I-Connect007

MAY 2022

DESIGN007 MAGAZINE

Designing in a Vacuum

DESIGN007
MAGAZINE

Are You Designing in a Vacuum?

For decades, this industry's top instructors and journalists have been preaching about the need to communicate with your fabricator. But for many of you, that's just not possible, and you spend most of your time designing in a vacuum. So, this month our contributors provide tips for designing in a vacuum, as well as the total costs of working in the dark, and some strategies for getting out of the Hoover.

FEATURE INTERVIEWS

12 **Tips for Designing in—and Escaping From—the Vacuum**
with Jen Kolar and Cory Grunwald

30 **Designing in a Vacuum Q&A: Mark Thompson**

48 **Designing in a Vacuum Q&A: Nick Barbin**

64 **Designing in a Vacuum Q&A: Carl Schattke**

FEATURE COLUMNS

10 **Are You Designing in a Vacuum?**
by Andy Shaughnessy

40 **Happier in a Vacuum: The Design Narcissist**
by Kelly Dack

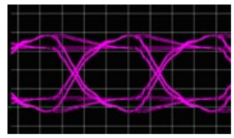
DESIGN007 MAGAZINE

WEBINAR REVIEW

52 **A Textbook Look: Signal Integrity and Impedance**
by Pete Starkey

COLUMNS

22 **Reflecting on Reflections**
by Barry Olney



36 **Enabling a Digital Thread Across IC/Package/PCB Design**
by David Wiens

44 **The Benefits of a Parts Library**
by Matt Stevenson

60 **Green Ambition for the Electronics Manufacturing Industry**
by Patrick Crawford

70 **Today's Preparations for Tomorrow's PCB Designs**
by Tim Haag

76 **Why Some Conformal Coatings Fail in Service**
by Phil Kinner

SHORTS

29 **U-Boat Worx Launches 9-person Flagship Lithium-ion Battery Submersible NEXUS**

39 **COLUMN EXCERPT: All Systems Go! Supply Chain Woes: Which Comes First, the Design or the BOM?**

79 **Book Excerpt: *The Printed Circuit Designer's Guide to... High Performance Materials*, Chapter 2: Glass Fabric**

DEPARTMENTS

93 **Career Opportunities**

108 **Educational Resource Center**

109 **Advertiser Index & Masthead**

HIGHLIGHTS

68 **PCB007**

74 **MilAero007**

90 **Top Ten Editor's Picks**

FLEX007

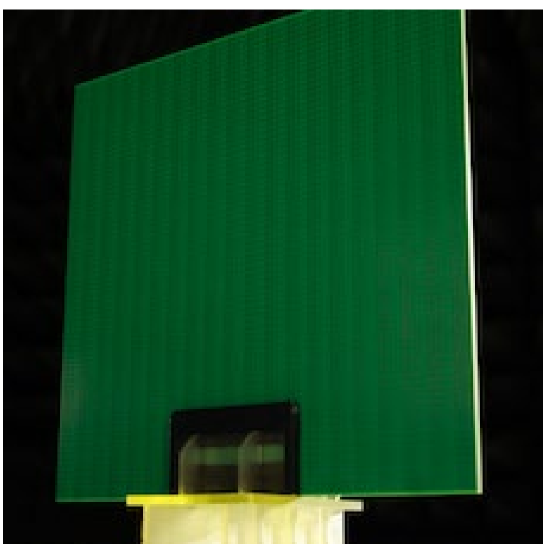
This Month in Flex

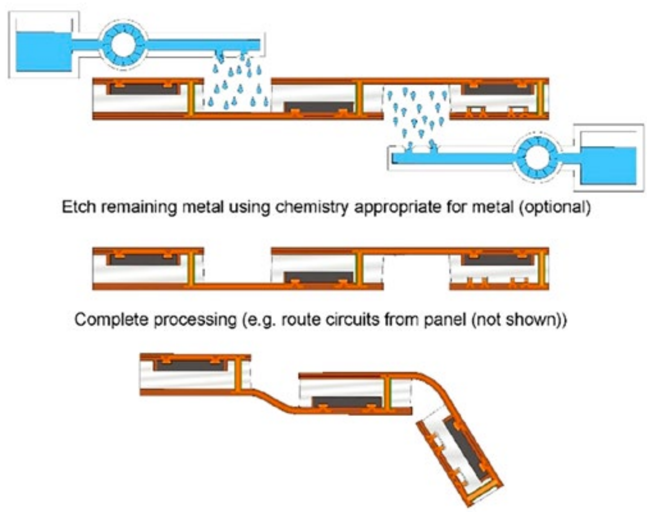
Aluminum has been around for hundreds of years, and we may have only scratched the surface of its many uses. In this column, Joe Fjelstad explains how this inexpensive metal may hold the key to making solderless rigid-flex circuits a reality. Aluminum may once again prove to be more valuable than gold.

FLEX007 COLUMN
80 Blue Skying it With Aluminum Rigid-Flex
by Joe Fjelstad

HIGHLIGHTS
88 Flex007

FLEX007 SHORT
86 KYOCERA Develops Transmissive Metasurface Technology





Are You Designing in a Vacuum?

The Shaughnessy Report

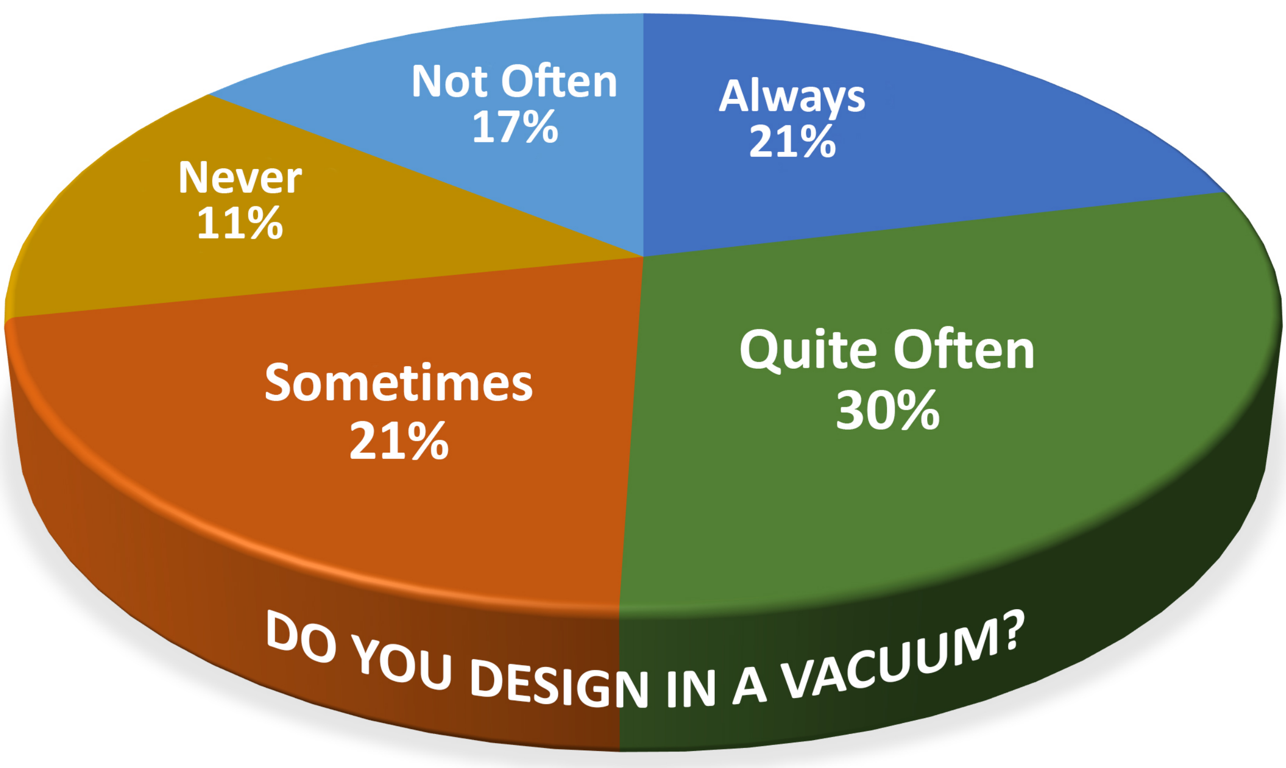
by Andy Shaughnessy, I-CONNECT007

Do you ever feel like you're designing in a vacuum? If so, you are not alone.

This month's topic came to us during an editorial meeting. We were discussing some of the PCB design classes at recent industry conferences—classes on signal integrity, supply chain management, DFM, and EMC. But after talking with designers who have grown accustomed to laying out boards without knowing where their boards would be manufactured, not to mention such info as impedance requirements, it dawned on us.

What the design community really needs to do is focus on designing in a vacuum. Is DIV the next industry acronym?

We surveyed our design and design engineer readers, and nearly half of respondents said they always or quite often design boards in a



vacuum. Only 11% said they never work inside a Hoover upright with HEPA filtration.

Check out some of the reasons designers cited for being kept in dark about where their board(s) would be fabricated:

- Client hasn't thought about that part of the project!
- We use CEMs with whom we've partnered for many years, and they will find the most appropriate PCB manufacturer for us.
- We are a service bureau so almost always ask but often the OEM design engineer doesn't know (yet).
- Purchasing department determines who will win the contract for the PBA.
- We do not know who is going to fabricate it.
- It has not yet been established as a practice to consider manufacturers before design.

Survey respondents also mentioned having to begin layout without impedance data, functional test info, panel drawings, tolerances, process limits, special design rules, drill size vs. finished plated hole, board thickness, and the height of components close to rails.

Fun fact: I attended the Supply Chain Roundtable at SMTA Atlanta. I asked everyone to raise their hands if they felt like they were working in vacuum. Almost everyone in the room—designers, fabricators and EMS providers—waved like Miss Universe. Everyone had a horror story.

If you watch *Intervention,* you know that the first step forward is admitting you have a problem. Well, we have a problem. For decades, this industry's top instructors and members of the media, including me, have been preaching about the need to communicate with your fabricator. But for many of you, that's just not possible, and our advice is not based in reality.

The design segment needs to have an ongoing conversation about designing in a vacuum, and how to make the best of it. This is not to say that designers shouldn't try to get out of the vacuum. But the current advice about always communicating with your fabricator is just not applicable to many designers.

So, this month our contributors provide tips for designing in a vacuum, as well as the total costs (in time, respins and other resources) of working in the dark, and some strategies for getting out of the vacuum.

We have an interview with Jen Kolar and Cory Grunwald of Monsoon Solutions, who discuss some of the tricks they've developed for filling in missing information, including the art of making accurate assumptions. Columnist Kelly Dack examines a breed of designer who actually thrives on working alone, often to the detriment of everyone else: the design narcissist. We also feature a variety of conversations about DIV with our experts, including Nick Barbin of Optimum Design Associates, Mark Thompson of Out of the Box Manufacturing, and Carl Schattke of PCB Product Development LLC. As Carl says, if you design the board correctly, not knowing where it's going to be manufactured shouldn't necessarily be a deal breaker.

Pete Starkey brings us a detailed review of a great webinar, and we have columns by Barry Olney, Dave Wiens, Matt Stevenson, Patrick Crawford, Tim Haag, Phil Kinner, and Joe Fjelstad.

Will we see a "Designing in a Vacuum" track at a conference someday? Instructors could borrow one from custodial staff...hey, it's a thought!

See you next month. **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.

Tips for Designing in—and Escaping From—the Vacuum

Feature Interview by the I-Connect007 Editorial Team

For decades, fabricators and PCB design instructors have preached a steady gospel: "Communicate with your manufacturer early and often." But many designers find themselves working with no idea who will be fabricating or assembling the board, not to mention dealing with the usual missing data, such as impedance requirements.

We asked Monsoon Solutions VP of Engineering Jen Kolar if she had any thoughts on "designing in a vacuum," and her response was, "Absolutely!" In this wide-ranging discussion, Jen and Senior PCB Engineer and Director of Designer Development Cory Grunwald share some tips and techniques for designing PCBs in a vacuum, and a few methods for getting out of the Hoover's dust bag.

As Cory points out, "In the end, the communication is going to happen anyway. It's just a matter of whether it happens at the end or in the beginning."

Barry Matties: For years, we've heard fabricators and design instructors recommend that designers communicate with their manufacturers, and as early in the design process as possible. However, we are often told that the PCB designer does not know who the fabricator is going to be when they embark on the design. As a design service, how do you approach that?

Jen Kolar: That can be one of the big tricks when we're working with a lot of different customers. If we're managing the fabrication, then

Jen Kolar

that's easy. Many of our customers, however, want to be that interface. They want to be the one who passes the data back and forth, and thus things get lost in translation, and that can add a lot of time. We've found that can be a real barrier if the designer isn't able to directly talk to the manufacturer, even if we're trying to. Our best practice is that our designers always verify information with the fab manager in advance.

Andy Shaughnessy: Everyone talks about DFM and the need for communication, but in the end, there's almost no communication with the fabricator because they don't know where the board will be built. So, instead of telling designers to talk to your manufacturer, maybe we should just embrace reality and call it "designing in a vacuum?"

Cory Grunwald: If I'm working with a customer and I don't know which fab shop I'm going to, I must make a lot of assumptions. I must work over what I expect a minimum trace and space or minimum via sizes will be. I'll start with a capability sheet from a known supplier that we'll go through. But then I must add a little bit to what I expect for those tolerances, because I don't know if they're going to a shop that can do 3-mil trace and space, or if a shop has to go to 5 mils.

Matties: When you're designing like that, what's the downside?

Grunwald: The downside is you can't be on the cutting edge. You can't be working on the smallest parts and the thinnest traces and spaces because you don't know who it's going to and who can handle it.

Matties: But is the design that you're allowing some tolerance for still a successful design?

Grunwald: Yes. It can be a successful design, but as you know, projects are getting smaller and faster, and those requirements usually require a top-end board shop. Without talking to your shop directly, you can't assume that you're going to a place that can do it.

Matties: That makes it tough to do your job.

Kolar: It can add a lot of churn. It can be, "Hey, you design the best with the information you have." Then the customer takes it through DFM. They come back and say, "We have a fundamental issue. We need 2-ounce copper on this outer layer, but we can't support this trace and space with that." It could lead to a full redesign because we've assumed we can do stacked vias and they won't do that. There are several things that can lead to redesigns, and it's even more fun when the customer wants to work with multiple vendors at once.

Matties: But the reality, Jen, is designers are designing in a vacuum all the time, right? This isn't an isolated case.

Kolar: I would say that the more complex the board, getting into motherboard scale, the more likely you're getting to have multiple reviews by component manufacturers, or you're doing SI/PI analysis, and iterating much more into DFM. At that scale, no, you're not. But for an average board, it's often, "Here are my controlled impedances. I'm going to do my best to calculate them in Polar. I'm going to put this together and send it off, and the fab shop will tweak as needed."

Matties: As Cory was describing, he's adding extra safeguards in the design to make sure it gets through. As you're pointing out, there are downsides to this approach. Why is it so difficult to get that communication up front?

Grunwald: It depends on the company structure and the way they work. We work directly with fab shops a lot. But in previous companies I've worked at, the design was basically handed over to purchasing, and purchasing found whoever can supply that board the cheapest. Not knowing who it ends up with was a big deal. There wasn't any talking with a board shop at the beginning because who knows where it would end up.

Happy Holden: The danger is that suppliers may be quoting a lower price because they think they can meet the design rules, but then they discover the process wasn't so robust. They have yield loss, which they conceal rather than talk about it. Eventually they've got to come back with this big price increase. Now you're stuck because of schedule. If you had gone with supplier B or C, that wasn't the lowest price, but they may have had more capability. When you're pushing close to the edge, a fabricator doesn't necessarily have the time and money to have 99% confidence that the process is robust enough to build it. And if they do, they're going to quote higher.

Matties: Cory, how do you alter your process when you are designing in a vacuum?

Grunwald: It depends on the end-product we're trying to develop. For a general type of board without a lot of high speed or a simpler design,

Cory Grunwald

you can go with bigger traces and spaces, and with bigger vias. But when you go into smaller and higher-speed designs, you just make assumptions and are willing to change the design at the DFM stage, if needed.

Matties: Those assumptions can be costly.

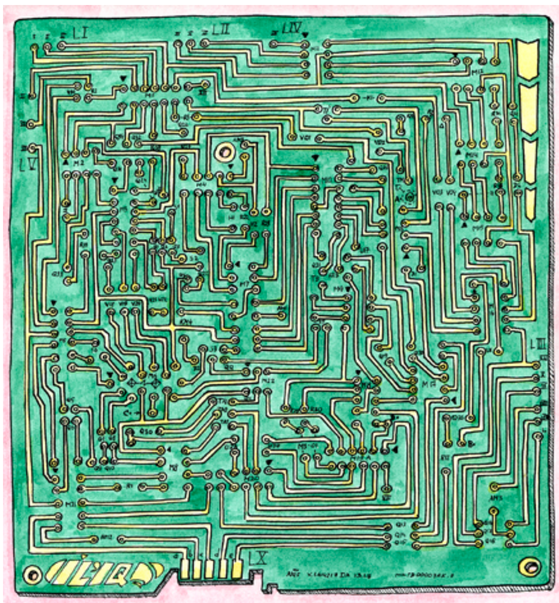
Kolar: It can be frustrating. Sometimes, we'll release the design and if we haven't been involved in the DFM and fab process, the customer may come back and say, "What the heck? This isn't buildable. Why did you design a non-buildable board? The vendor is saying, we can only do X, Y, and Z."

But we never had those requirements in the first place, so it can end up with a situation where it makes us look bad. We're also having to read the Magic 8-ball. We may not realize that they decided to change and go with a much less-expensive vendor that can't support what we thought they could, or they've changed vendors halfway through the process, and we weren't told.

Grunwald: In the end, the communication will happen anyway. It's just a matter of whether it happens at the end or in the beginning—and it costs more at the end.

Kolar: I think part of the barrier can be that, depending on your fab vendor, you're asking of their time to do pre-DFM. Depending on that vendor, they may or may not want to spend that time with you. Plan your questions well so that you can have your lightweight, upfront questions answered, such as, "What are some of the core requirements I must work around?" and, "Can you confirm the stackup is buildable?" Then you can move forward with a fair amount of confidence without wasting a ton of their time.

Shaughnessy: Jen, tell us about your pre-DFM process.



Kolar: If we are building the boards, we will work with our fab vendors in advance to get a stackup pre-approved. We'll usually do initial calculations with Polar on our own to get a starting point. Then we'll work with the vendor based on what they have in stock and get a stackup as early as possible in the process. If it's not a complex board, we're not typically sending a pre-DFM package, unless it's a crazy fast turn time. But we are doing that upfront iteration to confirm capabilities and the stackup.

Shaughnessy: In our past conversations, you described yourself as being a detective. That's really what working in a vacuum is like—you must be able to answer questions on your own. How do you approach that?

Kolar: You must dig for the information because you're not getting all the requirements.

Grunwald: That's right. From a customer's perspective, we need to have power requirements, current draw requirements, impedance requirements, and length matching. Those are a lot of things that we often must dig up from a customer. They may put it in a schematic or not even mark it up. From a board shop perspective, usually it's finding capability sheets from known shops that either I've worked with before or shops in the area, if you're lucky enough to have local shops. I'm basing my design decisions on their design specs, so I can expect that the end-shop will be able to meet those designs as well.

It can add a lot of time—making assumptions for where connectors need to be placed when it's going into a housing. I make sure I get that nailed down before I start working on anything else because that can drive so much of the rest of the design. Trying to get to the engineer to confirm placement is a big deal.

Kolar: The other place that can sometimes feel like a vacuum is trying to get feedback from the end-customer that we're going along, doing placement, trying to make sure that's approved and locked before we go to route. Meanwhile, we're not getting feedback, so we're having to make assumptions to keep moving forward. Then suddenly, we get a lot of feedback at once at the very end. It's that feedback along the way, trying to figure out how much review the end customer is really doing. How much of this is on us?

Shaughnessy: Some designers tell us they don't even know who to ask in their company to get the final answer about where the board is being built.

Kolar: Either that or everybody is too busy. Some smaller companies don't have an electrical engineer on staff. They're relying on us for more of what should be an internal function. Sometimes they literally don't know what they don't know.

Matties: What would you advise designers who are working in a vacuum?

Kolar: I think one thing would be having a good, consistent deliverable package. If you don't have any contact with a final fab vendor or assembler, make sure that your package is clean, and your fab notes are correct and thorough. Make sure your data is correct, the drawing is consistent with the net list, and the data is self-contained so that someone seeing it for the first time can pick it up and build it.

Matties: Great advice. How many DFM checks will you do on a design in-house?

Grunwald: It will usually be at the output stage. The internal tools will guide us as we're doing our routing. But once we do output, we do a DFM check internally, and if there are any errors or modifications, we go back and do a check again. The goal is that what we're outputting is clean, with zero errors.

Matties: Good communication skills are critical here.

Grunwald: For sure. Part of what Jen mentioned about having your output package being good is organizational skills. Not only should your data be the same outputs for any project you're doing, but having it organized in the same way so it can be found quickly and easily is critical. If you're going to the same shop with multiple projects, they know exactly where they can find the data they need.

Kolar: One of the other big vacuums where we rarely get feedback is about how the board worked or whether there were any issues in fab or assembly. When we go to the next build, we aren't getting that knowledge. We aren't learning anything. Closing that loop is very rare unless we're doing project to project. You don't want to propagate the same mistakes over and over.

It involves picking conservative best practices. You hope they won't throw you a curve ball by going to a low-end shop. I've seen designers trying to design rigid-flex in a vacuum, which is difficult.

Grunwald: You could run into situations where you end up with a design that's not "fab-able." If you go with the assumptions that your board shop can do a 3-mil trace and space, and this ends up going to a shop that can't do 3 mils, you now have an unfab-able design.

Matties: So, you know you're adding more cost, because you're having to compensate for a lack of information.

Kolar: I could say that it could take more time because you're essentially having to make educated guesses. That could add more time that you wouldn't have to add if you had clear information.

Matties: Exactly. Jen, we hear getting that end-user feedback is so important, but it doesn't happen very often. How do you solve that? Is there a way to find that feedback?

Kolar: It really depends on the customer, and on having good project management. We train our project managers to follow up with their customers, to reach back out and ask, "How did it go?" From our end, sometimes we won't necessarily do that, because they're busy and onto the next design. But ideally we ask, "Were there any issues we should keep in mind for next time, so that we can remove those pain points?"

Nolan Johnson: When you're dealing with the design part in the vacuum, do you start to actually calculate in the budget for some of that extra work for the design?

Kolar: Not really. When you're quoting something, you quote based on what you know, and what you think it will take, considering things go okay-ish. For some customers we say, "Oh, that customer is involved. Double the quote. They're going to swirl; they're not going to give us the info we need." But in most cases, a customer wants to get a quote that is aligned with what they expect. You're not going to get that bid if you buffer it like that. Here's another point in the vacuum—starting before the schematic is finalized or starting a quote before you have much data, when you're still guessing how things are going to evolve.

Grunwald: We will often get requests for quotes for a project that is in the schematic stage. That's difficult to quote because I don't know if these 10 pages of parts are going to be going on a board that's 12 by 12 inches or two by two inches. That's going to change the amount of time that design can take.

Kolar: There can be an information vacuum from different teams as well. You may be in a "mechanical requirements vacuum" when you're working with the electrical team. You may be waiting on the next STEP outline, but they're moving some holes around. Two weeks later, you find out they want a big mounting hole smack in the middle of where you put your processor.

Grunwald: I recently had a project put on hold because they didn't have a mechanical designer available to work on the design. After discussion with the customer, I told them that we didn't want to waste their time moving forward and then when mechanical comes back, having to make modifications for where connectors need to be placed. They agreed that we should wait until we have a mechanical designer.

Depending on the end-customer, they may like to get multiple bids from fabricators, and they don't necessarily want to decide which one they're going to use until the end.

Kolar: Depending on the end-customer, they may like to get multiple bids from fabricators, and they don't necessarily want to decide which one they're going to use until the end. With most fabricators, if you have a history of making RFQs and not going to them, they will get pretty burned out doing DFM and stackup requests. We don't want to burn that bridge.

Matties: What do you think is the most important skill for a designer to learn, especially younger designers?

Grunwald: Communication is a key skill to learn. It's not something you get in engineering school—learning how to manage a customer or manage your end-product. We've recently hired two new engineers. We're teaching them how to communicate with customers and get the requirements they're looking for.

Kolar: It's a lot of best practices. For us, we have a lot of manuals, guidelines, and best practices as the standard. Even if you aren't getting the information, being able to manage your customer, explain the impact of not getting the information, and the assumptions you are making, is very important. Otherwise, I would fall back on: quality matters most. Don't take shortcuts, and don't slap it together. If I knew nothing about this design, would my directions stand alone? Will the fabricator know what to do? Will the assembler know what to do without having to ask questions?

> Even if you aren't getting the information, being able to manage your customer, explain the impact of not getting the information, and the assumptions you are making, is very important.

Matties: Those are great checkpoints, for sure.

Grunwald: You must be able to look at it from the perspective of the fabricator and that's not common knowledge for the typical designer.

Kolar: That is true. We train our designers over time and get them involved in that process, so they understand how the board is built. What's the actual process? It's not just what you can do with your tool. It's how physics works and how the actual chemistry of the process works. Many young designers and engineers don't understand the "sandwich" of how a board is made or the impact and the cost of via structures.

Shaughnessy: You're basically brain-dumping onto these new designers because they can't possibly know most of this stuff early in their careers.

Kolar: Yes. We are in a unique position because we do a lot of manufacturing from our own designs. Since I tend to be the escalation point, if I get a phone call at night with a fab question, I try to share that "love" with the designers, so that they're learning from the process. We encourage our project managers to go back and tell the designer, "Here's what I encountered in DFM. Here's what we learned from this, and let's do this differently next time." We try to share that knowledge and complete that cycle, so they're learning.

Grunwald: Some of it comes down to the comfort level of the designer. If they haven't dealt with fab shops directly, they may not know what they need to ask or how they need to manage things. Making a relationship with fab shops that you use regularly helps in the end. If your purchasing department is the one that picks shops, knowing the couple shops you will use and building a relationship with them can be key to being able to feel comfortable asking those questions.

Matties: Any final thoughts?

Kolar: It can be very frustrating and demotivating to work in a vacuum, so just try to accept reality: "They didn't give me enough input, so they're probably going to have problems in fab. I will do the best I can do."

Matties: There are probably therapy groups for that (laughs).

Kolar: Those are called internal designer meetings (laughs).

Matties: That's really what we want to do here—acknowledge that this is the reality.

Kolar: It doesn't have to be that way though. It can work smoothly if you have the right collaboration or project management at the end. Also, a lot of fab vendors like to have the appearance of being easy to work with: "Just throw your data over the wall and we'll take care of it." Sometimes, you wonder if the fabricator is just going to make some assumptions vs. opening the cupboards a little bit and seeing what's messy.

Johnson: Those assumptions rarely get back into the original design so that the OEM knows how it's being built.

Grunwald: Right. It's also going to just add time to your manufacturing process. If your fab shop ends up having to come back with DFM questions, that's just added time. If you ask for a five-day spin, the five days may not start until that DFM is clean.

Matties: Happy, did you have any final thoughts here?

Holden: We've had to live with this for years, and every time we move forward, everything gets more complex, and we're still back to more vacuums.

Grunwald: One reason that you shouldn't make assumptions is that manufacturing processes are changing so quickly and improving. If you don't ask your board shop, you don't know what you might be missing out on.

Shaughnessy: This has been very good. Thank you both.

Kolar: Thank you all. DESIGN007

Reflecting on Reflections

Beyond Design

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

When a transmission line is perfectly matched to the driver and load, the signals propagating electromagnetic (EM) energy are totally absorbed by the load. This is the perfect scenario that all electronics designers strive for. However, this is rarely the case and reflections do occur whenever the impedance of the transmission line changes along its length. This can be caused by unmatched drivers/loads, layer transitions, different dielectric materials, stubs, vias, connectors, and IC packages. By understanding the causes of these reflections and eliminating the source of the mismatch, a design can be engineered to perform reliably.

There are many ways to dampen reflections in high-speed PCB design, as listed below:

Drive Strength

Reflections can be caused by using a drive strength that is too high for the load. Driver strengths typically range from 4 mA to 16 mA. A 16 mA driver is generally required for multiple loads—for instance, a DDR4 signal driving multiple SODIMM memory cards. In this case, the transmission lines are longer and the capacitive load higher, so simulation is necessary to confirm the required driver's current strength. To dampen the signal, terminations are typically placed on the memory card itself. However, if the signal is delivered to only one or two on-board memory devices, then the signal strength can usually be reduced to a minimum of 8 mA to prevent reflections.

The simulation in Figure 1 shows a clean signal with a wide eye opening, but there is a slight over-and undershoot of the signal (top and bottom) and a small amount of timing jitter (horizontal misalignment). The over/undershoot can be attributed to either over-driving the transmission line using a higher

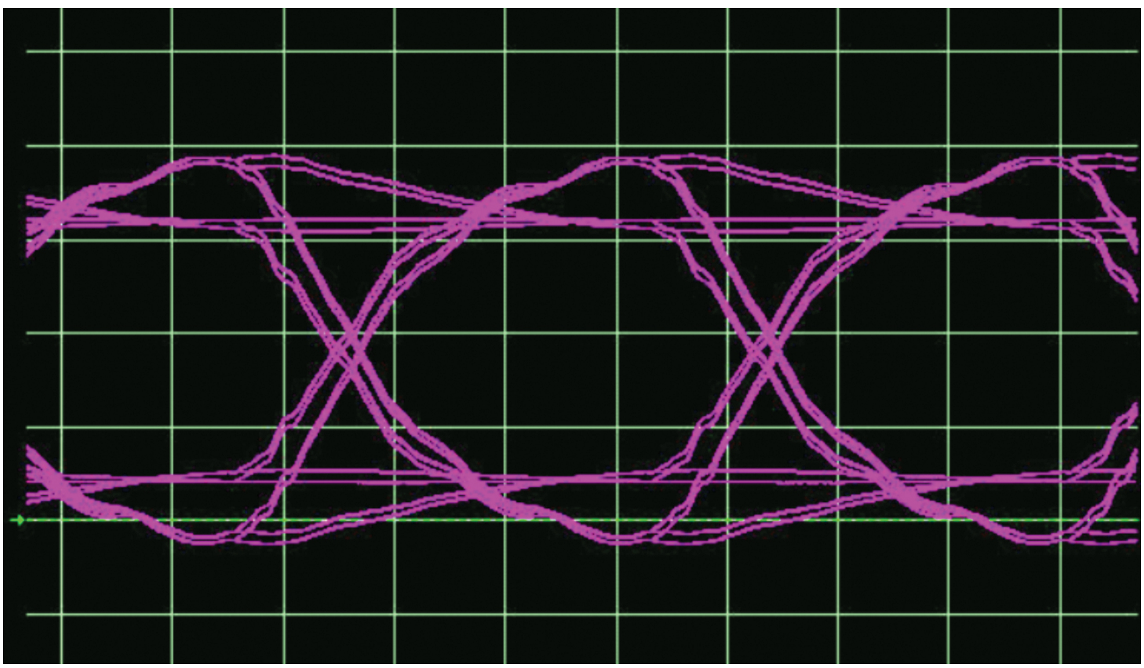


Figure 1: Reflections cause over/undershoot when the transmission line is over-driven.

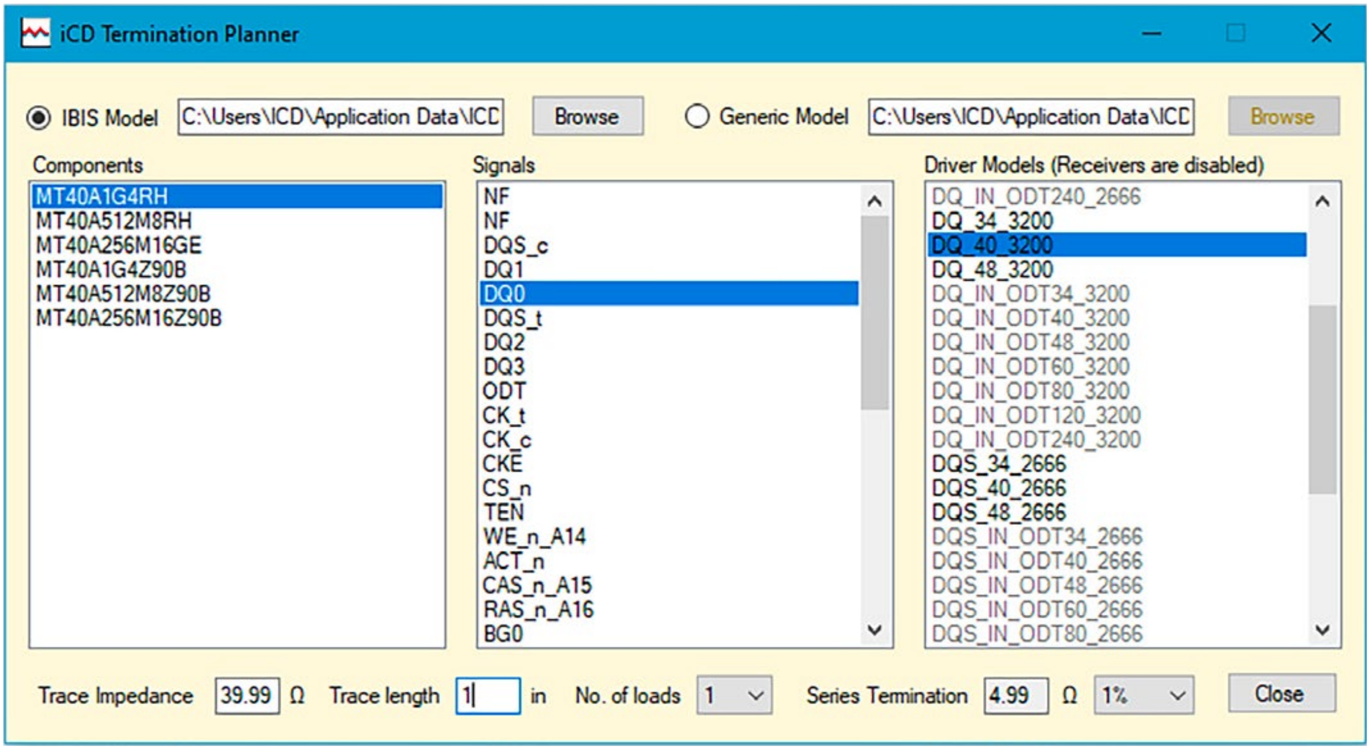


Figure 2: Micron 4 Gb DDR4-3200 SDRAM with ODT. (Source: iCD termination Planner)

drive strength than required or over-termination on the receiver side by using an external resistor value that is higher than the characteristic impedance of the transmission line. The degradation of the signal due to overdriving the transmission line might be sufficient to result in system failure.

On-Die Termination

On-die terminations (ODT) can be used with some memory devices to match the transmission line and dampen the reflections. Incorporating a resistive termination within the DRAM device improves the signaling environment by reducing the electrical discontinuities introduced with off-die termination. DDR4 and DDR5 technology, like DDR3, include ODT on the data I/O pins. This feature is controlled by the ODT pin and consumes additional power when activated. The ODT and the output driver on DDR4 devices include additional mode register settings over the previous DRAM to increase system flexibility and optimize signal integrity. The ODT impedance can vary from 34-240 Ω in receive mode and 34-48 Ω in transmit mode in Figure 2. DDR4 signal-ended signals are normally routed on 40 Ω transmission lines so, for short trace lengths with one load, a 40 Ω ODT is perfect. However, since the ODT and driver strength are software selectable, one can validate and tune the strength during the testing phase of development.

Series Termination

Unfortunately, drivers do not have the same impedance as the transmission line (typically 10–35 Ω) so series terminations are used to balance the impedance, match the line, and minimize reflections, particularly on long traces where ODT is not provided. Impedance matching slows down the rise and fall times, reduces the ringing (over/undershoot) of signal drivers, and enhances the quality of a high-speed signal. The ringing is dramatically reduced by adding a series terminator as in Figure 3. From this, we can see that the impedance has to be matched—but to what value?

In Figure 4, using a 12 mA LVCMOS 1.8V

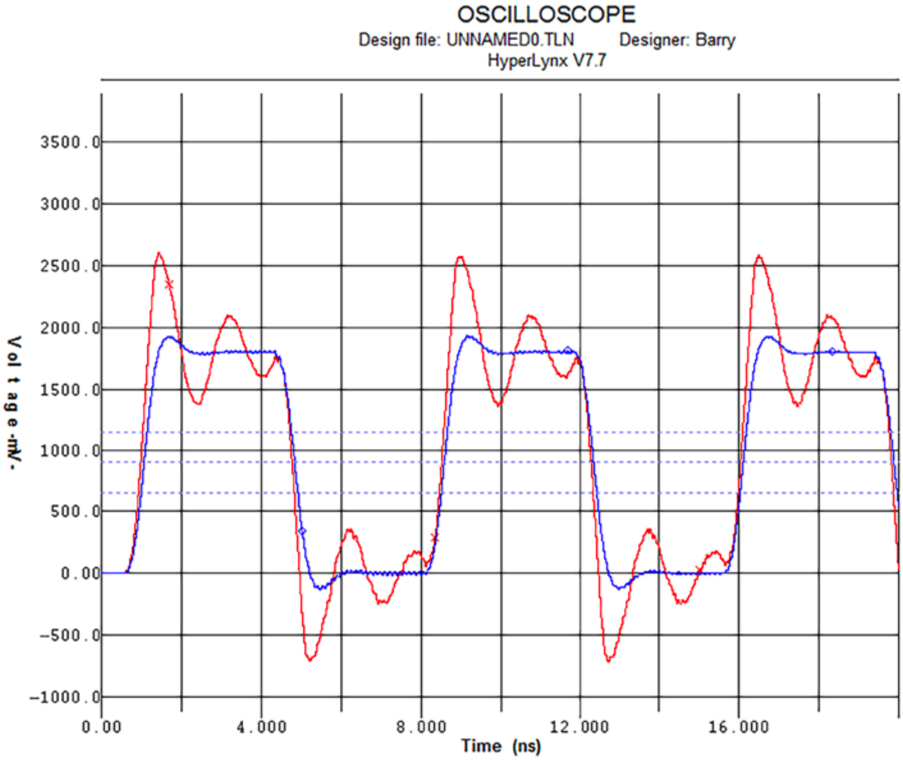


Figure 3: Unmatched (red) vs. matched (blue) transmission lines.

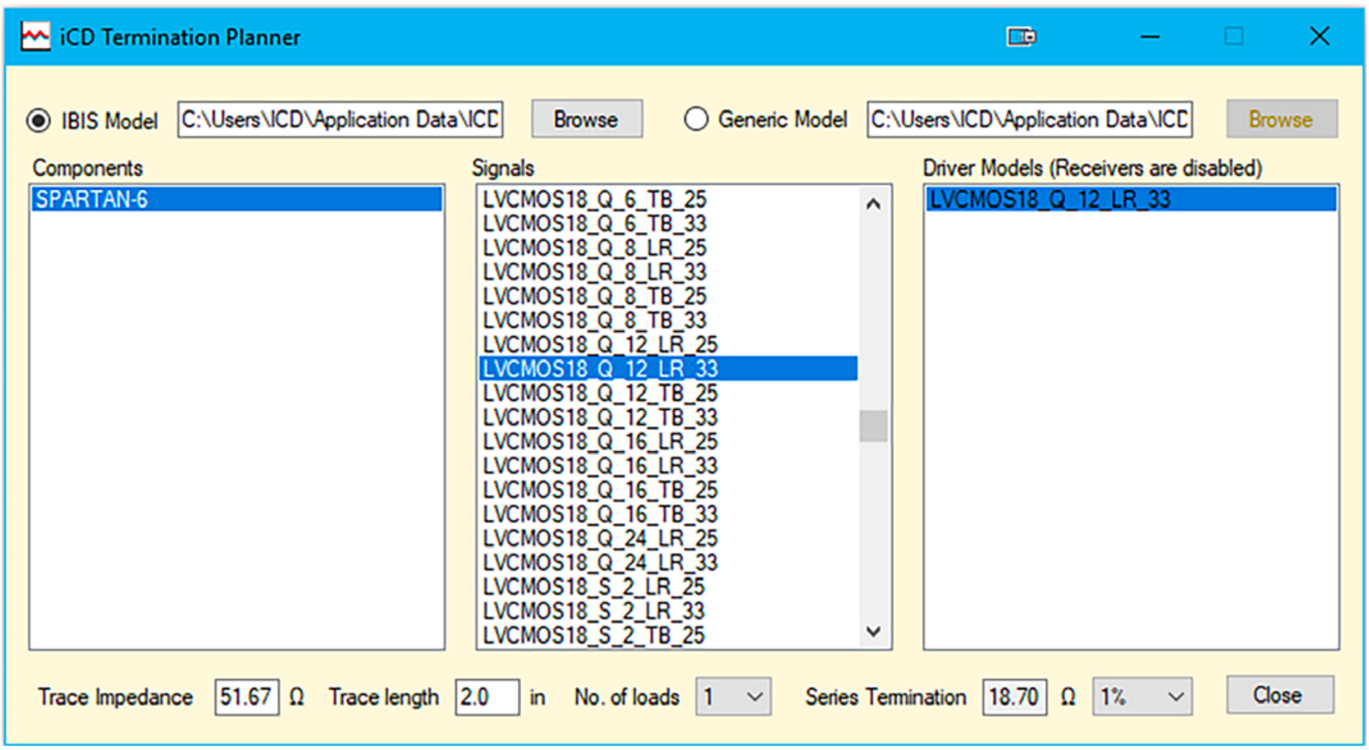


Figure 4: Matching the Spartan 6 driver to the transmission line. (Source: iCD Termination Planner)

driver of a Spartan 6 FPGA, an 18.7 Ω series resistor is required to match the driver to the 51.67 Ω trace on the outer layer. This is automatically derived from the IV curves of the Spartan 6 IBIS model by the iCD Termination Planner (Figure 4).

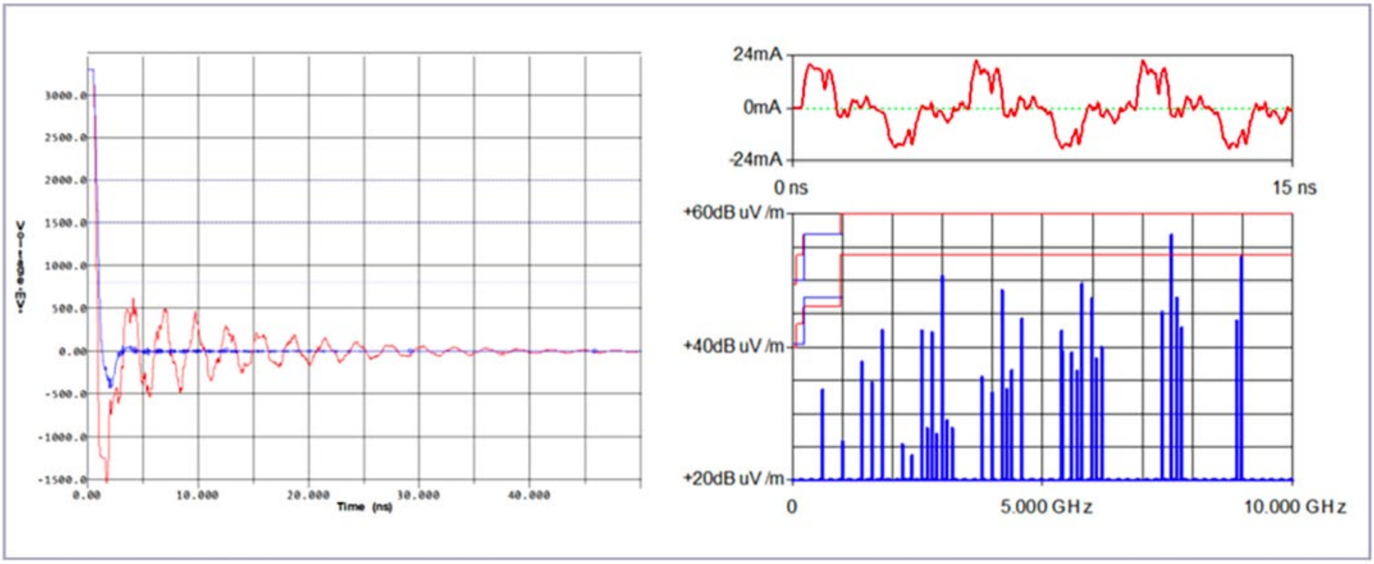


Figure 5: Ringing and radiation are reduced dramatically by adding a series terminator.

Figure 5 (left) illustrates the ringing (red) in an unmatched transmission line. This ringing, which also causes electromagnetic radiation (right), is dramatically reduced by terminating the transmission line with an 18.7 Ω series resistor.

Crosstalk

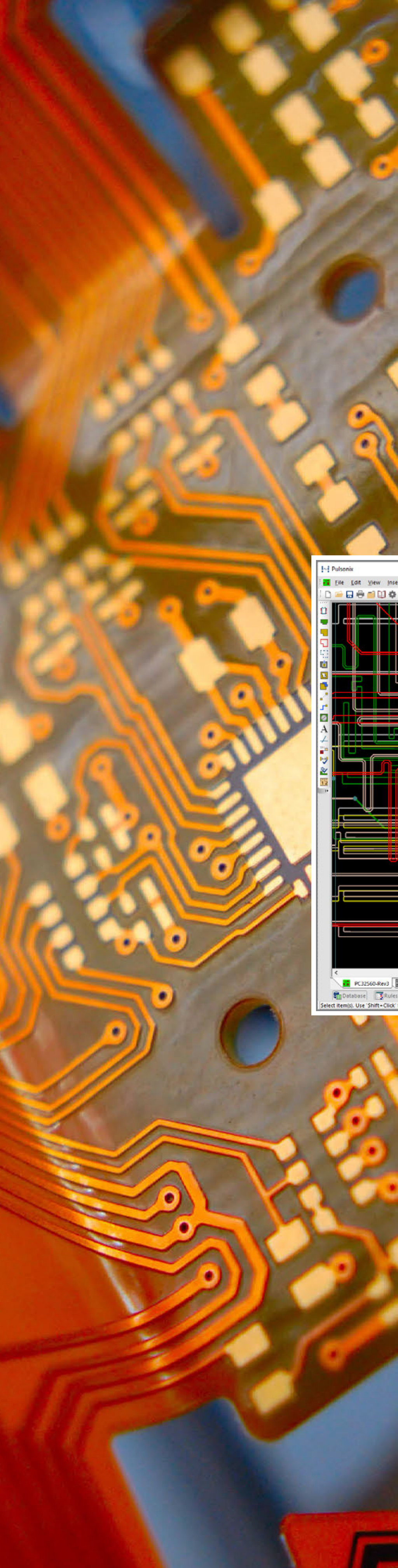
Another culprit is crosstalk, particularly on long parallel trace segments. Crosstalk arises as a result of the unintentional coupling of electromagnetic fields and causes both forward and reverse reflections. The easiest way to reduce crosstalk from a nearby aggressor signal, of course, is by increasing the spacing between the signals in question. Crosstalk falls off very rapidly with distance. Crosstalk plummets roughly quadratically with increased separation. Doubling the spacing cuts the crosstalk to roughly a quarter of its original level. A good rule of thumb for this is Gap = 3 x trace width. However, in today's complex designs it is not always possible to use up valuable real estate to satisfy the above. Reducing the signal trace to reference plane dielectric thickness can also reduce crosstalk while not requiring additional space. Also, different technologies should not be mixed as higher voltages create higher crosstalk. And long parallel trace segments should be avoided.

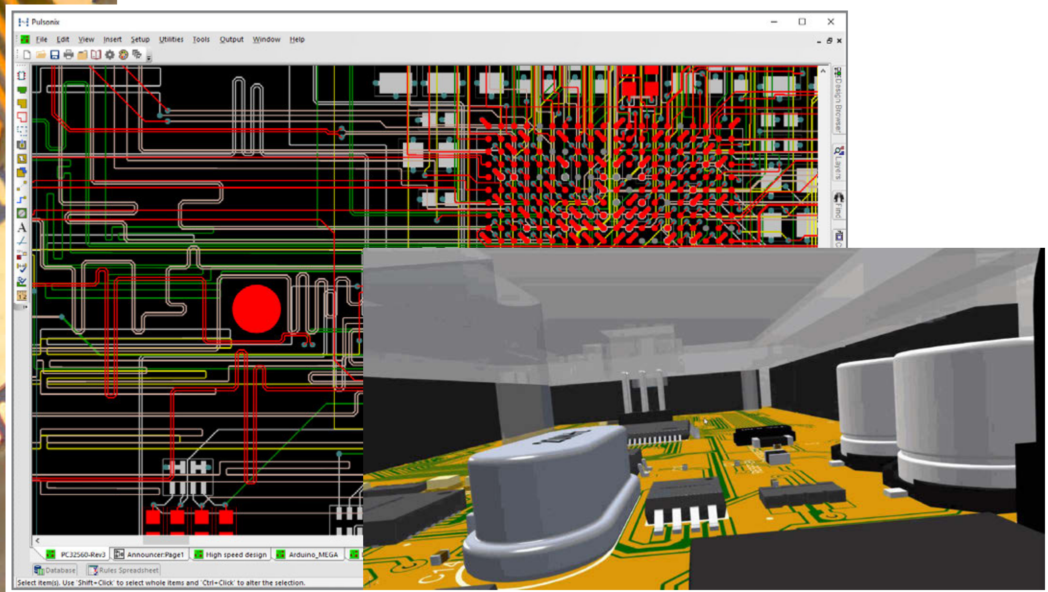
Crosstalk also depends on the load which may vary considerably when driving banks of memory modules, for example. Keep in mind that the total crosstalk on each victim trace is the total crosstalk from each of several nearby aggressors, all of which sum to produce the maximum value.

Plane Cavity Resonance

Plane pairs, in multilayer PCBs, are essentially unterminated transmission lines—just not the usual traces or cables we may be accustomed to. They also provide a very low impedance path, which means that they can present logic devices with a stable reference voltage at high frequencies. But, as with signal traces, if the transmission line is mismatched or unterminated, there will be standing waves—ringing. The bigger the mismatch, the bigger the standing waves and the more the impedance will be location-dependent.

When the cavity has open end boundary conditions, resonances arise when a multiple of half wavelengths can fit between the ends of the cavity. Figure 6 shows the cavity resonance of a plane pair with a resonant frequency of 1 GHz. If the signal clock frequency (or har-





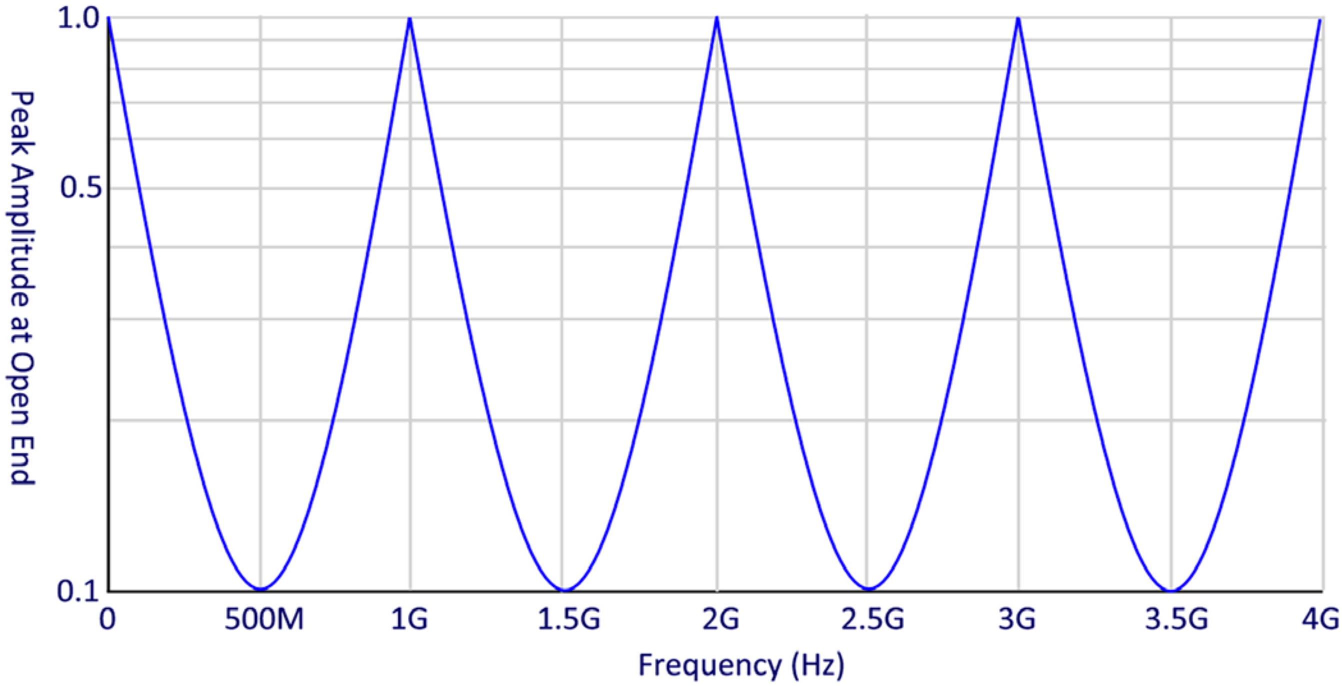


Figure 6: Amplitude, at the far end of planes, as the input frequency is swept. (Source: Eric Bogatin)

monics) is multiples of 1 GHz, then noise can be injected into the plane cavity. When the clock or data harmonics overlap with the cavity resonant frequencies, there is the potential for long-range coupling between any signals that run through the cavity, thus affecting signal integrity as a consequence of inadequate power integrity.

Plane cavity resonance and emissions can be reduced as follows:

1. A thin dielectric, in the plane cavity, is the most effective way of reducing the peak amplitude of the modal resonance. It reduces spreading inductance and the impedance of the cavity, and also reduces the resonance peaks by damping the high-frequency components.
2. RC terminations, of 3R5 in series with 10nF connected across the plane pair and placed along the board edges, are generally sufficient to reduce the resonance peaks.
3. The parallel resonant frequencies, of the cavity, can be pushed up above the maximum bandwidth of the signals by reducing the plane size and by adding stitching vias between (similar) planes of a cavity.

When a signal's EM energy propagates from the driver to the receiver along a transmission line, it changes along its length. The original signal will be received with varying degrees of distortion and degradation. This signal distortion happens due to factors such as impedance mismatch, reflections, ringing, crosstalk, dielectric loss, jitter, and ground bounce. The PCB designer's primary objective should be to minimize these issues at the source, so that any signal distortion is eliminated.

Key Points

- Reflections can be caused by using a drive strength that is too high for the available load.
- Over/undershoot can be attributed to overdriving the transmission line using a higher than required drive strength setting on the driver.

- On-die terminations (ODT) can be used with some memory devices to match the transmission line and dampen the reflections.
- Drivers do not have the same impedance as the transmission line (typically 10–35 Ω) so series terminations are used to balance the impedance, match the line, and minimize reflections.
- The easiest way to reduce crosstalk from a nearby aggressor signal is by increasing the spacing between the signals.
- Reducing the signal trace to reference plane dielectric thickness can also reduce crosstalk while not requiring additional space.
- Plane pairs in multilayer PCBs are essentially unterminated transmission lines.
- If the plane cavity is mismatched or unterminated, there will be standing waves—ringing. DESIGN007

Resources

1. Beyond Design columns by Barry Olney: "The Fundamental Rules of High-Speed PCB Design, Part 2;" "Stackup Configurations to Mitigate Crosstalk;" "Controlled Impedance Design;" "Plane Cavity Resonance;" "Dampening Plane Resonance with Termination;" "Signal Integrity, Part 3."

Errata: In the February 2021 issue of Design007 Magazine, the Beyond Design column, "Stackup Configurations to Mitigate Crosstalk," contained two errors in equations 1 & 2. The term Lm/Ctotal should have been Lm/Ltotal. Please download the revised PDF to replace your library copy today.

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.

U-Boat Worx Launches 9-person Flagship Lithium-ion Battery Submersible NEXUS

Dutch submersible manufacturer U-Boat Worx breaks the mould with the launch of the NEXUS series. U-Boat Worx is the market leader in private and commercial submersibles.

The NEXUS series comprises two models featuring an ultra-large elliptical acrylic pressure hull with unrivalled passenger comfort. Seating up to eight passengers and one pilot, the NEXUS provides 25% more interior space than competing models. The NEXUS subs are depth-rated to 200 meters (650 feet).

Equipped with the latest lithium-ion battery technology, the subs can operate for up to 18 hours and dive up to ten times a day. 80 guests can participate in an unforgettable sub-sea adventure daily.

The unique revolving seating arrangement featured in the Cruise Subs has been replicated in the NEXUS. Passengers therefore always enjoy the best view, regardless of the direction the submarine is travelling. The NEXUS subs are designed as multi-directional, meaning they can manoeuvre in any direction without compromise. This is a distinguishing capability only achieved in U-Boat Worx submersibles. Ten silent thrusters provide the power required for top performance and unmatched speed.

The experience and knowledge gained from the development of the Cruise Subs has evolved the new features in the NEXUS series, including an XXL entrance hatch, the largest ever incorporated on a submersible of this size.

The NEXUS is optimised for ship-based launch and recovery and can also be deployed from land and transported to dive sites.

(Source: ACN Newswire)

Designing in a Vacuum Q&A: Mark Thompson

Feature Interview by Andy Shaughnessy
I-CONNECT007

I-Connect007 Columnist Mark Thompson of Out of the Box Manufacturing has been in CAM engineering for decades, and he's also worked as a PCB designer, so we knew he would have a few things to say about working in a vacuum. As he explains, the designer isn't the only stakeholder in the process who feels like he's working inside the dust bag of a Hoover upright.

Andy Shaughnessy: How often do you design a board without knowing who is going to fabricate and/or assemble it?

Mark Thompson: That depends on the complexities of the given job. As an example, if the job/design has controlled impedances, a good design bureau or company will consult with the fabricator for the lines/spaces and dielectrics needed to achieve the impedances prior to design and layout. So, the designer knows who will be fabricating the job. If there are special considerations, the design engineer may even visit the facility to make sure they understand the intricacies of the job. Many companies will have a design engineer or a field application engineer pre-qualify a fabricator to make sure they can do the work and go over any details necessary to pave the way for success of the job.

I have heard of designers "camping out" with the fabricator for the entire fab process if the

Mark Thompson

board is extremely involved. Additionally, it helps for designers and fab engineers to have a solid working relationship. If the designer and fab engineer are working together for the first time, when the fab engineer has a question about the design, he will typically call the contact person (often a project manager or purchasing person), who will connect him with the designer. This is where a great fab engineer who asks unpleasant questions is indispensable.

Unfortunately, things may change, and the designer may not know who will be fabricating the part or even assembling it. For instance, the chosen fabricator may have a long lead time or some other previously unforeseen complication. In this case, the part may have to go with another fabricator. Please note: When this happens, if the job is controlled impedance, it is a good idea to have the new fabricator run the calculations to see if there will be any large trace or space swings that result in a redesign. This includes things like increasing traces when you've already designed to the minimum space allowance.

Other hurdles, such as choosing a different material type (due to supply chain issues), may result in a complete redesign, which, unless you are a glutton for punishment, is not so fun. Likewise, the designer may have an idea who will be assembling it, and he may not. This is where I get on my soapbox. If you genuinely want to make sure something gets done, communicate it clearly.

Shaughnessy: Why do so many designers not know who is going to manufacture their boards?

Thompson: I can think of two reasons. The first reason is company culture. If the designer works for a company that has a purchasing department that does not share information about where the part will be fabricated, designers must ask probing questions. Companies today have numerous design and engineering meetings where this information can be shared, but frequently it is not, for whatever reason. I see no reason a purchasing person would intentionally withhold information regarding the selected fabricator, but it could happen.

Remember that if cost is their sole reason to choose a fabricator, you get what you pay for. If a job requires multiple respins due to a fabricator's lack of quality control, you have lost more than just time. I think that a thorough company will always share such information with the designer and engineer on that project, especially if there are any specific items to be passed along to the fabricator relating to the success of the part.

There is one other possible reason for designers to be left in the dark about the chosen fabricator, also related to company culture: Sometimes, the purchasing person may not care where the part is fabricated, provided the result is as anticipated, and the cost and lead time are good. To me, this leaves a lot on the table, with the potential for miscommunications about the expectations and net result. What may be a "simple" board to a designer may be less than simple when it reaches the fabricator.

Shaughnessy: What other missing information or data do you find yourself having to "design around?"

Thompson: PCB design is a hierarchy. As a designer, you start with what is known and ask relative questions about the unknowns you can foresee at the first customer meeting. You will start by reviewing the schematic so that you have an idea about power and part placement.

Designers may have more questions when the design progresses. Speaking of power, power management ICs often raise questions for the designer. A former boss of mine said of PMICs, "I had one instance where I was able to follow the reference/input very closely, and I thought it would be good, but it still needed some difficult work to get it right. Adding or moving caps and trying to squeeze more copper area or vias is often not easy to do in crowded areas."

There is no shame in asking questions but try to catch as many of the relative questions in the first or second go-arounds. Customers don't appreciate an endless stream of questions and, more importantly, the resulting time lost for those additional questions.

The only things you can "design around" are the non-essentials, like mask color or impedance calculations. These things are done at layout and part placement occurs prior to layout. All stages of the design effort should have checks and balances, such as a part placement review by the end-user.

Let's review the typical design process and see where these gaps can be filled in:

1. **Kick-off meeting:** Schedule a meeting with the customer to discuss variables. This is the first meeting, so having your ducks in a row is a must. Review the schematic and formulate questions regarding the expectations. This is the customer's first chance to share any specific concerns about the part or specific placement concerns.
2. **Part placement review:** This is the second opportunity for the customer to change part location or orientations prior to trace layout. Many times, the customer had forgotten to tell the designer, or the designer did not ask in the kick-off meeting, and part placement changes must be made.
3. **Final review of part/traces prior to exporting the final package to the end user:** This is the final chance for the customer to make changes. Sometimes it gets all the way to the final review before it is realized that parts will have to change, due to availability, and a new schematic created. On a particularly tight design schedule, this may cause you some grief.

On the topic of missing information, customers invariably have some picture in their mind of what the layout should be.

On the topic of missing information, customers invariably have some picture in their mind of what the layout should be. Not only do we have to divine what that is, via additional conversations or other tools at our disposal, but we must balance that with our own knowledge of best practices for the given circuit and our customer's ideas on implementation. And then be able to create something that meets their expectations, even if it must be something different than what they thought it would be.

Some engineers hold everything close to the vest and divulge very little, but they should understand that a good designer can look at the parts list and schematic and tell what the product is, no matter how well the engineer conceals it. Remember the designer is your ally not your enemy.

Shaughnessy: When you find yourself designing in a vacuum, what tricks have you devised as workarounds to get the job done?

Thompson: Research, resources, and resolve. Again, there is no shame in not knowing something. The shame comes in when you don't even try to find out and give up.

As far as research and resources:

1. **The internet:** Many, many technical processes are discussed online. If you really search, you will likely find the answer.
2. **EDA software forums:** There's a lot of info on these EDA tool forums, but make sure you are using the latest rev of the software.
3. **Other designers:** Speak with other designers and you'll often find that they have experienced the same problem on a previous project and can help you with a workaround.

There is no shame in not knowing something. The shame comes in when you don't even try to find out and give up.

This reminds me of a presentation I did for HP years ago. There were at least 50 engineers in the audience. At the outset, I gave my normal disclaimer: "Do not discuss or ask questions relative to anything we have an NDA on." This cut out a lot of questions and conversation. NDAs can be intimidating.

When I finished my presentation, there were no questions. But I noticed that many of the engineers in the audience were looking to "Bob" to see if he would ask any questions. Noticing this, I made eye contact with Bob and spoke directly to him. After a while, Bob started asking questions, which opened the entire audience to do the same and made my visit worthwhile. My point is that sometimes simply having a conversation with another designer will be the spark that lights the fire.

Sometimes you get so entrenched with a particular design that you need to take time to revisit your options. It's time to reset and refresh. I often have the greatest epiphanies after I have gone to bed. If this happens to you, wake up and write down your thoughts. If you think you will simply remember them in the morning, you are mistaken.

Lastly, resolve yourself to finding the answer to get the job done.

Shaughnessy: What are some of the hidden costs, in addition to monetary, of designing in a vacuum?

Thompson: If you don't know something, you may be out of more than just money. As the adage says, "Time is money." This has never been truer than for board design.

Shaughnessy: What advice would you give designers who are trying to get out of the vacuum?

Thompson: Do the research. If you run into a snag, try another way. Try to find the answer elsewhere if you cannot ask the customer directly.

Shaughnessy: Thanks for your help, Mark.

Thompson: Thank you, Andy. DESIGN007

Enabling a Digital Thread Across IC/Package/PCB Design

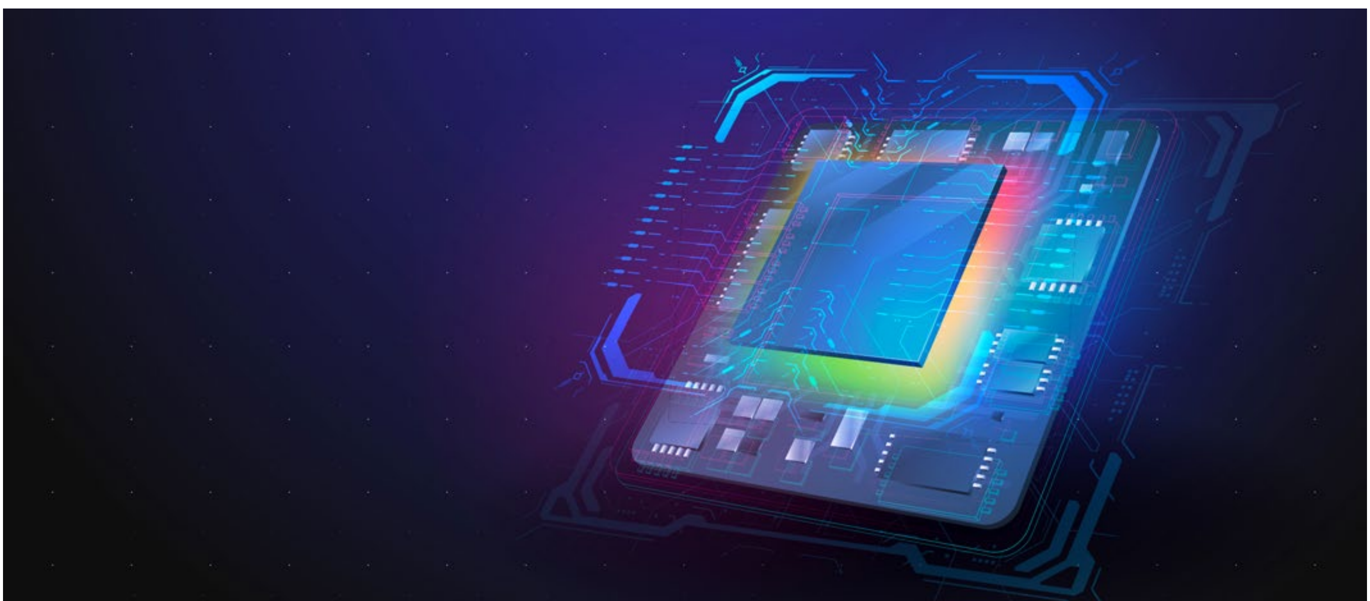
Digital Transformation

by David Wiens, SIEMENS EDA

If you've been keeping up with the electronics trade news, you're probably aware that there is a slow but steady growth of ICs now being implemented in 2.5D or 3D IC configurations. Over the last decade, these device configurations have been steadily growing in popularity in highest-capacity FPGAs, high-bandwidth memory devices, and processors targeting high-performance computing and datacenters. But with Apple's recent announcement of M1 Ultra—which will power its new generation of desktop and laptop computers—the age of 3D IC is quickly coming upon us and may become the norm rather than the exception. It behooves us to ask: What, if any, impact will this have on PCB systems design?

Typical 2.5D design processes design each substrate (e.g., the ICs and interposer) separately, then integrate at the end (often with derived data like spreadsheets), where errors are found, and time-consuming iterations begin. Optimized performance of 2.5D or 3D IC configurations requires a co-design methodology across the ICs and interconnecting interposer. Likewise, co-designing the PCB(s) in context with the IC package yields the same benefit. Co-design requires a digital thread across all fabrics, ensuring that all design teams are working with the same data, and enabling cross-substrate trade-offs.

Often the bridge between chip and system PCB, IC packaging offers the best opportunity for optimizing signal performance, power delivery/integrity, and interconnect optimization. Using a multi-substrate design approach, simulation becomes an achievable reality, from



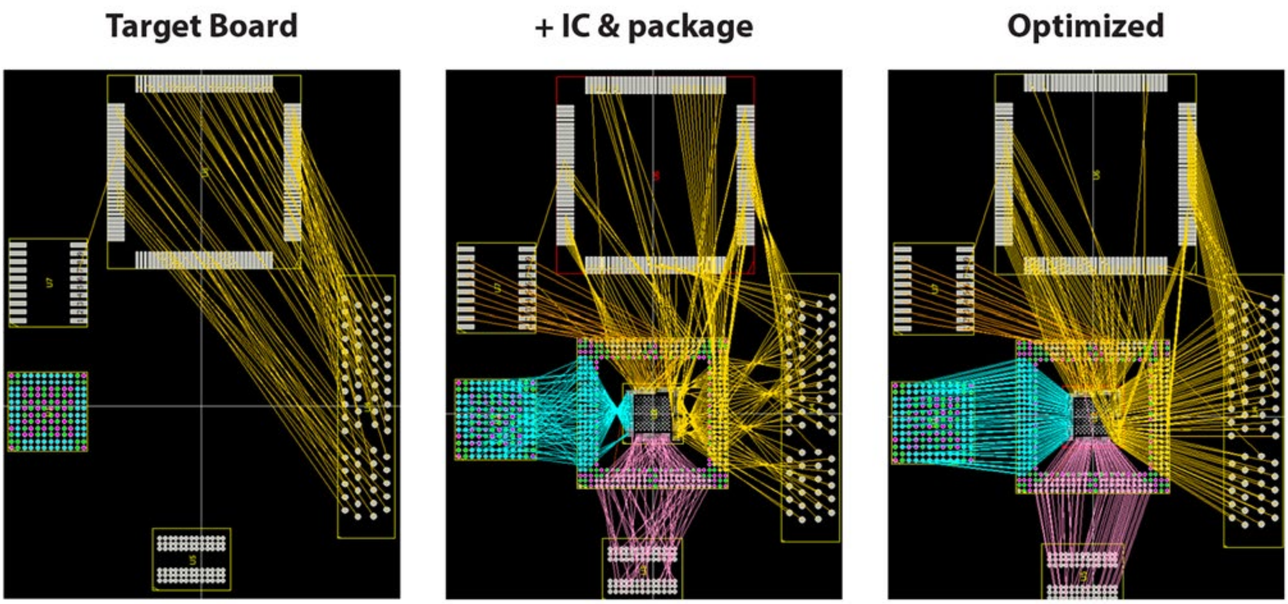


Figure 1: System path uncrossing and route optimization before layout.

the chip's buffer block through package routing and on to system PCB implementation. Optimizing critical signals down interface paths throughout the system will result in far fewer downstream issues that will require resolution.

The traditional path to system design is to know what you need to build, to identify what must be done, and then to design to those requirements. The defense and aerospace industries have had great success enforcing this method of constrained design, especially for complex system designs where the correct-by-construction approach speeds design time, improves quality, and lowers costs by reducing design iterations.

The correct-by-construction approach forces designers to make assumptions regarding the limiting issues in the design and then to simulate these assumptions in software before building a physical prototype. Before translating a conceptual design limitation into a physical design constraint, designers must understand the source of the challenge and where to go in the design to resolve it. This requires designers and engineers to see the big picture to make good decisions.

Visualizing and comprehending the entire multi-substrate design while keeping essential and critical design requirements in mind often overwhelms the designer and design team. Typically, the task of creating a new SoC/chip, its corresponding package, and a new system board(s) involves three different engineering teams with three different perspectives. Although it can be possible to get the different design disciplines together in a coordination meeting, a successful joint plan really requires a methodology, process, and usually some technology and automation for tying these worlds together.

There are three concepts that, when added to traditionally deployed design methods, can enable the creation of an optimal system product: multi-substrate visualization, system-level cross-substrate interconnect untangling, and cross-substrate co-design. Multi-substrate visualization means not solving micro, or single substrate design issues until you can see the whole picture. Once you step back and look at the whole design, the pieces of the solution start to become clear.

When designing a new die such as a processor, the system target is an industry-standard memory chip with a fixed pin-out. Unraveling starts from the system PCB and drives toward the die, as the PCB design domain drives the optimization. If you co-design the PCB, package, and die, pre-optimization enables ideal, easy-to-implement routing. Co-design provides design flexibility in all three domains, as the design of the target PCB, package, and die pads all begin at the same time.

System-level optimization isn't just about simplifying the life of the poor PCB engineer, who is often on the receiving end of a complex new 2.5/3D IC. A streamlined co-design methodology that seamlessly interchanges digital twins of ICs, packages and boards optimizes collaboration and minimizes rework. Attacking the full system problem at one time, rather than in discrete and serial chunks, enables performance optimization and cost reduction at each substrate level. **DESIGN007**

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

Additional content from Siemens Digital Industries Software:

The Printed Circuit Assembler's Guide to ...Smart Data: Using Data to Improve Manufacturing

The Printed Circuit Assembler's Guide to... Advanced Manufacturing in the Digital Age

Siemens' free, 12-part, on-demand webinar series "Implementing Digital Twin Best Practices From Design Through Manufacturing."

RealTime With ... Siemens and Computrol: Achieving Operational Excellence in Electronics Manufacturing

---

COLUMN EXCERPT: All Systems Go!

Supply Chain Woes: Which Comes First, the Design or the BOM?

By Nitin Bhagwath, Cadence

In an ideal world, when developing a printed circuit board (PCB) for an electronic product, decisions made during the design process should drive the bill of materials (BOM). We may think of this as an example of "the dog wagging the tail." In the real world, however, there has always been some small amount of the BOM driving the design, which we may think of as "the tail wagging the dog." A classic example of this is when an engineer's calculations indicate the need for a resistor of 123 kΩ—a 40-cent part—while a 120 kΩ resistor—available for only 4 cents—will provide an almost identical response.

Current realities have made such BOM-driving-the-design decisions more inescapable to ensure product manufacturability. Everyone around the globe—from small companies to mega-enterprises with trillion-dollar valuations, all the way to the U.S. government—is currently facing unprecedented supply chain challenges. Supply chain optimization has long been under pressure, involving as it does a sophisticated balance of low cost, future availability, and product needs. While it's true that difficulty obtaining all the components that are perfect for the task at hand is not totally unprecedented, alternative parts used to be plentiful. This meant that, even if you couldn't get exactly what you wanted, you could get something similar. But with supply chain disruptions around the globe, even next-best parts are hard to find, making it critical to ensure component availability for the design.

One part of the solution is for everyone in the organization to have visibility with respect to stock levels. It's important for this visibility to commence at the earliest stages of the design while the engineers are working on the initial high-level block diagram, thereby allowing for the design architecture to accommodate parts that can actually be sourced.

To read the entire column, click here.

Happier in a Vacuum: The Design Narcissist

Target Condition

Feature Column by Kelly Dack, CIT, CID+

1. Narcissist (Oxford Def): a person who has an excessive interest in or admiration of themselves. "Narcissists think the world revolves around them."

Older PCB designers from the aerospace industry will surely remember the fictitious MIL-TFP-41 spec (Make It Like the Freakin' Print for Once), which was a common expression of frustration by un-empathetic engineers regarding the status of their partially built or incorrectly-built designs. Sadly, missing a target condition on a run of printed circuit boards is a two-way street. Root causes may show a breakdown of process on the part of the supplier, but all too often perceived fab or assembly failure is due to missing or fuzzy design specification, abnormally tight tolerancing or warped expectations borne from prototyping in the lab using "unobtainium," which is unavailable to production suppliers.

Without a doubt, arrogant workplace attitudes have existed in this industry over the decades. Hopefully, as a profession, PCB designers are moving forward and accepting the fresh, positive signals from the many training leaders and trade organizations emphasizing the importance of stakeholder awareness. These visionary champions are conveying the message that we are better off when we look out for all our important industry stakeholders' functions collectively, as we perform within our personal areas of expertise.

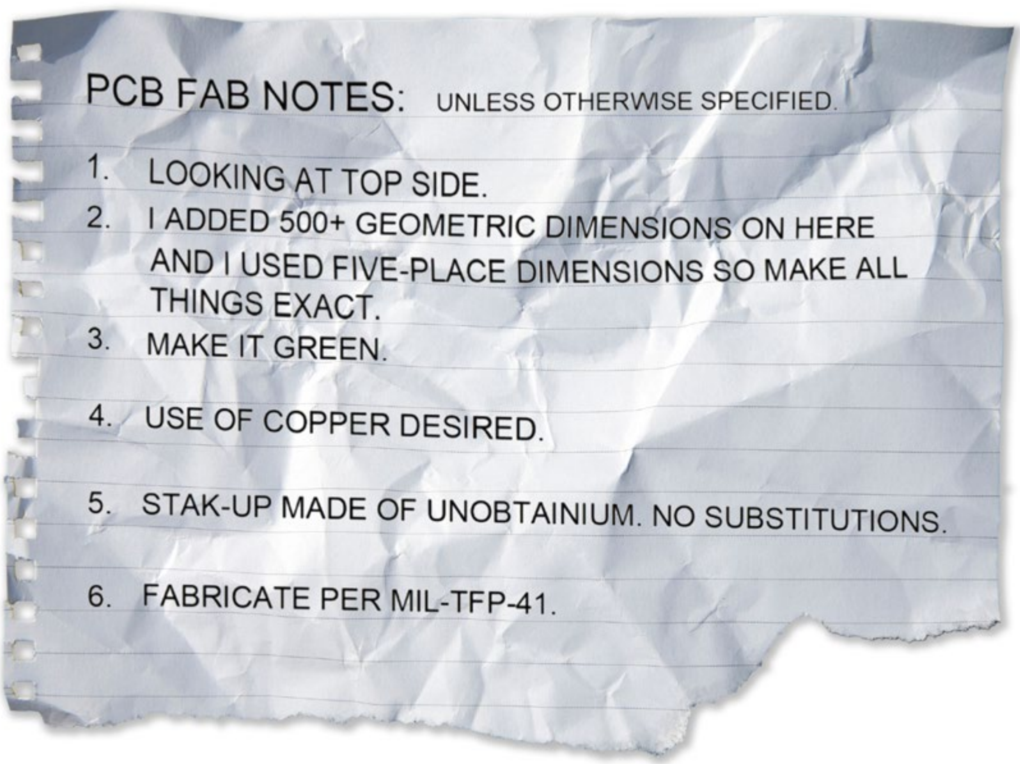


Figure 1: Fabrication notes of a Design Narcissist.

Now more than ever, electronics industry professionals—especially our newly graduated PCB design engineers—are learning the impor-

tance of communications and the need to understand the people, processes, materials, and equipment used in PCB engineering and manufacturing. This positive traction has been gained in part because of PCB design awareness programs such as the CID and CID+ curricula offered by the IPC.

However, not all PCB designers have taken the opportunity to be exposed to the workflows and capabilities of their stakeholder counterparts through such programs as these. Design data continues to show up on the doorsteps of production fabricators and EMS providers with inaccurate or missing elements of design for manufacturing (DFM) data. An afternoon spent in the office of a CAM engineer or an EMS provider's program management lead will convince any observer that many PCB designers are still basically going it alone. They simply do not associate their design practice with the requirements of the other stakeholders in the design and manufacturing cycle.

I believe there exists a dark place—a vacuum—within which a few PCB designers with narcissistic tendencies consciously choose to operate.

Yes, their boards are routed. But their manufacturing disconnects are, in essence, "unterminated leads" which were never conceptually connected to the workflow requirements of the PCB fab, test, and assembly team counterparts in the first place.

As the battle of good vs. evil plays out in Hollywood blockbusters, we may have to concede that along with all the good PCB designers, there may be a contingent of dishonorable counterparts. This is the designer who refuses to recognize the needs of others. I believe there exists a dark place—a vacuum—within which a few PCB designers with narcissistic tendencies consciously choose to operate.

Within this vacuum, requests for improvement from others are never heard. By choosing to work in such a vacuum, a headstrong, vain, stubborn, downskilled PCB designer could feel empowered to create PCB layouts solely to claim their own glory without being bothered by supplier feedback or other project stakeholder constraints. These PCB designers avoid teamwork but still feel comfortable taking credit for the work of the team.

Behold the design narcissist! It sounds like a great title for a Jerry Bruckheimer movie, doesn't it?

Narcissism is a human condition that affects 100% of the population to one degree or another. One statistic I read pegs the more serious condition of "Narcissistic personality disorder" to between 2% and 16% of the clinical population.

The last time I checked, PCB designers are still human, so it makes sense that during our PCB engineering careers there is a chance we could cross paths with a full-blown PCB design narcissist. But how do you recognize one and what can you do to improve such a situation?

Spotting the Potential Design Narcissist

We should all be on the alert for design narcissists. One may join your design team or be your company's next new hire. As members of an engineering community, we should be aware of the traits of the design narcissist so we can respond with an appropriate action, up to an all-out stakeholder intervention. Here's what to look for:

- Avoids checking in with fellow stakeholders, preferring to work in a vacuum. Lacks understanding and empathy for fellow stakeholders working on the same project.

- Avoids the design review cycle and even cheats the design rule check settings on their own layout tool at the expense of downstream stakeholders.
- Too self-absorbed to learn or upskill. Severely lacks both spoken and written communication skills required to understand or express the unifying language of industry standards.
- When a challenge arises, they are always the victim.
- Quick to require a corrective action report (CAR) of a supplier but would blow a fuse if one was ever issued by a supplier in return.

Dealing With the Potential Design Narcissist

If you have ever helped your daughter dump an abusive mate or couldn't figure out why the gal on your Tinder date couldn't stop talking about herself, you may have Googled "narcissism" to find out more. Because this is a PCB design and design engineering magazine, I have listed a few handy countermeasures to keep the design narcissist at bay:

- Educate yourself. Make sure you are up to date on all the PCB industry design and manufacturing standards so that you are in a position to help and stand your ground professionally.
- Create boundaries. Establishing and documenting capabilities, processes and workflows create useful peripheries for design and manufacturing teams to operate within.
- Speak up for yourself. Use the documentation in the previous point to be clear and concise. When a project begins, make sure that all project stakeholders are represented and are given an opportunity to express their requirement for success.
- Keep your distance but make certain to highlight the respect your position merits on the project team.
- Stick to the rules. A PCB design narcissist will try to bend the rules and push for favors which may rob success from another team stakeholder. As a project team, do not cave to these demands or you will feed the monster within.

If you're worried about being a narcissist or design narcissist, as the adage goes, you probably aren't one.

If you're worried about being a narcissist or design narcissist, as the adage goes, you probably aren't one. However, if someone you know is exhibiting disregard for commonly available manufacturing capabilities, with tendencies to design in a vacuum and avoid contact with fellow stakeholders, you may be in the presence of one. One sure sign: The designer utters the words "I alone can fix it!"

We must recognize the folly of designing or manufacturing alone in a vacuum. We must watch out for each other, locally and globally. We must push back against any who would look out only for their own success within a PCB project, a company, an industry, a nation, or a world. There is safety in numbers. We must check in with each other often. And keep reading *Design007 Magazine,* and not just my column. **DESIGN007**

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

The Benefits of a Parts Library

Connect the Dots

by **Matt Stevenson**, SUNSTONE CIRCUITS

To be effective at PCB design and layout, individuals need to become proficient with the different tools of the trade. Parts libraries are among the most important.

PCB design and prototyping is a critical component of electronic product development. Being faster to market has always been a competitive advantage and a focus for electronics manufacturers. With persistent marketplace uncertainty and supply chain disruption creating delays, in-house PCB design offers a way to accelerate electronic product development projects.

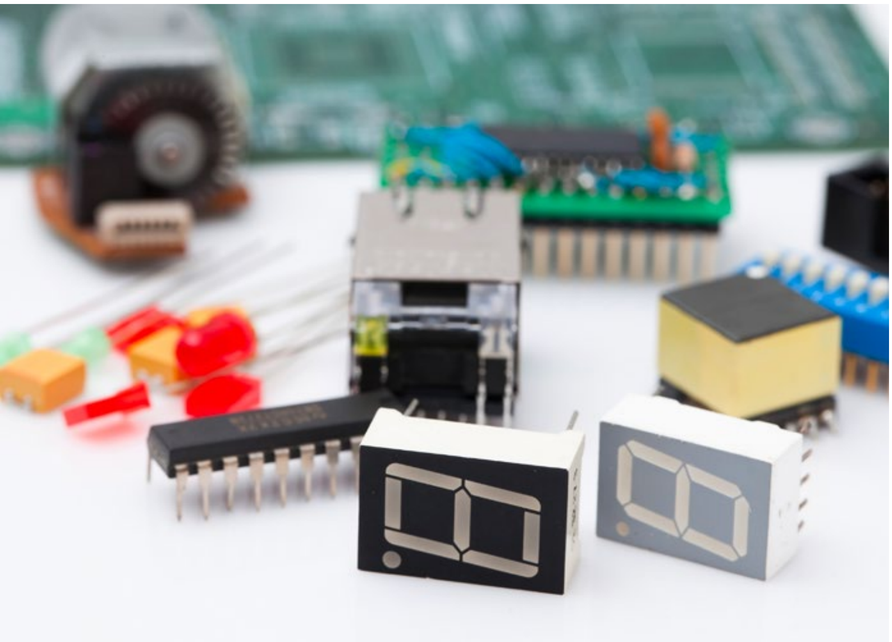
Not every product team is fortunate enough to have an experienced PCB designer on hand, so the task often falls to someone with limited experience who (hopefully) enjoys the process.

The first step in the design process is developing the electrical schematic and the accompanying bill of materials (BOM) needed to create the physical design. A well-planned schematic links to the layout tool, so the parts in the BOM are matched to what is available in a digital parts library. If all necessary parts are available, this can make the layout of the PCB much more efficient.

Parts can be hard to find sometimes. While the datasheet usually provides enough to locate the right part in a library, those datasheets can be very complex and challenging for less experienced designers or designers on small teams. Each part has numerous identifiers, including pin functional names, pin numbers, manufacturer part number, and tolerances. Even with so much data for each part, there can be information gaps. We are disappointed (but not surprised) to see discrepancies between the component footprints and the datasheet. Sometimes the symbol on the datasheet does not specify part orientation.

The PCB industry is now moving so rapidly that many parts are becoming obsolete and consolidated at a rate that makes it extremely difficult to keep libraries updated on a timely basis. Designers can waste hours on a fruitless search.

This often creates the need for designers to create parts themselves, parts that may already



exist outside their purview. This is one of the more common complaints that we hear from design engineers. Creating part footprints and symbols can be arduous and time-consuming using the CAD tool. Doing so also increases the number of opportunities for error.

Ideally, it takes a few hours to design a customer part to perform as needed. But if designers must deal with funky shapes, a unique size for a desired functionality, or an inventive footprint, the process can eat up a lot more time. All that manual configuration and design also increases the risk of error. It is hard to perfectly translate every design element from the datasheet. As the designer makes more manual entries, missing something that will create problems becomes much more likely.

Some teams are large and have enough resources devoted to part creation or update, as well as quality control and cataloging. Over time, this method results in a personal CAD library that can eventually reduce the number of parts needing creation. Another option is farming parts creation out to third party vendors.

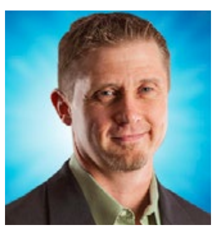
Both methods have one thing in common: they add cost to the product development. Is there a better way to manage parts for a small team or a single designer? As is often the case with PCB design, the answer is a resounding… probably.

Dealing with less-than-adequate parts libraries is a constant pain point for designers. For a small or overburdened design team, it makes sense to use the most up-to-date, comprehensive parts libraries available. There are several third-party library options available where the search for and download of parts is usually free. The two main players in this space are SnapEDA and Ultra Librarian. Both support a fairly wide range of CAD tool formats.

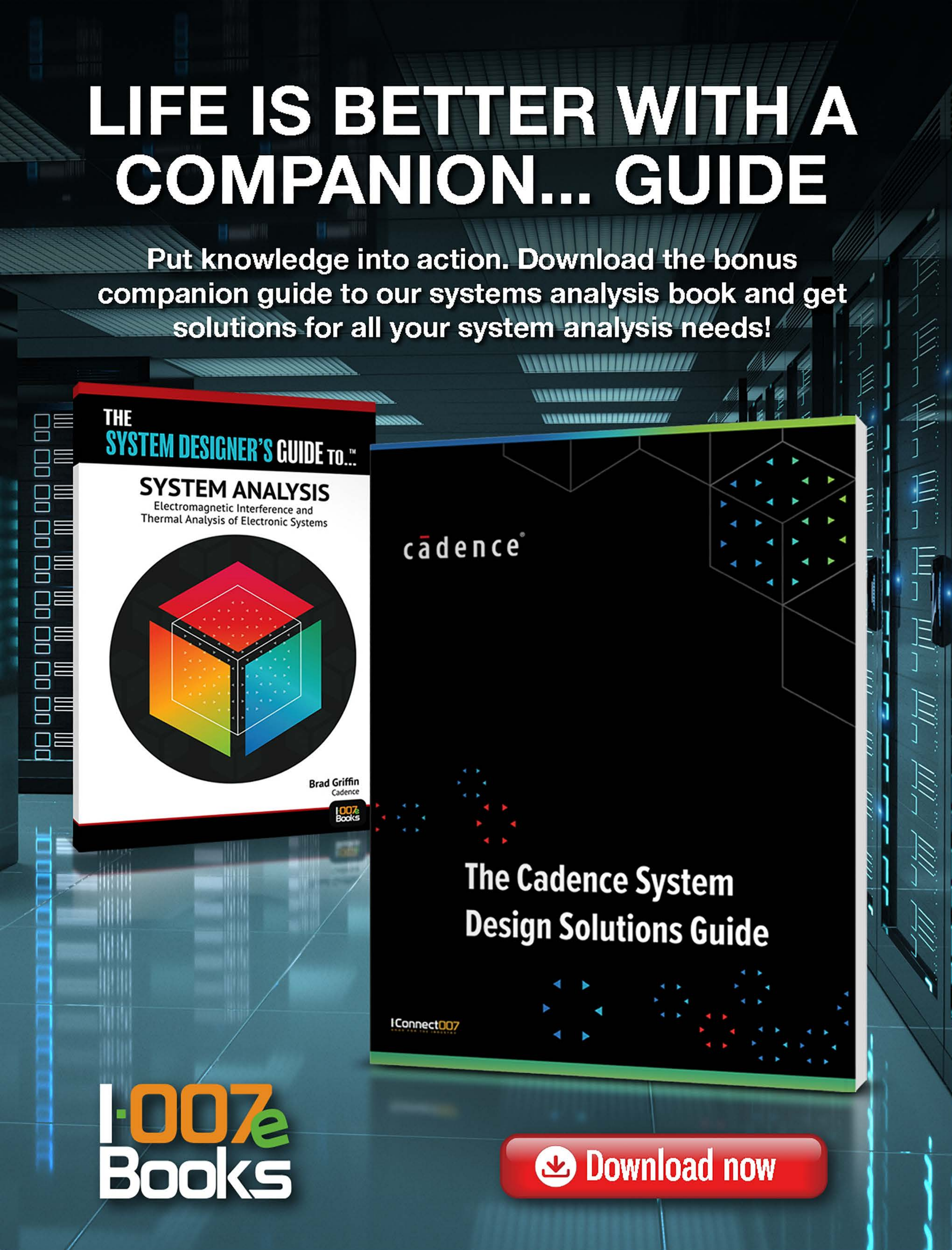
There are several CAD tools available that can integrate with one of these services. This makes the searching and saving of parts an intuitive part of the CAD tool. Designers should look for free-to-use CAD tools that integrate with their chosen parts library. Integrating the CAD tool with the parts library can put millions of parts at the designer's fingertips and help eliminate the need for building custom parts.

No matter how a design team is structured, there will be opportunities to bolster and streamline the process by which PCB part libraries are accessed and used. Even with parts library integration, the need for custom parts can never be eliminated. It makes sense to check all new or updated parts against the datasheet—be they created in-house or by a third-party vendor. Things like hole and pad spacing, as well as size, are very critical elements of the part footprint. Even a small error can mean the difference between the part fitting or not.

Always try to work smarter, not harder, and keep in mind that technology can be a very useful tool when it comes to parts libraries. **DESIGN007**



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


LIFE IS BETTER WITH A COMPANION... GUIDE
Put knowledge into action. Download the bonus companion guide to our systems analysis book and get solutions for all your system analysis needs!
THE SYSTEM DESIGNER'S GUIDE TO...™
SYSTEM ANALYSIS
Electromagnetic Interference and Thermal Analysis of Electronic Systems
Brad Griffin
Cadence
I-007e Books
cadence
The Cadence System Design Solutions Guide
IConnect007
I-007e Books
Download now


Nick Barbin

Designing in a Vacuum

Q&A: Nick Barbin

Feature Interview by Andy Shaughnessy
I-CONNECT007

Optimum Design Associates President Nick Barbin has worn a lot of hats in this industry, from PCB designer to EMS company owner. We asked Nick to share his thoughts on what it's like to design in a vacuum, but some strategies for escaping the vacuum.

Andy Shaughnessy: How often do you design a board without knowing who is going to fabricate and/or assemble it?

Nick Barbin: As a PCB design service provider, we are generally not always privy to this information unless we ask, and at Optimum, we do ask. It's not so much that we care about the "who" aspect; it's more about being in alignment with the specific manufacturer's capabilities to ensure we are providing a design that meets our customer's objectives for cost and reliability. This is especially important in the case of fabrication shops where capabilities tend to vary dramatically and can result in the shop requesting to modify the artwork to be able to meet the specifications called out.

Shaughnessy: Why do so many designers not know who is going to manufacture their boards?

Barbin: In the case of fabrication shops, once an OEM has qualified multiple vendors to build their boards, purchasing departments will consider that part number a commodity item, since it is generally one of the highest-cost items on any given bill of materials (BOM). In some cases, the OEM allows the commod-

ity manager (CM) to choose the fabrication vendor. Because of these dynamics, it is very difficult for designers to keep track of specific vendors. Designers should have a pretty good idea of who their CM partner is, as most OEMs have one strategic vendor they rely on. Global time differences and language barriers may also contribute to designers not knowing who's manufacturing their boards.

Shaughnessy: What other missing information or data do you find yourself having to "design around?"

Barbin: We believe designing around missing data is a mistake with unknown consequences. If there are missing datasheets or constraints that are truly missing, all efforts must be made to acquire the data or receive a satisfactory sign-off before moving forward. I cannot think of one instance where anything else would be acceptable.

We believe designing around missing data is a mistake with unknown consequences.

Shaughnessy: When you find yourself designing in a vacuum, what tricks have you devised as workarounds to get the job done?

Barbin: Whether we are designing in a vacuum, at Optimum, we have documented our DFM (design for manufacturing) guidelines in our Designer Handbook so each of our designers have best DFM practices at their fingertips. Additionally, each designer has a checklist of over 90 items, many of which pertain to DFM, that must be checked off as completed or considered for each design that is delivered. This is part of our ISO 9001-2015 quality control system. Lastly, each design must pass through a DFM check using the Valor NPI software from Siemens. It is Optimum's objective to ensure each of our designers deliver data for their respective designs to the downstream manufacturers that is both comprehensive and manufacturable to each of our customer requirements.

Shaughnessy: What are some of the hidden costs, in addition to monetary, of designing in a vacuum?

Barbin: Delayed time to market certainly comes to mind. The minute a released manufacturing package cannot be manufactured as-is and needs to be modified, the project is immediately put on hold waiting for approvals or answers. This can delay the development cycle by days and weeks as it progresses through many email mailboxes.

Shaughnessy: What advice would you give designers who are trying to get out of the vacuum?

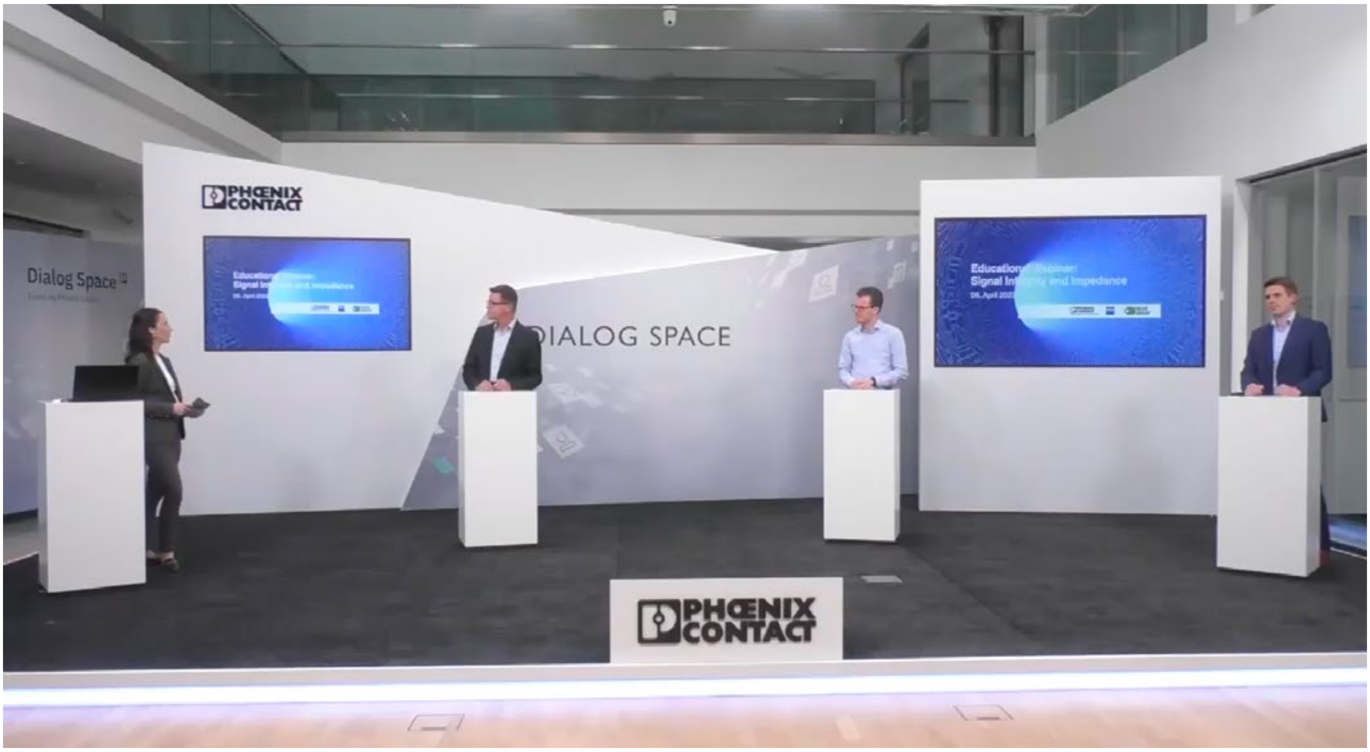
Barbin: We hear a lot about designers needing to "left shift" more into the electrical disciplines such as signal and power integrity, all of which is very true. However, I would advise designers to not necessarily shift left or right, but to stretch more and more in all directions. The designer's role is expanding and becoming more valuable in the product development process, and now is the time to become a more active member of the team vs. a passive participant.

Shaughnessy: I appreciate your insight, as always. Thanks, Nick.

Barbin: Thank you, Andy. DESIGN007

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

A Textbook Look: Signal Integrity and Impedance



Article by Pete Starkey
I-CONNECT007

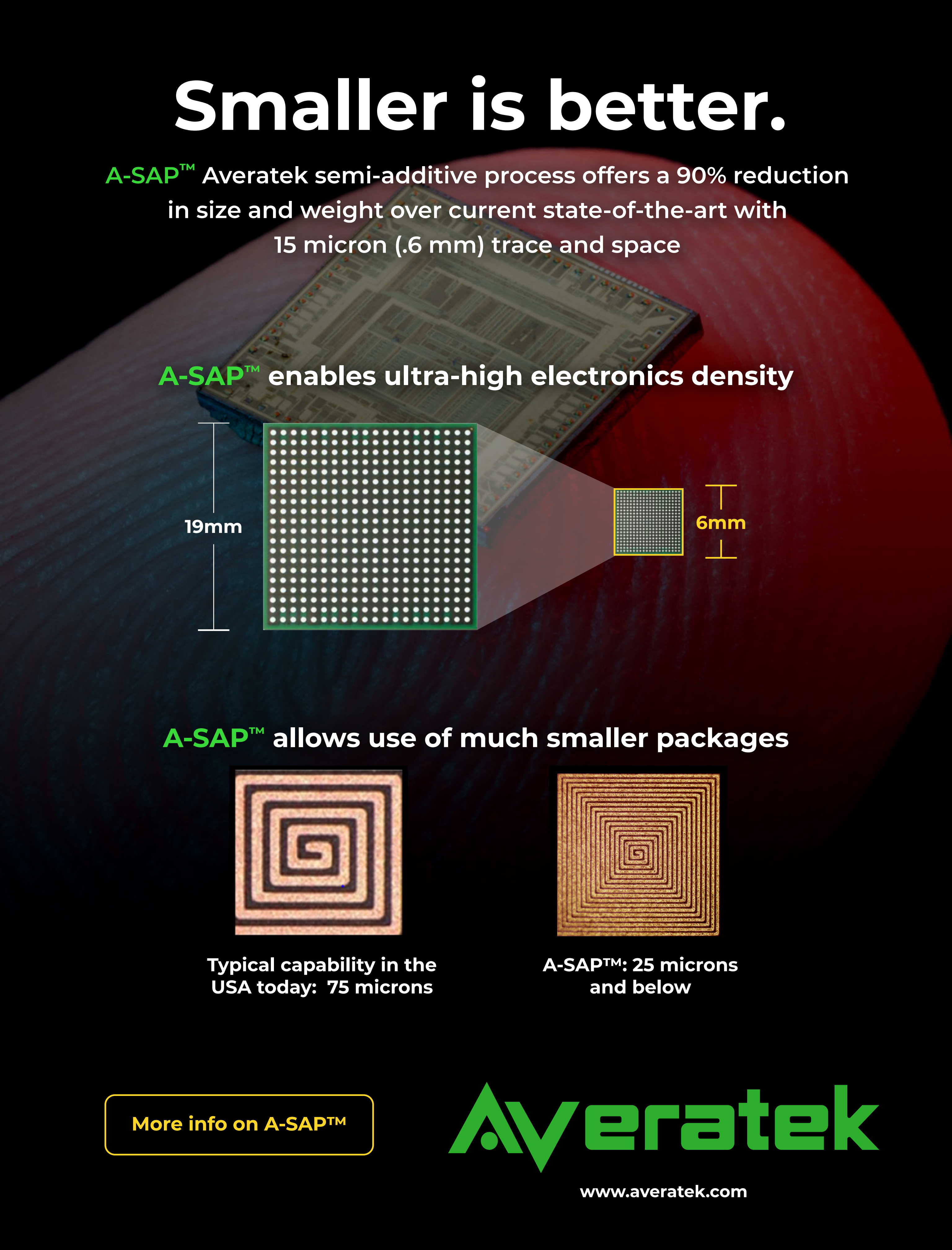
Believing that I knew a bit about signal integrity and controlled impedance, I was pleased to take the opportunity to connect with an educational webinar that I hoped would extend my knowledge. In the event I was surprised at how little I actually knew, and the webinar was an excellent learning opportunity.

Jointly organised by NCAB Group, Polar Instruments and Phoenix Contact, it brought together three expert speakers who shared their knowledge on the theoretical basis of signal integrity, the customer technical and engineering support provided by a global PCB supplier, and the design and optimization of high-speed data connectors.

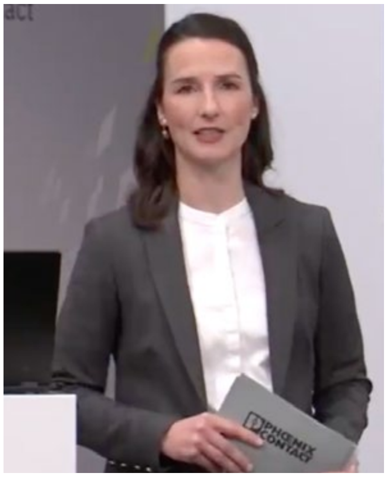
The webinar was introduced and expertly moderated by Anna Brockman, team leader campaigns and media management at Phoenix Contact in Germany.

Hermann Reischer, managing director of Polar Instruments in Austria, gave the clearest first-principles explanation of signal integrity and controlled impedance that I have experienced in many years. He made the topic intelligible and understandable by breaking it into its most basic elements and explaining each in plain language with meaningful illustrations.

Against a background of high-speed requirements such as 77 GHz analogue frequencies in automotive radar applications, and ever-rising


Smaller is better.
A-SAP™ Averatek semi-additive process offers a 90% reduction in size and weight over current state-of-the-art with 15 micron (.6 mm) trace and space
A-SAP™ enables ultra-high electronics density
19mm
6mm
A-SAP™ allows use of much smaller packages
Typical capability in the USA today: 75 microns
A-SAP™: 25 microns and below
More info on A-SAP™
Averatek
www.averatek.com

digital data rates with pico-second rise times, the printed circuit board ceased to be a passive interconnecting substrate and became a complex component in its own right. A PCB trace behaved like a transmission line when its physical length was of the same order as the electrical wavelength of the transmitted signal.



Anna Brockman

He examined the properties of a transmission line: in addition to its DC resistance it experienced a propagation delay depending on the dielectric characteristics of the material surrounding it, which also contributed to losses as well as those attributed to copper roughness and line length. The transmission line had a characteristic impedance determined by trace width, dielectric spacing, and relative permittivity. All of these parameters needed to be considered in order to ensure signal integrity, which became increasingly difficult to maintain as frequency increased. PCB material selection and stack-up, trace width, dielectric separation, design and layout, and component placement were all critical.

Reischer explained how a transmission line could be considered in terms of a series of inductors and capacitors and described how a signal was propagated along it, charging each successive capacitor element. The goal was to achieve constant impedance across the entire length of the line, with a uniform voltage and current wave-front in order to ensure good signal integrity. The propagation speed of an electromagnetic wave was a function of the speed of light and the permittivity of the dielectric, which for air or vacuum was 1.0. In typical PCB materials with relative permittivity around 4.0, the propagation speed was reduced from the speed of light to about half that value.

The "critical line length," beyond which a conductor would be considered a transmission line, was related to the rise time of the signal. Early semiconductor devices had rise times of around 5 nanoseconds, giving a bandwidth of 70 MHz and a critical length of 36 cm. Current gallium arsenide devices had rise times of 0.3 nanoseconds, giving bandwidth of 1.166 GHz and a critical length of 2 cm.

He clarified the differences in propagation characteristics between low-speed and high-speed signals, and he explained the ways in which electromagnetic waves were reflected at impedance discontinuities. Impedance matching offered a solution, and he gave an example of a typical transmission system where the impedances of the source, the transmission line, and the termination were matched at 50 ohms. This was generic in a radio frequency environment; 75 ohms was typical in video applications, 90 ohms in USB transmission, and 100 ohms for Ethernet.

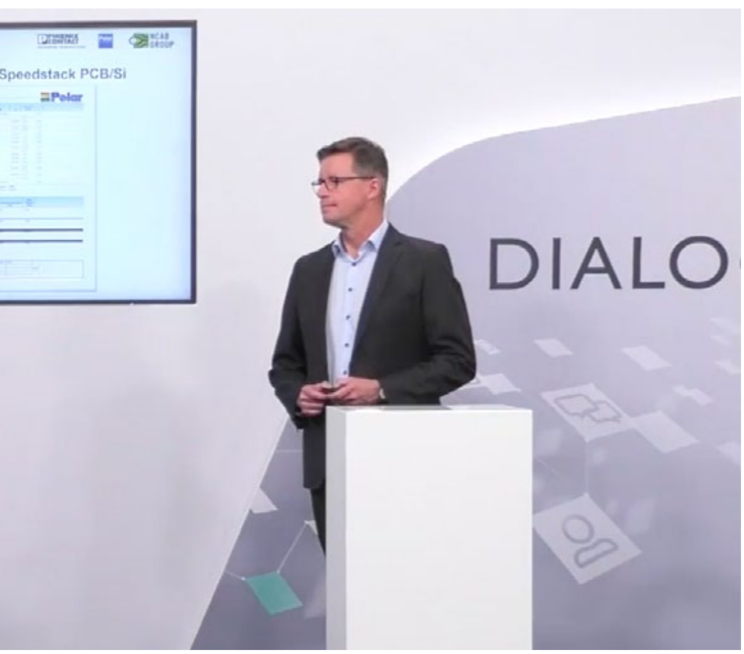
Reischer commented that his system example illustrated the essential interaction between the three companies contributing to the webinar: Phoenix Contact supplying interconnects, NCAB Group supplying printed circuit boards,

and Polar Instruments supplying simulation tools for calculating impedance.

So how to make sure that the transmission path had the correct impedance? He took a familiar example of a coaxial cable, whose impedance was determined by its geometry and the permittivity of the insulating material and demonstrated how the principles could be adapted to the planar structure of a PCB in the form of an offset stripline. The geometry and the permittivity of the material determined the impedance, and he showed a range of examples of "single-ended" structures. In contrast were "differential" structures, widely used in digital transmission systems, usually consisting of two traces running in parallel and coupled with a capacitor. The advantages of differential signalling included improved noise immunity, lower voltage requirements, and higher data rates.

Hermann Reischer

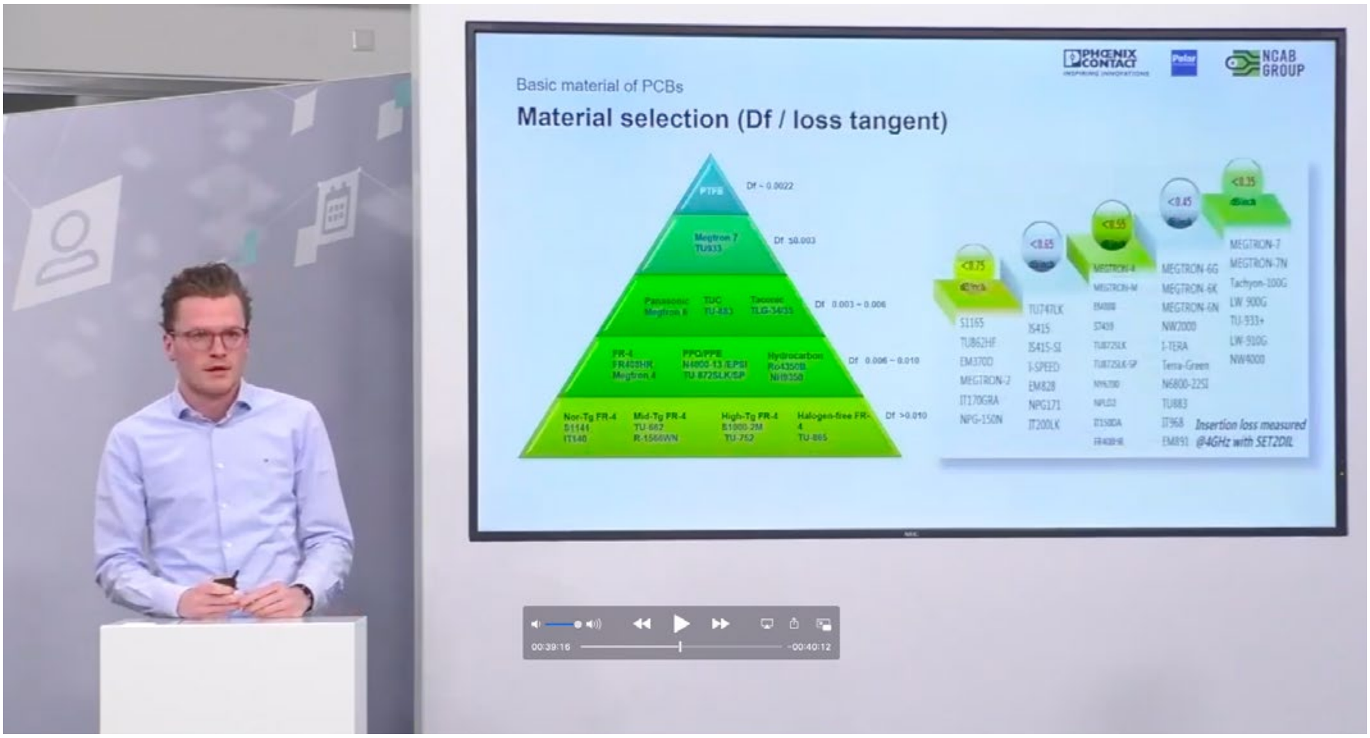
Reischer listed the factors influencing impedance. Trace width was the most significant and was simple to modify. Substrate height, relative permittivity, and copper thickness were determined by material selection, and etch-back was process-dependent. Solder mask thickness could have a significant effect on the properties of edge-coupled microstrip designs, as could the influence of local resin-rich areas if similar features were embedded within the structure of the PCB. He discussed the practical aspects of resin distribution and thickness control during multilayer pressing.



Vias gave some interesting effects; they were generally quite small compared with the wavelength of the signal, so they could normally be ignored except at extremely high frequencies. But on thick boards, as via stub-length approached a quarter of the wavelength of the signal, they could cause undesirable resonance. Back-drilling was a technique used to mitigate these effects.

Future challenges included tighter impedance tolerance specifications, lower dielectric thicknesses and narrower trace widths. Frequencies beyond 5 GHz resulted in "lossy" transmission lines; dielectric losses could be reduced by using new base materials and smoother copper could reduce skin effects, both at increased material cost. From a design point of view, Reischer recommended keeping traces as short as possible and ensuring uninterrupted signal return paths. For various reasons it was preferred to place critical lines on inner layers. And he emphasised the importance of consulting with the PCB vendor on stack-up design and material selection before commencing the layout.

Reischer's presentation delivered a comprehensive theoretical background to the concepts of impedance control and signal integrity. His closing comments about consulting with the PCB vendor before commencing the layout provided a perfect opening for Michiel Op den Camp, engineering manager for NCAB Group Benelux, to give an insight into the technical



Michiel Op den Camp

support that a major supplier like NCAB can offer to its customers during the design, quoting and production stages of a circuit board. In this instance the particular focus was on signal integrity.

Reflecting upon the emergence of demand for controlled impedance product, he observed that during the 1990s only 10-20% of designs had that requirement, whereas today's transmission rates increased the percentage to the 80-90% region and upcoming 5G networks operating at 28 GHz would require ever more critical PCB properties. His schematic balancing PCB design challenges against design tools and process over the last five decades indicated a rapidly growing area of risk as complexity increased.

What could be done to minimise this risk and to avoid over-specifying or under-specifying the design? The answer was to consult with the PCB supplier's applications engineering specialists as early as possible in the project and allow them to calculate and recommend optimum materials and stack-ups to achieve cost-effective performance and reliability and shortest time-to-market. He stated that NCAB set standards stricter than IPC, both in technical requirements and in quality assurance. And in the case of special requirements, they offered application-specific consultations with field engineering experts. Op den Camp gave examples of designs with data greater than 2.3 Gbps and impedance tolerances of 5%.

Another significant point he made was that many of the designs offered for volume production were based on the capabilities of prototype suppliers whose yields might not be economic in a production environment. Again, consultation at an early stage could improve the suitability of the design for volume manufacture and offer yield improvements and cost savings.

He gave examples of products designed for operation at high analog frequencies and high-speed digital applications and discussed hybrid constructions and low-loss materials.

His graphs of relative permittivity versus frequency indicated that for FR-4 materials, dielectric constant was "not really constant" as frequency exceeded 2 GHz, so there was a need for materials with better and more consistent dielectric properties. His triangular representation of low-loss materials had FR-4 grades at the base and PTFE at the apex, with many proprietary products occupying the intermediate levels. Not just price, but availability, could influence the choice and again he stressed the importance of consulting the PCB supplier before specifying a particular proprietary material.

Turning to the characteristics of woven-glass reinforced materials, he illustrated how glass weave and resin content could cause local variations in dielectric constant and hence the impedance uniformity of embedded conductors.

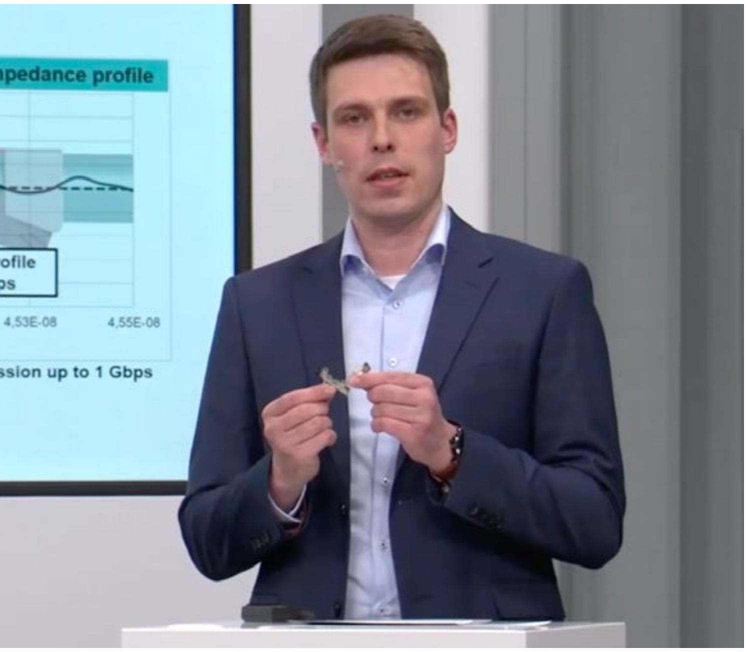
He discussed details of component placement and conductor routing, and gave several examples of good and bad practice in optimising signal integrity. It was clear that Op den Camp himself had high-level expertise as a designer in addition to his knowledge of materials and fabrication technology. He echoed Hermann Reischer's comments on the benefits of back-drilling via stubs, and also recommended the removal of non-functional pads on high-speed vias.

NCAB had a whole series of design rules and guidelines available for download from their website, and their applications engineers were always on hand to answer specific questions.

Having learned the theoretical background of signal integrity and controlled impedance, complemented by a review of the practical aspects of the design and manufacture of printed circuit boards for high-speed applications, it remained to explore the technology of ensuring the integrity of the signal as it passed through interconnecting components. The final presentation came from a Phoenix Contact specialist in the design and optimisation of data connectors, Sebastian Stamm.

He began by making it clear that although signal integrity was a very important consideration, it was one of many product requirements. Cost and installation space were factors that reduced design freedom. Therefore the target was not to achieve signal integrity at any price, but to produce application-oriented connectors with high signal integrity.

Sebastian Stamm



He pointed out examples of different applications with different requirements on a graphic showing transmission length versus transmission speed.

Each connector had its own impedance profile, which was location-dependent. Influencing factors included transmission frequency, geometry and material, and the arrangement of components. Whereas PCB impedances could generally be considered as two-dimensional characteristics, two dimensions were insufficient to describe the characteristics of a connector. Stamm showed the impedance profile of an industrial Category 5 connector as measured with a time-domain reflectometer and explained the deviations from the nominal 100 ohm value.

Discussing cause and effect mechanisms, he described how the impedance profile of a connector depended on the transmission frequency because relative permittivity was frequency-dependent and shorter discontinuities became visible with shorter pulse-rise times, and these caused larger deviations from the nominal impedance. He illustrated the effect graphically.

For standard board-to-board connector designs, the exact applications and transmission frequencies were not always known, but the target was to achieve the same impedance profiles within a product family so that the customer had the advantage of a scalable device design without influencing signal integrity.

In the case of wire-to-board connectors, where transmission frequencies and requirements such as return loss were specified, the target was to optimise impedance profiles until the transmission requirements were satisfied. He took the example of single-pair Ethernet connectors.

The impedance profile of a connector was influenced by the geometry of the signal lines and its effect on the capacitance and inductance per unit length, and small clearances between signal conductors or from ground planes could lead to lower impedance values. Sometimes, the impedance value of a connector was too low as a consequence of the small space available for installation. Stamm explained how impedances had been optimised by targeted adjustment of geometry, again with the objective of achieving a consistent impedance profile within a product family.

He went on to discuss various effects of material and component arrangement before finally mentioning an investigation during a development project of the effect of differences in contact lengths on run time and propagation delay. In fact, it was observed that the delay skew was 80 times less than the propagation delay, and negligible in practical terms.

After the formal presentations, Anna Brockman moderated an open discussion, inviting questions from the audience.

From a personal point of view, I found this webinar extremely informative and well-balanced in content. I learned a lot. It filled in many gaps in my knowledge and brought the whole topic together in a logical and effective configuration. My thanks to the organisers for a first-class event. DESIGN007

Click here to watch this educational presentation by NCAB Group, Polar Instruments and Phoenix Contact.

Green Ambition for the Electronics Manufacturing Industry

Design Circuit

by Patrick Crawford, IPC

Regulations, policies, and plans are being adopted around the world which require manufacturing companies of all kinds to implement sustainable engineering practices. These efforts are numerous and are intended to protect human and environmental health as well as the integrity of supply chains that may rely on non-renewable resources. While these regulations largely focus on limiting the release of chemicals—either during production or end-use—that contribute to anthropogenic climate change, most also address the potential hazards to both workers and consumers posed by the process chemicals and materials used to manufacture a product. The electronics manufacturing industry is no different, of course, and every electronics engineer is certainly at least aware of the Restriction of Hazardous Substances Directive (RoHS).

One of the most watched of these novel plans is coming from the European Union (EU) in the form of the European Green Deal. This sprawling framework is devised to help EU member states meet emissions targets—namely, the reduction of greenhouse gas emissions by at least 55% by 2030, as compared to levels measured in 1990—and achieve other circularity benchmarks regarding the recyclability, repairability, and reusability of products. This framework was bolstered by the European Climate Law, passed in July 2021, that legally binds EU member states to these benchmarks. To accelerate the changes mandated by the European Green Deal, the European Commission drafted the Circular Economy Action Plan, which details how governmental and private entities can minimize their environmental footprint while also minimizing the impacts on businesses that these progressive steps may incur. Ultimately, the European Commission writes that the Circular Economy Action Plan will create a "regenerative growth model that

gives back to the planet more than it takes, advance[s] towards keeping its resource consumption within planetary boundaries, and therefore strive[s] to reduce its consumption footprint and double its circular material use rate in the coming decade."[1]

Like most industries, the electronics manufacturing industry utilizes processes that inherently contradict the goals of the Circular Economy Action Plan.

Like most industries, the electronics manufacturing industry utilizes processes that inherently contradict the goals of the Circular Economy Action Plan. The fabrication and assembly of PCB devices is a power-consumptive process, and it cannot yet be guaranteed that this power comes from a green source (although efforts to reduce power use are ubiquitous and laudable on the part of tool vendors and factories everywhere). The shipment of raw materials, electronic sub-assemblies, and components through a supply chain still (usually) relies on carbon-positive logistical networks such as trucks, especially in the "last mile" between an OEM and their customers or vendors. There are other examples like these but suffice it to say that this enterprise-level waste and pollution generation is not a topic considered when discussing sustainable products.

The Circular Economy Action Plan provides a basic blueprint for "circular electronics," but the design of truly sustainable electronics requires concerted effort and focus to develop specific technologies, methodologies, and policies. Indeed, forward-thinking companies have since begun to nucleate around what is now commonly referred to as "eco-design for electronics" or just eco-design. Generally, eco-design is the design of products intended to have the greatest positive impact on society and a minimum impact on the environment and economy. In the electronics manufacturing industry, this can refer to the design of the whole product, especially for smaller consumer products such as ICT devices, but more usually refers to the electronics components themselves—the PCB assemblies, wire-harnesses, components, housings, etc.

Some of these forward-thinking companies have banded together to share expertise and resources to achieve shared eco-design goals. The International Electronics Manufacturing Initiative (iNEMI) is a not-for-profit research and development consortium consisting of leading electronics manufacturers, suppliers, associations, government agencies, and universities.[2] It leverages the collective experience of its member network to create and socialize roadmaps for future technology requirements. It should then be no surprise that one of iNEMI's key areas of research is sustainable electronics. IPC is a member of the iNEMI Eco-design Team and works with them to provide webinars detailing specific cases of eco-design by companies including Nokia, Samsung, Schneider Electric, Fairphone, ADVA Optical, Fronius, IBM, Framework, and Barco. These webinars are available for free on YouTube (via iNEMI) and are valuable resources for designers who would like to know more about specific examples of eco-design in electronics products. After all, one of the key takeaways from these webinars was that eco-design cannot happen in a vacuum, and that it is essential to share knowledge regarding specific advances in technology where possible (and not confidential, of course).

This webinar series is a wonderful resource for teams who are doing product design, including electrical and PCB designers. And while a few of the strategies presented therein may affect the whole product, not only the electronics—human-centric design, general modularity for reuse, use of recycled material in product packaging, the intelligent utilization of tools or ISO 14040 to conduct LCA—there are those that are essentially targeted toward the PCB design itself. These strategies mostly focus around three areas: design for repairability, design for recyclability, and design for reuse. Those of you who have used the IPC-2231 DFX Guidelines document called Design For eXcellence may think that at least one of those is familiar.

For the unacquainted, the IPC-2231 DFX Guidelines document provides a framework to establish a design review process for the layout of printed board assemblies. This design review assesses the manufacturability attributes of printed boards, namely design for manufacturing, fabrication, assembly, testability, cost, reliability, environment, and reusability. Ah, there it is: design for reus(ability)—DFR. Yes, at least one IPC document does currently touch upon a review of eco-design principles in the layout of printed boards. However, as of IPC-2231A, these sections (design for environment and design for reusability) are not as robust as they should be in the face of the European Green Deal or any of its tenets of circularity (and likely impending regulations thereof, as well).

I don't write this to disparage our document, but rather to signal that IPC is aware that electronics manufacturing companies will soon be responsible to regulatory bodies around the world—not only the EU—as well as to the environment. We are actively reviewing the IPC-2231A to revise it within the next two years. As part of that revision, we are making a point to include (among other things) discussion of PCB designers, and their teams can begin to implement review processes for creating and assessing eco-design benchmarks for their designs. As the IPC-2231 is a guideline, these will not be normative requirements, but rather best practice suggestions from the 1-14 DFX Subcommittee and our friends from iNEMI, who will be joining us to help review the reusability and environment sections of the document (and add sections as necessary).

But we won't stop there. While a revision of the DFX Guidelines document is IPC's priority, we anticipate that we may be able to expand IPC eco-design activities to include the creation of a new eco-design for electronics guideline that provides a holistic coverage of circularity concepts and how they touch every step of the electronics manufacturing pipeline. We feel that this is an excellent opportunity to make a real difference in eco-design for circular electronics, but again, it all starts with the IPC-2231 DFX Guidelines document.

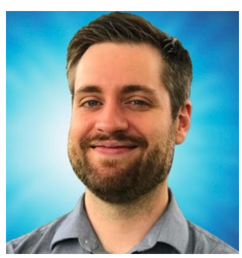
To that end, if you consider yourself to be an eco-design "expert," or know someone who is, please reach out to me. I would love to hear how you think IPC can help the electronics industry navigate the new circular economy.

The European Green Deal is likely just the beginning of an explosion of "greening" around the world. While IPC is determined to help electronics manufacturing companies become greener themselves, it will take continued input from industry, both through consortia like iNEMI and standards development activities through organizations like IPC, to produce meaningful change.

When it comes to confronting these new opportunities in sustainability and circularity, I think that we should embrace the spirit of the late, great Stan Lee and say, "Excelsior!"—or as we mortals would say, "Ever upward." DESIGN007

References

1. "Communication from the Commission to the European Parliament, the Council, the European Economic and Social Committee and the Committee of Regions," Eur-Lex.
2. inemi.org.



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

Designing in a Vacuum Q&A: Carl Schattke

Feature Interview by Andy Shaughnessy
I-CONNECT007

Not long ago, I caught up with Carl Schattke, CEO of PCB Product Development LLC and a longtime PCB designer, for his thoughts on "designing in a vacuum." As Carl points out, if you follow PCB design best practices, knowing the identity of your fabricator is not a "must-have." He also offers some communication tips for discovering the information you do need, including one old-fashioned technique—just asking for it.

Andy Shaughnessy: Carl, how often do you design a board without knowing who is going to fabricate and/or assemble it?

Carl Schattke: It's pretty often that the design will start and get completed, or nearly completed, without any board vendor involvement. This is far more common if it's a standard stackup and a routine process.

For sequential lamination, rigid-flex and flex boards, it's very rare to not involve the board vendor early on. Also, if there are multiple differential pair impedances on different layers, it's much more likely a request to the board vendor will be made at the start of placement before routing commences.

Shaughnessy: Why do you think so many designers have no idea who is going to manufacture their boards?

Schattke: It's nice to have this information, but it's not a must-have. By following general rules for volume manufacturing, you should be able to get the board built by the vast majority of UL-listed board shops. EEs and purchasing people are typically very busy and they don't have time to do tasks until they need to be done. The can that can be kicked down the road usually is.

It's up to the PCB designer to request a stackup with impedance values and HDI rules if the board is higher up on the high-tech scale. None of us want to waste time. If the board is a common low-risk build, the build and rework risk is low if you have incomplete data about who will build the board. When we start using exotic materials and sequential laminations with varied copper weights and high layer counts, it would be foolish not to involve the PCB vendor on the design start.

Dense designs require going right up to the edge of the rules without breaking them. To do that, you need to review the process and your expectations with the vendor to make sure they can build what you design. To do otherwise could put hundreds of hours of work at risk for rework, and no one wants that.

Carl Schattke

Shaughnessy: What other missing information or data do you find yourself having to "design around"?

Schattke: Where do we start with this question? I would say that the need to get something out the door ASAP drives many designs to start in a state of partial completion. The primary risk of starting PCB layout too early is that the optimal result may not be obtained. Common missing items include incomplete schematics, or incomplete sections of the schematics, and missing current information needed for calculating trace widths.

Other missing information includes voltage clearance rules from one class of nets to another, and missing nomenclature requirements like part numbers, assembly numbers, test marking, and any other special text. Board finish and copper weight are often not provided, and the designer will decide what is best then. The environment the product will be used in is very important for the board design, and this is often missing. Mechanical outline drawings are often incomplete and follow-up details have to be requested.

Most often missed are hole and connector locations, dimensions, board thickness, NPTH/PTH, screw head clearances, height keep-outs, pin 1 locations, connector numbering order. The quantity of the initial and lifetime build. This will influence the decision to have ICT pads, or place only ICT pads on one side of the board. Which pins can be swapped? Often, some simple schematic changes can greatly reduce board routing challenges.

Shaughnessy: When you find yourself designing in a vacuum, what tricks have you devised as workarounds to get the job done?

Schattke: If you are designing on this earth, you are not in a vacuum unless you let one be drawn upon you. You have to break out of your container with phone, email, or instant message, and talk to the stakeholders and find out what they want. Seldom are they so busy that they don't mind something being done wrong that they will have to live with later.

I find it really valuable to go over each page of the schematics with the electrical engineer. When I do this, I want to understand the circuit, what it does, what's critical about it (or not), what kind of current is involved, how much heat will certain parts need to dissipate, what sections need to be near or away from other sections, and whether the EE has a certain flow in mind for the placement.

Is termination drawn on the page it's used with or in the place it needs to be? Has the EE included all the decoupling caps? Ask questions; the only stupid question is the one that was not asked that could have saved an iteration, ECO or rework. To recap, interview the EE, give immediate feedback on missing information to the mechanical engineer and the EE, and pull in other professionals as needed to get the information that helps establish the scope of the project.

Shaughnessy: What are some of the hidden costs, in addition to monetary, of designing in a vacuum?

Schattke: If the requirements phase of the project is not done well and scope creep is allowed, that is going to be a hidden cost-adder in a captive design team structure, and a large adder if the work is outsourced and additional hours are required to meet the added scope. To save money and get a better design, the schematics should be fully completed with all the rules and board technology features defined. This is the "ready, aim, fire" method. Saving money on the design is not always the top goal; often it's getting the minimum viable board ready for the marketplace to get first-mover advantage. In this type of a design it's often "fire, aim, ready," and many shots are taken and the aim is improved as new information becomes available.

Shaughnessy: What advice would you give designers who are trying to get out of the vacuum?

Schattke: Get really good at asking the right questions. Seek to understand what will make things work, what will be potential risks, and what can be done to mitigate those risks. Ask what communication method (and what time) is best for the overworked team members you are supporting.

Asking at the right time and in the right way will lead to far more answers than the alternative. Respect the time of the other team members, but also remember that they save time by using you—but only if what you do is useful. No one on the team wants to go through redesign efforts. That being said, no EE or product manager wants to release a product they know they can make better or less risky by implementing a PCB design change. If the design team is so busy that it's hard to get information, take a best guess at what you think is right, research the data, or go to PCB and assembly vendors and get their feedback. Then ask your question with the possible solution in place for the design team. This is how you become more valuable and get better outcomes.

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

Schattke: I don't believe that any of us live in a vacuum. We all design and produce things of value for others. The better we do that, the better it is for everyone; less scrap is better for our environment. We owe it to ourselves and the organizations we work with to break out of any self-imposed vacuum we find ourselves in. We should actively seek out help from anyone who will help us build a better product.

The best ideas and products come to fruition from a rich soil that has lots of nutrients available. It's up to us to turn the soil and plant the seeds. We have to tend to the design garden until it's ready for market. Just as a gardener must water, keep pests away and pull weeds, we have to do the same as designers to get a completed design ready for manufacturing.

Shaughnessy: Thanks, Carl.

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

PCB007 Highlights

EIPC Technical Snapshot: High-end PCB Market Requirements and Technology Trends ▶

The European Institute for the PCB Community (EIPC) continues to provide an efficient platform for following and pursuing new developments. Its Technical Snapshot webinar has become established as a must-attend monthly event, and consistently delivers essential information of the highest caliber and relevance.

Improved Thermal Interface Materials for Cooling High-Power Electronics ▶

Heat has been a significant concern in electronics since the beginning of the electronics age when hot glowing vacuum tubes were first used to receive and transmit data bits. The transistor and integrated circuit effectively solved that basic problem but increases in integration resulted in increased concentration of heat, exacerbated by relentless increases in operating frequency.

Insulectro Gears up for More Business in Canada ▶

Insulectro, North America's largest distributor of materials for use in manufacture of printed circuit boards and printed electronics, has broadened its commitment to Canadian business. It has acquired more warehouse space in Scarborough, Ontario, significantly increasing its local presence and capabilities in Canada.

Testing Todd: Has Universal Fixture Testing Gone the Way of the Dodo? ▶

Although flying probe testers have become common place in today's manufacturing theatre, one must wonder if the fixture tester, specifically the universal grid or "pin in hole" fixture has any valuable use in the electrical test arena? The advancements in flying probe technology are undisputed with the new abilities to do many of the tests that benchtop testing historically required.

Measuring Multiple Lamination Reliability for Low-loss Materials ▶

Taiwan Union Technology Corporation (TUC) provides copper-clad laminates and dielectric resin composites used to manufacture PCBs. The enthalpy of these resin composites meets and exceeds customers' objectives and shows the deterioration of the resin's physical properties because of multiple lamination cycles.

The Plating Forum: How the Pandemic Impacted PCB Manufacturing ▶

Thanks to its classification as an essential business associated with national security, PCB manufacturing in the U.S. was exempt from pandemic shutdowns; it was not, however, immune to supply chain disruption. Raw materials shortages set the stage for higher prices. Companies that relied on just in time (Kanban) inventory management held back product, further burdening the supply chain.

Happy's Tech Talk #6: Looking at the Process of Repanelization ▶

I have spent many years in printed circuit fabrication, including nearly 20% of my career in Asia. One problem that concerns all fabricators is the issue of how many 'X-outs' are allowed per assembly sub-panel array. Here are a couple of solutions I have used and encountered in my travels.


Hmm, what is recommended **minimum distance for copper to board edge?**

Today's Preparations for Tomorrow's PCB Designs

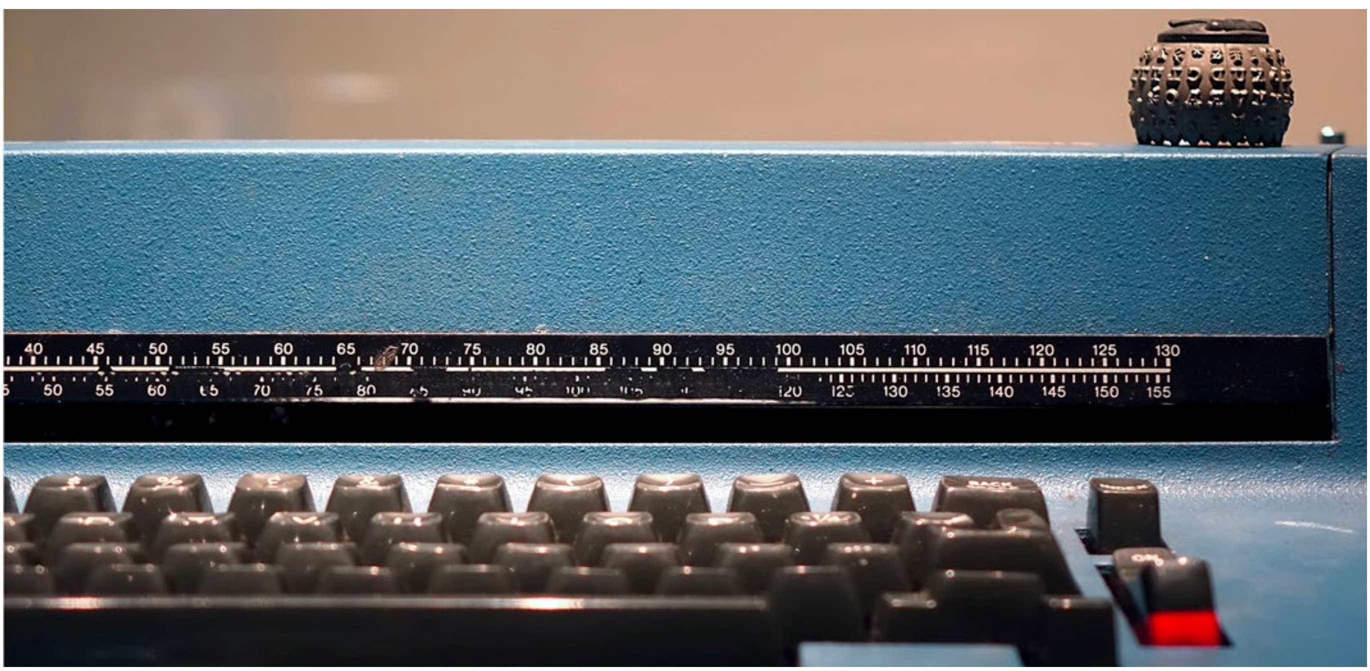
Tim's Takeaways

by Tim Haag, FIRST PAGE SAGE

For some strange reason that I still don't really understand, I abruptly decided to take a typing class in the eighth grade. This baffled my parents and teachers alike as everyone knew that I had absolutely no aptitude for any sort of literary or language skills. Not only was it a challenge for me to string a simple sentence together coherently, but I was (and still am) possibly the worst "spellar" in the history of writing. Honestly, and I don't think that I'm exaggerating much, I'm pretty sure I've seen a white flag pop out of the back of my computer as the spell checker admits defeat and surrenders in humiliation. It's not just my computer either. When I ask for help trying to sound out a word, Alexa will turn a deaf ear and my wife will politely ask if I have taken up a new foreign dialect as a hobby.

As you can see, it simply didn't make any sense for me to take a typing class, but I did it anyway. This was the era of the big typewriters that devoured paper by the ream and had zero editing capabilities, which seems barbaric by today's standards. But to tell the truth, I got pretty good at it and scored fairly high in the typing speed tests—as long as you ignored my "speling" mistakes. However, once I finished the class, any thoughts of typing disappeared from my head as I pursued music, sports, and girls—not necessarily in that order. Remember, this was still the eighth grade.

Who knew that eventually the computer generation would be upon us, and that I would start laying out circuit boards on various computer design systems? The CAD tools didn't necessarily require the ability to type, but the



old Unix networking tasks and the shell scripts I started writing certainly benefitted from my eighth grade typing class. Eventually I moved into management positions, which required writing reports and procedures, followed by a time of providing customer support for a PCB CAD company through emails and white papers. Now I am employed as a writer covering various technical subjects and clients. As I said, who knew? I can tell you, though, I am certainly thankful that I took that class.

My point is to prepare yourself for the career ahead of you. I was very fortunate that one random decision I made a very long time ago ended up being so helpful to my career, and by extension, to my life. However, if we were to entrust our entire future solely on the hopeful benefits of random choices, we would more than likely end up with a disaster on our hands. Therefore, it is important to make some informed choices when it comes to preparing for the future, so that we are ready for what the world of circuit board design will require of us. In that spirit, here are some ideas that can help.

Education

PCB designers are expected now to have a thorough background of engineering or technical training and education, and that should be your first focus. Beyond that, additional classes, seminars, and workshops will help to broaden your overall knowledge, while training in specific areas such as high-speed design are essential for most applications. It is also very helpful to obtain certification in different industrial standards.

Experience

There is no denying that industry "experience" is like the old expression: Which came first, the chicken or the egg? Potential employers are looking for experience, and you must be employed to get the experience they are looking for. However, there are always unique employment solutions waiting where you don't expect, such as surprise job openings, related positions that allow you to work into a design position, etc. Keep looking and knocking on doors. One of my first jobs in PCB layout came because I persisted in my contact with a design shop. When they finally needed additional entry level help, I was the first one they turned to.

Tool Savvy

The more experience you have with different design tools, the better. Yes, you should focus on just a few to refine your skills as a designer, but it still is a good idea to keep up with others. You never know when your company may decide to swap out a tool set, you need to look for another job, or an incredible new opportunity suddenly presents itself. Aside from those, there is another reason why it is beneficial to understand different tools. Each CAD vendor takes a slightly different approach to the structure of PCB design data. By familiarizing yourself with these different approaches, you will have a better understanding overall of how CAD processes work. This can be very beneficial when someone is looking for a designer with a more global perspective on CAD tools to help migrate to a new system, or looking to improve their own process and workflow.

Process

The more you know about the process of printed circuit board design, fabrication, and assembly, the better. As a young designer I had no idea why I had to follow some of the requests that engineers made of me, but as I came to understand manufacturing processes and DFM requirements, it all started to come into focus. This greater understanding gave me the ability to design for the process instead of correcting my work to bring it up to meet the requirements. But don't stop just there; keep on exploring new technologies and processes. Explore flex manufacturing if you don't already understand it, and learn about different board materials for RF, high-temperature, or other harsh environments. Our industry is constantly growing, and it is essential to grow with it.

Work Ethic

Oh, boy, here's the tough one. I'm just going to put these ideas out there, and if you think that I'm nuts on some of this, so be it. But I have found their relevance in my own career path, so hopefully they can be of help to you.

- **Be on-time and available:** Not only do you need to be at your workstation to do the job you've signed up to do, but you need to be available for meetings and consultations during your regular work hours. Many people don't realize how much this matters to managers who need a simple question answered or a task accomplished. To put it simply, if you aren't where you should be on a regular basis, your manager will soon turn to someone else to get the job done.
- **Be a team player:** More than likely you will be working with others throughout your career, and you will have to function in a team environment. Teamwork can be tough, especially in our industry where we often work on our own, and you may not always agree with each other. But it is essential to work collaboratively toward a common goal even if it isn't always in the direction you expected. Be prepared to stand up and make a compelling argument for what you believe, while at the same time prepared to make a reasonable compromise. Negotiation, communication, and mutual respect are all essential components of a successful team, and that general rule remains the same no matter what industry you are in.
- **Learn to take a blow:** Sometimes events just don't turn out the way we expected them to. We didn't get the job we wanted, somebody threw us under the bus at work, or we really messed up a layout that cost time, money, and embarrassment. Maybe we've even been laid off from a position that we really loved. I've been through all of these, and I know how tempting it is to want to hide from the world, hoping that it will all just magically go away. But don't do it. Pick up the pieces, make the corrections, and most importantly, believe in yourself again. Take control of the events surrounding you instead of yielding that control to the circumstances.
- **Produce beyond what is expected of you:** I realize that this is a very sensitive topic, but if you want to succeed, you must commit to what you are doing. Make sure your job gets done, volunteer for the tougher assignments, and don't overcharge your employers for what you are doing. The goal is to market yourself as the go-to person in your employer's eyes so they will see you as an asset to their team.
- **Protect yourself:** It can be very easy to over-commit while building your career. There are also unscrupulous employers out there who will abuse your honest efforts without anything given in return. To ensure that you can always do the highest quality of work, keep an eye out for signs of burnout and protect yourself, even if it means pursuing other opportunities.

As I look around at the work being done in our industry today, I am filled with anticipation of what is around the corner. Tomorrow really is going to be an amazing day. It is exciting to be a part of all of this, and I hope these ideas have helped in some small way to prepare the designers of tomorrow with the tools they will need to get there. See you all next time, and until then, keep on designing. DESIGN007


Tim Haag, 1973

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

MilAero007 Highlights

The Government Circuit: Robust U.S. Electronics Industry in Everyone's Interest ▶

I've been saying for months that decisions made in 2022 will be critical to the future of electronics manufacturing for years to come. After years of government policy neglect, we have unprecedented opportunities to make things better and position the industry for long-term success.

Sierra Space to Revolutionize Space Exploration With Siemens' Xcelerator ▶

Siemens Digital Industries Software announced that Sierra Space has implemented Siemens' Xcelerator portfolio of software and services as the foundation of its next-generation digital engineering program.

Zollner Elektronik AG—Nadcap Reaccreditation/MERIT Status Renewed ▶

In a total of five days of auditing, two German locations—Altenmarkt III and Untergschwandt—were thoroughly examined by an external auditor. In particular during the audit, the various manufacturing processes of electronics production were examined on-site, which must meet the stringent standards of the Nadcap program.

Prototron Circuits Installs Maskless Model 5600 LED Direct Imager ▶

Prototron Circuits of Tucson, Arizona, has recently installed a Maskless Model 5600 LED Direct Imaging machine. The Maskless Model 5600 allows for processing .003 mil lines on most applications.

SCS Announces Opening of New Vietnam Site ▶

Specialty Coating Systems, Inc. (SCS) is pleased to announce the addition of Specialty Coating Systems (Vietnam) Co. Ltd. The ISO 9001:2015-certified facility provides conformal coating services to customers across the consumer and industrial electronics, transportation, aerospace, and medical device industries.

Ultra and Sparton JV (ERAPSCO) Awarded $11.6M for U.S. Navy Production Contract ▶

Ultra Electronics Holdings plc (ULE) and Sparton DLS, LLC announce the award of a contract valued at $11.6 million to their ERAPSCO joint venture, against the $222 million competitive Indefinite Delivery Indefinite Quantity production contract for the manufacture of next-generation sonobuoys for the U.S. Navy.

Sikorsky Leverages Established High-Tech Manufacturing ▶

The Sikorsky RAIDER X® competitive prototype (CP) for the U.S. Army's Future Attack Reconnaissance Aircraft is more than 85% complete, progressing 50% faster through production and assembly compared to legacy programs and resulting in a cost-effective, transformational aircraft.

Lockheed Martin Demonstrates New Layered Missile Defense Integration ▶

Lockheed Martin, the Missile Defense Agency (MDA), and the U.S. Army successfully further integrated the PAC-3 Missile Segment Enhancement (MSE) interceptor into the Terminal High Altitude Area Defense (THAAD) Weapon System.

Why Some Conformal Coatings Fail in Service

Sensible Design

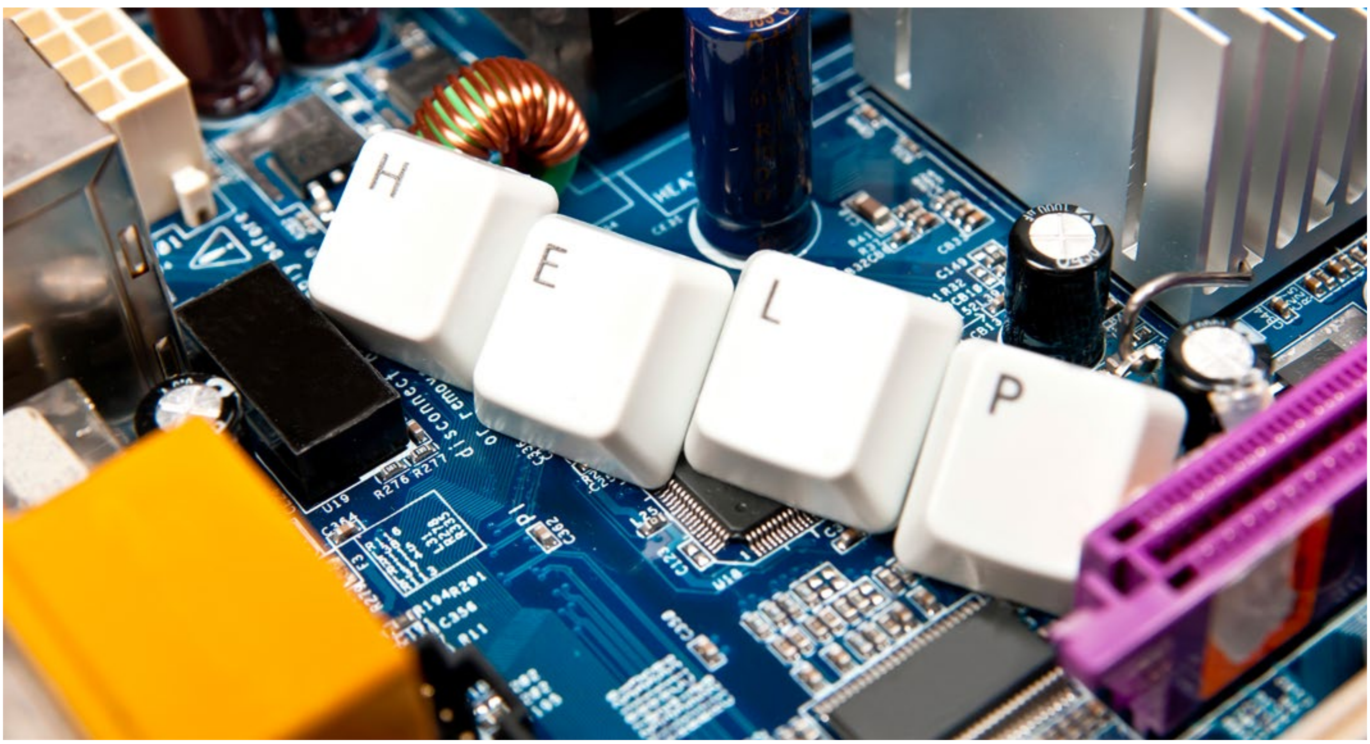
by Phil Kinner, ELECTROLUBE

In this month's column, I will examine why coatings sometimes fail in service and the steps you can take to avoid failure in the first place. At Electrolube, we take great care in our research, product formulation, and developing the appropriate processing and application techniques for our products because we know that, on occasion, conformal coatings can fail and it is our business to understand why.

Our investigations are thorough, and to list our findings here in any detail is somewhat beyond the scope of this column. However, some of the more common root causes of failure pop up frequently, and I'd like to run through these issues, offer some tips, and bust the odd myth. As in our previous columns, they will be presented in a question-and-answer format.

Are some PCBs impossible to coat due to their design?

Nothing is impossible if you have enough time and money, but design is important in determining suitable application methodology and, therefore, the cycle times and the costs involved. Some simple things, like trying to keep connectors or other no-coat areas on the same edge of the assembly, can make a huge difference to the ease of coating an assembly, the cost of coating that assembly and, of course, the overall reliability of that assembly.



Integrated Tools to Process PCB Designs into Physical PCBs

Visualize

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

Verify

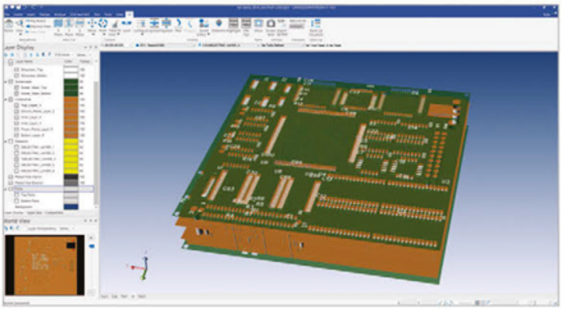
Ensure that manufacturing data is accurate for PCB construciton.

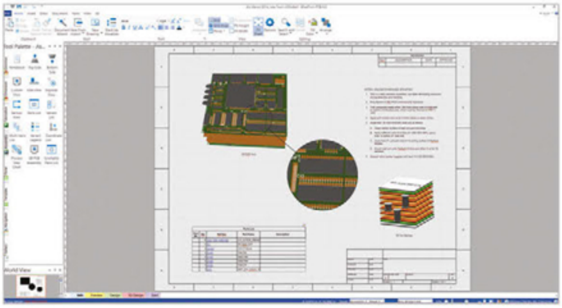
Panelize

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

What are the main reasons for coating failure?

Coating failure can happen for any number of reasons, some common and some not so common. Here are five fundamental reasons which are generally the result of poor product selection and/or application, or some underlying problem arising from insufficient surface preparation or some chemical activity going on beneath the coating that is entirely unrelated to the coating chemistry:

1. The formulation of the coating is insufficient for the job. Choosing an appropriate material selection at the outset can help avoid problems such as condensation occurring, and the coating fails to maintain an adequate level of insulation when the PCB is subjected to a condensing environment. There are some coatings that are much more resistant to these sorts of conditions; failure could have been prevented.
2. Application is crucial to coating success. If the conformal coating has not cured properly, it will not have a chance to develop its protective properties to the full. By getting the application right, you can resolve many issues in one hit.
3. Inadequate coverage or insufficient thickness. Sharp edge coverage can be difficult to achieve with many coatings and it can be hard to ensure sufficient thickness in these areas to maintain protection. A combination of material selection and application technique/workmanship will remedy these sorts of issues.
4. Unexpected interaction with other process materials used to prepare/build the PCB, such as flux residues, are particularly illustrative of this type of problem. In a "no-clean" process, for example, these can inhibit the cure of some types of coating or lead to a loss of insulation of the system, greater than either material in isolation.
5. Something else is going on that isn't directly related to the coating. Unless there has been meticulous attention to preparation or pre-coat cleaning regimes, potentially corrosive residues bridging the PCB's conducting tracks can, over time, cause failures. While the coating may delay failure for many years, at some point failure will inevitably happen.

Is the application method important to the reliability of coatings?

Application method is probably the number one determinant of success. Often a poor material, well applied, can be just as good (or sometimes better) than a material with great properties that is applied badly. Coating is about getting sufficient coverage of the sharp edges and metal surfaces without applying the material too thickly elsewhere. Some materials "apply better" than others and make this process more straightforward, but in the end, the performance of liquid applied coatings will always be determined by how well they were applied.

Would a thicker coating be less likely to fail?

To be honest, it depends on the coating chemistry. Thicker can be better up to a point but at some stage the coating material will be too thick and will either crack itself, or even cause cracking of the coated components themselves, for example, during thermal shock or thermal cycling. And depending on the type of coating material used, solvent entrapment (i.e., the solvent not having enough time to evaporate from the coating film before it hardens) can become an issue.

On top of this, by adding unnecessary amounts of coating material you are in effect wasting it, adding to costs and, importantly for some applications, adding weight. Compromised thermal management issues can also arise as it may prove difficult to dissipate heat away from thickly coated components.

It is good to remember that above a certain thickness, which does vary according to the material being applied, any increase is likely to deliver diminishing returns. Get the advice of a reputable supplier; they have laboured hard to establish optimum coating thicknesses for their products in all kinds of operating environments. There's a lot to consider for a successful conformal coating process. Getting it wrong could compromise the reliability of an electronic assembly, shorten its life expectancy, or even prove critical. First and foremost, seek some expert advice as there are new coating materials, such as our new bio-based UVCLX coating with 75% bio-organic content from renewable sources, which might solve your requirements more sufficiently. I hope my column has been useful and of course, we are always happy to help and advise. **ESIGN007**

Phil Kinner is the global business and technical director of conformal coatings at Electrolube. To read past columns or contact Kinner, click here. Download your 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!"

BOOK EXCERPT

The Printed Circuit Designer's Guide to... High Performance Materials

Chapter 2: Glass Fabric

Evolution of Glass Fabric

Materials such as cotton, paper and glass mat were used before woven glass fiber started mass production and became the dominant fabric used for PCB composites. Early PCBs were low layer count and used a heavy glass weave, such as 7628 or 7629, as the primary component. Since the mid '70s, glass manufacturers have introduced new and thinner glass weaves to enable ultra-thin laminate and substrate materials.

Woven glass fabric is impregnated with resin and when fully cured or dried, provides the mechanical, chemical, and electrical properties for rigid printed circuit applications. The characteristics and properties of glass fiber and fabrics are defined by IPC-4412 "Specification for Finished Fabric Woven from E-Glass for Printed Boards." There are compositions of glass fibers that will also be discussed including the next generation of lower-Dk glass fabrics now in development.

Glass is a mix of several inorganic minerals which are melted in a furnace. Temperatures are maintained above 1000°C so the components can be melted and homogenized. The molten glass is then drawn from platinum bushings to form fibers which are collected to make a yarn which is collected onto a bobbin. These bobbins are then processed into an electronics fabric designed to be used in production of laminate and prepreg. Figure 2.1 is a review of the whole process from raw material to the fiber and process to be woven as glass cloth.

Download this book to continue reading.

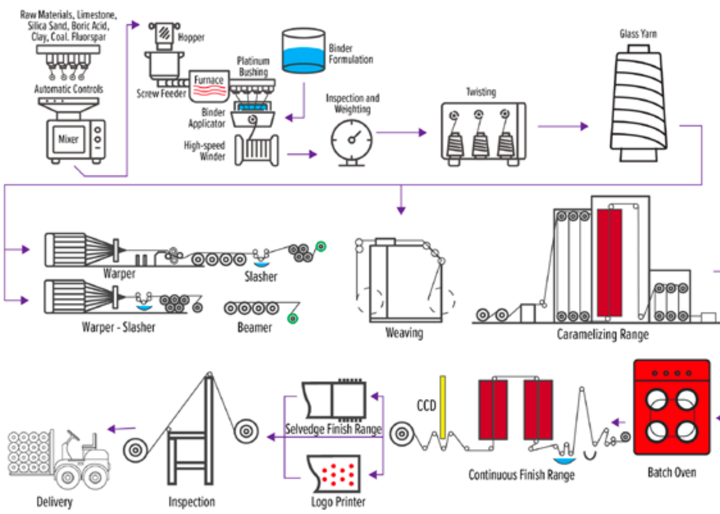


Figure 2.1: Glass fiber/glass cloth manufacturing process. (Source: Taiwan Glass)

Blue Skying It With Aluminum Rigid-Flex

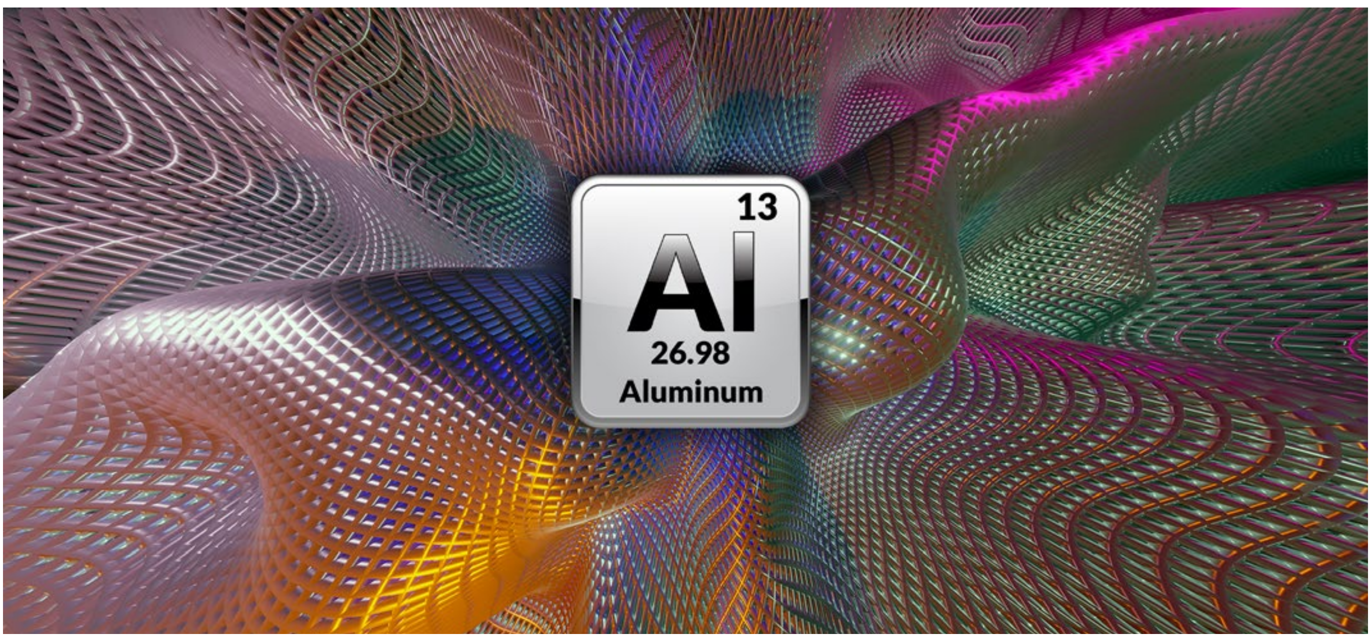
Flexible Thinking

by Joe Fjelstad, VERDANT ELECTRONICS

Aluminum is an amazingly versatile metal and has found its way into countless products since its discovery during the reign of Napoleon III of France. At the time, it was more valuable than gold, and at hosted dinners, the emperor and his honored guests dined using aluminum cutlery while the others had to make do with gold utensils.

It took some time for scientists to calculate that it was by far the most abundant metal found in the earth's crust at 8.3%, ranking third among all elements found in the crust, ranking behind oxygen (46%) and silicon (26%).

Today, aluminum is one of the most affordable metals and arguably, also one of the most useful. Aluminum has a long list of desirable properties. It is lightweight, and an excellent electrical and thermal conductor. It can be easily machined, stamped, chemically milled, and formed. Being a metal it is, by nature,




PREMIER GLOBAL SUPPLIER
to AUTOMOTIVE
FLEXIBLE CIRCUITS | EMS/ASSEMBLY | CONTRACT MANUFACTURING
Industry leading Flex Design Support
Flexible Circuits, Rigid Flex, Membrane Switches, Plastic Moldings
Specialized EMS/Assembly Services
Assemblies and Product Module Builds for Automotive
IATF 16949
EXPLORE the POSSIBILITIES!
FLEXIBLECIRCUIT Technologies®
Flexible Circuit Technologies
9850 51st Ave. N. | Plymouth, MN 55442
www.flexiblecircuit.com | +1-763-545-3333

dimensionally stable. It also expands isotropically (unlike composite materials which typically expand at different rates in X, Y, and Z dimensions). Aluminum can be anodized, which converts its skin to alumina (Al_2O_3), which is ceramic, and thus offers a potential to create an in situ distributed capacitance layer.

The metal can also be electrophoretically coated uniformly with polymer and enamel insulating materials (think automobile paint uniformity). It also has a CTE which is close to that of copper (22 ppm/°C for aluminum vs. 18 ppm/°C for copper) and is non-toxic (not a RoHS target). It is historically low cost at about $1 per pound. Those in PCB manufacturing will know that aluminum has for many years been a process consumable as drill entry material when drilling printed circuits to mitigate copper burr formation. Typical laminates, exclusive of copper foil, can be two to three times more costly on a by-weight basis and much more expensive to create.

Typical laminates, exclusive of copper foil, can be two to three times more costly on a by-weight basis and much more expensive to create.

With such a long list of benefits, it would seem aluminum might be an ideal candidate for making printed circuits (including rigid-flex circuits), except for one hitch. Being a great thermal conductor makes aluminum a lousy choice for soldering components in place due to the high risk of creating cold solder joints, and/or if one is to avoid thermally damaging components by using excessive heat during the soldering process.

What If Soldering Could Be Avoided?

Following is a "blue sky" thought experiment/description of one possible way to build a rigid-flex assembly with an aluminum base and no solder. Rigid-flex circuits were originally designed to be used in rugged military applications and thus are perhaps ideal as a candidate for being adapted to the use of an aluminum core. While such an assembly could be comprised of any suitable metal carrier (brass, bronze, steel, etc.), aluminum, for the many reasons cited at the outset, is arguably preferred.

The process that will be described is basically the reverse of traditional manufacturing, in that, rather than building a rigid-flex board and soldering components to the finished product, the aluminum cores which support the components are processed in a manner to accept the chosen components so that their terminations face outward from the body of the aluminum sections on one or both sides of the metal cores and later plated with copper to interconnect their terminations. It is advantageous, after creating cavities which hold the components in place, that the assembly be processed by anodizing or electrophoretic coating with a suitable dielectric to make the surfaces nonconductive. The "component board" can then be coated with a flexible dielectric. The components are preferably of a common thickness, but it is not imperative as there are ways to deal with height differences. More importantly, components should be fully tested and burned in, though tested bare die are a possible option.

At this point, a layer of flexible insulation is to be applied to one or both sides of the core (Figure 1). The assembly can now be processed as if it were a rigid printed circuit using a build-up process to make the circuits. Side-to-side connections can be made by drilling and plating through-holes but will require a process step to coat the exposed aluminum and a second drill through the dielectric to accept the electroless and electrolytic copper plating.

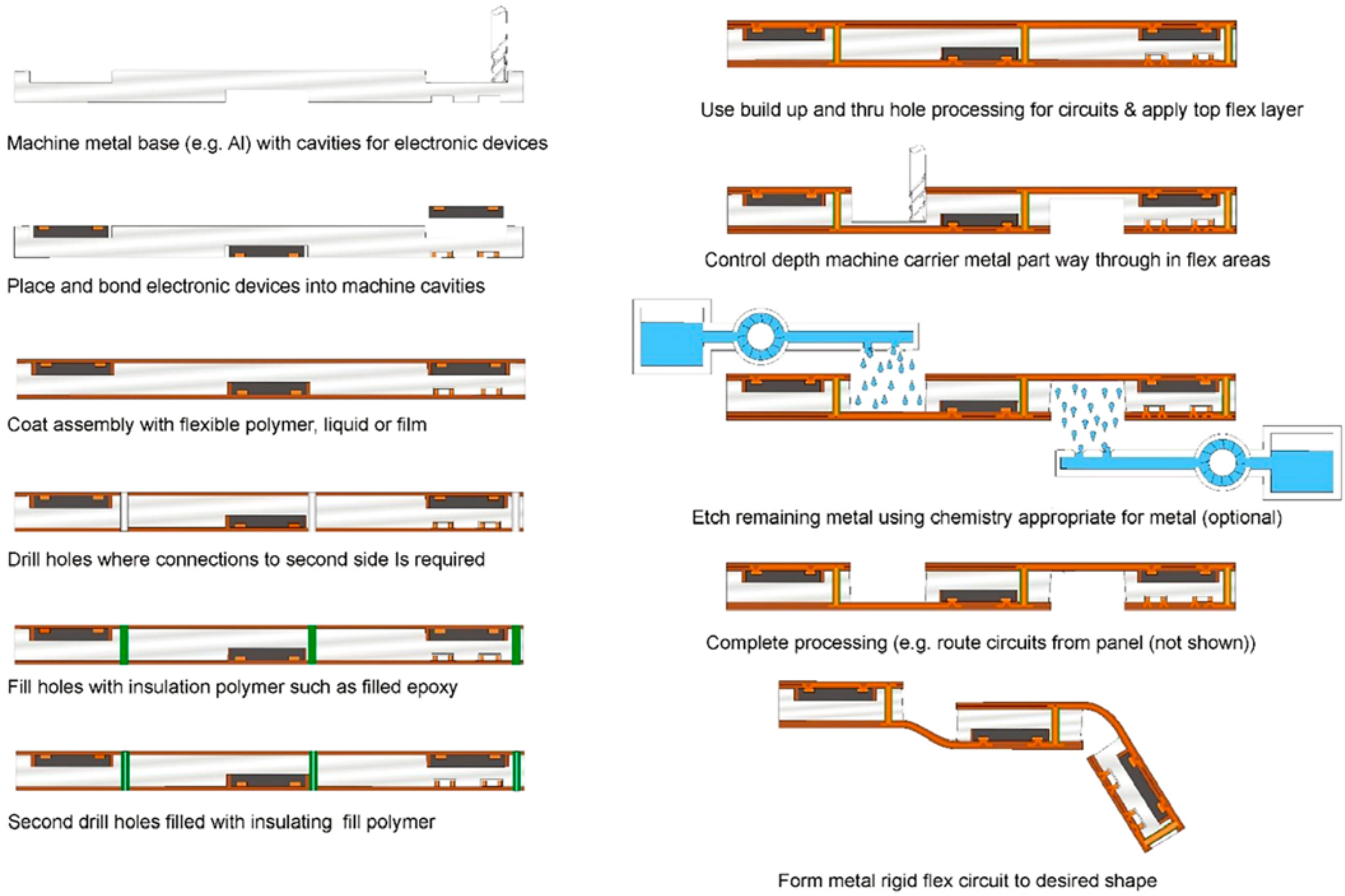


Figure 1: Example of a possible aluminum rigid-flex manufacturing process. (Source: U.S Patent 10,285,270)

An advantage of not soldering to make connections is that the component lands can be treated as microvias and the free space can be used for increased routing. When all the required interconnections and circuits are added, the assembly can be partially machined in areas where bending is performed to thin the metal carrier sufficiently to allow the residual metal to easily bend. If left, the residual metal can serve as a ground layer for the assembly. If isolation is required or desired, the residual metal can be etched using chemistry appropriate for the metal used as a carrier. Alkaline chemistries such as dilute sodium hydroxide and sodium gluconate solutions are suitable for aluminum, but other chemistries such as ferric chloride are more universal in their potential application. There are many proprietary chemistries which can be used as well. The patented process concept is illustrated in Figure 1.

Accepted flex circuit design practices suggest that the number of circuit layers through the areas where flexing or bending is required should ideally be limited to one or two metal layers. That aside, the processes described offer significant potential benefits for increased reliability and reduced overall cost.

Conclusion

The continuing challenge of making lead-free solder work in electronic assembly has given rise to new ways of looking at circuit assembly and fabrication, using methods that could eliminate solder and its myriad issues altogether. What has been offered for the reader to ponder is just the tip of the iceberg. My mind (and perhaps now yours) lights up daily with ideas and ways to integrate packaged chips with the interconnections they require to carry out their mission. Improved approaches to assem-

bly such as those that become possible using the Occam process offer both the independent designer and OEM a new option for products that are highly reliable and cost-effective. Such products will skirt the numerous concerns that have vexed manufacturers for the last decade.

Currently, solderless flexible circuit assembly technology is making headway in several ways using printed inks and conductive adhesives in place of copper traces and solder. These are important first steps, but it is still uncertain if the inks will ever compete with copper metal. Still those efforts hold out the promise of significant potential for the future. They are helping to broaden the range of material options and significantly expand the horizon of electronic interconnection design. Once interconnection technologists and product designers begin to open-mindedly climb this new "Occam technology tree," they will find branches that take them to many new types of processes, methods, and structures that they would not have anticipated or seen before they began the climb. The specific structure and process example provided here has not yet been reduced to practice, but it is possible. All that is missing is a willingness to try something, and unlike building a new semiconductor fab, it won't cost billions of dollars to try it out. It just requires a few more people to take up Steve Jobs' challenge to the industry issued at Apple some 20 years ago to "think different." **FLEX007**

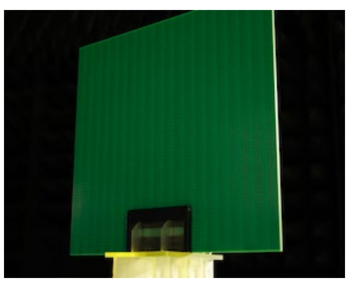
Joe Fjelstad is founder and CEO of Verdant Electronics and an international authority and innovator in the field of electronic interconnection and packaging technologies with more than 185 patents issued or pending. To read past columns or contact Fjelstad, click here. Download your free copy of Fjelstad's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

KYOCERA Develops Transmissive Metasurface Technology

Kyocera Corporation has developed a Transmissive Metasurface technology that can redirect wireless network signals in a specific direction to improve the coverage area and performance of 5G and eventually 6G networks. The Transmissive Metasurface will help deliver high-frequency millimeter-wave 5G and 6G to places where communication is impossible.

Development Background

The 28 GHz band used in 5G networks, and the higher frequency band being studied for 6G, have a high degree of rectilinear propagation. Signals often cannot reach locations where a direct line of sight to the base station is obstructed. Reflective Metasurface technology offers a very limited ability to change the direction of a signal to reach these areas. To solve this issue and expand performance, Kyocera developed a new Transmissive Metasurface technology that can redirect radio waves at smaller angles to extend targeted network coverage.



Features: Transmissive Metasurface Technology

1) Kyocera expands the direction in which radio signals can be redirected

Radio waves striking a conventional Reflective Metasurface device can be redirected at a wide angle, but not at narrow angles beyond the metasurface. Kyocera's new Transmissive Metasurface technology is able to bend at narrow angles in order to avoid obstacles that may block transmission, expanding 5G and eventually 6G coverage even further.

2) Proprietary Flexible Size Development

The area in which a Transmissive Metasurface can deliver signals is proportional to the size of the metasurface itself. This makes it possible to install metasurfaces in more places, such as a home patio or apartment balcony.

(Source: Business Wire)

Flex007 Highlights

S & P Flex Circuits Ltd Opts for Kaima Universal Fixture Tester From Gardien ▶

Gardien Group is pleased to announce the installation of the Kaima bare circuit board fixture tester to S & P Flex in Scarborough, Ontario, Canada.

Flexible Thinking: The Fascinating History of Wearable Electronics ▶

Wearable electronics have been capturing much attention in the press, both technical and business, over the past few years. Articles for consumption by the public, as well as technical research papers on the topic, have been increasing steadily in recent times. However, wearable electronics are far from new. Moreover, the term "wearable" is quite fungible and encompasses a broad spectrum of prospective embodiments.

Beyond Design: Designing for the SAP Fabrication Process ▶

PCB designers are continually challenged with demands for reduced product size. However, form factor-driven design pressures have been relieved, in part, by the increased use of high-density interconnects (HDIs), which enable more functionality per unit area than conventional PCBs. Leveraging finer lines, thinner materials, and laser-drilled vias, HDIs have played a crucial role in device miniaturization. However, the traditional PCB subtractive etch processing becomes very difficult for feature sizes below 3 mil trace/space. This forces PCB designs to become more complex as electronics packages shrink—adding extra routing layers, and microvia layers, and increasing the number of lamination cycles required, all of which impact yield, reliability, and thus cost.

Multek to Reduce Carbon Emissions by Half by 2030 ▶

Multek will work to reduce its carbon emissions by half by 2030 and continue to accelerate toward carbon neutrality.

Altix Receives Bundle Machine Order from Major Chinese FPC & PCB Player ▶

Altix is delighted to announce a significant order for both direct imaging and contact printer equipment. The bundle encompasses both panel and RtR solutions to be installed at a new plant in Jiangxi province.

FLEX Conference 2022 to Highlight Latest Flexible Hybrid Electronics Innovations ▶

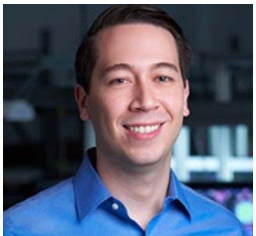
FLEX Conference will gather industry experts July 11-14, 2022, at the Moscone Center in San Francisco for keynotes, panel discussions, technical sessions, and product-based demonstrations highlighting the latest innovations in flexible and printed electronics.

Catching Up With Alpha Circuit's Prashant Patel ▶

There is plenty of evidence that the American PCB industry is going through a revitalization. While a few new companies are being established, others are being rejuvenated as investors gain more interest and confidence in domestic PCB companies. I reached out to Prashant Patel, owner and president of Alpha Circuit Corporation in the greater Chicago area. I wanted to hear about his investment and the unique path he took to owning a PCB shop.

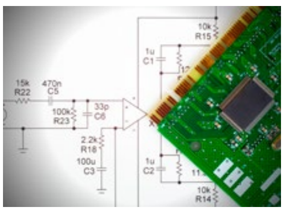
TOP TEN EDITOR'S PICKS

AltiumLive 2022: MacroFab Unites Design and Manufacturing in One Domain


Joey Rodriguez

I recently spoke with Joey Rodriguez, director of product management for MacroFab, which just announced a major partnership with Altium that brings supply chain information to Altium users much earlier in the design cycle. Joey and I discussed his AltiumLive presentation, now available online, which provides details about MacroFab's efforts to "left-shift" supply chain information, and the need for designers to think holistically about the supply chain as early as possible in the design cycle.

Connect the Dots: Six Key Considerations for Designers New to PCB Layout



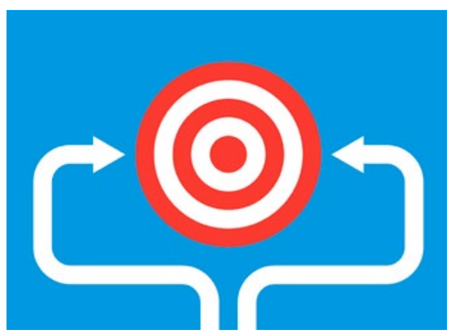
Demand continues to increase for boards used in consumer electronics, intelligent machines used in manufacturing, and smart devices for health services applications. Our industry needs more smart people designing PCBs to help drive artificial intelligence (AI) initiatives and power the Internet of Things (IoT), which is why we are welcoming new designers into the fold every day.

A Definitive Review of High-Performance Materials Book

The specialists at Isola have written an excellent book about high performance materials, now available for download from I-Connect007. Author Michael Gay, a 25-year veteran of laminate manufacturing, provides essential information for everyone, including the experts, because the materials and applications for laminates are constantly changing.

Additive Design: Same Steps, Different Order

We recently spoke with Dave Torp, CEO of Winonics, about the company's additive and semi-additive processes and what PCB designers need to know if they're considering designing boards with these new technologies. As Dave explains, additive design is not much different from traditional design, but the steps in the design cycle are out of order, and additive designers must communicate with their fabricators because so much of the new processes are still proprietary.

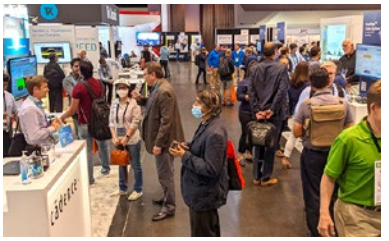


Samtec's Richard Mellitz Named Engineer of the Year at DesignCon 2022


Richard Mellitz

Richard Mellitz, distinguished engineer at Samtec, was selected by peers as recipient of the 2022 Engineer of the Year Award. He took home the well-earned title for his nearly 40 years as an engineer.

DesignCon 2022: Back to Business as Usual



I attended DesignCon 2021 at the San Jose McEnery Convention Center in July of last year, and I could see a big improvement in attendance as DesignCon moved back into its usual venue.

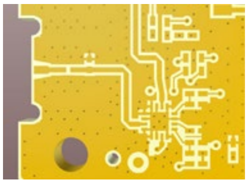
Sensible Design: What Can Manufacturers Expect from Bio-based Conformal Coatings?

Bio-based materials, obtained from biological resources (biomasses, feedstock, plants, and biological waste), deliver a viable and sustainable alternative to materials derived from petro-chemicals, and we have developed a new coating incorporating 75% bio-organic content from renewable sources. It's a global first for the industry and is completely free from solvents.

Setting Goals for Your PCB Design Education

For our March issue, we spoke with Bill Brooks of Nordson ASYMTEK, a long-time PCB designer, CID instructor, and fantastic sculptor. I asked Bill to share his thoughts on setting up a PCB design education career plan, and the need to stay on top of your game as a PCB designer.

Does Copper Pour on a Signal Layer Decrease Signal-to-Signal Isolation?



Does putting a ground pour on PCB signal layers make the isolation better or worse? It can go either way, but with the proper knowledge and application, this technique will improve your designs.

Elementary, Mr. Watson: The Five Pillars of your Library, Part 4—Review

Now that we have a single library managed using our revisioning, and we have lifecycle schemes organized so that we can easily find something in the component category, family, and subfamilies, we are now ready to look at one of our library's most vital principles and pillars.

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

Career Opportunities

Career Opportunities

Polar

US-Based Technical Sales Specialist

Polar Instruments, Inc. (Beaverton, OR) is looking for an additional US-based technical sales specialist to assist in selling our growing family of PCB signal integrity tools for the PCB fabrication/design industry and the electronics supply chain.

A background in the PCB industry and fabrication knowledge are an advantage. A B.S. in engineering or physics (or other science or technology degree) and related technical sales experience preferred. Willingness to travel 8-10 times a year for a week at a time and 1-2 weekends.

Polar has been in business for over 40 years and are a small, eclectic group of people spread around the world. Our values are centered on enjoying what we do–which is working to make things just right for our customers. We approach challenge as an opportunity, frequently an adventure. We work together as a team, without politics, and with our team's welfare at the forefront of our concerns. We are successful. We have grown year-upon-year, and we take great pride in pretty much owning our niche in a very large marketplace. We command great respect from our users and industry partners.

If interested, please contact Lupita at
jobs.usa@polainstruments.com

apply now

ALTIX
Direct Imaging & Contact Printers

Sales Technical Engineer

ALTIX, a French company, designs, manufactures, markets and services exposure equipment for the printed circuit board, flexible circuit, metal etching, touch panel and other industries. The U.S. subsidiary, focused on the sale and service of Altix equipment in North America, is looking for a sales technical engineer to support their growth.

Responsibilities

- Promote Altix's products by visiting customers
- Serve as a technical lead & product expert to provide technical recommendations to customers
- Gather on-the-ground market intelligence through customer contact
- Ensure sustainable growth in sales, profits, and market presence
- Develop new business and achieve targets for market penetration, sales and profit
- Manage sales partners

Skills & Qualifications

- Minimum 2 to 5 years' experience in sales for capital equipment in the PCB market or related industries
- Business development and marketing background preferred
- 5+ years' North American business leadership experience in related field
- Strong leadership, decision-making and communication skills.
- Proficiency in standard computer software applications such as Microsoft Office
- Excellent written and oral communication skills
- Willingness to travel within the US, Canada and to France for training

Email contact: sylvain.dromaint@altix.us

apply now

Career Opportunities

Global Account Manager, e-Mobility & Infrastructure

Location: AZ, CA, TX (remote)

Job Summary:

The Global Account Manager, e-Mobility & Infrastructure is a key position for the sales organization, serving as Indium Corporation's lead sales contact responsible for developing targeted accounts in the e-Mobility and related infrastructure space.

Responsibilities will include:

- sourcing for new global business opportunities
- implementing effective sales strategies
- interfacing with customers' senior management
- execution of action plans through the regional teams
- interaction with internal customers (R&D; Inside Sales; Technical Support; Logistics; Product Management; Operations; Engineering; Quality; etc.) resulting in evaluation, qualification, specification, and maximum customer share for designated customers
- providing risk assessment of the business to senior management

apply now

Director of Operations

State College, PA

Chemcut Corp., a world leader in wet processing equipment for the manufacture of printed circuit boards and chemical etching of various metals, is seeking a Director of Operations.

Objectives of the Role:

- Collaborate with the CEO in setting and driving organizational vision, operational strategy, and hiring needs.
- Oversee manufacturing operations and employee productivity, building a highly inclusive culture ensuring team members thrive and organizational outcomes are met.
- Directly oversee manufacturing operations, production planning, purchasing, maintenance & customer service (product support) and partner with the CEO and controller on sales management to budget for sufficient investment capital to achieve growth targets.
- Aggressively manage capital investment and expenses to ensure the company achieves investor targets relative to growth and profitability.

Qualifications:

- Bachelor's degree in mechanical, electrical, or related fields
- 5+ years' experience in leadership positions
- Leadership skills, with steadfast resolve and personal integrity
- Understanding of advanced business planning and regulatory issues
- A solid grasp of data analysis and performance metrics
- Ability to diagnose problems quickly and have foresight into potential issues

Preferred Qualifications:

- Master's degree in business or related field
- International business experience

To apply, please submit a cover letter and resume to hr@chemcut.net

apply now

Career Opportunities

Career Opportunities

ventec
INTERNATIONAL GROUP
騰輝電子

Operations Supervisor

Elk Grove Village IL, USA

As operations supervisor at Ventec USA LLC, you will have a hands-on and quality-driven approach to coordinating and overseeing the operations in Elk Grove Village, Illinois. You will plan, organize, and implement the day-to-day warehouse activities to ensure customer expectations are met. Tasks will include planning, implementing performance improvement measures, procuring materials and resources, and assuring compliance to the Quality Management System. You will be a mentor to team members, find ways to maintain and improve the highest quality of customer service, and implement best practices across all levels to help the company remain compliant, efficient, and profitable.

Skills and abilities required:

- Proven experience as operations supervisor or similar role
- Knowledge of organizational effectiveness and warehouse management
- Experience with ISO9001 or similar QMS
- Experience in budgeting and forecasting/ familiarity with business and financial principles
- Excellent leadership ability and communication skills (English)
- Outstanding organizational skills
- Qualification in distribution, logistics, transportation, or business studies is preferential

What's on offer:

- Excellent salary & benefits commensurate with experience

This is a fantastic opportunity to become part of a successful brand and leading team with excellent benefits. Please forward your resume to **HR@Ventec-usa.com** and mention (Operations Supervisor) in the subject line.

apply now

ventec
INTERNATIONAL GROUP
騰輝電子

European Product Manager

Taiyo Inks, Germany

We are looking for a European product manager to serve as the primary point of contact for product technical sales activities specifically for Taiyo Inks in Europe.

Duties include:

- Business development & sales growth in Europe
- Subject matter expert for Taiyo ink solutions
- Frequent travel to targeted strategic customers/ OEMs in Europe
- Technical support to customers to solve application issues
- Liaising with operational and supply chain teams to support customer service

Skills and abilities required:

- Extensive sales, product management, product application experience
- European citizenship (or authorization to work in Europe/Germany)
- Fluency in English language (spoken & written)
- Good written & verbal communications skills
- Printed circuit board industry experience an advantage
- Ability to work well both independently and as part of a team
- Good user knowledge of common Microsoft Office programs
- Full driving license essential

What's on offer:

- Salary & sales commission--competitive and commensurate with experience
- Pension and health insurance following satisfactory probation
- Company car or car allowance

This is a fantastic opportunity to become part of a successful brand and leading team with excellent benefits. Please forward your resume to **jobs@ventec-europe.com.**

apply now

Career Opportunities

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

Wet Process Engineer

ASC, the largest independent PCB manufacturer in the Midwest, is looking to expand our manufacturing controls and capabilities within our Process Engineering department. The person selected will be responsible for the process design, setup, operating parameters, and maintenance of three key areas—imaging, plating, etching--within the facility. This is an engineering function. No management of personnel required.

Essential Responsibilities

Qualified candidates must be able to organize their own functions to match the goals of the company.

Responsible for:

- panel preparation, dry film lamination, exposure, development and the processes, equipment setup and maintenance programs
- automated (PAL line) electrolytic copper plating process and the equipment setup and maintenance programs
- both the cupric (acid) etching and the ammoniacal (alkaline) etching processes and the equipment setups and maintenance programs

Ability to:

- perform basic lab analysis and trouble-shooting as required
- use measurement and analytical equipment as necessary
- work alongside managers, department supervisors and operators to cooperatively resolve issues
- effectively problem-solve
- manage multiple projects concurrently
- read and speak English
- communicate effectively/interface at every level of the organization

Organizational Relationships

Reports to the Technical Director.

Qualifications

Degree in Engineering (BChE or I.E. preferred). Equivalent work experience considered. High school diploma required. Literate and functional with most common business software systems MS Office, Excel, Word, PowerPoint are required. Microsoft Access and basics of statistics and SPC a plus.

Physical Demands

Exertion of up to 50 lbs. of force occasionally may be required. Good manual dexterity for the use of common office equipment and hand tools.

- Ability to stand for long periods.

Work Environment

This position is in a manufacturing setting with exposure to noise, dirt, and chemicals.

Click on ‘apply now’ buttton below to send in your application.

apply now

Career Opportunities

MacDermid Alpha
ELECTRONICS SOLUTIONS

R&D Scientist III
Orange, CT

Job Description: The scientist will be a leader in technology for plating chemistry development, electrolytes, and additives. The position is hands-on, where the ideal candidate will enjoy creating and testing new aqueous plating processes and materials to meet the most demanding semiconductor applications related to Wafer-Level Packaging and Damascene. The qualified candidate will work as part of the R&D team while interacting with scientists, product management, and application engineers to commercialize new products for the advanced electronic solution business.

apply now

Technical Marketing Specialist
Waterbury, CT

This position provides information from the product team to the marketing communications team. It is a multifunctional role that requires some experience within electronics manufacturing supply chain or knowledge of how electronic devices are manufactured, specifically PCBs, semiconductors, and the chemical processes utilized therein. The primary function of this role is to help in the generation of product marketing collateral, but also includes assisting in tradeshow content development, advertising, and launches.

apply now

KYZEN
Where Science and Care Converge.

Regional Manager
Midwest Region

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

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

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

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

fernando_rueda@kyzen.com

apply now

Career Opportunities

Printed Circuits, a fast-growing printed circuit board fabricator, offers:

- Excellent opportunities for advancement and growth
- Dynamic manufacturing environment
- Excellent health, dental and other benefits
- Annual profit-sharing plan
- Signing bonus
- Additional incentives at the leadership level
- Clean facility with state-of-the-art manufacturing equipment
- Highly collaborative corporate and manufacturing culture that values employee contributions

Laminator Technician

Nature of Duties/Responsibilities

- Layup cover lay
- Layup rigid flex
- Layup multilayer/CU core boards
- Oxide treat/cobra treatment of all layers/CU cores
- Shear flex layer edges
- Rout of machine panel edges and buff
- Remove oxide/cobra treatment (strip panels)
- Serialize panels
- Pre-tac Kapton windows on flex layers (bikini process)
- Layup Kapton bonds
- Prep materials: B-stage, Kapton, release sheet
- Breakdown: flex layers, and caps
- Power scrub: boards, layers, and caps
- Laminate insulators, stiffeners, and heatsinks
- Plasma cleans and dry flex layers B-stage (Dry)
- Booking layers and materials, ready for lamination process
- Other duties as deemed necessary by supervisor

Education/Experience

- High school diploma or GED
- Must be a team player
- Must demonstrate the ability to read and write English and complete simple mathematical equations
- Must be able to follow strict policy and OSHA guidelines
- Must be able to lift 50 lbs
- Must have attention to detail

Wet Process/Plating Technician

Position is 3rd shift (11:00PM to 7:30AM, Sunday through Friday)

Purpose

To carry out departmental activities which result in producing quality product that conforms to customer requirements. To operate and maintain a safe working environment.

Nature of Duties/Responsibilities

- Load and unload electroplating equipment
- Fasten circuit boards to racks and cathode bars
- Immerse work pieces in series of cleaning, plating and rinsing tanks, following timed cycles manually or using hoists
- Carry work pieces between departments through electroplating processes
- Set temperature and maintains proper liquid levels in the plating tanks
- Remove work pieces from racks, and examine work pieces for plating defects, such as nodules, thin plating or burned plating
- Place work pieces on racks to be moved to next operation
- Check completed boards
- Drain solutions from and clean and refill tanks; fill anode baskets as needed
- Remove buildup of plating metal from racks using chemical bath

Education and Experience

- High school diploma or GED required
- Good organizational skills and the ability to follow instructions
- Ability to maintain a regular and reliable attendance record
- Must be able to work independently and learn quickly
- Organized, self-motivated, and action-oriented, with the ability to adapt quickly to new challenges/opportunities
- Prior plating experience a plus

Production Scheduler

Main Responsibilities

- Development and deployment of a level-loaded production plan
- Establish manufacturing plan which results in "best possible" use of resources to maximize asset utilization
- Analyze production capacity of manufacturing processes, equipment and human resource requirements needed to produce required products
- Plan operation manufacturing sequences in weekly time segments utilizing production labor standards
- Maintain, align, and communicate regularly with internal suppliers/customers and customer service on key order metrics as per their requirements
- Frequently compare current and anticipated orders with available inventory and creates replenishment plan
- Maintain master distribution schedule for the assigned facility, revise as needed and alert appropriate staff of schedule changes or delays
- Participate in periodic forecasting meetings
- Lead or participate in planning and status meetings with production, shipping, purchasing, customer service and/or other related departments
- Follow all good manufacturing practices (GMPs)
- Answer company communications, fax, copy and file paperwork

Education and Experience

- High school diploma or GED
- Experience in manufacturing preferred/3 years in scheduling
- Resourceful and good problem-solving skills
- Ability to make high pressure decisions
- Excellent written and verbal communication skills
- Strong computer skills including ERP, Excel, Word, MS Office
- Detailed and meticulous with good organizational skills
- Must be articulate, tactful and professional at all times
- Self-motivated

apply now

Career Opportunities

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

Career Opportunities

Career Opportunities

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

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

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

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

Career Opportunities

I-007e EDUCATIONAL RESOURCE CENTER

Latest I-007eBooks

The Printed Circuit Designer's Guide to... High Performance Materials

by Michael Gay, Isola

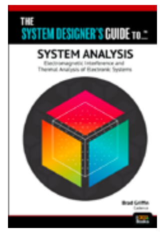
This book provides the reader with a clearer picture of what to know when selecting which material is most desirable for their upcoming products and a solid base for making material selection decisions. **Get your copy now!**

The Printed Circuit Designer's Guide to... Stackups: The Design within the Design

by Bill Hargin, Z-zero

Finally, a book about stackups! From material selection and understanding laminate datasheets, to impedance planning, glass weave skew and rigid-flex materials, topic expert Bill Hargin has written a unique book on PCB stackups. **Get yours now!**

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



by Brad Griffin, Cadence

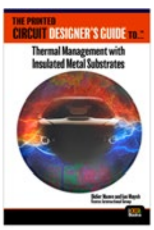
In this book, the author, Brad Griffin of Cadence, focuses on EM and thermal analysis in the context of data center electronics systems. Be sure to also **download the companion guide** for end-to-end solutions to today's design challenges.

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

Thermal Management: A Fabricator's Perspective

by Anaya Vardya, American Standard Circuits

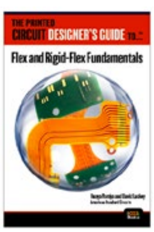
Beat the heat in your designs through thermal management design processes. This book serves as a desk reference on the most current techniques and methods from a PCB fabricator's perspective.



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.



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) 365-1727; barb@iconnect007.com

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

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

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

TECHNOLOGY SPECIALIST: **BRYSON MATTIES**

COVER: **SHELLY STEIN**

COVER IMAGES: **ADOBE STOCK © ANDREY CHMELYOV & SERGEY NIVENS**

DESIGN007 MAGAZINE

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

May 2022, Volume 11, Number 5
DESIGN007 MAGAZINE is published monthly, by BR Publishing, Inc.

ADVERTISER INDEX

Accurate Circuit Engineering........................ 5
Altium.. 49
American Standard Circuits........................ 61
APCT... 23
Averatek... 53
Burkle North America.............................. 85
Cadence Design Systems............................ 7
Candor Industries.................................. 37
Downstream Technologies..................... 17, 77
EMA Design Automation........................... 35
Flexible Circuit Technologies..................... 81
I-Connect007 eBooks................... 2, 3, 47, 87
In-Circuit Design Pty Ltd.......................... 59
IPC... 65
MacDermid Alpha Assembly Solutions.......... 89
Miltronix... 9
NCAB Group................................... 69, 71
Precision Technologies............................ 31
Prototron Circuits.................................. 75
Pulsonix... 27
Siemens Digital Industries Software............ 41
Taiyo America....................................... 83
US Circuit... 51
Ventec International Group.................. 13, 45