade organizations that specialize in educating their members.

Presently serving as PCEA's chapter liaison and Orange County chapter president, Scott coordinates all the PCEA chapter leaders and inspires them to move in a consistent and helpful direction. Local PCEA chapters are actively planning their annual itineraries to serve our industry. As a leader of the Orange County chapter, Scott has worked to





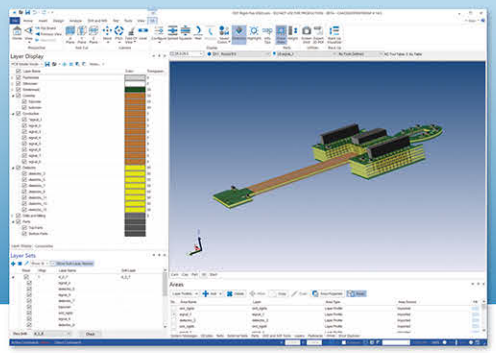


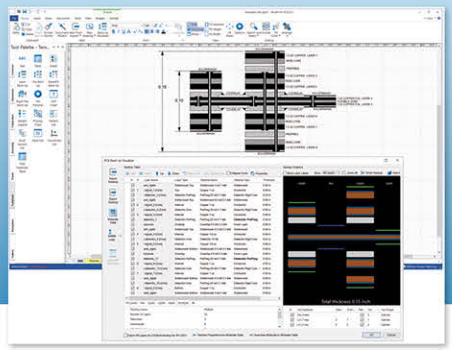
Support For Flex, Rigid Flex and Embedded Component Designs Now Available.

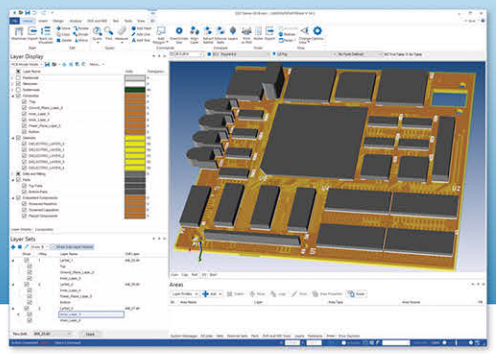
BluePrint-PCB®

CAM350®

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







DownStream Technologies

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

capture his local chapter activities. From the start of PCEA and throughout the pandemic of 2020, Scott worked with chapter members and industry educators to create a PCEA YouTube channel to spread educational presentations.

Message from the Chairman

by Stephen Chavez, MIT, CID+

This month's topic on education is a good one. I feel it's important for many of us, especially those of us involved in printed circuit engineering, fabrication, assembly, and testing of PCBs. For me, continued education/professional development is a must to stay relevant and competitive within our industry. When I think of this topic, there are a few things that immediately pop into my mind. First, "Who's responsible for your continued education and professional development?" Is it your employer? Is it you as an individual? These are questions I hear, and they are debated often.

Overall, I get mixed responses. Some say it's your employer's responsibility to provide continuing education and professional development, while others say that it's ultimately your own responsibility. I strongly feel it's you as an individual who is ultimately responsible for your own continuing education and professional development, not your employer. I also feel that a good company in today's industry will view their employees as their own internal best assets and best long-term investments to be successful.

Therefore, such companies will provide opportunities and look for creative ways to provide continuing education and professional development for their employees so that the company will get its best return on investment for the overall success of the company. This also helps to cultivate a positive culture and working environment so that the potential for long term success is most likely to be achieved. Sadly, not every company takes this approach. The negative side of a company not providing these opportunities may potentially lead to a slowdown or lack of evolution, longer than normal project schedules, excessive budgets, and in worse cases, a high employee turnover rate leading to lack of long-term company success.

Stephen Chavez

For those who feel that it's their employer's responsibility to provide for continuing education, they are at the mercy of their company for their individual evolution, which may happen at a slower pace, not at all. This will definitely have a negative impact on the company as well. It becomes a double negative. No one wins if both the company and the employee feel they are not responsible for continuing education and professional development.

Here's my advice: Take ownership of your own continuing education and professional development, whether your company provides the opportunity or not. Invest in yourself because you are your own best asset. In today's industry, especially as we have adapted and evolved during the pandemic, there are so many opportunities to take advantage of. Between all the available online webinars and course offerings, to the many conferences now getting back to "face-to-face" engagements, there is simply so much more opportunity today than there was a decade ago.

PCEA is one of the many industry associations where you find awesome opportunities for continuing educational and professional development. A great example was the recent PCEA-Orange County chapter event led by Scott McCurdy. This event had strong attendance because it shared valuable industry content. This led to tremendous collaboration, education, and inspiration for the betterment of those who attended; this greatly benefits the industry overall. PCEA will continue to do its part of collaborating, educating, and inspiring within the industry by providing such opportunities. Be sure to stay tuned for up-to-date opportunities and industry events listed on the Events page of our website.

As always, if you have anything to do with printed circuit engineering, I highly recommend you get involved with and join the PCEA collective, if you have not already. By joining PCEA, your percentage of long-term professional development increases significantly.

I continue to wish everyone and their families health and safety. Best of success to you all in 2021.

Warmest regards,
—Steph

Next Month

There are a lot of items on the PCEA roadmap which were presented in our July 9 meeting. As we evaluate our important to-do list, I'm not sure which of the items will be moving forward first. I'm sure next month's PCEA column topics will be a potpourri of subject matter coverage as the PCEA moves into trade show season.

Upcoming Events

Below is our list of upcoming events. Hope to see you there.

DesignCon 2021
August 16–18, 2021
San Jose, CA

PCB West 2021
October 5–8, 2021
Santa Clara Convention Center
Santa Clara, CA

SMTA International 2021
Nov. 1–4, 2021
Minneapolis, Minnesota

PCB Carolina 2021
Nov. 10, 2021
Raleigh, North Carolina

productronica
Nov. 16–19, 2021
Munich, Germany

Spread the word. If you have a significant electronics industry event to announce, please send the details to kelly.dack.pcea@gmail.com, and we will consider adding it to the list. The effects of the pandemic are still in a state of flux, so watch this column and our website for the most current information on scheduled events. If you have not yet joined PCEA, please visit our website, pce-a.org, and find out how to become a PCEA member.

Conclusion

Educating the industry is one of the key goals of the PCEA. Our leaders have been delighted to find so many educators in our industry willing to provide their knowledge and allow us to capture and present it in the spirit of altruism. As we continue to collaborate, inspire, and educate as a nonprofit, please take time to watch, comment, and thank the many presenters we bring to you as a service.

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

Kelly Dack, CIT, CID+, is the communication officer for the Printed Circuit Engineering Association (PCEA). To read past columns or contact Dack, click here.

The Benefits of Understanding Engineering Disciplines for PCB Design

Lightning Speed Laminates

Feature Column by John Coonrod, ROGERS CORPORATION

I started my PCB industry career in 1987 in the Flexible Interconnect Division of Rogers Corporation. That division of Rogers was sold many years ago, but I learned a tremendous amount working there. My first responsibility was learning flexible circuit design and I was in the ideal surroundings to learn.

I had responsibility for a new product from beginning to end of life. I designed flexible circuits, many times from scratch using schematics and mechanical drawings. In the beginning phase of the product life cycle, I designed the circuit and panel layout, designed the tooling, and established the build sequence for the prototype. In my learning stage and after all the tooling arrived, I went to the prototype department and helped them build the circuits from beginning to end. I cannot express how valuable that experience was, seeing the tooling I designed, working with the various process steps in the build sequence, and finally shipping the parts.

After multiple revisions of prototype circuits, the customer would usually go into high-volume production, and it was a very different circuit design discipline for designing the circuits and tooling for mass production than it was for prototype circuits. High-volume production requires the designer to consider all customer circuit properties and specifications, and perform CpK analysis to ensure the circuit will have good manufacturing yield at high volumes.

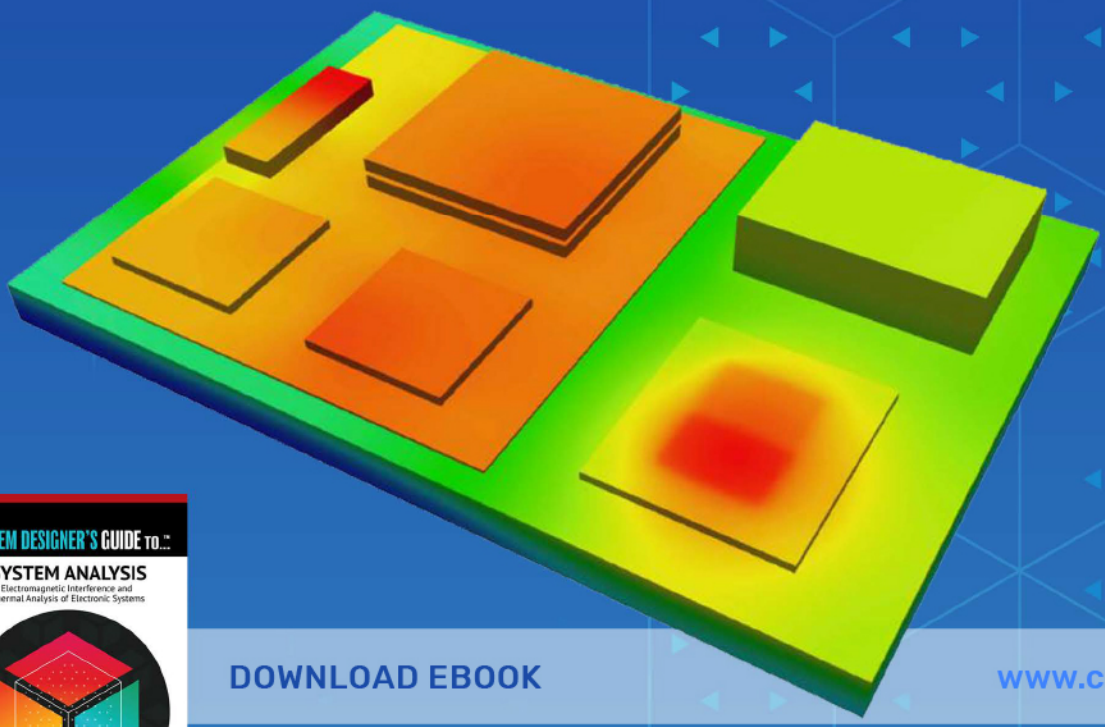
Over the years, I've taught several engineers circuit design. I am convinced that if the designer has a very good understanding of the different process steps it takes to make a circuit, they will be a much better designer. As an example, if a designer encounters a need for exception when designing for annular ring, the designer who knows the circuit fabrication process well will make a much better design decision than a designer who only knows the annular ring design theory.

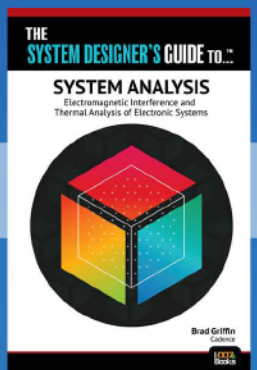
cadence®

Thermal and Stress Analysis of 3D-IC Systems

The explosion of data, both in volume and speed has increased the need to detect and mitigate thermal issues early in the design process to reduce the number of design iterations. This requires complete thermal analysis and signoff from chips to enclosures in a system. On a 3D-IC system, it is even more compelling. The massively parallelized Celsius™ Thermal Solver with its FEM and CFD engines and transient and steady-state capabilities offers the most complete thermal analysis, up to 10X faster and with virtually unlimited scalability.

WATCH NOW





DOWNLOAD EBOOK

www.cadence.com

My circuit design experience also included rigid board design, which was different than flex circuit design, but the basics were the same. After my circuit design experience, I was a technical support engineer for the flex circuit materials division of Rogers and later a technical support engineer for the high frequency circuit materials division of Rogers. In the technical support role, I worked directly with our circuit fabricator customers and that caused me to work with many processes for making flex circuits and rigid high-frequency PCBs. Also, because I worked at a materials company, I had to understand the material engineering aspects of the circuit material, how it interacted in different PCB processes, and its end-use. Additionally, I was a part-time student in electrical engineering at ASU during most of my engineering experience in the industry. My diverse engineering experience was influenced by the engineering theory while I was working in the industry. The overall diversity of my engineering background has proven to be extremely beneficial for me at Rogers Corporation.

My point is that having a very diverse engineering background is extremely beneficial for understanding PCB designs.

My point is that having a very diverse engineering background is extremely beneficial for understanding PCB designs. I think most engineers today probably do not have the opportunities I had. I would strongly recommend design engineers get very involved in circuit processing, manufacturing, and circuit materials engineering. When a PCB designer understands these different aspects of PCB technology, they will certainly be able to deliver circuit designs that will be more robust and have better manufacturing yields. The high-volume manufacturing yields of a PCB are directly related to profit and that is the connection between good circuit design and a successful circuit-product.

An aspect related to PCB design is software. There are many different software programs that the circuit designer needs to work with, and some of these programs allow the designer to run simulations. This is another tool for designers to build on their design expertise. The simulation software can be used to model a circuit design, change different features of the circuit, run the simulation again, and learn what the design change will do for circuit performance. The benefit of running many different simulations can be very educational for a circuit designer.

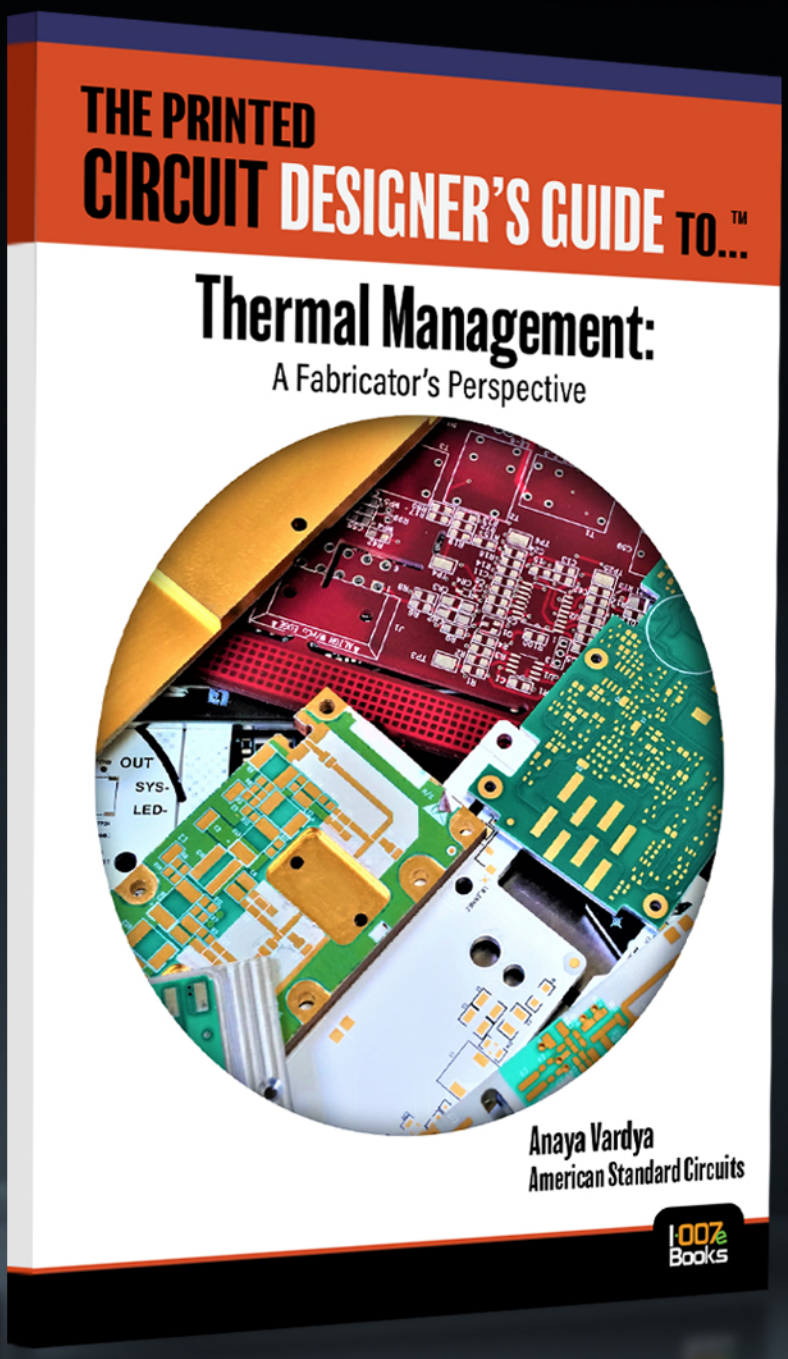
Another aspect of engineering that can be beneficial for the PCB designer is having a good understanding of practical statistics. Since PCB design is focused on the customer's specifications, it is beneficial for the designer to have a good understanding of standard deviations, CpK analysis, histograms, box plots, etc. These statistical tools help with the circuit design process, but they can also help with optimizing yields, troubleshooting, failure analysis, and design modifications.

In summary and in my opinion, the following topics need to be well understood for a PCB design engineer to gain expertise: circuit materials, major processes in PCB fabrication, defining stackups, understanding the PCB build sequence, statistics, and simulation software. Additionally, there are many resources for PCB design at IPC.org, IEEE.org, and the websites for companies that make PCB software for routing and stackups. **DESIGN007**

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

Master the art of keeping cool!

Learn how to beat the heat in your designs with techniques and methods from a fabricator's perspective.



I-007e Books

I-007eBooks.com

DOWNLOAD NOW

Selecting Resins for Effective Resistance in End-Use Environments

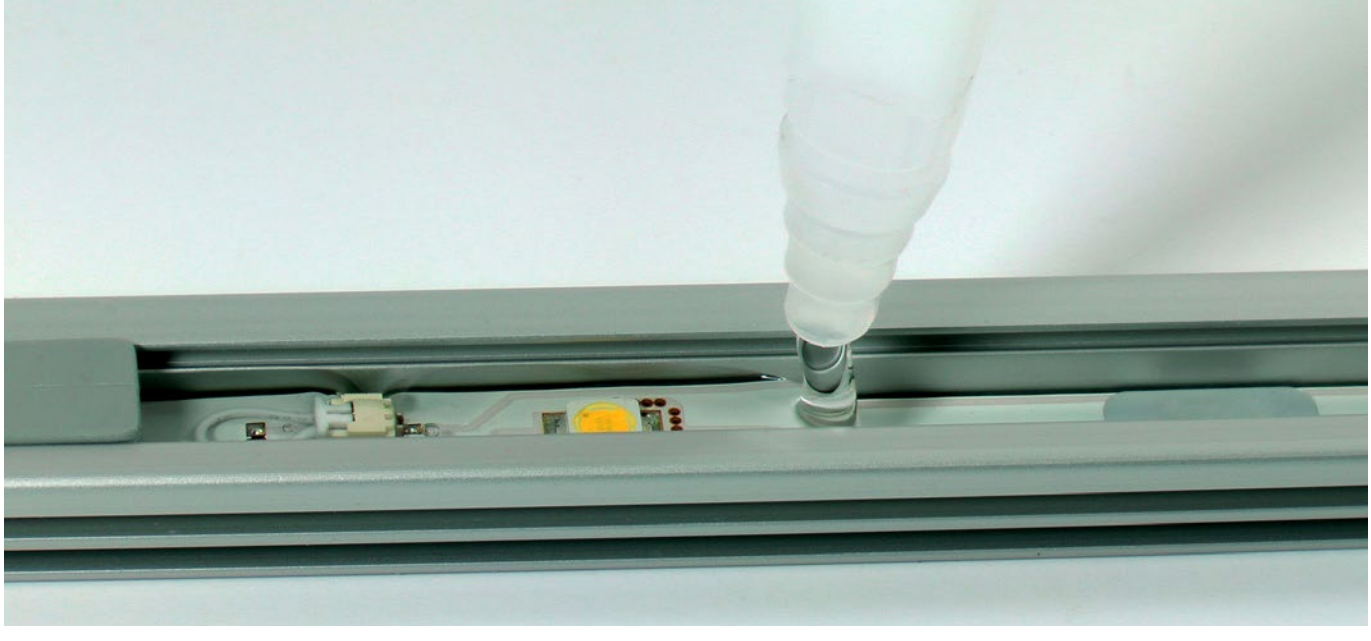
Sensible Design

by Beth Turner, ELECTROLUBE

Last month, I started this series of columns on resins by going back to basics, questioning the core rationale for resin choice and application. I will be exploring this area in more detail over the forthcoming months, to assist you in your pursuit of steadfast circuit protection. For this month's column, let's take a closer look at the fundamental chemistries of potting and encapsulation resins, and examine how each resin type offers individual properties that can be exploited to maximise performance under a wide range of environmental conditions. When you embark on resin selection, there will be critical points that arise for effective resistance in end use environments, so let's explore these in a bit more detail.

First, let's begin by looking at the archenemy of electrical and electronic devices, the dreaded "moisture." On top of producing short-circuits, moisture also causes corrosion, which leads to premature deterioration of components. So, what resin is best suited to combatting moisture? Polyurethane resins. These are usually supplied as two-part products that achieve the desired cure when mixed, and provide that all-important moisture resistance, as well as excellent electrical insulation, flexibility, and good adhesion to most substrates, both metal and plastic.

For more intensive moisture resistance, such as marine applications, there are polyurethane resin systems available that provide exceptional resistance to seawater and temperature extremes, making them an ideal choice for marine applications such as sensor encapsulation. Another good example of a polyurethane



IPC BUILD ELECTRONICS BETTER®

PCB DESIGN COURSES

DRIVE QUALITY AND INNOVATION

Introduction to PCB Design II	Military & Aerospace Applications	Rigid-Flex Boards
Upcoming class:	Upcoming class:	Upcoming class:
8/31–10/21	8/30–10/6	8/30–10/6

Look for these additional courses!

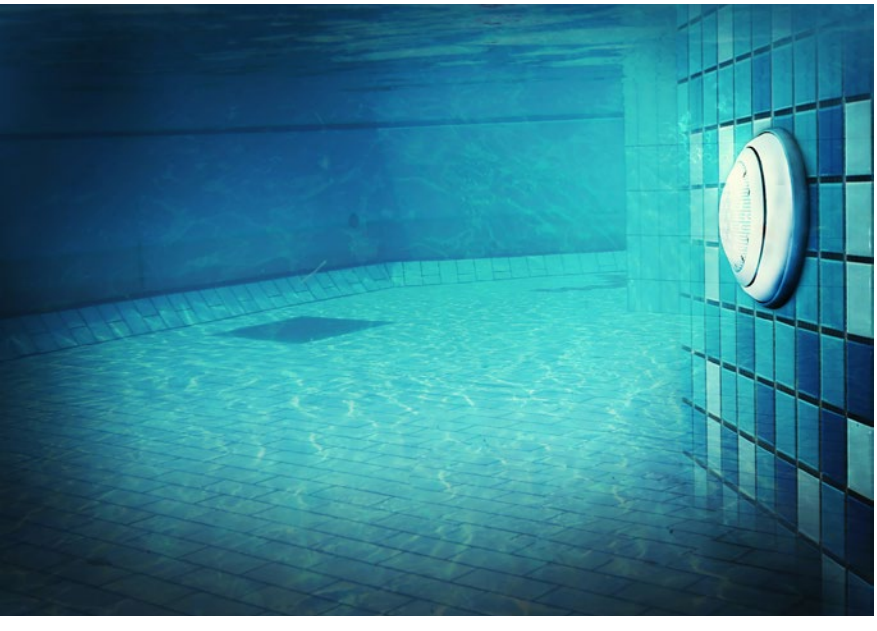
- PCB Design for Manufacturability
- PCB Design Embedded Components
- Introduction to PCB Design I
- PCB Design for Advanced Packaging
- PCB Design for Extreme Environments
- PCB Design for Printed & Wearable Devices

Learn more about our PCB Design Courses.

resin-encapsulated component that requires maximum protection against water ingress outdoors is the LED lighting unit; these resins are also optically clear and UV stable, making them ideal for the task.

Electrolube was tasked with solving an underwater LED lighting application. This is a prime example where encapsulation resins are critical for long-term performance. Water naturally corrodes most metals due to electrochemical oxidation; when a saltwater medium is used, the rate of oxidation significantly increases due to the presence of dissolved sodium and chloride ions.

Technical Support swung into action to resolve the problem. The customer was already using an epoxy resin from a different supplier; this was causing deformations to the plastic housing due to the high exotherm and needed to be replaced. The resin had to withstand water temperatures between 5–40°C, as well as be flame retardant, and needed to be light blue for aesthetic purposes. Importantly, the unit required a flexible encapsulation resin that could resist attack from constant immersion in pool water. Critically, this water could be saltwater or fresh water. The unit to be potted was complex, with tiny gaps in the casing. It was essential the resin did not bleed through these gaps and cover the LED lens.



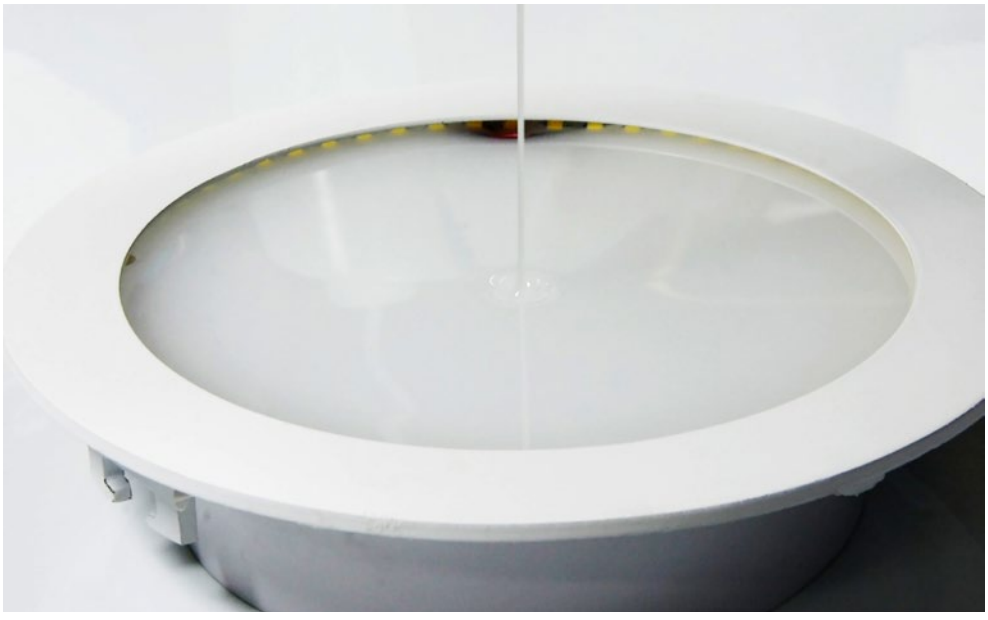
The product used had a history of providing protection in underwater applications. The only problem was the colour—black—and the fact that it is designed to flow through tight spaces. With a bit of lateral thinking, the logical solution was to increase the thixotropic nature of the resin, which would prevent the resin bleeding through the gap and simply change the pigment in the resin.

The R&D team produced a material that was still very easy to mix and pour into the unit, with sufficient thixotropy to prevent flow through the gaps and over the LED lens, and altered to the desired colour shade. The solution worked extremely well. Importantly for this application, the resin can withstand both fresh and saltwater ingress and prevent the transmission of metal ions that could attack any metal present. The polymer used in the resin is highly resistant to water even at the high pressures experienced as water depth increases. This example demonstrates anything is possible when it comes to finding resin solutions that provide effective resistance to their end-use environments.

If protecting components from mechanical shock and vibration are a concern, then a more flexible cured resin is likely to be the best solution as it adds a level of compliance that helps to insulate the potted components against adverse mechanical movements. In addition to their moisture resistant properties, polyurethane resins also provide this desired level of flexibility and so they are often the first choice for shock and vibration protection. Polyurethane resin finds use in Tyre Pressure Monitoring System (TPMS) sensors; this tough moisture resistant resin easily passes accelerated life testing, critical to resin qualification for TPMS used in the automotive industry. Silicone resins also offer excellent flexibility and will tolerate even higher operating temperatures.

There are epoxy resins that offer good moisture resistance, as well as

a fast cure, hardness, and high stability, particularly in variable temperature environments. These epoxies also adhere well to most substrates, even difficult substrates with a very low surface tension, such as PTFE, ensuring a watertight encapsulation.

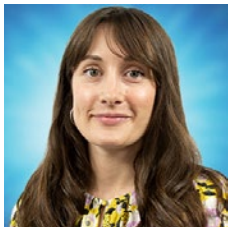


You may also need to protect electrical or electronic components from encountering chemicals, including acids, alkalis, solvents, and other substances that pose a threat to delicate circuits and components. Chemical resistance is very much the territory of epoxy resins, although some of the stronger polyurethane products, as well as some silicone-based formulations, will provide a degree of protection. Many epoxy resin products are effective at protecting units that undergo frequent or permanent immersion in solvents. Some are specifically designed to offer high temperature stability and excellent resistance to a wide range of chemicals and oils. Used in fuel level sensors, which are exposed to some of the most hostile conditions, they can be critical to the reliability and functionality required in automotive applications.

Thermal shock can have a devastating effect on electronic components, shortening their operating lives at best and completely destroying them at worst. The adverse effects of thermal stress can be vastly improved by using two-part epoxy resins that have a low coefficient of thermal expansion (CTE), the best solution for those applications that undergo severe temperature cycling or which are likely to suffer thermal shock. Low CTE resins mitigate the mismatch in thermal expansion between the resin, substrate, components, and solder joint. Low CTE epoxy resins can provide dimensional stability, protecting the integrity of the whole unit during high and low temperature thermal cycles.

Lastly, it is worth remembering that, aside from providing all the protections listed, opaque potting and encapsulation resins also conceal what lies beneath. This could provide an effective foil against counterfeiters or those wishing to copy a circuit layout, helping to protect your intellectual property.

Over the following months I hope to provide some useful tips and design advice that will help make life a bit easier for those who are responsible for making the decisions on protecting components and circuitry. I hope the foregoing will help you regarding selecting the right resin for its end use environment. Watch 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!"

FLEX007
A SPECIAL DESIGN007 MAGAZINE SECTION

The Importance of Asking 'Why Not?' When Inventing

Flexible Thinking

Feature Column by Joe Fjelstad, VERDANT ELECTRONICS

Regular readers hopefully do not need to be convinced that flexible circuit technology is one of the most enabling technologies for making interconnection. In the world of electronics, flexible circuits have continued their steady climb from relative obscurity half a century ago to center stage in the world of electronic interconnections. The reasons for their popularity are numerous:

- They are thin and light
- They can be bent, folded, or flexed
- They can offer superior electrical performance
- They can provide a highly reliable interconnection structure
- They can make possible structures that cannot be achieved by any other method (at least not as easily or cheaply)

With such an impressive list of benefits, it might seem as though the technology has reached its improvement limits. However, the principle of continuous improvement does not rest, and it demands that we persist in our efforts to do and make things better over time.

The great Irish playwright, critic and Noble Prize-winning author George Bernard Shaw is

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

FLEXIBLE CIRCUITS | EMS/ASSEMBLY | CONTRACT MANUFACTURING

Industry leading Flex Design Support

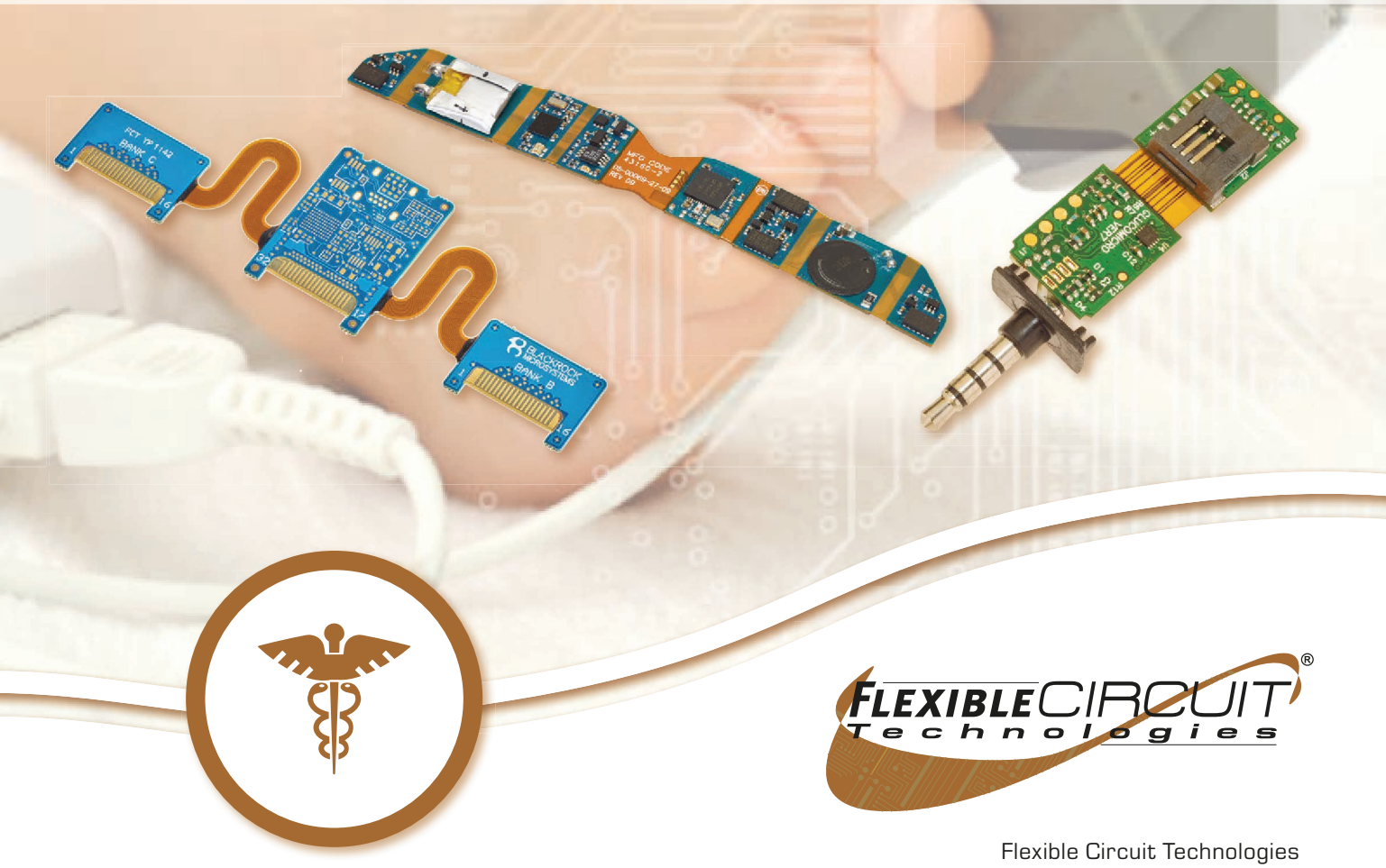
Flexible Circuits, Rigid Flex, Flexible Heaters Membrane Switches, Plastic Moldings

Specialized EMS/Assembly Services

Product Module to Complete Product Box Builds

ISO 13485 – FDA registered Box Builds

EXPLORE the POSSIBILITIES!



FLEXIBLE CIRCUIT® Technologies

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

credited with saying, "Some look at things that are and ask why. I dream of things that never were and ask why not?"[1] It is clear from the order of his statement that Shaw appreciated the importance of "why" and that it is one of the most often used by children trying to understand the world. Without asking that important and fundamental question of "why?" the question of "why not?" (which is the one that truly sparks the inventive spirit) cannot logically follow. Indeed, innovation is commonly a product of observation and these "why" and "why not" questions.

Such questions pressure the curious mind into action, hopefully resulting in one seeing (or dreaming of, as Shaw suggests) either the missing piece or alternatively the boundaries of empty space that define the missing piece. To be certain, these questions will, at some point, also cause a cascading chain of other common questions: what, how, where, who, when, and how much, which need also to be answered to complete the circle. Thus, the all-important first challenge that confronts us is to try and see what is not there and to then set about turning it from a concept to something concrete—from vision to reality.

With that preamble staging, and our collective challenge to "fill in the blank," it is worth taking a very short "mind walk" to uncover or illuminate the missing pieces that await their moment of discovery. For an exercise, we can apply this notion to flex circuit technology and see what it yields.

First, it is worth noting that once we become familiar with something, no matter what its nature, we become wedded to our perceptions of it—and technology is not immune. Flexible circuit technology, we know, is a highly enabling technology and it has many facets to it: materials, design tools and practices, manufacturing processes and methods, assembly tools and methods, testing equipment and protocols, etc. While the interdependence and interplay between these elements must be considered (changing one thing will normally impact another), it should not be an initial constraint. One should not be fearful of (and indeed should be encouraged to) wandering off the beaten path. To be sure, by not staying on the main streets there will be some apparently blind alleys found, but these alleys can sometimes yield some unexpected treasure that, while not of value to the current effort, could be useful in an unrelated effort later. Another thing to avoid early on is any consideration of cost. It is often the case that a process or device is very expensive at the outset but that the price will come down with experience and greater numbers of participants.

> ...questions pressure the curious mind into action...

Those points aside, we can now quickly apply the "why" questions to some aspects of flexible circuit technology to see what it yields. Don't look for answers to follow here; there is no ability or intent to answer the questions the reader might ask. They are yours to ponder on your own and hopefully come up with some "why not" ideas of their own. Consider then the following:

- Why do we use only certain materials?
- Why do we need holes?
- Why do we use cover layers?
- Why do we need lamination?
- Finally for a little bit of controversy, why do we even need flexible circuits?

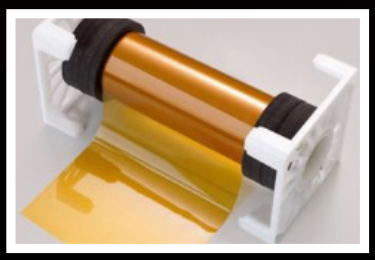
There are no right or wrong questions or answers; they are simply questions, ones that might help us all to break loose from our mental chains and think in new directions and dimensions. Since this column offered up an early quote from a genius, it seemed appropriate that

BENDING THE POSSIBILITIES

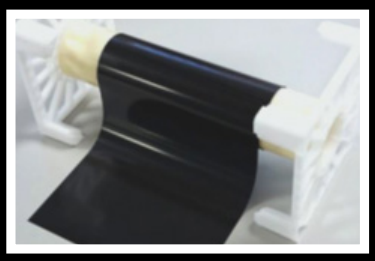
FLEXFINER

BY TAIYO

AVAILABE IN TWO COLORS



AMBER



BLACK

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

OUR BIGGEST FLEX YET!

Contact your local sales representative today!

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

TAIYO AMERICA

UNMATCHED INNOVATION

Follow us on social media

it should end with a quote from another great mind, Supreme Court Justice Oliver Wendell Holmes, who astutely observed, "Man's mind, stretched by a new idea, never goes back to its original dimensions." It seems doubly fitting, knowing that stretching also helps one to stay flexible in body as well. **FLEX007**

References

1. John F Kennedy paraphrased Shaw in his memorable first and, sadly, only inaugural speech and often is given credit as the first to present the idea.

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."

Scientists Discover How High-Energy Electrons Strengthen Magnetic Fields

Electrons and ions. The motion of these charged particles produces magnetic fields that form an interstellar magnetic web. These magnetic fields are important for a wide range of processes, from the shaping of galaxies and the formation of stars to controlling the motion and acceleration of high-energy particles like cosmic rays—protons and electrons that zoom through the universe at nearly the speed of light.

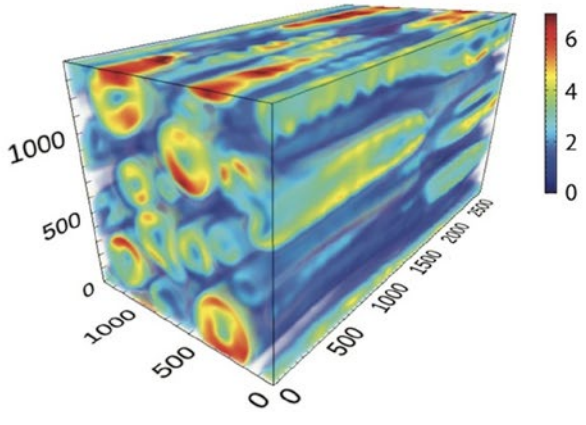
Until now, the way energetic particles affect magnetic fields was not well understood. In a paper published on the cover of Physical Review Letters, researchers from the Department of Energy's SLAC National Accelerator Laboratory show how electrons can amplify magnetic fields to much higher intensities than were previously known.

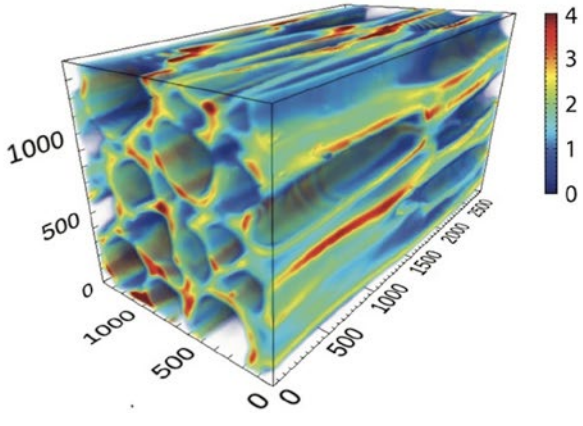
The motion of electrons carries an electrical current, which produces magnetic fields. Usually, charges from background plasma interfere with this current by moving in a way to cancel it, making strong magnetic fields difficult to produce. Using numerical simulations and theoretical models, the researchers found that high-energy electrons can actually expel the background plasma to create a hole, making it harder for the plasma to cancel their current.

"As the current is exposed, strong magnetic fields are produced that further push the background plasma away, creating bigger holes, leaving more of the current exposed, and producing even stronger magnetic fields," says Ryan Peterson, a PhD student at Stanford University and SLAC who is the first author of the publication.

The researchers hope to find ways to reproduce it in a laboratory experiment, which would be an important step in developing compact high-energy radiation sources. Those sources would allow scientists to take pictures of matter on the atomic scale with extremely high resolution for applications in medicine, biology and materials research.

(Source: SLAC National Accelerator Laboratory)



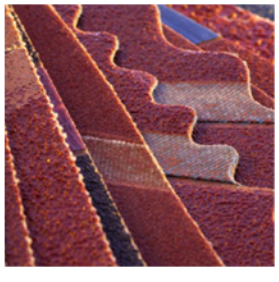


WE WERE PRINTING ELECTRONICS LONG BEFORE 'PRINTED ELECTRONICS' WAS A THING.

From simple switches to complex, functional, multi-layer prints with printed components, we have the capability to meet your needs and can enable your ideas moving into the future. Request a consult today!

Book a consult






Sheldahl®

A Flex Company

www.sheldahl.com

Flex007 Highlights

Self-powered Implantable Device Stimulates Fast Bone Healing, Disappears Without a Trace ▸

In 2017, Green Bay Packers quarterback Aaron Rodgers broke his right collarbone in a game against the Minnesota Vikings. Typically, it takes about 12 weeks for a collarbone to fully heal, but by mid-December fans and commentators were hoping the three-time MVP might recover early and save a losing season.

First-ever Transient Pacemaker Harmlessly Dissolves in Body ▸

Researchers at Northwestern and George Washington (GW) universities have developed the first-ever transient pacemaker—a wireless, battery-free, fully implantable pacing device that disappears after it's no longer needed.

IPC Offers Training Course on PCB Design for Rigid-Flex Boards ▸

IPC will be conducting a six-week online training course focused on PCB design for rigid-flex boards. The course will provide the skills necessary to effectively implement designs requiring flex and rigid-flex circuits in accordance with product requirements.

Unbroken: New Soft Electronics Don't Break, Even When Punctured ▸

A team of Virginia Tech researchers from the Department of Mechanical Engineering and the Macromolecules Innovation Institute has created a new type of soft electronics, paving the way for devices that are self-healing, reconfigurable, and recyclable.

Stanford Researchers Develop New Manufacturing Technique for Flexible Electronics ▸

Ultrathin, flexible computer circuits have been an engineering goal for years, but technical hurdles have prevented the degree of miniaturization necessary to achieve high performance.

Flexible Thinking: The Calf Path—Redux ▸

When I first read the poem many decades ago, it immediately struck me with its simple yet profound wisdom. Since that fortunate discovery, the poem has informed often my conscious thinking. I'm sure it will for you as well.

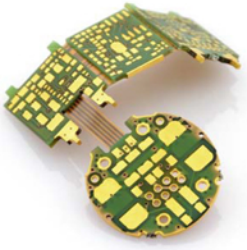
Flexium 1H 2021 Sales Up 27% YoY ▸

Taiwan-based flexible printed circuit (FPC) maker Flexium Interconnect Inc. has posted sales of NT$2.26 billion ($80.75 million at $1:NT$28.00) in June, up by 7.4% year-on-year, but down by 5.76% from the previous month.

Calling All Couch Potatoes: This Finger Wrap Can Let You Power Electronics While You Sleep ▸

A new wearable device turns the touch of a finger into a source of power for small electronics and sensors. Engineers at the University of California San Diego developed a thin, flexible strip that can be worn on a fingertip and generate small amounts of electricity when a person's finger sweats or presses on it.

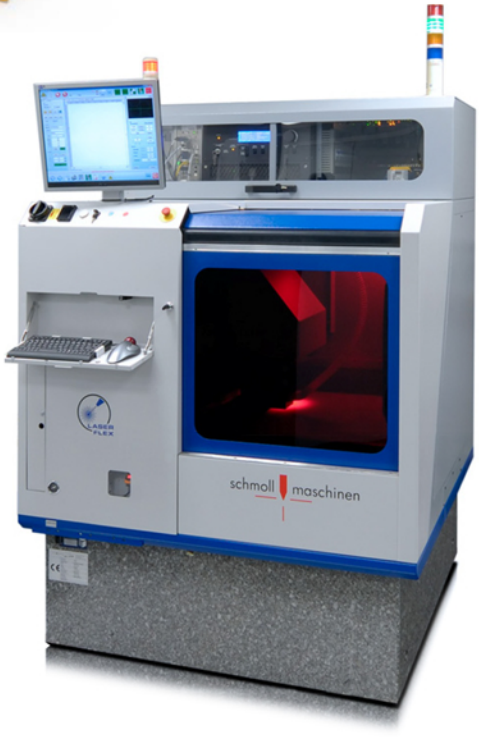
Focused on Flex



schmoll optronics

LaserμFlex series

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



Typical applications:

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

1991
2021

BURKLE
NORTH AMERICA, INC.

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

burkleamerica.com

1 Averatek Pushing Boundaries of Additive and Semi-Additive Processes ▸


Tara Dunn

At the IEEE International Microwave Symposium show in Atlanta, Andy Shaughnessy met with Tara Dunn, I-Connect007 columnist and VP of marketing and business development for Averatek. We sat down and Tara gave me an update on Averatek's continuing research into additive and semi-additive technologies.

2 New I-007eBook Highlights System Analysis ▸

In this latest title from I-007eBooks, authored by Cadence, readers will learn how system-level analysis of complex and high-speed electronic designs is critical to solve electromagnetic, electrothermal, and electromechanical simulation challenges and to ensure that the system works under wide-ranging operating conditions.

3 Sensible Design: Thermal Management—Good Design Practice for Heat Dissipation ▸

Trial and error is an essential process in the development of new and innovative products; however, excessive testing can be unnecessary and costly. Incorporating thermal management at the preliminary stage of your design process will ultimately lead to more reliable and cost-effective end products.

4 My Life in PCB Design ▸


Cherie Litson

In a cold (62°F), semi-dark room, there are banks of mainframe computers along one wall; a soft light glows upward from the table in the corner. I take my kit—a variety of sizes of black, red, and blue tape, decals, and an X-Acto knife—and set it on the side table.

5 Rogers Technology Update ▸


John Coonrod

At the IEEE International Microwave Symposium (IMS) show in Atlanta, Andy Shaughnessy met with John Coonrod, technical marketing manager with Rogers Corporation and a Design007 columnist. He asked John to discuss the papers he presented at IMS and to give us an update on Rogers' materials and technology.

6 Quiet Power: Ask the Experts—PDN Filters ▸


Istvan Novak

In recent years I have been getting a lot of questions about PDN filters from my course participants and from friends, colleagues and even from strangers. Long gone are the days when the essence of power distribution design recommendation was "place a 0.1µF bypass capacitor next to each power pin."

7 Electronic System Design Industry Reports 17% Revenue Growth in Q1 2021 ▸

Electronic System Design industry revenue increased 17% to $3.15 billion in Q1 2021, the strongest first-quarter growth ever, the ESD Alliance, a SEMI Technology Community, announced in the Q1 2021 Electronic Design Market Data report.

8 It's FR-4, Jim, But Not as We Know It! ▸

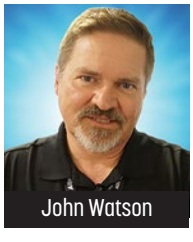

Alun Morgan

As Star Trek was responsible for adding terms such as photon torpedo, dilithium crystal, and warp drive to our language, so NEMA were responsible for the terms G-10, FR-4, and FR-5. "FR" was the NEMA abbreviation for "flame retardant" and materials so classified were marked with a red manufacturer's logo.

9 Ask the Experts: 12-Layer Stackups With Multiple Powers and Grounds ▸

Question: We're having lots of problems designing good stackups for 6-, 8-, 10- and 12-layer boards with multiple powers and grounds. Any suggestions?

10 Elementary, Mr. Watson: Time to Market, from Ludicrous Speed to Plaid ▸


John Watson

Mel Brooks may have something to teach us about going "ludicrous speed" in getting our designs to the finish line. I-Connect007 columnist John Watson explains.

Career Opportunities

Is your team growing?

Find industry-experienced candidates at I-Connect007.

For just $750, your 200-word, full-column ad will appear in the "career opportunities" section of all three of our monthly magazines, reaching circuit board designers, fabricators, assemblers, OEMs, and suppliers.

In addition, your ad will be featured in at least one of our newsletters, and your posting will appear on our jobConnect007.com board, which is also promoted in every newsletter.

Potential candidates can click on your ad and submit a resume directly to the email address you provide or be directed to the URL of your choice. If you wish to continue beyond the first month, the price is the same per month.

No contract required. We even include your logo in the ad, which is great branding!

To get your ad into the next issue, contact:
Barb Hockaday at barb@iconnect007.com or +1 916.365.1727 (-8 GMT PST)

I-Connect007
GOOD FOR THE INDUSTRY

smt007 pcb007 pcb design007

Career Opportunities

Application Engineer (m/f/d)—Fulltime, Germany

Our company is expanding its product portfolio into custom made products. This creates the need for an Application Engineer to provide technical support to our existing sales team and customer base.

Responsibilities:

- Analysis of incoming technical data and handling of engineering questions
- Technical consultation of customers (incl. new customer specifications and discuss with relevant technical and quality teams worldwide)
- Support and consultation for new projects
- Lead and/or participate in local, cross-location/global cross-department projects of various scale
- Develop and provide function-related trainings to existing and new staff in order to transfer and optimize know-how
- Provide technical solutions

Skills:

- Technical expertise in battery power solutions and technologies for Rechargeable and Primary cells and Battery Packs
- Mechanical background or knowledge to be able to discuss and manage other products, like custom made connectors, cable assemblies and keypad touch panels.
- Written and spoken English and German, any other European language a plus.
- Highly technical with a commercial flare.
- Self-motivated, ambitious, and eager to grow in a dynamic organization.

Interested? We are looking forward to your application!

Please send your application to **hr@cmit.support**. For any inquiries, please contact Mrs. Amélie Filler. For more information visit www.cml-globalsolutions.com

apply now

Sales Manager (m/f/d)—Worldwide Locations

CML Group is a leading provider of Printed Circuit Boards. We develop tailor-made sourcing concepts for our customers worldwide creating strong partnerships and reliable connections.

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 Profile:

- Profound sales and technical expertise in printed circuit board industry
- Local market knowledge and ideally a customer base of contacts in one or more of the listed countries
- Have successful track records in developing new business opportunities
- Excellent command in spoken and written English and one additional local language
- Highly self-motivated, ambitious, eager to grow in a dynamic organization
- Able to work independently and have good communication skills and leadership skills
- Self-employed/contractor/commission-based agent also welcome

Your Target Markets:

- Europe: Spain, France, Germany, Netherlands, UK, Denmark, Sweden, Norway
- USA: New Jersey, Florida, Georgia, Michigan, San Jose, Bay area, Pacific Northwest and Canada
- Others: Singapore, Thailand, Malaysia, Australia, Brazil, Turkey, Russia, and South Africa

Interested? We are looking forward to your application!

Please send your application to **hr-china@cml-eurasia.hk**. For any inquiries, please contact Ms. Grace Feng. For more information visit www.cml-globalsolutions.com

apply now

Career Opportunities

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

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

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 stand-alone boundary scan and flying probe desired.
- Some travel required. Positions are available in the Hunt Valley, Md., office.

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

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

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

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

EAGLE electronics Inc.
MANUFACTURERS OF QUALITY PRINTED CIRCUIT BOARDS

Maintenance Technician

Inspects work-related conditions to determine compliance with prescribed operating and safety standards. Operates power-driven machinery and uses equipment and tools commonly used to maintain facilities and equipment. Replace filters, belts, and additional parts for repairs and preventive maintenance. Moves objects weighing up to 150 lbs. using a hand truck or pulley. Cleans work area and equipment. Works with cleaning fluids, agents, chemicals, and paints using protective gear. Works at elevations greater than ten feet, climbing ladders, while repairing or maintaining building structures and equipment. Assists skilled maintenance technicians/workers in more complex tasks and possible after-hours emergency repairs. Must meet scheduling and attendance requirements.

apply now

Plating Operator

Plating operator for printed circuit boards. No experience necessary, will train. Must be able to work with chemicals, lift up to 50 pounds, and have good math skills. Minimum high school/GED or equivalent. All shifts (1st, 2nd, 3rd), 8 hours per day minimum, Monday thru Friday. Saturday and Sunday work is common allowing for steady overtime pay.

apply now

EAGLE electronics Inc.
MANUFACTURERS OF QUALITY PRINTED CIRCUIT BOARDS

Water Treatment Operator

Responsible for operating waste treatment plant, our operation that converts wastewater in drains and sewers into a form that's metal free to release into the environment.

Control equipment and monitor processes that remove metals from wastewater. Run tests to make sure that the processes are working correctly. Keep records of water quality and pH. Operate and maintain the pumps and motors that move water and wastewater through filtration systems. Read meters and gauges to make sure plant equipment is working properly. Take samples and run tests to determine the quality of the water being produced. Adjust the amount of chemicals being added to the water and keep records that document compliance.

apply now

Drilling Operator

Drilling operator for printed circuit boards. Minimum 2 years of experience. Minimum high school/GED or equivalent.

All Shifts (1st, 2nd, 3rd), 8 hours per day minimum, Monday thru Friday. Saturday and Sunday work is common allowing for overtime pay.

apply now

Career Opportunities

FirstPageSage

Technical Writer
(Full-time, Remote)

SEO company seeking a technical writer in the area of PCB design & manufacturing. We provide Search Engine Optimization and Thought Leadership services for well-managed, ethical companies. Our team produces high-level content for companies who are leaders in their industry. We are seeking writers who are well-rounded researchers with a particular interest in electrical engineering and impeccable writing skills.

Ideal candidates are seasoned writers with practical experience in electronic systems development in one or more of the following areas:

- Circuit simulation
- Schematic capture
- PCB layout
- Design analysis
- PCB fabrication
- PCB assembly
- PCB testing

Qualifications:

- Technical writing experience with a proven track record of independent research and content development. Experience in data sheet, report, or white-paper writing preferred.
- Electronic technician or engineering degree, ideally in electrical engineering, computer science, or mechanical engineering.
- Industry experience in PCB design, testing, or manufacturing.
- Punctuality, professionalism, and excellent time management skills.
- A reliable internet connection and computer

Salary & Benefits:

This is a full-time telecommuting position with a starting salary range of $62,000 to $68,000 annually. Benefits PTO, sick time, 401K, health and dental coverage, and more!

To see the full job description and apply, please click the link below.

apply now

SIEMENS

Marketing Coordinator/Writing Strategist: Embedded Software

Location: Portland, Oregon or USA (remote)
Job Number: 242982

Seeking a technology communications change maker! Siemens Digital Industries Software is looking for a content creator for its embedded software group. The ideal candidate for the Brand Marketing coordinator/writing strategist position will work closely with engineers and managers to write, edit and produce compelling technology marketing content (magazine articles, blogs, technology papers, multi-media, customer success stories and promotional materials). Do you possess creative energy and enjoy storytelling with an energetic team?

Requirements:

- Strong writing and editing skills
- Education and/or experience in technology, science, journalism and/or English
- A technical background or experience (such as a BS or an associate's degree in engineering or computer science) is preferred
- 1-3 years of experience in writing about technology solutions
- Basic knowledge of online publications, digital platforms and social media is useful to meet project specifications in a fast-paced environment
- Ability to research and collect data, repurpose existing materials, collaborate with subject matter experts, and translate technical information into compelling marketing communications content that engage audiences

Creative materials will be used globally, in a high-energy environment, supporting the world's leading industrial software company.

apply now

Career Opportunities

MivaTek Global

Product Manager

MivaTek Global is preparing for a major market and product offering expansion. Miva's new NG3 and DART technologies have been released to expand the capabilities of Miva's industry-leading LED DMD direct write systems in PCB and Microelectronics. MivaTek Global is looking for a technology leader that can be involved guiding this major development.

The product manager role will serve as liaison between the external market and the internal design team. Leadership level involvement in the direction of new and existing products will require a diverse skill set. Key role functions include:

- **Sales Support:** Recommend customer solutions through adaptions to Miva products
- **Design:** Be the voice of the customer for new product development
- **Quality:** Verify and standardize product performance testing and implementation
- **Training:** Conduct virtual and on-site training
- **Travel:** Product testing at customer and factory locations

Use your 8 plus years of experience in either the PCB or Microelectronic industry to make a difference with the leader in LED DMD direct imaging technology. Direct imaging, CAM, AOI, or drilling experience is a plus but not required.

For consideration, send your resume to N.Hogan@MivaTek.Global. For more information on the company see www.MivaTek.Global or www.Mivatec.com.

apply now

MivaTek Global

Field Service Technician

MivaTek Global is focused on providing a quality customer service experience to our current and future customers in the printed circuit board and microelectronic industries. We are looking for bright and talented people who share that mindset and are energized by hard work who are looking to be part of our continued growth.

Do you enjoy diagnosing machines and processes to determine how to solve our customers' challenges? Your 5 years working with direct imaging machinery, capital equipment, or PCBs will be leveraged as you support our customers in the field and from your home office. Each day is different, you may be:

- Installing a direct imaging machine
- Diagnosing customer issues from both your home office and customer site
- Upgrading a used machine
- Performing preventive maintenance
- Providing virtual and on-site training
- Updating documentation

Do you have 3 years' experience working with direct imaging or capital equipment? Enjoy travel? Want to make a difference to our customers? Send your resume to N.Hogan@MivaTek.Global for consideration.

More About Us

MivaTek Global is a distributor of Miva Technologies' imaging systems. We currently have 55 installations in the Americas and have machine installations in China, Singapore, Korea, and India.

apply now

Career Opportunities

Sheldahl®

A Flex Company

Sheldahl, a leading provider of flexible interconnect products and electronic materials, is seeking candidates to join their diverse and skilled team.

We are looking for people who demonstrate:

- Intense collaboration
- Passionate customer focus
- Thoughtful, fast, disciplined execution
- Tenacious commitment to continuous improvement
- Relentless drive to win

Positions in America include:

Project Manager – Northfield, MN
Candidate will provide timely cost estimation and project budget definition, be responsible for maintaining customer relations, participate in meetings, etc.

apply now

Program Manager – Specialty Films
Candidate will work with our Specialty Films in the Aerospace, Medical, and Commercial Aviation markets providing timely cost estimation and project budget definition, maintaining customer relations, participate in meetings, etc.

apply now

Business Development Manager – North America
Candidate will provide leadership in the planning, design and implementation of customers' specific business plans and will provide vision, penetration strategies and tactics to executive managers in order to develop and drive external and internal senior-level relationships.

apply now

Sheldahl®

A Flex Company

Sheldahl, a leading provider of flexible interconnect products and electronic materials, is seeking candidates to join their diverse and skilled team.

We are looking for people who demonstrate:

- Intense collaboration
- Passionate customer focus
- Thoughtful, fast, disciplined execution
- Tenacious commitment to continuous improvement
- Relentless drive to win

Positions in Europe include:

Business Development Manager — France
Seeking out-of-the-box thinkers to help us take the ordinary to the extraordinary by cultivating current customer relationships and developing new business opportunities with our European team, based in France.

apply now

Business Development Manager — Germany
Seeking out-of-the-box thinkers to help us take the ordinary to the extraordinary by cultivating current customer relationships and developing new business opportunities with our European team, based in Germany.

apply now

Career Opportunities

ventec INTERNATIONAL GROUP 騰輝電子

Technical Support/ Sales Engineer, UK

We are looking to expand our UK technical & sales support team. As a technical support/sales engineer (home office/Leamington Spa) you will assist potential and current customers in appreciating the benefits of using--and optimizing the use of--Ventec materials in their printed circuit board manufacturing processes, and so enhance customer loyalty and satisfaction, spread the use of Ventec materials, and grow sales. You will provide a two-way channel of technical communication between Ventec's production facilities and UK/European customers.

Skills and abilities required for the role

- HNC, HND, degree or equivalent in a technical/scientific discipline
- Sales experience/negotiating skills
- Printed circuit board industry experience an advantage
- Good written & verbal communications skills
- Ability to work in an organized, proactive and enthusiastic way
- Ability to work well both in a team and independently
- Good user knowledge of common Microsoft Office programs
- Full driving license essential

What's on Offer

- Excellent salary and 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
anthony.jackson@ventec-europe.com

apply now

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

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

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

Career Opportunities

Now Hiring

Director of Process Engineering

A successful and growing printed circuit board manufacturer in Orange County, CA, has an opening for a director of process engineering.

Job Summary:

The director of process engineering leads all engineering activities to produce quality products and meet cost objectives. Responsible for the overall management, direction, and coordination of the engineering processes within the plant.

Duties and Responsibilities:

- Ensures that process engineering meets the business needs of the company as they relate to capabilities, processes, technologies, and capacity.
- Stays current with related manufacturing trends. Develops and enforces a culture of strong engineering discipline, including robust process definition, testing prior to production implementation, change management processes, clear manufacturing instructions, statistical process monitoring and control, proactive error proofing, etc.
- Provides guidance to process engineers in the development of process control plans and the application of advanced quality tools.
- Ensures metrics are in place to monitor performance against the goals and takes appropriate corrective actions as required. Ensures that structured problem-solving techniques are used and that adequate validation is performed for any issues being address or changes being made. Develops and validates new processes prior to incorporating them into the manufacturing operations.
- Strong communication skills to establish priorities, work schedules, allocate resources, complete required information to customers, support quality system, enforce company policies and procedures, and utilize resources to provide the greatest efficiency to meet production objectives.

Education and Experience:

- Master's degree in chemical engineering or engineering is preferred.
- 10+ years process engineering experience in an electronics manufacturing environment, including 5 years in the PCB or similar manufacturing environment.
- 7+ years of process engineering management experience, including 5 years of experience with direct responsibility for meeting production throughput and quality goals.

apply now

Now Hiring

Process Engineering Manager

A successful and growing printed circuit board manufacturer in Orange County, CA, has an opening for a process engineering manager.

Job Summary:

The process engineering manager coordinates all engineering activities to produce quality products and meet cost objectives. Responsible for the overall management, direction, and coordination of the engineering team and leading this team to meet product requirements in support of the production plan.

Duties and Responsibilities:

- Ensures that process engineering meets the business needs of the company as they relate to capabilities, processes, technologies, and capacity.
- Stays current with related manufacturing trends. Develops and enforces a culture of strong engineering discipline, including robust process definition, testing prior to production implementation, change management processes, clear manufacturing instructions, statistical process monitoring and control, proactive error proofing, etc.
- Ensures metrics are in place to monitor performance against the goals and takes appropriate corrective actions as required. Ensures that structured problem-solving techniques are used and that adequate validation is performed for any issues being address or changes being made. Develops and validates new processes prior to incorporating into the manufacturing operations

Education and Experience:

- Bachelor's degree in chemical engineering or engineering is preferred.
- 7+ years process engineering experience in an electronics manufacturing environment, including 3 years in the PCB or similar manufacturing environment.
- 5+ years of process engineering management experience, including 3 years of experience with direct responsibility for meeting production throughput and quality goals.

apply now

Career Opportunities

LENTHOR ENGINEERING
MAKING THINGS FLEXIBLE IN A RIGID WORLD®

Sales Account Manager

Sales Account Management at Lenthor Engineering is a direct sales position responsible for creating and growing a base of customers that purchase flexible and rigid flexible printed circuits. The account manager is in charge of finding customers, qualifying the customer to Lenthor Engineering and promoting Lenthor Engineering's capabilities to the customer. Leads are sometimes referred to the account manager from marketing resources including trade shows, advertising, industry referrals and website hits. Experience with military printed circuit boards (PCBs) is a definite plus.

Responsibilities

- Marketing research to identify target customers
- Identifying the person(s) responsible for purchasing flexible circuits
- Exploring the customer's needs that fit our capabilities in terms of:
  - Market and product
  - Circuit types used
  - Competitive influences
  - Philosophies and finance
  - Quoting and closing orders
  - Providing ongoing service to the customer
  - Develop long-term customer strategies to increase business

Qualifications

- 5-10 years of proven work experience
- Excellent technical skills

Salary negotiable and dependent on experience. Full range of benefits.

Lenthor Engineering, Inc. is a leader in flex and rigid-flex PWB design, fabrication and assembly with over 30 years of experience meeting and exceeding our customers' expectations.

Contact Oscar Akbar at: hr@lenthor.com

apply now

LENTHOR ENGINEERING
MAKING THINGS FLEXIBLE IN A RIGID WORLD®

Senior Process Engineer

Job Description

Responsible for developing and optimizing Lenthor's manufacturing processes from start up to implementation, reducing cost, improving sustainability and continuous improvement.

Position Duties

- Senior process engineer's role is to monitor process performance through tracking and enhance through continuous improvement initiatives. Process engineer implements continuous improvement programs to drive up yields.
- Participate in the evaluation of processes, new equipment, facility improvements and procedures.
- Improve process capability, yields, costs and production volume while maintaining safety and improving quality standards.
- Work with customers in developing cost-effective production processes.
- Engage suppliers in quality improvements and process control issues as required.
- Generate process control plan for manufacturing processes, and identify opportunities for capability or process improvement.
- Participate in FMEA activities as required.
- Create detailed plans for IQ, OQ, PQ and maintain validated status as required.
- Participate in existing change control mechanisms such as ECOs and PCRs.
- Perform defect reduction analysis and activities.

Qualifications

- BS degree in engineering
- 5-10 years of proven work experience
- Excellent technical skills

Salary negotiable and dependent on experience. Full range of benefits.

Lenthor Engineering, Inc. is the leader in Flex and Rigid-Flex PWB design, fabrication and assembly with over 30 years of experience meeting and exceeding our customers' expectations.

Contact Oscar Akbar at: hr@lenthor.com

apply now

Career Opportunities

Manncorp™

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

Manncorp™

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

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

Innovative Circuits, Inc.
Real Quality Is Innovative!

Multiple Positions

Innovative Circuits, a quick-turn, high mix, low-volume PCB manufacturer located in Alpharetta, Georgia, is growing and looking for talented individuals to join the team.

Front End Engineering Manager
Oversee CAM, programming/production engineering and quoting departments. Ideal candidates will have 15 years' experience working in a printed circuit board front-end department with flex and rigid flex circuit board construction.

Process Engineer
Responsible for the implementation and maintenance of chemical and/or mechanical processes used to produce flex circuits, rigid flex and rigid printed circuit boards.

Third Shift Production Manager
Oversee third shift productions workers, product schedule and reporting.

Wet Lab Tech
Perform all lab analysis using burettes, pipettes, pH/ion meters, atomic absorption spectrophotometer, laboratory balance, hydrometers, hull cells, CVS, and all other lab-related equipment.

CAM Operator
Inspect, modify, and contribute to the initial development of producing flex circuits, rigid flex and rigid printed circuit boards based upon customer requirements and data files.

Quality Inspector
Responsible for verifying that the product meets customer requirements prior to shipping.

Wastewater Technician
Operate, monitor, maintain and troubleshoot the wastewater treatment facility and its processes.

Production Worker
Machine operator and light chemistry in a PCB manufacturing environment.

Please visit the link below to view our opportunities and apply.

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

APCT
Passion | Commitment | Trust

APCT, Printed Circuit Board Solutions: Opportunities Await

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

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

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

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

apply now

Career Opportunities

EAGLE electronics Inc.

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

eptac

TRAIN. WORK SMARTER. SUCCEED.

Become a Certified IPC Master Instructor

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

Qualifications and skills

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

Benefits

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

apply now

Career Opportunities

CAM / Process Engineer

The JHU/APL PCB Fabrication team is seeking a Computer Aided Manufacturing Engineer to support front-end data processing of APL manufactured hardware. You will directly contribute to hardware fabrication in support of National Security, Military Readiness, Space Exploration, National Health, and Research related to fundamental scientific advancement. This position includes a variable mix of core CAM work scope with additional opportunities for hands-on support such as bare board electrical testing, laser drilling, and mechanical CNC drilling and routing.

Responsibilities:

1. Computer Aided Manufacturing for rigid PCB, rigid-flex, and flexible circuits
   a) Perform design checks, panel layout, coupon generation, file generation, stackups
   b) Support manufacturability reviews with internal APL engineers (customers)
   c) Generate work travelers
   d) Communicate status to supervisors and internal customers
2. Support transition of software tools (Genesis 2000 to InCAM Pro)
   a) Edit design rules checks and generate automation scripts
   b) Develop new ideas to further the technical progress of our product
   c) Develop CAM area through continuous improvement initiatives
3. Interface and inform APL Engineers on PCB design for manufacturing guidelines
4. Operate bare board electrical tester
5. Backup operator for CNC drilling, routing, laser drilling (on-site training)

For more details and to apply:
www.jhuapl.edu/careers and search for CAM.

apply now

Sales Representatives (Specific Territories)

Escondido-based printed circuit fabricator U.S. Circuit is looking to hire sales representatives in the following territories:

- Florida
- Denver
- Washington
- Los Angeles

Experience:

- Candidates must have previous PCB sales experience.

Compensation:

- 7% commission

Contact Mike Fariba for more information.

mfariba@uscircuit.com

apply now

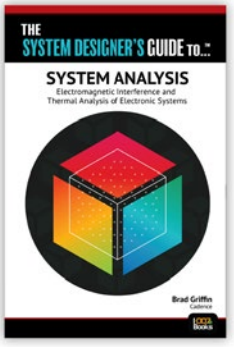
I-007e EDUCATIONAL RESOURCE CENTER

Introducing:

The System Designer's Guide to... System Analysis

Electromagnetic Interference and Thermal Analysis of Electronic Systems

In this latest title from I-007eBooks, readers will learn how system-level analysis of complex and high-speed electronic designs is critical to solve electromagnetic, electrothermal, and electromechanical simulation challenges and to ensure that the system works under wide-ranging operating conditions. Get your copy now!

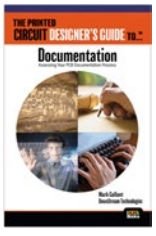


I-007eBooks The Printed Circuit Designer's Guide to...

Thermal Management: A Fabricator's Perspective

by Anaya Vardya, American Standard Circuits

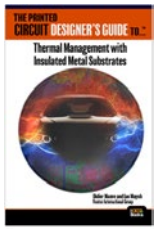
Beat the heat in your designs through thermal management design processes. This book serves as a desk reference on the most current techniques and methods from a PCB fabricator's perspective.



Documentation

by Mark Gallant, Downstream Technologies

When the PCB layout is finished, the designer is still not quite done. The designer's intent must still be communicated to the fabricator through accurate PCB documentation.



Thermal Management with Insulated Metal Substrates

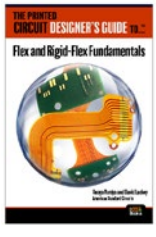
by Didier Mauve and Ian Mayoh, Ventec International Group

Considering thermal issues in the earliest stages of the design process is critical. This book highlights the need to dissipate heat from electronic devices.

Fundamentals of RF/Microwave PCBs

by John Bushie and Anaya Vardya, American Standard Circuits

Today's designers are challenged more than ever with the task of finding the optimal balance between cost and performance when designing radio frequency/microwave PCBs. This micro eBook provides information needed to understand the unique challenges of RF PCBs.



Flex and Rigid-Flex Fundamentals

by Anaya Vardya and David Lackey, American Standard Circuits

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

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

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

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

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

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

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

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

SALES MANAGER: **BARB HOCKADAY**
(916) 608-0660; 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 © DAVID PIMBOROUGH**

DESIGN007 MAGAZINE

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

August 2021, Volume 10, Number 8
DESIGN007 MAGAZINE is published monthly,
by BR Publishing, Inc.

ADVERTISER INDEX

Problems solved!