MAY 2020

DESIGN007 MAGAZINE

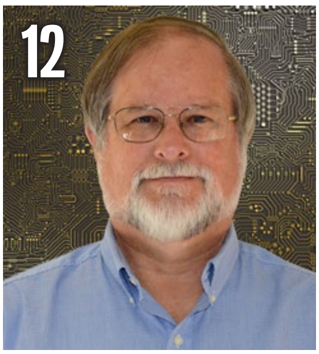
I-Connect007


Is your PCB documentation process outdated?
Today's automated documentation solutions can eliminate post-processing errors and speed up time to market.
THE PRINTED CIRCUIT DESIGNER'S GUIDE TO...™
Documentation
Assessing Your PCB Documentation Process
Mark Gallant
DownStream Technologies
I-007e Books
DOWNLOAD NOW
I-007e Books

DESIGN007 MAGAZINE

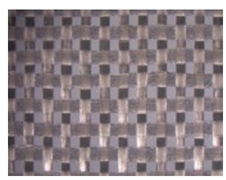
What Do You Expect From Me?

How many times have you found yourself asking this question during your career? You're not alone. Almost everyone involved in PCB design and manufacturing—especially fabricators—has had to deal with unmet expectations. This month, our experts discuss the expectations of technologists throughout the design cycle, the ramifications of not meeting expectations, and how to define requirements so that unmet expectations are (eventually) a thing of the past.



FEATURES:

FEATURE COLUMNS:

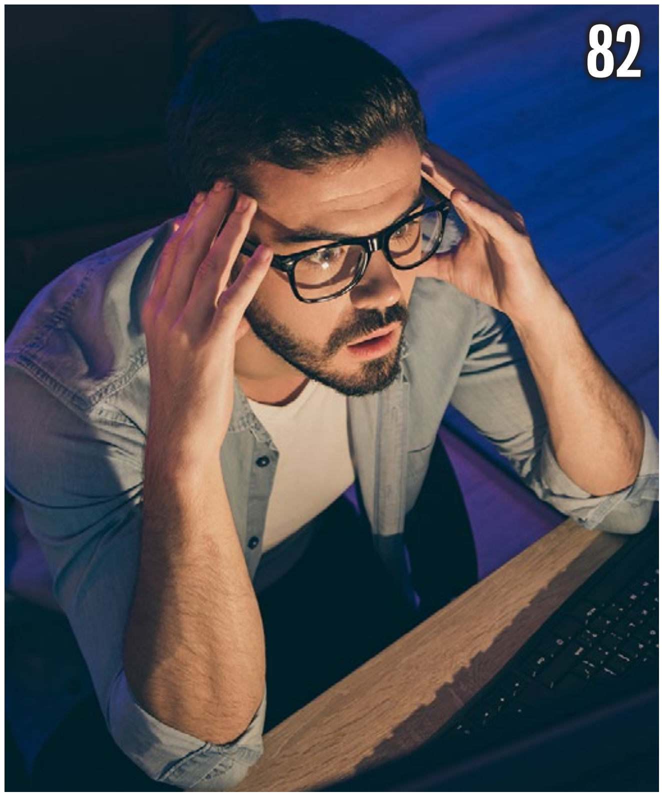


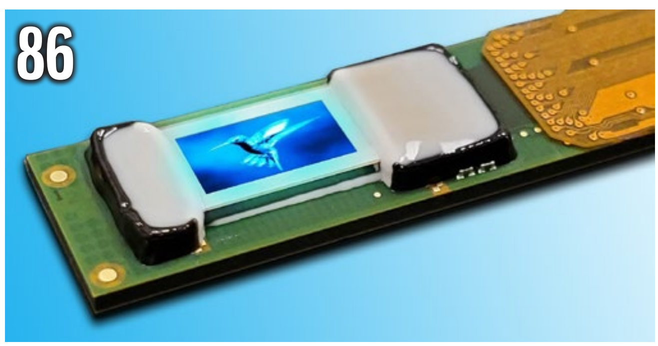
DESIGN007 MAGAZINE

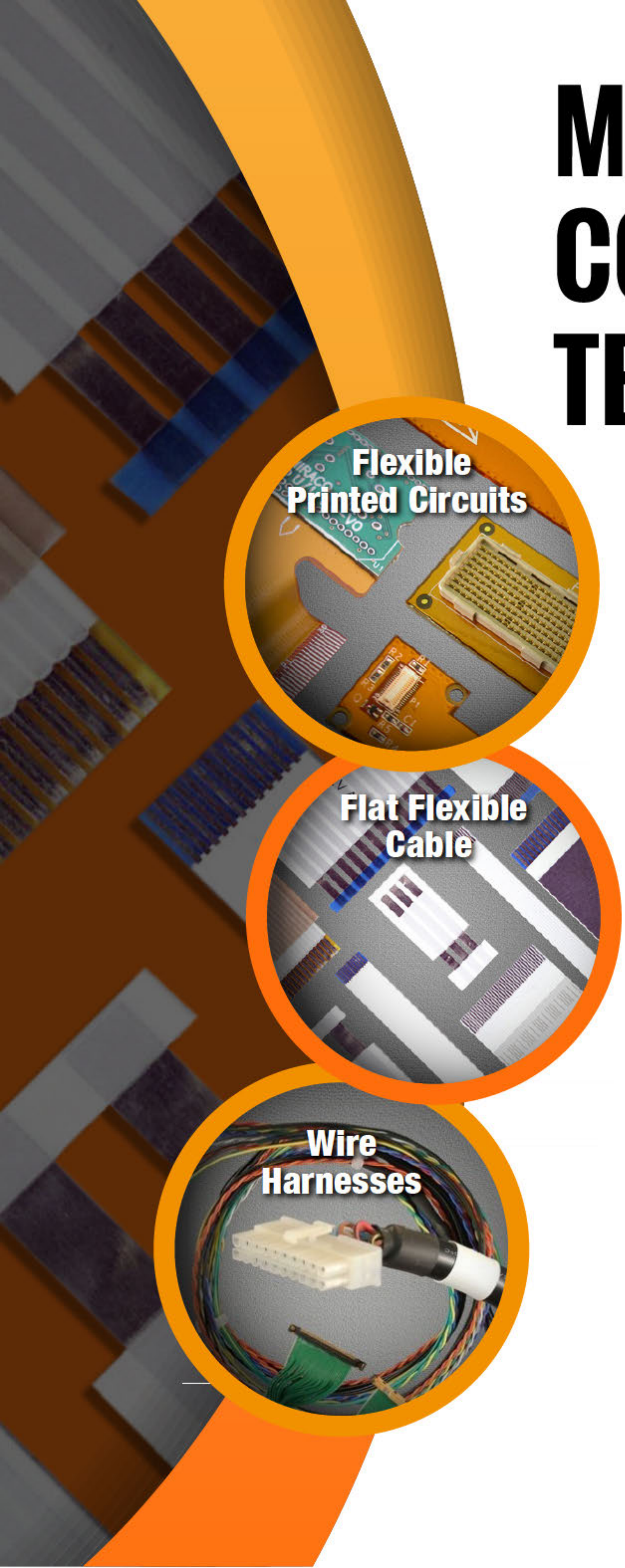
FLEX007

This Month in Flex

It's tough enough to achieve the results you expect when designing and manufacturing rigid PCBs. But flexible circuit designers face even more hurdles, and in three dimensions. This month, our flex experts discuss various ways to meet expectations upstream and downstream, and what to do when results just don't match the requirements.






Flexible Printed Circuits
Flat Flexible Cable
Wire Harnesses

Great Expectations

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

"What do you expect from me?" How many times have you found yourself asking this question during your career?

Like a lot of ideas, the concept for this topic came to me at a trade show (remember when we used to attend trade shows?) in January. I was chatting with engineers at DesignCon—EDA vendors, OEM engineers, and consultants—and they explained that many of their challenges were not technical in nature. No, their problems were often the result of unmet expectations—either on behalf of the designer, other EEs, the fabricator, the final customer, or all of the above.

I could understand if we were talking about designing wearable health monitors or another new industry, but this is a mature industry. How can anyone in PCB design and design engineering not know what is expected of them? Most designers and EEs have been doing this for 30–40 years, so there shouldn't be too many surprises in the process.

No one sets out to make mistakes, but it seems that many people involved in the PCB design process have gotten used to having unmet expectations. This holds especially true for fabricators; CAM departments have grown so accustomed to receiving incomplete or bad data packages that most of them won't even complain. They'll just fix the data errors and move on without telling the designer. Bad data is now part of their expectations.

In a recent Design007 survey, 84% of respondents said that their fabricator's primary expectation is a "complete, accurate data package." And some CAM shops will tell you that a similar percentage of incoming data packages require some kind of attention before manufacturing can begin.

Then, there are the customers' expectations to contend with. In that same survey, we asked readers how customers' expectations have changed over time. Here are a few of the replies, slightly edited for clarity:

- Today's customers are less likely to be aware of the requirements for a completed design
- Faster delivery times, even for compact designs with many high-speed signals
- Tighter, smaller designs without increased cost—seems unrealistic but it's true
- Customers require wider expertise now—HDI, high-speed, flex, etc.
- It follows the evolution of IC integration, with new packaging like BGA, BTC, and fine pitch

- Do more with less—better performance for less cost and time, with more parts on smaller boards
- We're not just connecting the dots anymore

Yes, you're not just connecting the dots anymore. Designers have to know more about engineering than ever before while doing more with less board real estate, educating the customer about PCB design, and keeping an eye on cost. That sounds like a daunting task, doesn't it?

We asked our expert contributors to discuss meeting expectations at every level of the design cycle. We start with an interview with Dana Korf of Korf Consulting, the former chief PCB technologist for Huawei. He describes what every fabricator expects from a designer—a complete data package that can be used to build a board right the first time. Mentor's Todd Westerhoff describes a variety of reasons that expectations are often not met, starting with a poor definition of requirements.

Tim Haag explains why designers should not let "negative expectations" affect their thought process, and why expectations must be reasonable and unambiguous. Next, Martyn Gaudion outlines the need for bidirectional communication between designers and fabricators. Albert Gaines of HiGain Design explains why some fabricators are nearly shocked to receive a perfect data package and why fabrication and assembly providers should provide designers feedback when there is a problem.

We also bring you columns from our regular contributors Kelly Dack, Vern Solberg, Phil Kinner, Barry Olney, Bob Tise and Matt Stevenson, and Joe Fjelstad. Plus, we introduce a new column—"Design Circuit"—written by Patrick Crawford of IPC. Patrick offers an update on the new organization IPC Design, as well as plans for an international design competition next year.

Ah, next year. Will 2021 mark a return to normal? I'm glad to see so many of the companies in our industry building ventilators and doing everything they can to help beat this thing while keeping their employees safe and healthy. We'll continue to bring you COVID-19 updates from the electronics industry. See you next month! **DESIGN007**

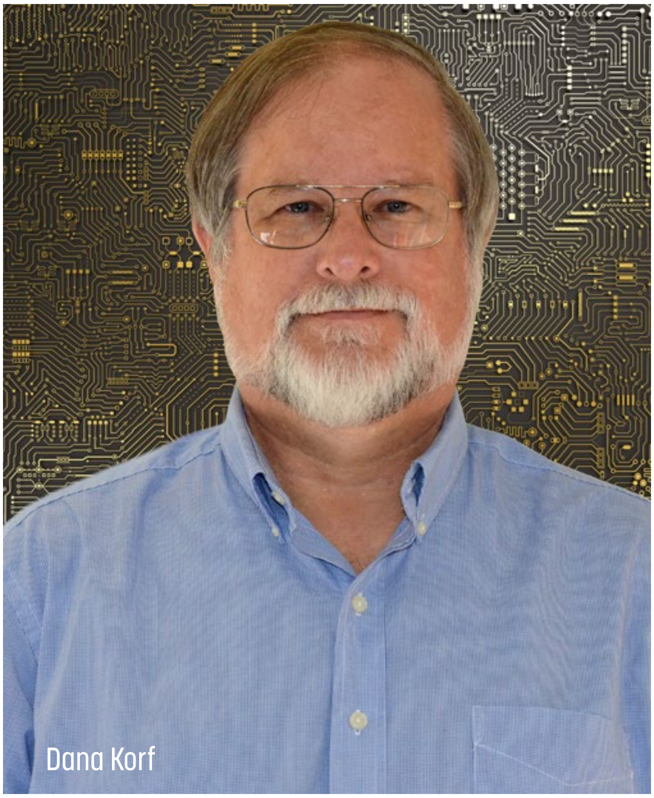
Andy Shaughnessy is managing editor of *Design007 Magazine*. He has been covering PCB design for 19 years. He can be reached by clicking here.

Realtime Robotics Unveils Realtime Controller

Realtime Robotics, the inventor of responsive motion planning for industrial robots and autonomous vehicles, has launched its Realtime Controller. The solution dramatically reduces and simplifies the programming required to safely integrate robotic workcells, speeding up the time to deployment.

As the pressure intensifies to improve margins, it will accelerate the adoption of robotic automation. However, integrating robotic operating systems currently is complex, time-consuming, and cost-prohibitive, which has limited where and how the technology is deployed. With the Realtime Controller, multi-robot cells are much more flexible, as it automates many core processes and can dynamically adjust to variable production conditions.

Using Realtime's Controller, manufacturers can now quickly and easily plan, simulate, and validate automation through the entire deployment. The Realtime Controller connects with the customer's PLC and robot controller so that they can autonomously calculate, communicate, and execute collision-free motions. Both development time and cycle time are significantly reduced with automated interlocks and interference zone-free multi-robot workcells enabling robots to be deployed more quickly and expand into new areas that were previously cost-prohibitive. (Source: Business Wire)



Dana Korf

Dana Korf: What Fabricators Expect From Designers

Feature Interview by the I-Connect007 Editorial Team

Andy Shaughnessy and Barry Matties spoke with Dana Korf, former chief PCB technologist for Huawei and currently principal consultant of Korf Consultancy, about the breakdown in communication between manufacturers and designers. Dana discusses exactly what a fabricator expects from a PCB designer, why these expectations are often not met, and the need for designers to make mistakes so that they can learn from them.

Andy Shaughnessy: Dana, we're going to talk about expectations and the problems with unmet expectations. Can you give everybody a quick rundown of your background?

Dana Korf: I started out post-Vietnam War in the mini-computer industry where we were trying to displace mainframe computers; it was said that you couldn't do it, but at the end of a few years, our machine came out, and we could beat them in some benchmarks. Then, PCs came out and displaced that industry. At that time, I was working my first PCB in the fab shop, which was a six-layer board and hand layout with tape with Mylar paper two times the size. Over the years, I worked in fabrication on design for a telecommunications company. I co-founded Interconnect Technology Inc. (ITI) in the early '80s with some people from Tektronix, where we had a service bureau set up doing the laser blind via drilling PCB layout, fabrication and surface-mount assembly. Unfortunately, the company didn't make it, but I made a lot of friends in various groups that scattered out around the industry, and some of them are still around.

After that, I went into telecom design and worked on one of the first fiber-to-the-home distribution systems, and then I went back to the board fabrication industry, where I've been there pretty much the last 25+ years. I've been involved in high-end circuit board manufacturing, run R&D, and was involved with buried capacitance technology when it first came out. It was a radical idea that enhanced the existing decoupling capacitors with higher-frequency plane-to-plane capacitance, which was critical for signal integrity and power dis-

tribution. We marketed it for many years then it finally took off. That was with Zycon, then we got bought by HADCO and subsequently by Sanmina, so we had a lot of board shops around the world. I was responsible for the front-end engineering for all the shops globally, where the key challenge was to be able to transfer designs from North America to Europe and Asia; that was difficult because the materials and plant capabilities were not always the same.

Then, I got intrigued by an offer to be the chief PCB technologist for Huawei in Shenzhen, China, which resulted from a conference call. They asked me to talk to them about high-speed design for 30 minutes. I had to ask them about what part; I could talk for hours on that. At the end of the phone call, they said, "Are you going to come work for us in China?" I laughed because I was in California. Two days later, they called me back, "Are you?" I said, "Oh, you're serious." A few weeks later, I moved over there, and that was a great experience. My focus was on developing the designs and PCB fabrication supply chain for 5G technology with a continual focus on cost reduction.

After my contract expired, I went to work for Multek in Zhuhai China for a few more years, doing the same thing—front-end engineering with a lot of focus on trying to improve design quality, which goes both ways, and we'll discuss that later. Then, last year it was time to leave them, return to the U.S. and semi-retire. Now, I'm in Bremerton, Washington. I formed Korf Consultancy, and my challenge is to improve design transfer from design to manufacturing. I want to get to where there are absolutely no issues. It can be done. I've done it on spot occasions with various companies by working on both ends of the candle and try and help fix up the issues. That's my end game job. I'll keep running with this until I can't anymore.

Shaughnessy: That ties right in with this topic. With your manufacturing hat on, what do you expect from the designer, or what should manufacturers expect from the designers?

Dana spent almost eight years leading engineering and DFM teams at Huawei and Multek in China.

Korf: I expect the designer to send me a file to build, and I can build it as they sent it to me. I can compensate for etching and drill hole sizes, etc., but I expect the design to be correct. I shouldn't have to determine if they designed it right. Did they meet their own specifications? You send me a Gerber plus an ODB++ data, and they don't match, so which one is right? The fabrication print is different than the graphical file, and the design violates their own specifications. I break these issues out into three areas. One is the design doesn't match the capability of the supplier; we can talk about that extensively. The second is the design violates its own specifications. For example, they require four mils between layers, while they send the design in with two. The third is conflicting data and requirements in the data package.

In the '60s, some board fabricator must have built a board that didn't work because they didn't find the designer's error, and they offered to pay for it. They said, "I'm sorry. I'll build it for you for free." That's when all this started. We got into a culture of, "Get the design close, throw it over to manufacturing, and they'll find all of our capability issues and fix it for free. We don't have to worry about it." That's kind of the standard process today. Get it close, and throw it to them because they'll do it for free since it's so competitive. I expect to have a design to come in that I don't have to modify. I hated having front-end engineering/CAM people modify a customer's design as they might mess it up accidentally.

And I know that for board layout—especially for a poor layout person who is laying that board out—it's not like they have to just worry about DFM. They have schedule, SI constraints, schematics, timing, thermal, cost, and mechanical constraints. There are lots of constraints involved when you lay the board out, and because I've had quite a few designers work for/with me over the years, I appreciate how hard their job is and how the software tools aren't up to speed enough to allow them to do all this automatically.

Shaughnessy: Right. And it's funny because the designers will say, "Don't mess with my design," but almost every CAM person I talk to says that, unless it's a long-time customer, they're going to have to do some tweaking. It's just accepted.

Korf: It is accepted to a certain point, and there are various levels of that; it's interesting. You look at the broad industry overall, and this has been true in North America 20+ years ago and very true in Asia; someone will get a design in, edit it, and not tell the customer they changed something. They build it, send it back, and the customer goes through all the testing—including development and software testing—and it works. Then, they send the design out to a higher volume shop that has more financial risk for it if it doesn't work, and they come back with a long list of questions. A lot of times, the designer responds, "My last supplier didn't have that issue." One of my favorite responses was, "This is a law of physics problem. They couldn't have built the board if they didn't fix this. It wouldn't have worked." I guarantee you they modified X, Y, and Z to make it work.

Another example: A customer with a via fill board wanted a 100% via fill guarantee. I told him, "That is physically not possible." Number one, it's not possible because there are always air bubbles when you print any kind of ink, cure it, and the bubble gets trapped in the material. Number two, how do you even inspect it? You can't inspect the inside of every hole. They said, "My vendors never had the issue." And I said, "Go ahead, give me a board. Let me cross-section one via on a board randomly from one of your current vendors." About three months later, it was an automotive application, and he came back and said, "What should the specification be?" He went and tested some boards and found that they weren't 100% either. But they were working just fine. Sometimes, there are laws of physics that are real.

One big problem is, and I wrote a column about this a few months ago, that board shops consider design rules proprietary. They don't want to tell the designer what the rules are because they will know all my processes, and it's secret. At one company, they wouldn't let me give the designers our design rules. I said, "How do you expect us to spend less time in quick-turn if you don't tell them what we want?" They finally let me do it, and the designs got better. We have to break this mentality of design rules being kept secret and not giving them to layout people. We should tell them everything that we can do—not how we do it, but what we can do.

Shaughnessy: I'm wondering how this thing has gotten so askew because most of these designers have been doing it for 30–40 years.

Korf: People have various forms of running DFM during design, post-placement, and whatever various stages of the layout process, but the problem is if you don't know what the rules are, you're always going to be wrong. You're always going to have at least one issue, probably. This is why it's a two-ended problem. It's a CAD tool problem; it's a sourcing problem. You see a lot of designs where a person lays the board out, a CM will do the assembly, and they'll pick the fabricator based on whatever their criteria are. The designer has no clue who's going to build it, let alone what the rules are. A lot of times, the CM will impose constraints on top of the design that the designer didn't have and add another layer of complexity to the problem.

To your point, over the years, my fabrication pre-engineers have "DFM-ed" hundreds of thousands of part numbers—probably more than a million part numbers over my career. Some of these would be for a company they've been building for for over 10 years, and they're still coming back bad. Why? "It takes too long to update the libraries." "It takes too long to update the rules and get them all approved." "We have three vendors, and we can't get them all to agree," which is false. I've been through many successful standardization exercises on that one. There are lots of excuses why they can't do it.

Another generic issue which is kind of interesting: Say I get revision A of a board. We go back and forth and resolve all the issues during the DFM process. Then, we all agree on the design changes we're going to have to do as the fabricator. When revision B comes in, the issues are still there. A lot of companies don't generally go back and update their designs with the issues so that revision B comes out clean. It's a standard practice of board fabricators to run the DFM review again on revision B, compare that against the down rev, and you'll see all the same issues—maybe a few more, maybe a few less. One of the TQs that's sent back on revision B is, "Can I do all the edits I did on revision A that you approved? Here's what you approved." It's kind of silly.

Barry Matties: What's the incentive for the designer to send complete and accurate documentation to a fabricator?

Korf: I'll give you another good example of why your question is valid. I had a large customer request a pre-laid out, pre-released DFM request. The Gerber data had a round hole in it, and the fab print had a slot. My DFM engineer sent back a TQ, questioning, "Which one do you want? Do you want the slot, or do you want the hole?" Unfortunately, the customer replied in the same email with two attachments—one reply was the hole, the other reply was the slot. Two different answers in the same email. We filed the response away because we weren't going to build the board for a while. An order comes in a few months later with the same design documentation error still there. Unfortunately, the engineer selected the hole; they didn't notice there were two answers. We built the board, and it came back saying that there was supposed to be a slot, not a hole. They said, "How could you possibly do that?" I said, "We asked you which one it was, and you said both of them were okay."

We responded that we should have caught it and got back to you, but it's your design mistake; your documentation was wrong. I wrote a corrective action back to the customers saying the root cause was their design documentation. Again, it was a large company, and I wrote to them that the root cause in the report that their documentation was wrong. Oh boy, everyone freaked out on that one because, in Asia, you don't tell your customers they're wrong—especially big ones. We argued over that. We paid for the mistake, even though they were the root cause. As I mentioned earlier, in the '60s, some fabricator decided to rebuild a board for free because they didn't catch the customer's error, and that became standard practice.

Matties: How do you incentivize it? A fabricator could say, "We're going to give you X% discount if you send us the complete and accurate data package, and we're going to conversely try to do X% more if you don't."

Korf: I've tried that in a couple of ways. One, we'll charge you X% if you don't send me a good one. They went, "What? The other suppliers don't add money, sorry, we're gone." They won't come back.

Matties: Is that necessarily a bad thing?

Korf: It is in bad times. There are two approaches that have worked in the past. One is, I've presented to small companies, and the VP of engineering is sitting in there, and I say, "You pay me more money to do a three-day turn, but the cycle time starts after the approval of the TQs. We both agree what the final layouts are, then the clock starts. If we take three days to go back and forth to get all the answers right, you are paying me three days of money for a six-day turn. They say, "Oh, that's interesting. It's actually a six-day turn from our standpoint." The other way is we've worked with companies that don't want to have issues because they understand it's a cycle time impact. Their management wrote it into their reviews saying, "Part of your KPI is to reduce TQs by X% every year." This way, they had some skin in the game also. That has worked very well.

I also worked with a consumer company. We went from 20-some TQs per part number—and these were little tiny thumbnail-sized boards—down to about two or three. It was a tremendous improvement. We were getting near zero, and then, all of sudden, we're back up to 15 or 20 again. I flew to the U.S., sat down with them, and said, "What happened?" They looked to each other and said, "Oh, we went from our in-house design to subcontracting design. We forgot to tell them what the rules are." They made an unintentional mistake, then corrected it.

Matties: You keyed in on an important aspect of this: time. We did that issue on design economics, looking at the cost of design. The cost of re-spins is incredibly expensive, and it seems to me that people would be highly motivated to avoid that at almost any cost—the old adage, "There's no time to do it right but plenty of time to do it over."

Korf: I've done this—and what I'm going to say is not a negative opinion—but there are shops that all they do is quick turn, meaning 1–3 days, for instance; get it in and get it out. By default, some of them don't have time to do the back and forth with you. The order must be on the floor in an hour. They will go ahead and look at it and say, "I'll yield enough to get four pieces out. I'll put four panels in the process, and if we get one panel to work, we're okay. It's factored into our pricing." The other approach is, "We'll change it to make it work and not tell them. We don't have time." The customer gets a board back, and it's different than what they sent, but they weren't notified that it was modified. Then, when they go to the volume shop, and it is thoroughly reviewed, the same issues are brought up this time for resolution. But the customer says, "Wait a second. Our prototype shop didn't have that problem." Actually, they did, or they changed it, and you didn't know and couldn't tell during testing. Sometimes, it's just a capability mismatch between the proto and volume plants.

Matties: What's built in prototype will often not work in production.

Korf: The yields may be too low and won't meet the production cost goals, etc. I would re-state it that they may not work. It depends on how close the design is to the fabricator capability.

Matties: Or material supply is perhaps different as well.

Korf: Yes, I had a customer that went through full FDA approval based on a prototype shop they used in the U.S. They transferred it to Asia, and we said, "We can't get that material in Asia because they don't ship that here." We got it for them. They paid an arm and a leg for that board because we had to ship to Asia because no one uses it there. They didn't know that. It wasn't their fault.

Shaughnessy: Designers say that sometimes they don't know where it's going to be prototyped, much less where it's going to be built in volume.

Korf: Sometimes, purchasing waits to the last second to pick a supplier based on delivery, cost, etc. They may have three, four, or even five vendors that they send that technology to a lot. I call it the plus or minus one mil rule; everyone is within a mil of each other. Get a compromise set of rules, I've done this multiple times, including stackups. Everyone has a different way they do stackups, but if a couple of companies can make it, the other companies can too. They haven't done it that way before, or it's not the normal way they do it, but they can build it. Part of the problem is decoupling the supply chain from the designer. How are they going to know what rules to use? They don't.

Dana at Tiananmen Square in Beijing.

Matties: When it comes to the designer, what should the expectation be?

Korf: You send me a data package, and I build it as is. I can do my manufacturing, etch compensations, and that kind of editing, but I shouldn't have to run a netlist. Why should I? Your data should be good.

Matties: The expectation is complete, accurate, and correct.

Korf: Right.

Matties: Is there an expectation that they have a basic understanding of process knowledge?

Korf: That's a very good question. If I'm designing a board that hits the middle of the technology of the fabricator I'm using—their sweet spot—it shouldn't matter how they build it. If I know I'm running at the edge of the technology and there are some unique feature attributes, or I have to do something special in a process, I may want to know how they do it only because if something goes wrong on that board, I want to know what didn't work. Generally, designers shouldn't have to worry about how it's done. Like I told my front-end people, you shouldn't have to worry about how they design. There are a lot of factors. Why did they do that? I don't know, and it doesn't matter—just do what they said.

Matties: And then there are those "Oh, you did it just as I asked" moments where you think, "What was I thinking?"

Korf: I worked through this with a large OEM. They were doing their first 5G boards many years ago. They were doing the anti-skew trace routing. You put a differential pair on, and you take one leg and snake it around to compensate for the skew. I told them that section is not a transmission line anymore. "Let's look at your TDR here, see those little bumps? That's where you snaked the trace. That's not a transmission line, and it's not 50 ohms anymore. This one says 90 ohms, but it's 40 here and 60 here." I've seen an impedance trace over a power plane, and suddenly, there was a gap in that power plane for some isolation reason. The trace runs over a plane, over a gap, and back to the plane. That gap, now the reference plane, is two layers down lower, and the impedance went up. You're going to ding me if the whole trace isn't 50 ohms. You didn't design it to be 50 ohms; you designed it to be 100 ohms for two inches. Why am I paying for that design mistake?

Matties: The designers' expectation is that you're the backstop to quality check to their work.

Korf: It's their assumption versus their expectation. I don't know if there's a designer out there that designs a board intentionally wrong;

that's my firm belief. I'm sure there is, but most of them don't come to work, thinking, "I'm going to make this one unmanufacturable." They don't know. It's a lack of communication.

Matties: There are finally some fresh new designers coming into the industry who don't have much experience. What should we expect from them if they don't have the experience?

Korf: I once made a presentation to the top technical/political person in Asia. He asked, "What do you think about these engineers here?" I said, "They're highly trained, very young, and very smart, but they haven't made enough mistakes yet to know what's wrong." He was in a chair, and he looked up at me and said, "Sir, you're absolutely right. This is why we need experienced people like yourself here to help us because we don't have the experience yet." A new designer comes in and doesn't know what kind of mistakes they're going to make because they haven't made them yet. The training doesn't have time to cover the kind of issues you might run into.

Matties: Is there an advantage for a supplier, a fabricator, or an assembler to be able to identify the level of experience that this design is coming from?

Korf: It's funny. Sometimes, a design would come in, and I'd say, "Is this designer 22 years old?" They'd say, "How did you know?" I'd say, "Because they specified a 5-ohm impedance (laughs)."

Matties: There's a lot of reliance on tools; that reliance is growing, and the demand for more features to support them is certainly there.

Korf: I've seen designers with 30 years of experience or engineers that lay their own boards out and always build it at their U.S. board shop that now have been forced to go to Asia. The capability is different, and changes are required. The response to the proposed modifications is, "No, you can't do that." "I've always done it that way." The rules in the tools need to match manufacturing capability.

Matties: That might be the other flag—the ones of 30 years (laughs).

Korf: I can be stubborn at times.

Matties: We all can. Now, you brought up a good point. You mentioned DFM, SI, power distribution, and all these disciplines that a designer needs to be knowledgeable in. There's quite a bit of weight or expectation is too great that they would know all these disciplines. Part of the problem is there is an expectation that they should know or will know it, and it's overwhelming.

Korf: Yes. To your point, the designers should have various levels. There are people designing a single-layer board, consumer products with no SI or anything—just safety just punch it out. Then, there's the person who's designing a router board, a medical device, or something mission-critical. There are radically different skill levels and knowledge required because they have to worry about more things as they move up that skill chain/ladder. I don't think that the industry pays them in such a way, or they get acknowledged or certified in such ways. It's so obvious when a layout is not performed by the appropriately skilled person.

Matties: That goes to another point of expectation. We talk about manufacturing process knowledge, but there are also a couple of barriers—communication with material suppliers before taking on the design and communication with your fabricator. It seems to me that you often hear that there needs to be more communication, and there is a level of expectation there, but oftentimes that communication does not exist.

Korf: Over the years, material suppliers have done a very good job of getting into the designers' camp—spending a lot of time with OEMs getting their specific resin designed in, which is a very good business model, and it has worked very successfully for them. Issues will arise when the designer picks a specific vendor and resin, and they move it to a shop,

Dana checks out the pandas.

say in China, and they say to use vendor X, but the manufacturer uses vendor Y's material, which could be the equivalent. The price goes up because they have to special order the material for special deliveries, and they don't have good contract pricing or have a validated process for it, yet.

I hate using the phrase 5G, but 5G designs are getting us into more loss-sensitive designs; changing FR-4 resins can mess up the design. And it's not obvious to a CAM person, quoting person, or the layout person when they're changing the resin, and changing the loss tangent a little bit can throw the design out of whack. It's getting a lot more complicated now, even from the CAM side, as to what you can or cannot change. I was teaching teams the types of designs to watch out for, and the ones you don't even ask to change certain features or specifications because you just can't. It's so tightly designed, you can't change it.

Matties: If you were to go out and hire a designer, what would be the most important attribute in your search?

Korf: Good visualization is important because you have to visualize what you're going to do before you start it. They would also have to be very adaptable, able to handle a lot of inputs at one time, and able to figure out what the best thing is to do; there's a lot to consider. There are SI, power, safety, and fabrication rules, and they have to balance all that to tell the engineer what they can do. They almost need to be, in my opinion, an engineer skill level type of person—not just a drafting person, per se, from a talent standpoint.

Shaughnessy: Do you have any final suggestions or recommendations for designers?

Korf: For new designers, use your available resources early on. A lot of board shops have field engineers who are happy to come to your site and work with you out there. They're happy to do stackups before it gets finalized or look at a preliminary lay out, maybe layout a few traces, and they can see what your routing strategy is. There are a lot of material suppliers and fabricators who are more than willing to help you out before you get too far down the design path. Use them; don't be afraid to because there's no stupid question.

Shaughnessy: I'm surprised that so much of this works out when we hear about all the potential pitfalls. There are so many steps to fabricate a simple board. Isn't it about 148 different decisions? It's amazing that it works at all.

Korf: As I mentioned earlier, I worked at a minicomputer company back in the '70s. One summer, I went down to Disney World, where our computers were controlling the Magic Mountain ride. Our engineering joke was we got twice the thrill as the average person because we knew what could go wrong (laughs).

Matties: That's a great quote. Dana, I always appreciate our conversations and your knowledge. It's enlightening to chat with you.

Korf: Great. Thanks for asking me. This is an area I have a lot of passion for, and I'm going to help fix this. It can be fixed. DESIGN007

What Did You Expect From Me, Anyway?

Feature by Todd Westerhoff
MENTOR, A SIEMENS BUSINESS

As engineers, we work in the middle of a (usually long) process chain. Product requirements come from the front of the chain (marketing), the products we create are physically realized at the back of the chain ("production"), and hopefully get sold to customers who enjoy them and buy more.

It's sort of like working on an intellectual assembly line—we get requirements and data as input, perform our particular task, and then provide our output as requirements and data to the next person on down the line. It seems easy enough. So, why is it that so many of the requirements we're supposed to meet and so much of the data we receive is downright bad?

To be fair, "hard data" is usually okay. Component dimensions, material properties, pinout definitions, etc., all tend to be correct because, without that, reliable manufacturing would be impossible. It's the soft stuff that tends to be the problem. In the case of signal integrity, it's the simulation models we receive from vendors or clear guidelines on just what is and isn't achievable from a board layout standpoint or manufacturing cost standpoint that make the analysis job difficult. Sorting through problems with data requires time that no one seems to have—at least, in the world as it existed three months ago—and any delay associated with vetting and reacquiring data presents a huge problem.

Example Issues

Why does this happen? While not an exhaustive list by any means, these are some of the issues I've seen.

Poor Definition of Requirements

Simulation models are a great example—even though the syntax for simulation models is well defined, there's really no good way to assess simulation model quality or fitness for a particular analysis. In my hardware design days, I noted that just about every analysis project had some simulation models show up past the period we had reserved to test them, so we just gave them a quick test and got to work instead. Without fail, we'd use the models for a week or so before problems popped

up and realized our analysis had been compromised. It was a 2–3 week hit, every time.

Sometimes the list of items to be delivered just isn't complete or clear enough, and only part of the information needed is provided. The time it takes to go back to the source and acquire the additional data (and justify why they should take the time to provide it) causes project downtime.

Domain Expertise and Lack of Global Focus

Most of us interact with experts from other domains who consult on our projects. Signal integrity specialists are a good example; they will help out on a specific part of a design. They're keenly focused on a few critical aspects, but may not see the larger picture or understand how their guidance affects it. For example, the signal integrity specialist may maintain that a low-loss dielectric is required to meet channel loss budgets without understanding that the additional cost makes the design untenable.

Sometimes the experts we consult don't understand their own process well enough. My team once contacted another signal integrity group in my company to get placement and routing rules for a custom ASIC they had analyzed for their boards. They responded by asking for our board's netlist—an odd request, but we provided the data anyway. Their response came back, "Your net names are wrong. You need to fix your schematic."

We were baffled; we had asked for layout guidelines, after all. After digging deeper, we realized that the person we were dealing with only knew how to run an automated process that took a design's netlist and added layout constraints based on a net naming convention the other group had established. The process had become so deeply ingrained within their group that they had lost sight of the bigger picture. In the end, we obtained a copy of their board database, reverse engineered the layout constraints, and proceeded with our own design.

Lack of Attention to Detail

There are lots of reasons for this—many of us are learning on the job, the people who can mentor us are usually busy, schedules are tight with too much to do, and mistakes happen. Everyone is overloaded and swapping between tasks, so consistent attention to the smallest details is tough, requiring a combination of persistence and downright stubbornness.

Let's be truthful here: when the chips are down and the hour is late, there's a huge temptation to take whatever you have, pack it up and ship it, hoping that whoever using that data down the line will be able to figure out what to do with it. "Correcting S-parameters for causality" wouldn't be such a big deal if people were fastidiously checking and correcting the data they produced in the first place.

Poor Communication

I think that we engineers routinely underestimate the value of good communication skills, believing that simply stating the facts as we see them is good enough. We don't appreciate how difficult good communication actually is and don't recognize the need to confirm that others have actually heard and understood. When others don't really hear, problems can get swept under the rug until they become too large to ignore.

Addressing These Issues

Now, what can we do about these issues? Just articulating a set of problems with no plan for improvement isn't helpful, so here's what I try to do. You'll note that none of these are simply technical; they're more about being human and technical at the same time.

Define Requirements Clearly With a Way to Verify Them

I try to give others a detailed list of what I expect as input with descriptions that are clear enough to act on and examples of what the data should look like. Ambiguity in requirements is not your friend, and I try to be as explicit as possible. I try to be just as thorough with the data I provide to others, making sure I have a complete list of what to provide and the format to provide it in.

Look for the Big Picture, Continually Asking, "What Problem Are We Trying to Solve Here?"

As engineers, we love details, but context is critical. If you can't chart the path from the top-level goal to the particular detail you're considering at the moment, you're in trouble. I try to make sure that each level of the problem hierarchy is clearly stated to ensure that whatever is consuming our collective creative energy is a problem that's really worth solving.

Practice Craftsmanship Seriously

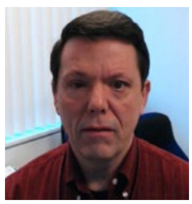
I once found an unattributed quote that said, "Craftsmanship is the quality that comes from creating with passion, care, and attention to detail." Much has been said about craftsmanship as a lost art, and I try to do what I can to bring it back by trying to understand the tools and processes I use as best I can, taking the end user's view and letting that drive the type of results I produce, and experimenting so that I can try to improve my processes. Just because a process works doesn't mean it can't be better. Learning, creating, and taking pride in my work is why I got into engineering in the first place.

Communicate, Communicate, Communicate

I've become a big fan of what I'll call "business psychology" books. I've learned (the hard way) that human communication is a discipline in its own right—one that requires study and practice, and is something that I'll never get exactly right. Simply spewing technical jargon is easy; simplifying complex concepts and communicating them in a way that brings others into the conversation is hard. One of my co-workers used to say, "You have to understand an issue really well to be able to summarize it in a few understandable sentences," and I couldn't agree more. There's a certain satisfaction in taking a complex issue and breaking it down to its simplest and most understandable form, and that's one of the things that makes me an engineer.

Conclusion

These are just a few things that we should all keep in mind, but I'd probably rank communication as the most important point of all. If we communicate with our customers and downstream partners, we'll be less likely to find ourselves asking, "What did you expect from me, anyway?" **DESIGN007**



Todd Westerhoff is the product marketing manager for high-speed design tools at Mentor, a Siemens Business.

Aldec's HES FPGA Accelerator Board Targets HPC, HFT and Prototyping

Aldec Inc., a pioneer in mixed HDL language simulation and hardware-assisted verification for FPGA and ASIC designs, has launched a new FPGA accelerator board for high performance computing (HPC), high frequency trading (HFT) applications and high speed FPGA prototyping. The HES-XCKU11P-DDR4 is a 1U form factor board featuring a Xilinx Kintex UltraScale+ FPGA, a PCIe interface and two QSFP-DD connectors (providing a total of up to 400 Gbit/s bandwidth), and which hits the ideal sweet spot between speed, logic cells, low power draw and price.

The new product, which joins Aldec's popular range of FPGA accelerators and prototyping boards, also features an FMC HPC connector for interfacing with Aldec's FMC daughter cards; the industry's widest range of FPGA mezzanine cards and the newest addition of which is the FMC-NVMe high-bandwidth, low-latency memory extension card. In addition, the HES-XCKU11P-DDR4's FMC HPC connector is compliant with the ANSI/VITA 57.1 standard and provides easy extension to similarly compliant peripherals.

"When developing this new FPGA accelerator board, our goal was to achieve an ideal compromise between performance, expandability and price, and thus help users rise to some of today's most advanced computing and/or networking challenges using FPGA acceleration," said Zibi Zalewski, General Manager of Aldec's Hardware Division.

(Source: Business Wire)

Navigating Industry Expectations

Tim's Takeaways

Feature Column by Tim Haag, FIRST PAGE SAGE

In case you didn't know, columns like this are usually written well in advance of when the magazine is published. Of course, there are times where something important gets rushed at the last minute in order to make it into the next issue. ("Hold the presses! We're remaking page one.") Normally, though, writing and publishing are carefully planned around precise schedules that must be followed. Even though you are reading this sometime in the middle of May or later, I am writing this during the first days of April while we are still observing the shelter-in-place restrictions due to the coronavirus.

I hope that by the time this column is published, the restrictions on gathering in public have been lifted, and life is returning back to normal. But, for now, everyone is doing their best to limit their contact with each other. This means that a lot of people across the world—especially in the PCB design industry—are working from home for the first time in their careers.

If this is the position that you find yourself in, and you are looking for some ideas on how to work successfully from home, I shared some recommendations in a recent column that you may find helpful. Thankfully, our technology today is better prepared for remote offices than ever before, and many EDA tool vendors and suppliers are offering work-from-home options to help their customers as well. Who would have guessed, though, that when we planned a theme for this month's edition of exploring industry expectations that the biggest expectation would end up being, "Stay home!" But that's the hand that we've been dealt.

Thankfully, everyone is doing what they can to help keep each other safe. Soon, I hope the

expectation of "stay at home" will turn around to become "get back to work," and that will be a welcome relief for everyone. There are a lot of other expectations, however, that designers deal with regularly, too. While some expectations are normal—and, well, expected—in the workplace, there are also those that do more harm than good. Do any of these three sound familiar to you?

1. **Schedules:** Not having enough time to get the job done, sliding delivery dates, or changes without corresponding schedule adjustments top the list. These are just some of the frustrations that we experience with schedule expectations, but there are plenty of others.

2. **Deliverables:** If your job is to lay out a PCB, it's pretty obvious what the overall expectation is. However, there are many details of the layout that can get lost in the shuffle, such as manufacturing drawings, output file requirements, and internal documentation. How these details are to be completed and who is responsible for their content and acceptability often gets changed without advanced notice, and yet it is the designer that is often held accountable.

3. **Processes:** A team member or a vendor may change, but you aren't notified, and you end up waiting for information that never comes. You're expecting answers while your boss is expecting results, and everyone ends up being disappointed.

This probably won't come as a big surprise, but the frustration over unmet expectations like this is common not only in our industry but everywhere. A recent informal survey conducted by our editors at DesignCon revealed that users and vendors alike often voiced the same concern: "I never know what is expected from me."

I have found that a lot of these problems can be resolved by working at improving communication with our co-workers, managers, and customers. I was careful to use the word "working" in that last sentence because good and effective communication rarely happens organically; it usually takes a lot of planning and effort to set up mechanisms that will promote good communication, but the results are well worth it.

Schedules, deliverables, and processes can be better managed, and everyone can take a big step backward from that cliff edge of uncertainty and anxiety over wondering just exactly what is expected of them. The next logical question is, "How do we do improve our communication in the workplace?" Here are three ideas that might help:

1. **Document the workflow:** By outlining the basic process steps and who the key reporting and decision-makers are, you can quickly help team members to know exactly what is expected of them. Obviously, you can't document every single tiny detail, but by giving everyone a clear path to follow, they will be less likely to get lost along the way.

2. **Multiple reviews in the workflow:** A lot of confusion around expectations occurs because of design changes. Changes will happen, of course, but the key is to catch them as early as possible. With regularly scheduled reviews and check-ins during the design process, you can hopefully avoid full redesigns because of a component change that should have been caught during placement.

3. **Include team members in the process:** Unmet expectations often happen because someone didn't feel empowered to ask a simple question. This can happen when team members don't feel like they are part of the process, so get them involved. By encouraging participation and feedback, your team members will feel more ownership of the process and will be more likely to get fully involved.

Many years ago, as a PCB designer for a large electronics manufacturer, I was stunned when on my first day of work, I was pulled into an impromptu meeting about a problem with the mechanical housing of one of our products. Since I was brand new to the company and didn't even have computer access yet, I couldn't figure out why I was being asked about a product that I had never seen before. I guess that they just wanted a fresh pair of eyes on it, but for me, the effect went much further beyond simply giving my opinion. I realized that I was in a work culture that wanted and encouraged my participation. This sent a very clear message to me that I had value, and the company's expectations of me were to be an active part of the corporate process and workflow. That moment still lingers in my memory as being a pivotal point in my growth as a designer and becoming part of a productive team.

The lesson here is that these kinds of expectations—to grow, ask questions, and be part of the team—result in freeing up people to operate at their best and take the initiative in their careers. On the other hand, expectations set around unrealistic goals do just the opposite; they tear down a person's confidence and self-esteem, making it much more difficult for them to be willing to take the steps and risks needed to expand and grow in their professional careers. It is safe to say that negative expectations—whether they are unobtainable, unrealistic, or ambiguous—should be avoided at all costs. We shouldn't put them on anyone else, nor should we accept them from others. It's just better for business that way. The next question, though, is, "Are we putting negative expectations like that on ourselves without realizing it?"

Just as being blamed by a co-worker for not meeting their ambiguous or unrealistic expectations can create stress, you may be doing the same thing to yourself without knowing it. There are so many different ways that we can set ourselves up for this negative expectation trap that it would be impossible to list them all here. Instead, let's turn this around and look at it from the other direction. Here are five ideas on what kind of positive expectations that we could set for ourselves:

1. **Trust your qualifications:** It can be easy to slip into doubt, especially when the going gets tough. Remember, though, that you were hired for your job because of what you can do, and you should expect yourself to live up to those qualities.
2. **Own your job:** Don't be content to merely do your job; own it. Make sure that you know your responsibilities in and out and expect yourself to succeed.
3. **Stay on target:** Be careful about becoming lackadaisical in what you do. Set expectations for yourself to maintain a consistent level of performance and excellence.
4. **Forgive yourself:** No matter what, you will probably make some mistakes along the way. It can be easy to dwell on those problems and derail your motivation and momentum. Set an expectation that you will forgive yourself in order to learn from those mistakes and move on.
5. **Believe in yourself:** I hate to sound like a greeting card here, but believing and expecting that you will succeed and perservering is an important foundation for any successful career.

Conclusion

We all have expectations in our jobs that have to be met for success, and more than likely, we are going to encounter ambiguous and unrealistic expectations as well. But we can choose how to respond to these by communicating with those we work with and setting clearly defined objectives and goals. Don't let negative expectations drive you. Instead, grab the wheel firmly and take control over those expectations so that you can drive them. Stay safe, everyone, and keep on designing. **DESIGN007**

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

Life During COVID-19 'Wartime'

The Digital Layout

by Kelly Dack, CIT, CID+, PCEA

Introduction

This month, I'll share updates on the progress the PCEA staff is making toward establishing a web presence and report on what the PCEA is preparing in support of the PCB engineering community.

Though PCEA chapter activities have been limited because of restrictions due to COVID-19, PCEA Chairman Steph Chavez offers his perspective on the challenges of staying connected without using traditional means. Lastly, I share our most updated list of professional development and event opportunities, although some may be affected by the COVID-19 outbreak. Stay tuned for more updates.

PCEA Updates

"This ain't no party, this ain't no disco, this ain't no foolin' around."—"Life During Wartime," Talking Heads

As we physically isolate ourselves to prevent the spread of COVID-19, we know that our work cannot stop. Many still have day jobs to tend to; you might be working from home, or you might be required to mask-up and join your co-workers on the front lines of an essential business. Our industry's needs and domestic responsibilities have not changed, but how we process them has. Adaptation is a key to those of us working at the PCEA to build a new organization. Now, more than ever, we are relying on the internet and online meetings to evolve our ideas and act on them.

The PCEA Website

As we approach a target release date for the PCEA website, I joined an online meeting from my basement office to review content for the site. As I donned my ear pods and plugged in my cam-

era, I couldn't get the tune "Life During Wartime" by the Talking Heads out of my head. As each invitee checked in on the screen, I felt an appreciation for the depth of experience that even this small portion of PCEA members brought to the table. Vice Chairman Mike Creeden led the meeting, and I quickly discerned that this group was not here to party, disco, or fool around; they were meeting to execute and accomplish. They were armed with cameras, mics, and headsets, using the best means possible to share their ideas for planning to serve the PCB engineering community with a viable website.

Starting any effort, such as developing a new website, can be a daunting task. An important purpose of our site is to establish a base platform to continue our growth as an organization. Most of us think we know what needs to be basically communicated on a website, but section headers and click logic for a site like ours requires knowledge of electronics and industry topics and resources, as well as experience in psychology, etc. The PCEA is blessed to have a master web communicator like Altium's Judy Warner on board.

Keep an eye out for the rollout of our new website set for around May 15 at www.pce-a.org (Figure 1). At the site, you will see that the PCEA is a non-profit organization and can read our mission statement and raison d'etre. Dive deeper, and you will find information on our international engineering network chapters, including seven new ones that will be established soon. There is a feedback page, which allows you get onto our mailing list and join us, as well as an education area, which will help you to fulfill your professional development by offering many resources; there are too many to list here, but they are all too valuable to pass up!

Moving Forward

Writing on behalf of all of the PCB chapter leadership, we hope you are all doing well. This has been quite a month, and the isolation

Figure 1: The PCEA's new website will be up and running soon.

challenges affecting printed circuit engineering and our local chapters are real. I've read from some of our chapter leadership about postponements of chapter meetings and some stagnation of activities due to our present shelter-in-place conditions.

Speaking with Steph Chavez, we discussed the realities of how our chapter members are being challenged. While some are working from home, some may have no work at all. There are a lot of unique conditions that the COVID-19 restrictions have cast upon our community. We have asked our chapter leaders for feedback or stories from our members, which we would like to share with you in the coming months. As the PCEA continues to organize, we will find our niche for helping our members as they "hunker down" with their own unique local conditions. We are actively forming means by which our members and the entire PCB engineering communities can use this time to tap into resources that will help them prepare for when the world reopens.

Share Your Stories and Photos

In the meantime, we are looking for heartfelt stories from any of our readership who would like to provide personal stories with regard to how your PCB engineering, manufacturing, or connected job has been affected and how you are coping and adapting. Considering your personal accounts might help all of us to adjust our focus on how we can help.

We would love to read any content or reports you have to share, which we may include in this column. Photos? Heck yes! We'll take all of the isolation photos of you performing your PC engineering tasks that we can print.

Please send your stories and photos to kelly.dack.pcea@gmail.com.

Message From the Chairman

by Stephen Chavez, MIT, CID+

CHAIRMAN OF THE PCEA

The COVID-19 pandemic has reshaped our daily lives and work environments. Most of us are working remotely or in isolation in one form or another these days. The industry as a whole is getting hit hard. Our country and the world are going through some tough times. Many feel like they're going stir crazy from staying in our homes almost 24/7 or with very limited outside social interactions due to "social gathering restrictions" in response to fighting the spread of this virus.

I, too, feel cooped up. My only outside interactions these days is that not-so-frequent trip to the local grocery store. Wearing a face mask in publicly populated areas is not something I have gotten used to yet. Part of our communication as humans is facial expressions. I feel that wearing these face masks makes our public interactions with one another a bit cold and impersonal. It takes away that simple gesture—a smile—that we usually give without hesitation to others. Sometimes, a simple smile can make someone's day.

I miss in-person, face-to-face chats that always included a handshake or a hug because I am definitely a people person. This isolation may be tough, but I know it is helping. Please join me in sending prayers to all our first responders who are going above and beyond to help and keep us safe. THANK YOU! God bless you and keep you safe.

The negative impact of this pandemic is being felt all over. From many business closures, cutbacks, furloughs, and layoffs unfolding across the board throughout the globe, these are tough times we are going through. But during these tough times, we do what we always do: We adapt and overcome.

Now, virtual meetings—such as WebEx, GoToMeeting, and Zoom, just to mention a few—seem to be our way of life. It was once thought that only certain professions used these types of tools for remote collaboration; today, these tools are almost as common as cellphones. Almost everyone is adapting and using these tools. It has gotten to a point where we now have kids in grade schools using them daily to continue their education.

With domestic and international travel at an extreme low, many have taken to this virtual lifestyle. I, like many others, find myself jumping from one online meeting to another, and in some instances, actually participating in two meetings going on at the same time. I have never used my laptop and cellphones as much as I do now. I live on them!

However, I'm not alone in doing this. There are so many others who have adapted to this new "normal" as well, such as members of the PCEA Executive Board. With most of us on this board having day jobs, a lot of passion, hard work, dedication, and extra effort are going into getting this right. Things are moving fast, to say the least.

We continue to solidify the foundation of the PCEA. Our website is a work in progress with a new and improved version ready this spring. The team working on this is doing an outstanding job. I'm truly looking forward to its official release. The PCEA business structure is also coming together as planned. We should be ready for an official virtual open house in the near future.

As always, I highly encourage you to get involved. Join the PCEA by visiting our website and registering as a member to become part of the PCEA collective. Reach out to me (stephen.chavez.pcea@gmail.com) or Kelly Dack (kelly.dack.pcea@gmail.com) to get more information, as well.

Professional Development and Events

It has been our custom to highlight all up-and-coming industry events to look out for in 2020. We will continue this; however, with the challenges brought upon our industry by the COVID-19 outbreak, we can only remain hopeful that these events will not be affected. If you have an interest in any of these events, please search and contact the event coordinators directly for the latest updates.

- June 9–10: PCB2Day—**SMT Assembly Boot Camp** (Austin, Texas)—***Postponed***
- June 11–12: PCB2Day—**Design Essentials for PCB Engineers** (Austin, Texas)—***Postponed***
- June 14–20: **IPC SummerCom 2020** (Raleigh, North Carolina)—***Canceled***
- June 22–25: **Realize LIVE 2020** (Las Vegas, Nevada)—***Virtual***
- September 8–11: **PCB West** (Santa Clara, California)
- September 16–17: **Del Mar Electronics & Manufacturing Show** (San Diego, California)
- October 7–9: **AltiumLive 2020** (San Diego, California)
- November 11: **PCB Carolina** (Raleigh, North Carolina)
- May 11–13, 2021: **IPC High-Reliability Forum 2021** (Baltimore, Maryland)

Spread the word. If you have a significant electronics industry event that you would like to announce, please send me the details at kelly.dack.pcea@gmail.com, and we will consider adding it to the list.

Conclusion

No matter what your part is in this industry, let's continue to move forward together by finding solutions for the unknown and having hope for the future. DESIGN007

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

Communicating Materials From PCB Design to Fabrication

The Pulse
Feature Column by Martyn Gaudion, POLAR INSTRUMENTS

Designer and fabricator communication—especially for high-speed PCBs—should be a bi-directional "thing." It is so easy for a designer to say, "Just build this," and hand over a challenging design to a fabricator who could have performed better with some preliminary conversation or dialog before placing the order.

Materials always matter when it comes to PCB fabrication; however, as designs progress and industry changes and morphs, the PCB materials or specific material characteristics that impact the day-to-day lives of designers, fabricators, procurement specialists, and PCB technologists change. Characteristics that could safely be ignored in the past may now creep up and trip you over—catching you unaware—and yet other characteristics that may have been critical on legacy products may only have a second-order influence on your PCB specification and the chances of it performing as expected.

Depending on design requirements, a designer has a choice to entrust material specification to their fabricator or to lock down the materials far more rigidly before ordering or perhaps somewhere in between where only critical layers are deployed with mandated materials (Figures 1 and 2).

Important in guiding you through this are the specialists in the PCB base material supply industry, which has transformed from an industry with only a selected handful of material choices to a whole range of suppliers providing a complete gamut of materials with a range of purpose. This adds complexity and perhaps confusion as designers, fabricators, procurement specialists, and technologists work together to select appropriate base materials. Most material suppliers will have OEM teams whose aim is to guide you towards the correct base material or family of materials.

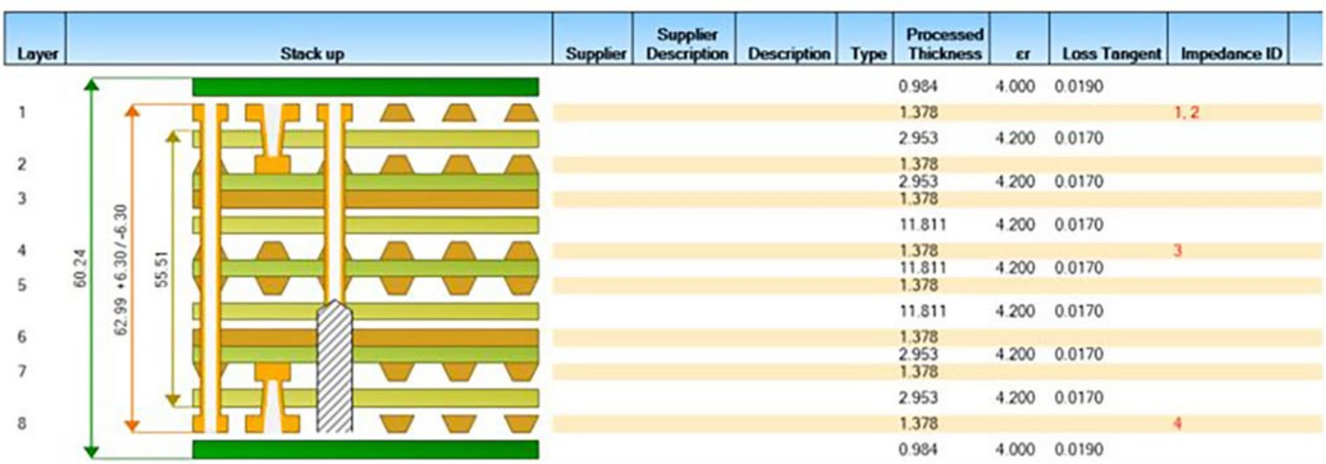
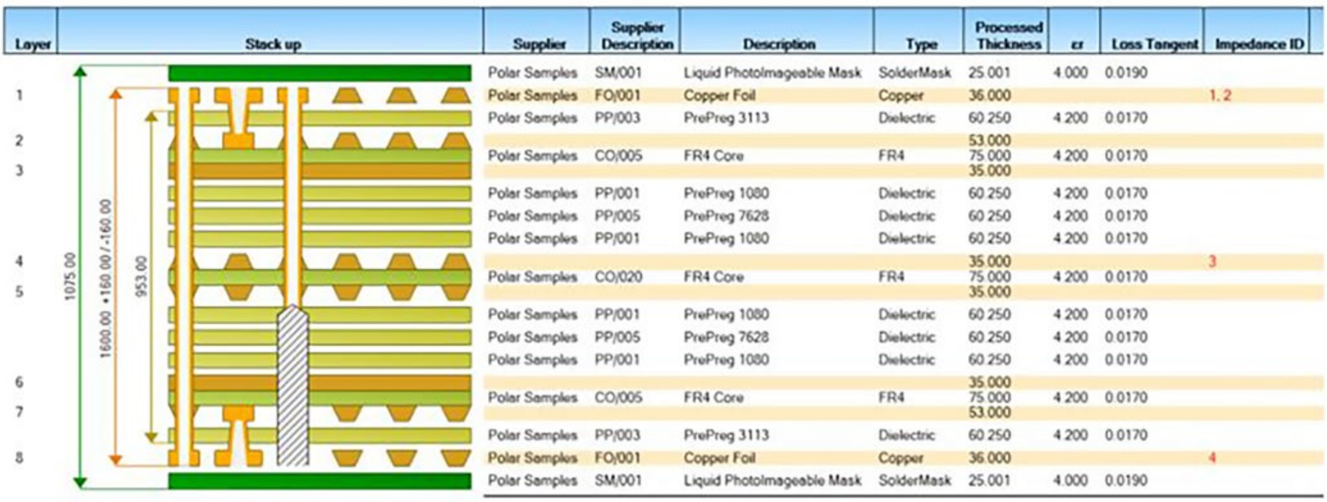


Figure 1: Virtual or generic materials (Report: Polar Speedstack).



Layer	Stack up	Supplier	Supplier Description	Description	Type	Processed Thickness	εr	Loss Tangent	Impedance ID
		Polar Samples	SM/001	Liquid PhotoImageable Mask	SolderMask	25.001	4.000	0.0190	
1		Polar Samples	FO/001	Copper Foil	Copper	36.000			1, 2
		Polar Samples	PP/003	PrePreg 3113	Dielectric	60.250	4.200	0.0170	
2						53.000			
		Polar Samples	CO/005	FR4 Core	FR4	75.000	4.200	0.0170	
3						35.000			
		Polar Samples	PP/001	PrePreg 1080	Dielectric	60.250	4.200	0.0170	
		Polar Samples	PP/005	PrePreg 7628	Dielectric	60.250	4.200	0.0170	
		Polar Samples	PP/001	PrePreg 1080	Dielectric	60.250	4.200	0.0170	
4						35.000			3
		Polar Samples	CO/020	FR4 Core	FR4	75.000	4.200	0.0170	
5						35.000			
		Polar Samples	PP/001	PrePreg 1080	Dielectric	60.250	4.200	0.0170	
		Polar Samples	PP/005	PrePreg 7628	Dielectric	60.250	4.200	0.0170	
		Polar Samples	PP/001	PrePreg 1080	Dielectric	60.250	4.200	0.0170	
6						35.000			
		Polar Samples	CO/005	FR4 Core	FR4	75.000	4.200	0.0170	
7						53.000			
		Polar Samples	PP/003	PrePreg 3113	Dielectric	60.250	4.200	0.0170	
8		Polar Samples	FO/001	Copper Foil	Copper	36.000			4
		Polar Samples	SM/001	Liquid PhotoImageable Mask	SolderMask	25.001	4.000	0.0190	

Figure 2: Mandated materials (Report: Polar Speedstack).

Choosing Materials

Knowing what you need is always a good place to start. The choices seem endless. High Tg or low Tg? Halogen-free? High speed? Controlled impedance? Is insertion loss critical? Rough or smooth copper? Reliability and susceptibility to thermal cycling and thermal stress? Expected lifetime of the product? Will the PCB go in a short-lived consumer product, a satellite or undersea cable, or automotive with harsh environments and be highly-price sensitive but with a lifespan midway between mobile phone and aerospace applications? Will the PCB be laser drilled? Sequential laminated or HDI and subject to repeated press cycles? Once designed, will the board be produced in low quantities by one fabricator, or will it ramp up and be sourced from a volume fabricator in another country? Maybe the OEM will handle procurement directly or—as often happens—the procurement will be outsourced to a specialist broker. What happens to the carefully crafted materials specification as the specification wings its way across the supply chain?

Cost is almost always there to intervene, and getting the PCB specification correct for the specific application—neither over- or under-specified—is important if you wish to minimise the cost for a specific application.

Conversations with technical buyers often raise the following question: "When I send out a PCB specification for quotation or purchase PCBs from multiple suppliers, why is the construction of the stackup different from the same stack specification?" In more than one case, a technical buyer expressed frustration that the same PCB repeatedly purchased from the same supplier was realised with different material stackups. The question is, "Does this matter?" And the answer is sometimes yes, sometimes no, and sometimes maybe; however, the more the procurement team knows of the ultimate application, the better they are able to specify the PCB to an appropriate level.

Looking at two extremes of PCB specification, here are some examples of the requirements, from simple to very demanding, that can be asked of a design. At the lowest end, the PCB could simply be a carrier for low-power LEDs for an indicator in machine status lights; the PCB may be wired into the machine, and the PCB fitted with screws to a backplate. Thickness isn't critical—neither is power handling—and the environmental stress on the board could be pretty benign, too. Likewise, the volume produced may be low, so, really, all that matters is that the board be the correct X and Y dimensions, have holes in the correct place, and the correct data imaged on the appropriate layer. It might need to be lead-free and halogen-free to comply with environmental regulations and minimal specifications required.

At the highest end, a large aerospace PCB will need to be built by a fabricator with all the correct approvals and likely from material which is certified for the job, too. The layer stackup will be tightly controlled, and any changes will have to be thoroughly investigated and signed off by the appropriate parties.

In between these two extremes, there is far more space for PCB technologists and procurement teams to make intelligent decisions to gain the best commercial advantage, and this does not necessarily mean just a price negotiation. A well-specified PCB will help a fabricator get a better yield with appropriately priced materials, there will be less waste in the process, and everyone wins. Good communication can allow businesses to work together to maximise profits and to minimise scrap and waste. Here's a sample checklist:

1. Speed: Low, medium, high, or ultra-high
2. Volume: Low, medium, or high
3. PCB size: Small or large
4. Traces: Short, long, or high-speed (Shorter traces make life easier for high-speed signals for a given material type.)
5. Layer count
6. Sequential lamination
7. HDI
8. Back-drilled
9. End-use: Consumer, industrial, automotive, or aerospace
10. Single supplier or multiple sourced
11. Laser or conventional drilled
12. Environmental/regulatory requirements

Taking all of the above into account, you may choose to band the type of specification you provide from a low-end "layer count/thickness" to a medium "layer count/thickness/IPC slash sheet material description" to a high medium "layer count/thickness/generic material family/impedance requirements" to a high "layer count, specific build, material vendor, glass style, specific base material, foil type—treatment type" depending on how "nailed down" you need the specification to be for the specific application.

Understanding Materials

Material is fascinating, and the more you understand, the more you can work with the characteristics rather than fighting with them. In previous decades, rarely did designers have to concern themselves with such matters as glass style and resin content, but here is an example of just how this has changed.

Let's take the case of spread glass cloth. You may be aware of a property called fibre weave effect, which frustrates high-speed designers. This stems from the fact that glass cloth and epoxy resin have widely different electrical properties.

Park that in your mind and consider something unrelated and from a completely different arena: laser drilling. When drill companies developed laser drilling for PCBs the composite nature of base materials limited the precision of laser drilling techniques. You can imagine why. You are trying to accurately ablate holes in a material, which is a composite of relatively easy to remove polymer and woven e-glass cloth (Figure 3)—not the easiest of materials to ablate. It must be like mechanically

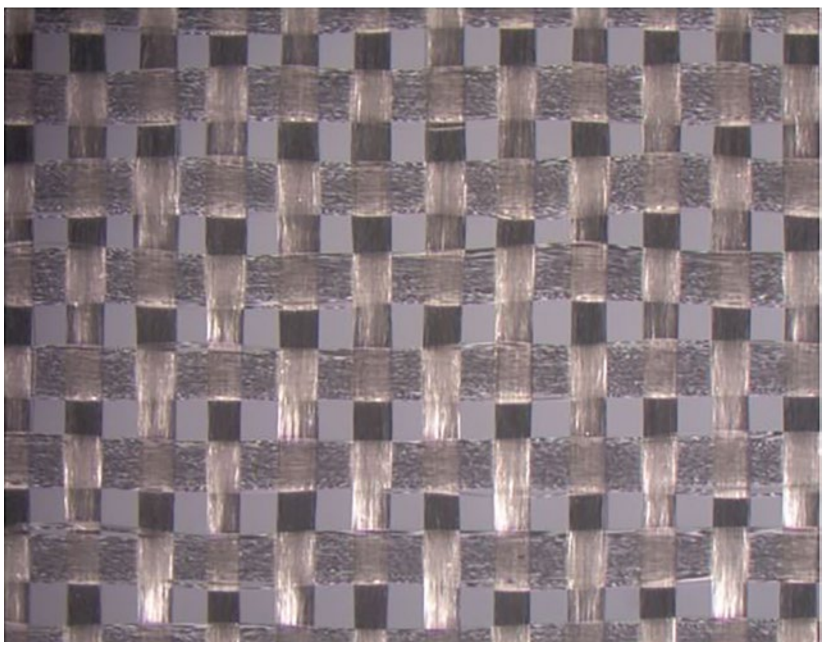


Figure 3: Traditional glass cloth/epoxy resin weave. (Source: Ventec International Group)

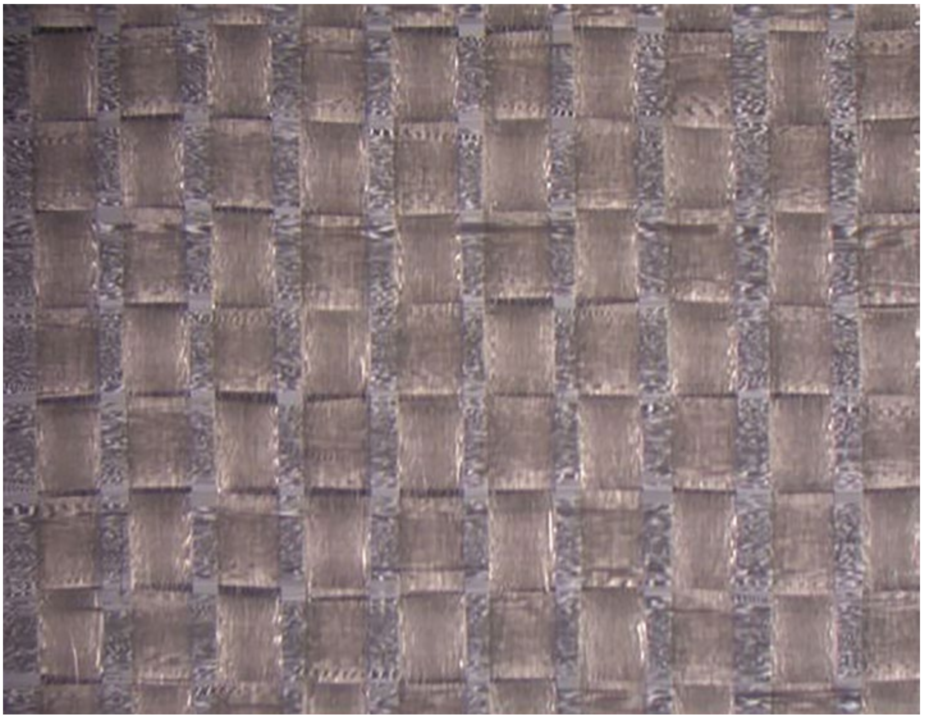


Figure 4: Flattened glass cloth/epoxy resin weave. (Source: Ventec International Group)

drilling into a block wall with steel rebar inserts. The lack of consistency would give even the best mechanical drill a tough time.

On the PCB front, the material suppliers innovated by developing flat or spread glass cloth; this modified the traditional warp and woof (weft in the U.K.) of the fibre and mostly eliminated the gaps in the cloth where the threads cross over (Figure 4).

This is achieved by mechanical or water jet blasting sometimes combined with threads that are "low twist" (i.e., the glass yarn has fewer twists per metre, making it easier to flatten out). This step-function improvement in materials for laser drilling has a hidden side effect. The more even lay of the glass also helps to eliminate the variations of dielectric constant that used to be experienced by signals transiting along traces, which ran predominantly over fibre or predominantly over resin. The win for drilling presented SI engineers with a win for signal integrity too. (There is a "but" coming, which isn't a problem, but is something you should be aware of.)

A third benefit of flat glass is that the cloth lies flatter and gives overall more control over finished thickness and thickness variation (here comes the "but"), but a benefit of the "old school" glass styles with the apertures between the crossover of the warp and woof of the cloth is that the apertures allow the easy passage of resin from one side of the cloth to the other—so more of the total resin content on top and below the cloth could flow from one side or the other into the gaps between the traces to ensure a reliable lamination process and minimal risk of resin starvation.

For the same resin content, flat glass may prevent the resin flowing across the glass boundary, so before you change glass styles, you should consult with your material supplier and take advice on the resin content requirements for successful lamination with flat glass. This is not a problem with the material; it simply means that when using the newer material styles, you need to ensure you understand what questions you need to ask to make sure your material supplier can help you specify the optimum build materials.

Having a material and stackup discussion before submitting the design or—even better—before starting to lay out the board is critical for complex designs. Moving stackup design as early in the process as possible gives both designers and fabricators time to discuss the trade-offs and benefits of different material choices. **DESIGN007**

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

Albert Gaines: Fabricators Almost Expect Bad Data


Albert Gaines

Feature Interview by Andy Shaughnessy
I-CONNECT007

Albert Gaines is the founder and senior design at HiGain Design Services in metro Atlanta. I usually chat with Albert at SMTA Atlanta each spring, but with all trade shows canceled because of the COVID-19 outbreak, we decided to conduct a virtual interview.

I asked Albert to discuss what fabricators and customers expect from a designer and vice versa, and what can happen when expectations are not met. Much of this problem can be precluded with good communication on all fronts. As Albert once told his kids, "They can't take away your birthday for asking."

Andy Shaughnessy: I'm glad to hear you're doing well in this strange time, Albert. As a design services provider, what do your customers expect from you? Do you provide them a checklist at the beginning?

Albert Gaines: Let me answer the last question first. When I opened HiGain Design Services, I did provide a checklist/questionnaire to my clients before starting a new project. I found I spent more time explaining the questions on the form than the information I received. I now have a phone call with the client to ask what their end-product is and then walk through my perceived requirements for the product that they are designing.

Most of my clients do not know about a board's technology from a fabrication standpoint. They do not understand board classes, board finishes, or requirements and processes of their fabrication/assembly teams. Their expectation from me is to fill in the gaps and provide a layout, including all the data for that board to be fabricated and assembled. They want a board ready for initial testing.

Shaughnessy: How often do you have to educate the customer about PCB design and manufacturing?

Gaines: Most of my clients do not know what they do not know. That's fine; that is what I am here for. I have trained many engineers over the years. They have taught me a lot in return. We all need to be brave enough to ask questions. I used to tell our children that if you do not know, ask. I told them, "They can't take away your birthday for asking."

I tell my clients that the only stupid question is the one not asked. We need more sharing of


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

knowledge across our industry. Things are better now with the use of the internet to find information—good and bad, depending on the application.

I have a client who I have worked with for many years. The engineering manager and I go back to the early 1980s. He has called me many times to make me aware of a new project with a new engineer that I would be working with. I jokingly say that I will train them well. I have started giving the new engineers who I help with their first layout a certificate stating their accomplishment. That may sound kind of silly to some, but that is a milestone they will remember forever; hopefully, they have learned something along the journey. I am willing to teach those that have a desire to learn. Maybe that is an issue in itself. Do those people designing a board really have a desire to learn about the process, or do they just want a board to start testing?

Shaughnessy: What do fabricators expect from you?

Gaines: Enough information to build a board accurately. I do not think board fabrication/assembly shops spend enough effort on telling their clients what their expectations are. They are just making it happen. Maybe if they billed their clients every time a question is raised, it would get someone's attention!

I believe someone is absorbing the cost somewhere. I have learned over the years what they may want data-wise, so I just output the same data for every job, whether it is used or not: Gerber, ODB++, pick-and-place, test reports, PDFs, etc.

Shaughnessy: What do you expect from customers and fabricators?

Gaines: I am not sure "expect" is the correct word here. Maybe "wish" is more appropriate. I wish all of my customers were more aware of the processes that are required to build a board. Therefore, when I need to discuss design options, I could discuss them without having to explain everything. I generally work directly with the project engineer. They already have so many hats they are wearing that expecting them to be informed on the board fabrication/assembly issues may be too much to ask. That is where a good layout designer comes in to guide and share knowledge.

I wish fabricators/assemblers would pass back to the engineer/designer any issues at the end of the job. This is an opportunity for knowledge to pass back to those who can learn from the previous job. It really is a pain when you get ready to move toward production and find there may be an issue that the fabricators or assemblers have worked around. Maybe they contacted someone and got an approval on an item, but that got lost—or they assumed it was fixed—and never made it back to the layout designer.

Shaughnessy: You came up in a drafting environment. Do you think that experience contributes to you being a better designer, more detail-oriented, etc.?

Gaines: Yes. Again, as an engineering team, we are creating documents to send to another team so that they can build what we desire. Our product is documentation, not a prototype in the lab.

All boards should have a part number and a revision code. That part number goes on the fabrication/drill drawing and is the number that represents that board in a parts list or a purchase order. The drawing should contain information needed to build that board using notes, reference specifications, charts, and dimensions. This drawing should be able to drive a quality inspection of the board/part. Basically, if it is something that could be in question, it should be answered with this drawing.

Our designs are nothing but details put together into a bigger detail. I do believe to be a good designer you need to be very detail-oriented. Layout designers are the link between the imagined and the producible.

Shaughnessy: Give us some examples of what happened when expectations were not met.

Gaines: I get few if any questions on our designs. As I stated before, I have learned what they may want and send them everything initially. I do not want to be contacted about missing data any more than they want to take the time to ask and delay a job.

I provide a folder structure broken into subfolders that indicate who the files are for: one for bare board fabrication, one for assembly, and then others for PDFs of the drawings. There can be delays in schedules or even an incorrect product delivered because of unknown or incorrect expectations. Most expectations are not met by people because they did not know what was expected. Ask your board shop and assemblers what they need. Make a list and add things as needed.

Shaughnessy: You said you recently worked with a fabricator who couldn't believe your complex data package was accurate and complete. They were used to being let down. How did people in this industry get so accustomed to being disappointed?

Gaines: Over the years, they have been desensitized to the point that they just make it happen. I can imagine that a CAM person has learned by name whose job will have an automatic delay, with questions to be asked. These questions have to impact someone's schedule. Some shops have learned to set the delivery clock to start after all issues are resolved.

In the dusty past, we had someone responsible for documentation in an engineering environment. In today's world, the circuit engineer has become the layout designer, parts engineer, purchaser, and anything else that can be put on them. These great design minds are being put into a world totally new to some. People have learned that they may not be getting the best information and to adapt and figure it out.

Shaughnessy: What advice would you give a new designer just entering the field today, as far as making sure he or she met expectations?

Gaines: Do your best to learn the expectations. Ask questions! Ask any possible fabricator/assembler what their expectations are. Make a list and follow it to send the same data out with every job. Also, as you grow in your job, come up with a list of what you expect from those supporting your efforts so you can quickly convey your expectations to others.

Shaughnessy: Thanks for the insight, Albert. Stay safe.

Gaines: You too, Andy. Thanks for the opportunity. **DESIGN007**

Panel-level Semiconductor Package Design Challenges

Designers Notebook

by Vern Solberg, CONSULTANT

Semiconductor package specialists continually work to improve high-volume manufacturing process efficiencies while reducing manufacturing costs. A majority of the commercial semiconductors are built-up on the surface of a circular-shaped silicon wafer with metalized terminal features at their perimeter to accommodate wire-bond interface with a lead-frame or package substrate. To enable direct, face-down mounting to a package substrate or host PCB, the aluminum wire-bond sites on these products must first be processed to provide an alloy (typically copper) that will accept traditional solder ball or bump termination features.

When die have a perimeter terminal spacing that is considered too close for efficient circuit routing, it is common practice to employ an additive metalization process on the active surface of the die to provide a conductive interface from the wire-bond lands at the die elements perimeter to a wider-spaced and more uniform array configured terminal pattern. Commonly referred to as fan-in, wafer-level package (WLP), the process for adding a metalized redistribution layer (RDL) connecting the original wire-bond lands to an array-configured terminal feature is accomplished while the die elements remain in the original silicon-based wafer format (Figure 1).

While the majority of the semiconductors selected for face-down interconnect can utilize the RDL processes noted, an expanding number of small-outline, very-high I/O, silicon-based processors are really not suited for conventional fan-in RDL processing.

Fan-out, Wafer-level Packaging (FOWLP)

The most practical solution for mounting and interconnecting very-high I/O, small-outline die to a substrate or PCB is to expand the terminal pattern outward. The process commonly utilizes a prepared silicon wafer base with metalized RDL designed to mount and interconnect the higher density die element to an array-configured terminal feature is outside the perimeter of the die. Metalization provided on the silicon wafer base redistributes the terminal features on each die to a pattern of plated micro-via holes that interfaces with metalized

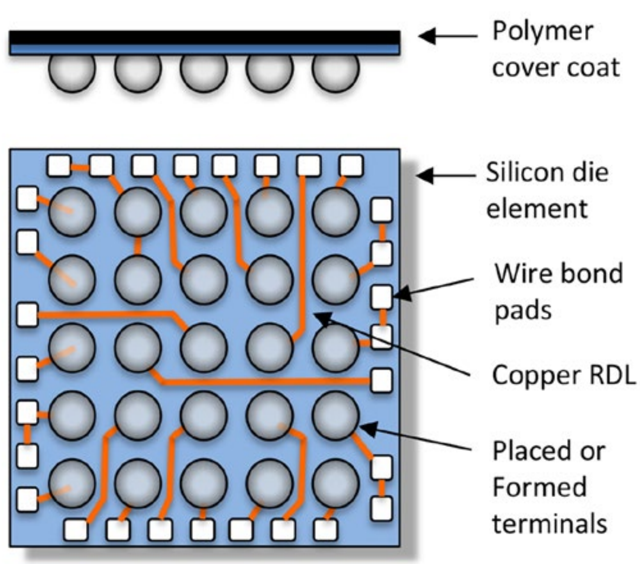


Figure 1: Fan-in with RDL WLP.

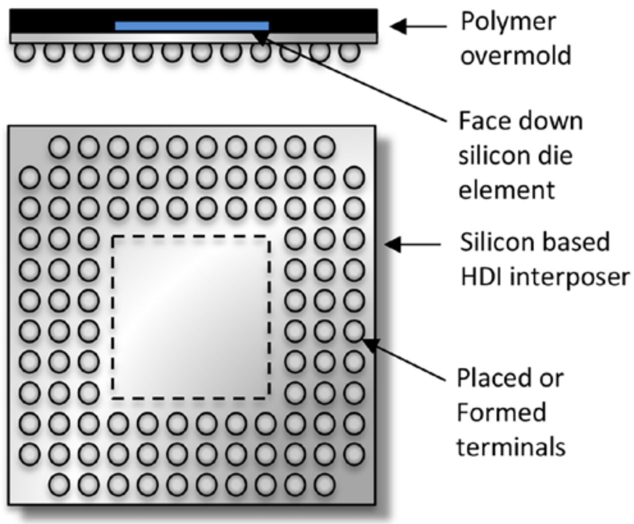


Figure 2: FOWLP (die-on-silicon).

land pattern features on the opposite surface of the wafer (Figure 2).

The die elements terminated on the wafer's upper surface are frequently encased with a polymer mold compound followed by solder ball or bump terminal formation, marking, and singulation to enable unit-level electrical testing.

Fan-out, Panel-level Packaging (FOPLP)

The semiconductor package development specialists are always striving to find solutions for improving manufacturing efficiency and trim manufacturing costs. Although the FOWLP process has proved robust and reliable, the cost associated with silicon-based interposer fabrication has been a primary roadblock. In the effort to trim overall packaging expense, a number of alternative packaging methodologies have emerged. Both independently and through consortia between academia and industry, several viable solutions have evolved that provide the same fan-out interface without the need for a silicon wafer as a base.

Panel-level package development utilizing alternative, lower-cost base materials actually began prior to 2016 to address several high volume market segments. By 2019, four prominent supply sources in Asia—Powertech, Samsung, and Nepes—had already achieved volume-manufacturing capability followed by ASE Group beginning production in early 2020. The timetable for additional offshore and domestic companies' availability of FOPLP in volume is forecast for 2021 and 2022.

FOWLP vs. FOPLP

When compared to the silicon-based FOWLP, developers implementing the panel-configured process have realized significant cost savings, greater process efficiency, and the economies of scale. Equating the high material cost of silicon wafers to the significantly lower-cost panel material is a key issue, but the greater die population potential for panel-level processing has proved most beneficial.

In regard to establishing standards for the basic panel structure, several manufacturers and material supplier members of SEMI (Semiconductor Equipment and Materials International) 3D Packaging and Integration Technical Committee have developed SEMI 3D20-0719, which is a specification for panel characteristics for panel-level packaging (PLP) applications. The organization's position is that "standards increase industry efficiency by reducing or eliminating duplication of efforts, help to define new markets, and promote competition by lowering barriers to entry."

The purpose and scope for the SEMI PLP specification were written to include four process-compliant base materials for carrier panels and establish standard panel dimensions. Revision A of the document (currently submitted for member approval) establishes two standard panel sizes: 510 mm x 515 mm (~ 20" x 20.3") and 600 mm x 600 mm (~ 23.6" x 23.6"). The four optional base materials noted for carrier panel preparation are glass, silicon, ceramic, and metal; however, silicon and ceramic materials have a significantly higher material cost.

The semiconductor-packaging specialists will select the material and panel size that will best meet their particular assembly, molding system, and plating process capability. In regard to panel size, the half-panel may be more suitable for processing within the commercial circuit board manufacturing environment while a quarter-panel size will likely be most compatible with semiconductor wafer processing specialists currently utilizing systems configured to handle the 300 mm silicon wafer format.

In a presentation by Fraunhofer IZM [1] at a recent International Electronics Commission (IEC) standards meeting in Japan, the speaker noted several key drivers supporting FOPLP technology:

1. Reduced overall package form factor
2. A thinner package profile
3. Improved electrical performance
4. Enhanced thermal management
5. The potential for greater component integration and design flexibility

The packaging process variations described in the Fraunhofer presentation primarily focused on a mold-first procedure where the singulated die elements are placed, with the active surface face-up or face-down, onto the carrier panel that is pre-coated with an adhesive.

Mold-first FOPLP

Following die placement, the populated carrier panel is overmolded with a reinforced polymer compound, fully encasing all die elements. The material selected for the carrier panel must closely match the CTE of the silicon-based die elements (2.3 ppm/K) to minimize the occurrence of die shift during the mold cure process,

- Face-down FOPLP: After mold cure, the encased die elements are separated from the carrier panel exposing the active surface of the die elements. Surface metalization follows with pattern plating from the die terminal sites to a fan-out terminal pattern (Figure 3a).
- Face-up FOPLP: The overmolded die elements mounted in the face-up orientation (Figure 3b) is somewhat more complex because it requires the removal of the polymer mold material to expose the terminal features for surface metalization and circuit pattern plating typical of that described above.

The final process sequence for both process variations includes terminal formation (typically a solder-compatible alloy ball or bump), marking, saw or laser singulation, and package-level electrical test.

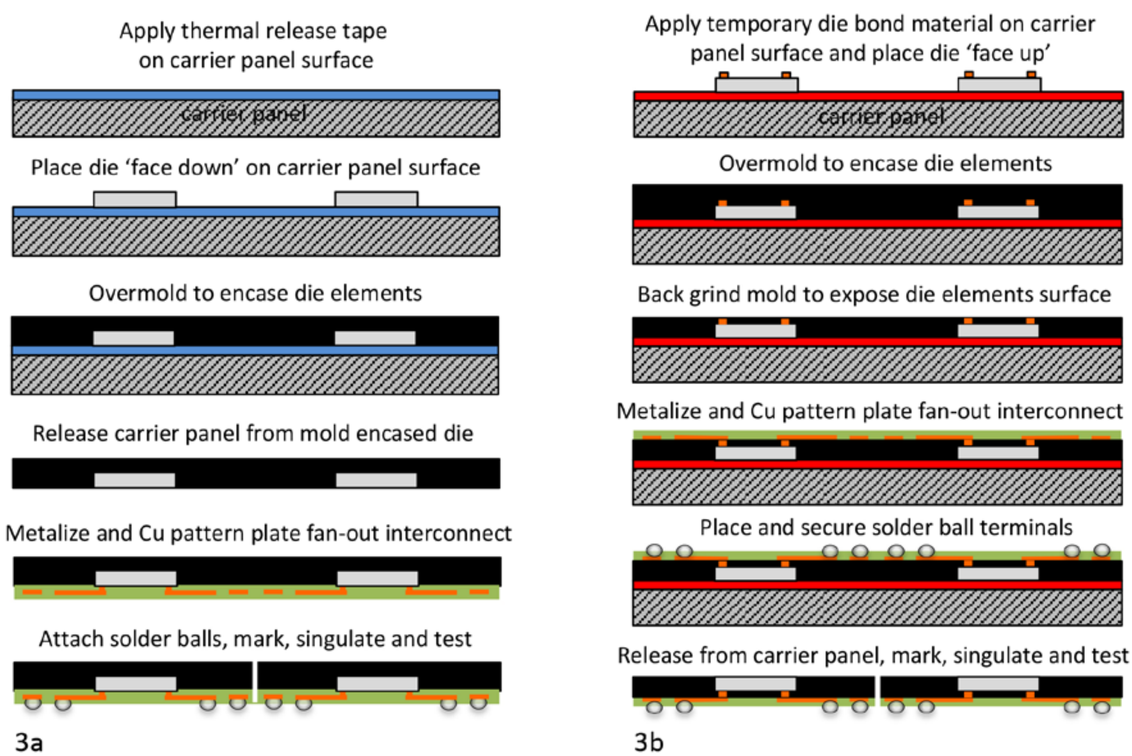


Figure 3a: Face-down, mold-first FOPLP assembly sequence. 3b: Face-up, mold-first FOPLP assembly sequence.

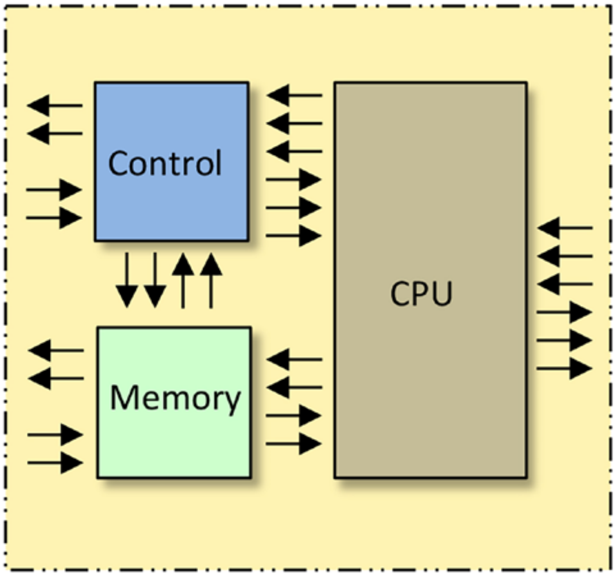


Figure 4: Heterogeneous three-die FOPLP.

For very fine pitch (> 3.0 mm) applications, the developer may employ a secondary selective plating process to form raised, solid copper terminals. The so-called "micro pillar" is significantly smaller in diameter than the ball or bump variations, enabling a considerably narrower (< 0.3 mm) terminal pitch.

Multiple-die FOPLP

Other process variations have naturally evolved, including multiple-die configurations. The benefit of clustering and interconnecting two or more associated heterogeneous or homogenous semiconductor die within the confines of a single package outline enables very close coupling and the potential for enhanced electrical performance (Figure 4).

The target package outline for a broad number of wearable and wireless products is 100 mm^2 to 140 mm^2. Ideally, the die elements selected for both single and multiple-die applications will be able to achieve equivalent finished package outline goals while maintaining the uniform terminal pattern required for efficient PCB-level circuit routing. Although the interconnect design process may initially remain within the package developer organizations, PCB design specialists will likely have the opportunity to contribute their talents as well, especially in developing the multiple-die FOPLP configurations.

Design Rules for FOPLP

In regard to design rules for single and multiple-die package applications, those currently familiar or becoming familiar with high-density additive and semi-additive circuit design criteria will be prepared to put their talents to good use. It's really a matter of scaling. The design guidelines furnished in Table 1 illustrate the expectations for interconnecting semiconductor die in the FOPLP format.

Because process capabilities will often vary between one supplier or another, the circuit density and feature sizes noted in Table 1 may be more or less than those factors shown. Before beginning the development of the fan-out circuit interconnect pattern, the designer is advised to confirm compliance with the manufacturer's imaging and plating capability.

Qualification Testing

With shorter development cycle time and more frequent introduction of new package technologies, a comprehensive qualification methodology will remain paramount in order to identify reliability weaknesses during the qualification of new package variations and material sets. A study by members of iNEMI (International Electronics Manufacturing Initiative) consortia [2] concluded that "New integrated

FOPLP	Standard	Advanced
Number of Die	1 – 3	≥ 5
Package Outline	10 x 10 mm 12 x 12 mm	20 mm x 20 mm 30 mm x 30 mm
Die Terminal Pitch	200 µm	150 µm
RDL Lines/Spaces	10/10 µm	5/5 µm
Buildup Circuit Layers	2	4
Package Terminal Pitch	0.5 mm – 0.3 mm	0.5 mm – ≤ 0.3 mm

Table 1: General design guidelines for FOPLP.

circuit package technologies can be qualified using procedures and test conditions based on experience with similar technology previously qualified."

For example, semiconductor package developers are currently applying established JEDEC standards that basically require the end-product to be subjected to three solder reflow cycles at 260°C for preconditioning followed by 1,000 cycles through temperatures that range between –40°C and +125°C, as well as a highly accelerated temperature and humidity stress test (HAST) lasting 96 hours at 121°C with 85% relative humidity. Some experts, however, are concerned that current test standards may not identify reliability risks for all commercial-use environments. The iNEMI study noted, "While previous experience is important to consider, it cannot be the only criterion and relying too much on past experience may result in overlooking new failure modes and/or new wear-out mechanisms."

In regard to package manufacturing process refinement, reaching satisfactory levels of throughput while maintaining quality objectives are not trivial issues. Developers have had to overcome a number of obstacles for each process variation. Matters that needed to be resolved include the selection of the most suitable carrier panel material, achieving precise die placement capability, dealing with mold material shrinkage and die shifting during the mold curing process, overcoming panel warping during the metalization process, and defining the most robust die thickness and mold cap thickness ratio. **DESIGN007**

References

1. Dr. Tanja Braun, "Status FOPLP," Fraunhofer IZM, Germany.

2. C. Grosskopf, F. Xue, D. Locker, S. Thomas, J. Zheng, & M. Tsuriya, "Benchmarking of Qualification Methodologies for New Package Technologies and Materials," 2019 *International Conference on Electronics Packaging (ICEP)*, Niigata, Japan, 2019, pp. 1-6.

Vern Solberg is an independent technical consultant, specializing in SMT and microelectronics design and manufacturing technology. To read past columns or contact Solberg, click here.

Intel Joins Georgia Tech in DARPA Program to Mitigate Machine Learning Attacks

Intel and the Georgia Institute of Technology (Georgia Tech) announced today that they have been selected to lead a Guaranteeing Artificial Intelligence (AI) Robustness against Deception (GARD) program team for the Defense Advanced Research Projects Agency (DARPA). Intel is the prime contractor in this four-year, multimillion-dollar joint effort to improve cybersecurity defenses against deception attacks on machine learning (ML) models.

"Intel and Georgia Tech are working together to advance the ecosystem's collective understanding of and ability to mitigate against AI and ML vulnerabilities. Through innovative research in coherence techniques, we are collaborating on an approach to enhance object detection and to improve the ability for AI and ML to respond to adversarial attacks," said Jason Martin, principal engineer at Intel Labs and principal investigator for the DARPA GARD program from Intel.

The goal of the GARD program is to establish theoretical ML system foundations that will not only identify system vulnerabilities and characterize properties to enhance system robustness, but also promote the creation of effective defenses. Through these program elements, GARD aims to create deception-resistant ML technologies with stringent criteria for evaluating their effectiveness.

In the first phase of GARD, Intel and Georgia Tech are enhancing object detection technologies through spatial, temporal and semantic coherence for both still images and videos. Intel is committed to driving AI and ML innovation and believes that working with skilled security researchers across the globe is a crucial part of addressing potential security vulnerabilities for the broader industry and our customers.

(Source: Intel)

Competition as a Tool for Growth

Design Circuit

by **Patrick Crawford,** IPC—ASSOCIATION CONNECTING ELECTRONICS INDUSTRIES

At IPC APEX EXPO in early February 2020, IPC announced our newest offering for the printed board design engineering community: IPC Design.

In the months since, we have made great strides in crafting IPC Design as a platform for the global printed board design engineering industry to connect and grow professionally, regardless of their skill level, country of origin, company, or age; all are welcome. The entirety of the IPC Design program model is well beyond the scope of what I would like to talk about here, but in a nutshell, printed board design engineers can affiliate with IPC Design by forming or joining an IPC Design Chapter or affiliating as an individual, member of a company group, or member of a STEM-focused student group—all at no cost. We are utilizing IPC's existing web infrastructure to enable a "collaborative content model" in which designers can create and access design content—media, forums, recordings, virtual meetings, etc.

IPC Design is advised by the Design Community Leadership—a group of industry experts with decades of experience in printed board design engineering and related technologies. This group has been instrumental in guiding IPC as we build the structure and initiatives of IPC Design, and I am excited to continue to unveil these initiatives here as they become available.

The construction of IPC Design is scaffolded by the 3Cs: content, career, and competition. These refer to the mindset by which both the Design Community Leadership and IPC approach any initiative within IPC Design. Put simply, does an initiative produce or provide *content* to better the industry, produce or pro-

vide experiences and opportunities to better a board design professional's *career,* or establish *competition* to engage and compel board design professionals? By examining IPC Design through this threefold filter, we have also identified primary initiatives that will act as "flagships" for IPC Design.

Earlier this month, we began work on one of these primary initiatives that was suggested by the Design Community Leadership and was fully embraced by IPC—an international printed board design competition.

Before we start building any stackups, we should stop to examine the high-level reasoning behind why competitions are so darn good at enabling professional growth.

By focusing on the needs of the printed board design engineering industry, we can build a competition that is a venue for design engineers to showcase their skills and learn from others.

By focusing on the needs of the printed board design engineering industry, we can build a competition that is a venue for design engineers to showcase their skills and learn from others. Skill transfer is a diffusive process by which information flows from areas of high concentration to low concentration. Even the most evenly matched competitive landscape will have this information gradient; the loser of a competition can always learn something from the winner by reviewing the winner's work.

Professional networking is a key goal of IPC Design and is imperative for the growth of the industry, as new contacts can mean new career opportunities, or simply the exchange of new ideas between design engineers; after all, we get by with a little help from our friends.

To that end, I would like to highlight the word "international" in the title "international printed board design competition." IPC is an international organization, and IPC Design is an international program. The best ideas for a given design challenge may not live in one country or continent, and we would like to facilitate the exchange of those ideas.

Another benefit of design competitions is the possibilities for interfacing with student competitors to help inspire the next generation of printed board design engineers. The Design Community Leadership has made it a key goal of IPC Design to engage with academia, and establishing a student division of the competition will allow for seasoned industry veterans to provide insight and guidance to students as they learn about, and try their hands at, the design of printed boards. (More on that below.)

This will not be the first time that IPC has conducted printed board design competitions, and we are leveraging the past experience of IPC India and IPC China, who have successfully hosted these competitions since 2012.

Additionally, IPC and the Design Community Leadership have agreed on high-level goals for the competition. We believe that achieving these goals will enable the competition to benefit a large demographic of designers in terms of both geography and experience with board design engineering.

The student division will be open to teams of college-level STEM students who can enter as representatives of their institution or as representatives of a collegiate electronics engineering-focused student group, such as the IPC Education Foundation. The professional division will consist of printed board design engineering professionals who self-report as such.

Second, to accommodate international competitors and the restrictive travel budgets anticipated for much of 2020 and 2021, we're planning for virtual challenges and remote judging. We envision that the student division competition will operate in the same manner except that the design requirements will be much simpler, and the students will be educated on printed board design topics beforehand and then connected with industry volunteers who can help with designs and field questions. Travel restrictions permitting, finalists will then be invited to defend their designs during an in-person event at industry events in key regions around the world.

As an example of one of these challenges, a professional challenge might consist of the library creation, mechanical creation, component placement, setup of constraints, routing, and generation of an IPC-2581 output for a 10-layer high-speed complex board based on IPC Class-2 standards. The competitors would be given four weeks to complete this challenge and then submit their design for judgment by a panel of industry expert judges.

Third, to complete their challenges, we would like to make sure that design engineers can use the design tool of their choice and are seeking and establishing relationships with tool vendors to accomplish this.

By sharing our vision, I hope that any printed board design engineer excited by the ideas put forth in the preceding paragraphs considers donating their time and talent to be part of the competition planning committee, a future competition judge, or a competitor.

I look forward to sharing future updates on IPC Design here. If you have any questions, or if you would like to get in contact with me regarding the international printed board design competition, please email me at PatrickCrawford@ipc.org. **DESIGN007**

Patrick Crawford is the manager of design programs and related industry programs at IPC and will provide updates on IPC Design activities in this column series.

BAE Ramping Up Efforts to Deliver PPE to NHS Frontline

BAE Systems recently made the following statement regarding their efforts against COVID-19: All of our industrial-scale 3D printers are now producing face shields, with supplies being delivered every day, directly to frontline medical staff around the UK. Our smaller scale 3D printers are also coming online to support the PPE effort, producing face shields at a smaller volume as well as innovative "door claws" that help care homes reduce the spread of infection through door handles.

To rapidly increase the numbers arriving on the frontline, we are sourcing tens of thousands of additional face shields through our suppliers. Together with our 3D printed face shields, we have delivered more than 24,000 units to date, with another 40,000 due to arrive in the next week to local communities in Bristol, Glasgow, London, Kent, Manchester, Portsmouth, Preston and Southampton.

Richard Hamilton, Manufacturing and Support Director at BAE Systems, said, "Employees across our air, maritime and electronic systems sectors, as well as our suppliers, are all rallying to play their part in the national endeavour as we work together to ramp up and further increase the supply of vital protective equipment to the frontline.

"As a result of everyone's collective efforts, we've been able to respond to requests to send equipment directly to medical staff in our local communities working in GP surgeries, the ambulance service and hospices, as well sending supplies to NHS hospitals and trusts."

(Source: BAE Systems)

Picking a Prototyping Strategy

Connect the Dots

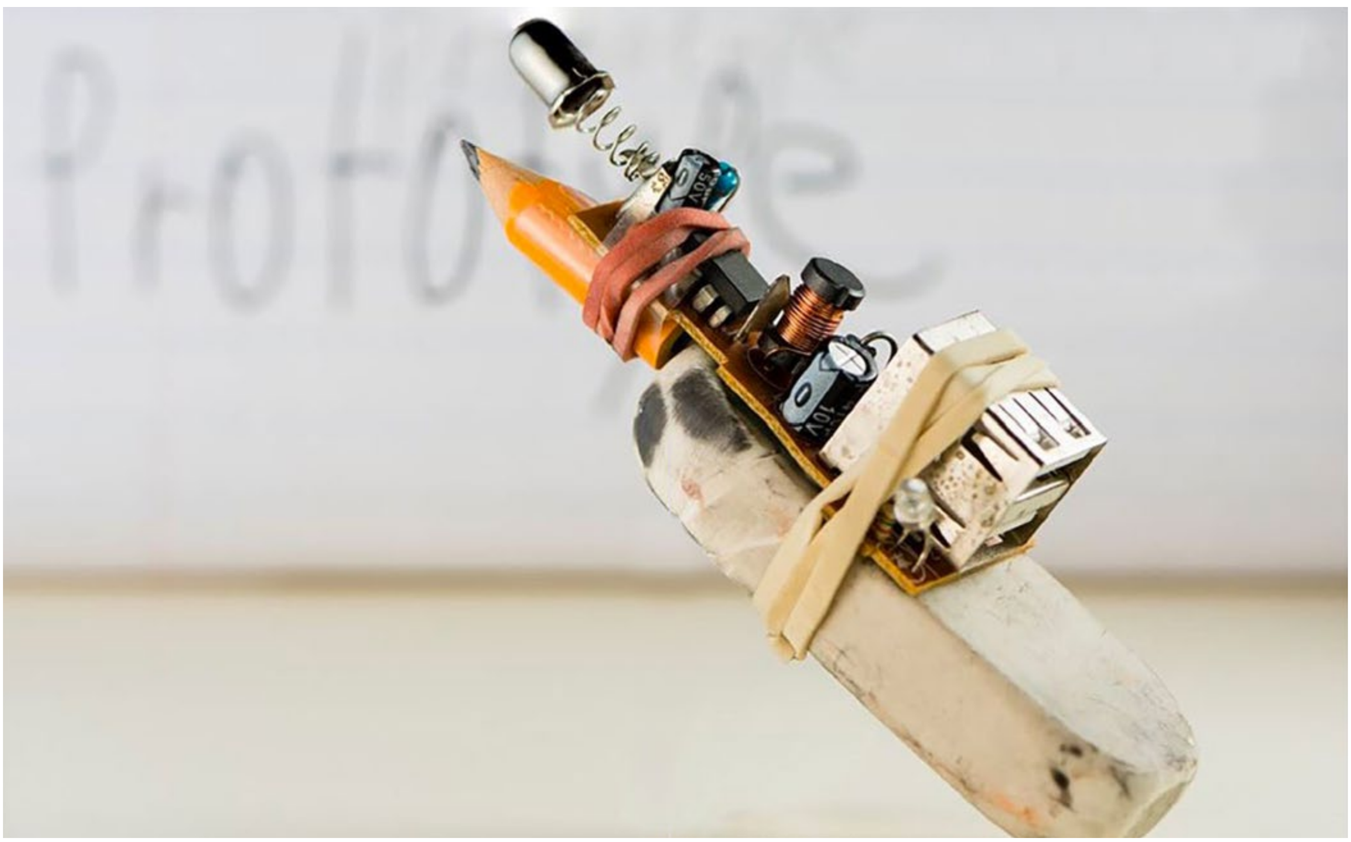
by Bob Tise and Matt Stevenson, SUNSTONE CIRCUITS

No matter how simple or complicated your electronic project, PCB prototyping is part of its journey from concept to reality. This process of turning the design into something physical can teach you a lot about what needs to be tweaked and improved before your PCB is ready for full production. There are a few different approaches, ranging from various DIY options to commercial prototype fabrication.

But before you can prototype, you have to design.

PCB design and initial testing usually happen with CAD tools and circuit simulator software. These tend to target high-speed, high-frequency, mixed signals, and even RF designs. However, a lot of these software packages have a big price tag that can be prohibitive for a part-time designer or hobbyist hoping to take advantage of their testing features. Fear not. There are still cost-effective approaches to prototyping available.

Since no two projects are the same, your approach to prototyping should be flexible. Your needs will be different every time. Are you going for fast-and-dirty prototyping? Do you need a quick turnaround time? Will it be easier for you to use a DIY prototyping technique?



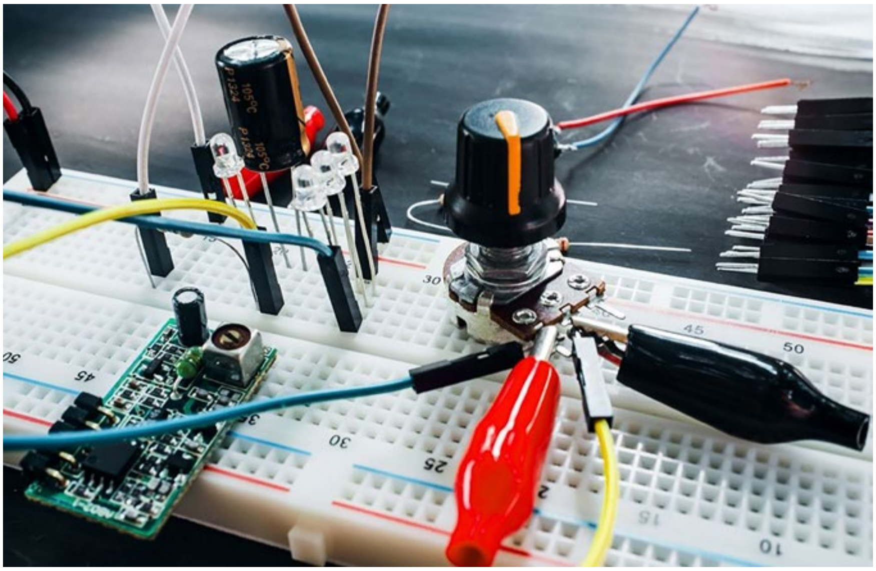


Figure 1: Breadboard.

Iterative Prototyping

Prototyping often takes multiple iterations. Your first design and your first PCB may not work. With iterative prototyping, you lay out your physical components to make sure your design actually does what you want. If it doesn't work, you make adjustments and try again. You will use actual ICs and components and try various designs until a prototype finally works. This working prototype will then be copied back into your CAD software.

Today, most small projects start off with something like an Arduino, a handful of components and jumper wires, and a solderless breadboard. The resulting rat's nest can be difficult to turn into a PCB design, however, which is why you will want to quickly move to something closer to a finished PCB (Figure 1).

Pro Tip

Note that solderless breadboards are great because you can easily reuse components. However, once you move to prototyping with other techniques, it's likely that your components will be permanently attached to boards that don't work. This can get expensive, so you will need to plan ahead carefully.

Doing It Yourself

Solderable prototyping boards are perfect for your early DIY prototype. They come in both set form factors and perforated sheets that allow you to pick a custom size for your prototype board. You can find good walkthroughs for planning IC and component layout for your first prototype using a blank board and Kynar wire [1].

If you're in the mood for really taking your DIY prototype to the next level, you can always try making your own PCB at home [2]. This requires getting a blank PCB, special transfer paper, etching chemicals, a power drill, and a lot of patience and careful work. You can also create your own PCB at home with a CNC mill, which allows you to skip the chemical etching step.

Pro Tip

If you do decide to make your PCB at home, just be careful with the chemicals. They require special handling and disposal, meaning you can't just pour them down the drain and hope they don't eat a hole in your pipes. You'll end up subject to massive fines and find yourself in desperate need of a plumber (Figure 2).

Fabricated PCB Prototypes

When your design has progressed far enough, it's time to turn your design into a profession-

Figure 2: Don't pour chemicals down the drain.

ally fabricated PCB. When preparing your design for printing, you will need to understand that every manufacturer has different capabilities, requirements, and turnaround time. You'll need to take all of these into account to manage your project efficiently.

If you already have your manufacturer figured out, keep all of this in mind from the very beginning of the design process. You don't want to get to the fabrication stage of prototyping and realize at the last minute you've overstepped your manufacturer's limitations.

Picking the Right Partner or Service

If your project is complex, especially if it's a brand new design, you will want to consider expert help. It makes sense to leverage such help with layout review.

Rapid prototyping fabrication services can help shorten the turnaround on complicated designs, which saves money during production while helping you stay on schedule. However, just because you can get boards quickly from a manufacturer doesn't mean you always should.

Look for manufacturing partners that can provide detailed, comprehensive feedback on your PCB design before your prototypes are fabricated. This will reveal yield or reliability issues with your design, improve the integrity of your production schedule, and ultimately improve the quality of the boards. The manufacturer will also be able to tell you if your design fits within their capabilities and limitations, saving endless prototyping headaches later.

Some PCB manufacturers offer multiple prototyping services. For example, we offer our ValueProto service through custom PCB manufacturing with access to experts, as well as a free DFM check with an online custom report for each job.

Conclusion

Prototyping is a necessary step on the road to production, so it's important to pick the right strategy for your project. An approach that works for one board may not be right for the next, and it will take time and experience to know which strategy is the best fit for each case.

Be flexible in your approach to prototyping options. Whether you choose to etch your own prototypes in your garage or engage professional experts from your PCB manufacturer, remember to plan ahead and don't get discouraged if one prototype is not enough. **DESIGN007**

Further Reading

1. M.B. LeBlanc, "How to Prototype Without Using Printed Circuits Boards," Instructables Circuits.
2. O.N. Dahl, "DIY PCB: How to Create Your Own Printed Circuit Boards," Build Electronic Circuits.

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

Bob Tise

Matt Stevenson

PCB007 Highlights

Accurate Circuit Engineering Staying Ahead of the COVID-19 Curve ▶

On April 16, Andy Shaughnessy spoke with Accurate Circuit Engineering's James Hofer. James provided an update on the company's responses to the challenges of fabricating PCBs while much of the country is under COVID-19 quarantine and explained that he is dedicated to keeping his employees safe and healthy while continuing to meet the needs of his customers, many of whom are medical and military OEMs. He also discussed his current workload, which includes one customer developing a ventilator that can serve four patients at the same time.

Photography Slideshow by Columnist Mehul Davé ▶

At I-Connect007, we like to work hard, play hard, and highlight our contributors. Mehul J. Davé is a great example of someone who is passionate about the industry as well as his other interests outside of work—especially photography. Here, we feature some of his work.

Jolly Holden: Expert Insights Into Distance Learning and Webinars ▶

With COVID-19 leading to increased distance learning throughout the U.S. and world, the I-Connect007 editorial team spoke with Jolly Holden, Ed.D., who is an expert on distance learning (and also Happy Holden's brother). In this wide-ranging interview, they discuss what distant learning is, how to do it well, what not to do, differences between academic and job skills training, and much more.

Punching Out! Stress Testing Deals ▶

Most M&A transactions fall apart several times before closing, even during normal times. When a major crisis occurs—whether internal or external—a deal can truly be stress tested. Tom Kastner shares 12 tips his company has used (and are currently using) to keep deals rolling.

North American PCB Industry Sales up 3.7% in March ▶

IPC—Association Connecting Electronics Industries announced today the March 2020 findings from its North American Printed Circuit Board (PCB) Statistical Program. The book-to-bill ratio stands at 1.15. Total North American PCB shipments in March 2020 were up 3.7% compared to the same month last year.

Gene Weiner: Lessons Learned From COVID-19 ▶

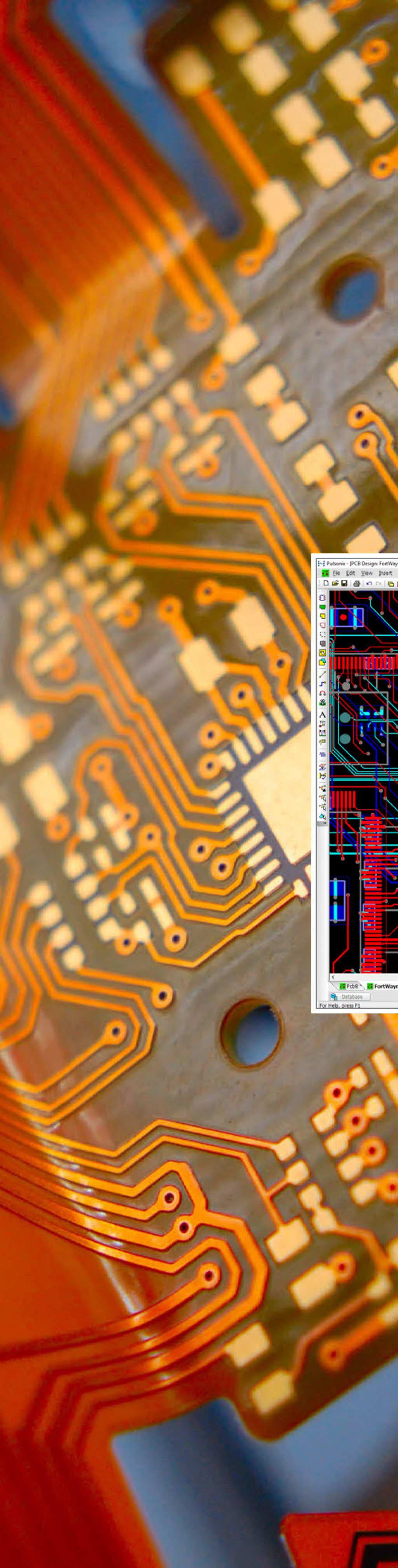
Barry Matties speaks with Gene Weiner, president and CEO of Weiner International Associates, about the electronics industry's continued operation under COVID-19 restrictions and some of the lessons learned during this pandemic. One such lesson: Companies and nations must do a better job of sharing information to help prevent this from happening again.

Isola's Travis Kelly: Maintaining Continuity of Supply ▶

On April 9, Barry Matties spoke with Travis Kelly, Isola president and CEO. Travis gave an update on Isola's responses to the COVID-19 challenges in materials manufacturing and pointed out that Isola's products fill a critical need in the switch to medical equipment manufacturing for ventilators and related products. He also spoke to the three keys to maintaining continuity of supply.

Ucamco Releases Latest Version of UcamX ▶

Amid the current worldwide situation, Ucamco is proud to bring positive news and announce the v2020.03 release of UcamX.



Conformal Coating Enemies: Challenges Sabotaging Your Process

Sensible Design

by Phil Kinner, ELECTROLUBE

Coatings fail for any number of reasons—some common, and some not so common. I have more than 20 years' experience and knowledge of the conformal coating business, and I'm still learning, but there are certain production-related issues that cause problems time and time again—some of which could have been easily resolved. In my previous columns, I have examined a number of coating failure issues, but this month, I'm going to talk about some of the arch enemies of conformal coatings, the challenges posed by sensitive areas and higher profile parts, and the best ways to avoid wicking to help you achieve a uniform and defect-free finish. Hopefully, it will provide useful background information to help you with your coating-related activities.

1. Why do taller/vertical pins on PCBs represent more of a challenge to coat, and can you suggest some key pointers for success?

Tall and vertical surfaces present challenges to achieving coverage with liquid conformal coatings, predominantly due to gravity and the application method, to a certain degree. Traditionally, liquid conformal coatings rely on solvent evaporation to increase their viscosity and prevent the material from flowing away from sharp corners and vertical surfaces by the combination of gravity or capillary forces. This process is relatively slow, and some degree of coverage issues is always likely to be present. UV-curable materials solidify very rapidly under UV radiation of the correct wavelength and intensity; however, there is nearly al-

ways sometime between the application of the material and the exposure to radiation, and during this time, the material will still be subject to gravitational and capillary forces, again resulting in some degree of coverage issues.

Given that selective coating with an XYZ-capable robot is now the dominant application technique, and these robots use a valve that is traditionally mounted perpendicular to the board, these issues are exacerbated, as the coating material is also projected with momentum perpendicular to the board. This reduces the chance of actually landing on a sharp corner or vertical surface and facilitating the gravitational and capillary flow towards the board surface. Tilting the application nozzle to a 45-degree angle increases the chance of actually hitting the vertical surfaces with material and reduces the vertical momentum and, consequently, the degree to which the material will flow. Modern, more viscous materials will also resist the gravitational and capillary forces when sprayed in this fashion to ensure the areas that were coated remain coated.

2. What steps should I take to protect particularly sensitive parts to be coated?

This depends on what the parts are sensitive to. If you are trying to protect parts from the effects of humidity, condensation, or immersion in water, then thickness and coverage are the most important factors to consider, along with the choice of conformal coating. Some coatings are inherently better at achieving coverage at higher thicknesses and have inherently greater barrier properties than others. Sometimes, parts are sensitive to being coated, and either the component itself or the solder terminations can be damaged by conformal coating materials.

This is especially true when the glass transition temperature (Tg) of the material falls in the operating range of the board in question. As the coating material passes through its Tg, there is a delay between the decrease in elastic modulus and the increase in the coefficient of thermal expansion (CTE). This lag results in maximum stress being applied to the component or solder termination and can damage the component or solder. Over a sufficient number of cycles, solder can even be extruded by the coating, leading to shorts.

Therefore, the Tg of the coating material should ideally lie outside of the operating range of the PCB or at least towards the lower end of the operating range. Occasionally, taller components can be more susceptible to damage by vibration, even though coatings can be used to help mitigate the effect of vibration, but this is a very complex, multi-faceted issue and probably a separate column in itself.

Usually, a staking compound or adhesive is used to glue the component body to the board to minimise its ability to move with vibration. However, the material has to be carefully selected to minimise CTE and needs to be sufficiently soft and elastic to dissipate vibration without being either too stiff or too elastic to avoid exacerbating CTE issues over the operating conditions of the assembly.

3. Is coating/board failure more common on higher-profile parts of the PCB, and if so, why?

This depends on the design of the assembly and the conditions to which it is exposed in the field. As discussed in point 1, there are numerous challenges of actually coating taller devices effectively. A lack of coating thickness and coverage make taller components more susceptible to failures by tin whiskers, arcing, corrosion, condensation, or immersion. However, under conditions of high humidity, the board level terminations are often more susceptible to corrosion due to the presence of flux residues, which exacerbates the tendency for localised condensation due to their hygroscopic nature and subsequent corrosion due to their ionic nature.

4. Can you name five "enemies" to a successful coating process?

Lack of cleanliness or the presence of contamination on the board before coating can lead to open defects in the coating due to dewetting or non-wetting. The contaminants can affect or compromise the ability of the coating to provide long-term protection by increasing the rate of moisture uptake and interaction with water, causing corrosion beneath the coating and compromising the adhesion of the coating or a combination of all of these factors.

Variations in surface energy of the incoming bare boards will be exacerbated by the assembly process (reflow/wave thermal cycles). The surface energy of the board will determine the way that the liquid coating interacts with the substrate and the degree to which the solid coating will adhere to the board. The higher the value of incoming bare board surface energy, the more repeatable and consistent the application of the liquid coating will be, and the greater degree of adhesion. The process will achieve a decent degree of repeatability and reproducibility (R&R), enabling a high degree of process control and consistent performance in the end-use environment. Conversely, the lower the incoming surface energy, the less repeatable and consistent the process will be, and the likelihood of field failures will be increased.

The actual design of the board often determines the success, or otherwise, of the coating process. Boards that are well designed for conformal coating will have good separation between areas that must and areas that must not be coated. Taller components will be grouped together away from low-profile components, and low-profile components will not be clustered together close to vias or SOIC, QFP, QFN, or BGA components, which will tend to pull material beneath them. This can lead not only to potential reliability issues with these components themselves but also leave low-profile SMT discretes short of coating or with bridging bubbles.

Conformal coating is a process. Ensuring a repeatable and consistent conformal coating result depends on the way the process is designed, just as much as the material selected. Manual processes, such as spray or brushing, will inherently have more variation in the end result due to the human operator, whereas machines will consistently perform the same routine over and over. However, machines currently have no feedback loop, so if the material behaviour changes, the end result will change, whereas a human operator may compensate for this (either deliberately or possibly subconsciously). As previously discussed, the surface energy of incoming boards, board design, and pre-existing contamination can all affect the way the liquid material behaves. In addition, the temperature of the material will affect the viscosity of the material.

The wider the change in ambient factory temperatures throughout the day (or even using material that has been stored somewhere else, such as outside), the greater the effect of these changes in material properties and the wider the variation in material application. Controlling the temperature of the material or compensating by machine measurements and process control equipment can be important in ensuring a repeatable machine-based process.

Airflow can affect the rate at which solvents evaporate and the material viscosity will build. Inconsistent airflow can lead to differential flow rates and unexpected issues, such as orange peel. If an oven curing process is used, the application of temperature can initially lead to a decrease in viscosity and increase the spread of the coating before the viscosity starts to build, possibly leading to coating areas intended to be left free from coating or a higher degree of bubbling than expected. Similarly, the rate of change of temperature can lead to excessive bubbling, so controlling the ramp rate or providing a solvent "flash-off" period can be necessary to ensure a uniform and defect-free finish.

5. What is wicking, and how do I avoid it?

Wicking is often used interchangeably with capillary flow, especially in a conformal coating context, to describe the ability of liquids to flow into narrow gaps without the assistance of any external forces. The narrow gaps are most often unsealed connectors or the standoff between component bodies and board substrate. In the case of unsealed connectors, switches, etc., the main issue is the material "wicking up" and coating the connector mating surfaces, therefore reducing the contact or potentially insulating the mating surface and preventing either the form, fit, or function of the connector or switch.

In the case of low-standoff devices, issues can be caused by material penetrating beneath the component package, leading to stress due to CTE mismatch (and potentially early failures) or displacing air, which can lead to the formation of open bubbles and can be an initiation site for corrosion or other types of failure.

Typical best practices would entail using gel versions of coating materials to seal the base of connectors switches and so forth before the application of the liquid coating. Masking materials can be used to achieve the same effect, although the application, curing, and subsequent removal of masking materials are all wasted processes. New materials are available, which inherently minimise the extent to which the materials are affected by capillary flow, potentially eliminating the requirement for masking or sealing before application.

Conclusion

Understanding the major enemies of conformal coatings will go a long way in implementing a successful conformal coating process. I hope I have heightened your awareness of some of the potential pitfalls you might face so that you can hopefully avoid any coating disasters.

Look out for my next column, where I will explore more conformal coating issues. In the meantime, please stay safe and well. **DESIGN007**

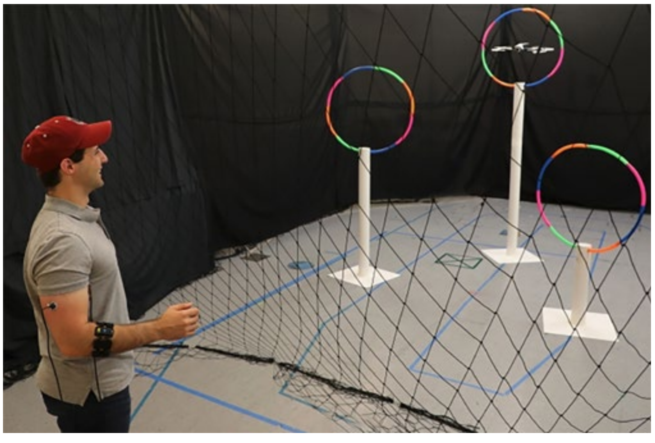
Phil Kinner is the global business and technical director of conformal coatings at Electrolube. To read past columns or contact Kinner, click here. Download your free copy of Electrolube's book, *The Printed Circuit Assembler's Guide to... Conformal Coatings for Harsh Environments*, and watch the micro webinar series "Coatings Uncoated!"

Muscle Signals Can Pilot a Robot

CSAIL's Conduct-A-Bot system uses muscle signals to cue a drone's movement, enabling more natural human-robot communication.

Albert Einstein famously postulated that "the only real valuable thing is intuition," arguably one of the most important keys to understanding intention and communication.

But intuitiveness is hard to teach—especially to a machine. Looking to improve this, a team from MIT's Computer Science and Artificial Intelligence Laboratory (CSAIL) came up with a method that dials us closer to more seamless human-robot collaboration. The system, called "Conduct-A-Bot," uses human muscle signals from wearable sensors to pilot a robot's movement.



"We envision a world in which machines help people with cognitive and physical work, and to do so, they adapt to people rather than the other way around," says Professor Daniela Rus, director of CSAIL, deputy dean of research for the MIT Stephen A. Schwarzman College of Computing, and co-author on a paper about the system.

Muscle signals can often provide information about states that are hard to observe from vision, such as joint stiffness or fatigue.

For example, if you watch a video of someone holding a large box, you might have difficulty guessing how much effort or force was needed—and a machine would also have difficulty gauging that from vision alone. Using muscle sensors opens up possibilities to estimate not only motion, but also the force and torque required to execute that physical trajectory.

(Source: MIT News Office)

The Impact of Signal Rise Time on Bandwidth

Beyond Design

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

The term bandwidth was first used years ago in the RF world to represent the range of frequencies in a signal. In digital electronics, we also use the term to describe the signal spectrum since square waves are made up of numerous sine waves (harmonics) of the fundamental frequency (Figure 1). For digital signals, the lowest frequency is always the DC component (zeroth frequency) and the highest frequency component is the maximum frequency that is significant (typically the fifth harmonic). The shorter the rise time in the time domain, the higher the bandwidth in the frequency domain, and the more closely the waveform resembles an ideal square wave. In this month's column, I will look at the relationship between signal rise time and the bandwidth of a digital signal.

Rise time describes how quickly a digital signal can change from a low state to a high state. A signal must have a fast enough rise time to accommodate the data being processed. Otherwise, information in the waveform (circuit timing) may be lost. However, a signal does not have to have a faster rise time than is required by the system. Faster is not necessarily better as it may create ground bounce, reflections, crosstalk, and electromagnetic radiation. In an ideal world, one should limit the bandwidth so that the system performs to expectations, but at the same time, avoid high-frequency effects, which cause electromagnetic compatibility issues.

This is exactly what a series impedance termination does. The resistor, close to the source, combines with the input capacitance of the receiver IC(s) to create a low pass filter (Figure 2). This filter rolls off the high-frequency components of

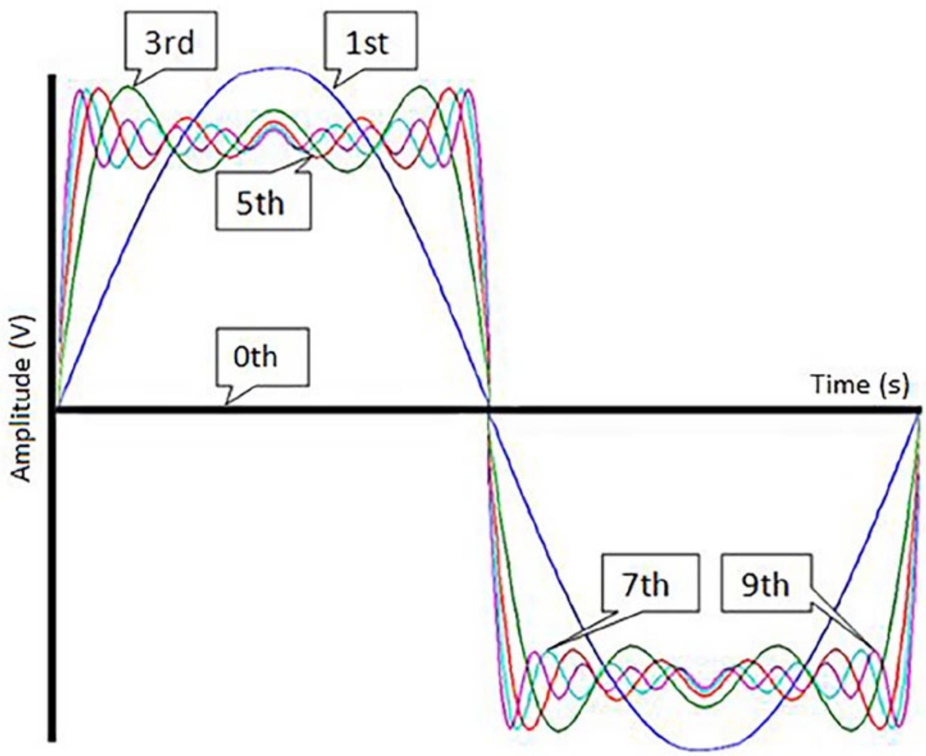


Figure 1: Harmonic content of a square wave. Higher harmonics have faster rise times.

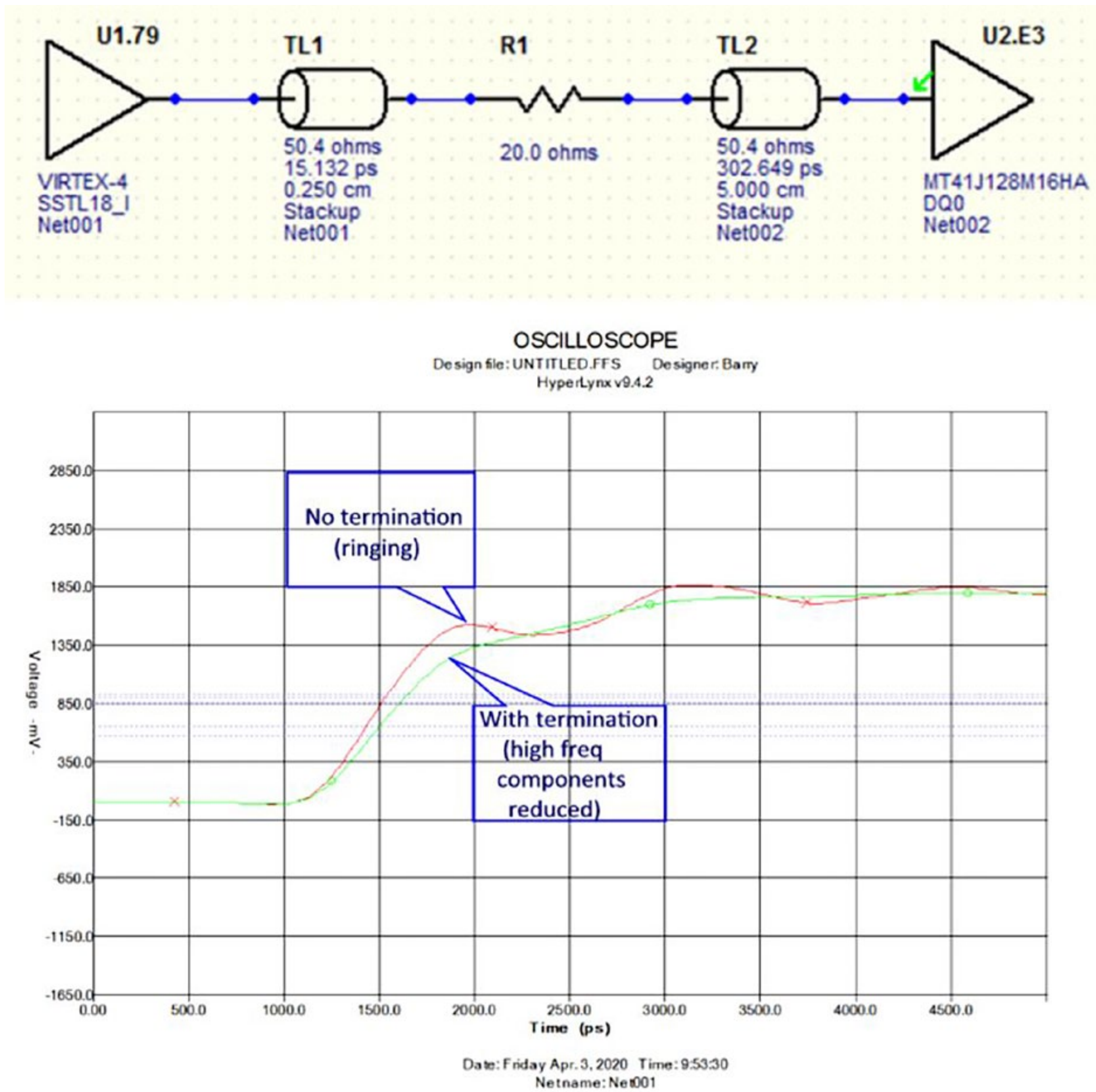


Figure 2: Series terminator rolls off the high-frequency components to reduce ringing (simulated in HyperLynx).

the waveform, reducing reflections. Providing we choose the right value to match the impedance of the transmission line, the circuit functions perfectly, although the higher frequencies are dampened.

Depending on the logic family, the fall time is usually slightly shorter than the rise time. This is due to the design of typical CMOS output drivers. For the same feature size transistor, an n transistor can turn on faster than a p transistor. This means switching from high to low, the falling edge will be shorter than the rising edge. In general, signal integrity problems are more likely to occur when switching from a high-to-low transition than from a low-to-high transition. This is something to look out for when using double data rate (DDR) memory, where the device is clocked on both the rising and falling edges of the waveform.

In the time domain, the most important merit for a signal is the rise time. The rise time is typically measured from the 10% point to the 90% point on the rising edge of the signal, although the industry is moving toward 20–80% to eliminate the distortion at the extremities. The shape of the rising edge strongly influences the interpretation of the rise time. The rise time of a signal is inversely proportional to its bandwidth.

$$BW\ (3dB) = \frac{0.35}{Rt} \qquad \text{Equation 1}$$

BW is the bandwidth in GHz, and
Rt is the rise time in ns.

The bandwidth of the signal in the frequency domain is where we arbitrarily define the bandwidth as the -3 dB point. This is the frequency at which the signal amplitude, through the circuit, is reduced down to 70% of the amplitude of the input signal. Higher frequencies are attenuated even more.

In practice, the signal rise time has an impact on the maximum signal bandwidth. Understanding the frequency band, that really matters, for digital design is very important. Traditionally, we used Equation 1 for the upper bandwidth, assuming one can ignore all the frequency components above the bandwidth and not lose any valuable information about the time domain response.

$$BW\ (6dB) = \frac{0.5}{Rt} \qquad \text{Equation 2}$$

However, as rise times become faster, a more accurate approach is to use the upper knee frequency (which is down -6.8 dB), as in Equation 2. This forms a useful translation between

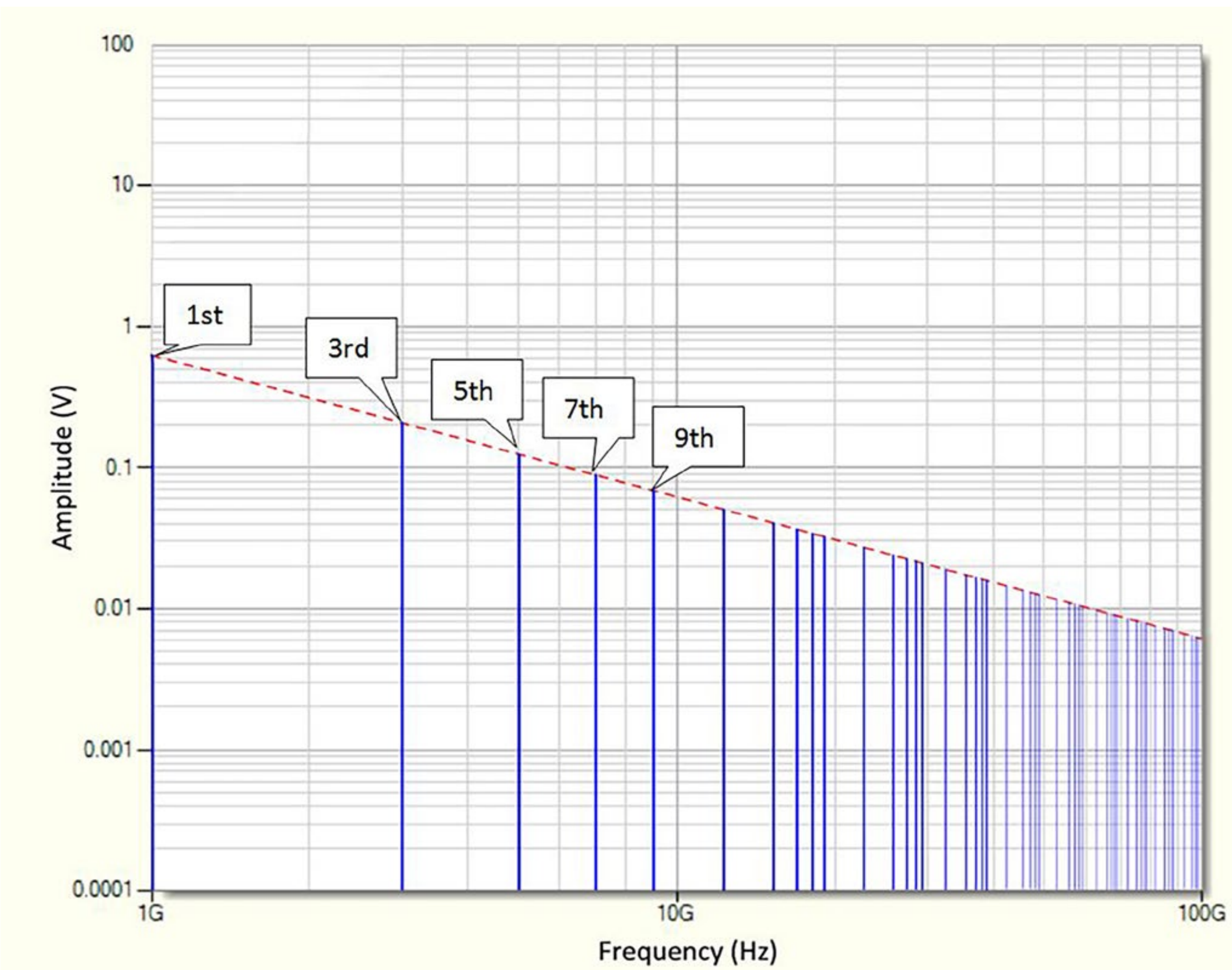


Figure 3: Odd harmonics of a 1 GHz fundament clock in the frequency domain.

the time and the frequency domains. The knee frequency is an estimate of the highest frequency content of the signal, which depends upon the rise time of the signal. For instance, if the rise time is 100 ps, then the upper bandwidth is actually 5 GHz regardless of the clock frequency. It is possible to have two different waveforms, with exactly the same clock frequency but different rise times and, therefore, different bandwidths.

When selecting the most appropriate dielectric materials for a stackup design, one should consider the bandwidth up to the fifth harmonic. For the 1 GHz fundamental frequency in Figure 3, the maximum bandwidth to consider is the fifth harmonic at 5 GHz if the rise time is unknown. This assumes the worst case of a 100 ps rise time. For high-frequency signals, it is more accurate to use the actual rise time to evaluate bandwidth, but unfortunately, we do not always have the luxury of that information.

It is important to note that the concept of bandwidth is inherently an approximation. It is roughly where the amplitude of the frequency components in a waveform begins to drop off faster than an ideal square wave.

One simple but often overlooked method of minimizing noise in a system is to limit the system bandwidth to only that required by the intended signal. The use of higher bandwidth than required allows additional noise into the system. The same principle also applies in the case of digital circuits. High-speed logic with a fast rise time is much more likely to generate and be susceptible to high-frequency noise than its lower speed counterpart.

Key Points

- The shorter the rise time in the time domain, the higher the bandwidth in the frequency domain.
- A signal must have a fast enough rise time to accommodate the data being processed.
- A fast rise time may create ground bounce, reflections, crosstalk, and electromagnetic radiation.
- One should limit the bandwidth so that the system performs to expectations, but at the same time, avoid high-frequency effects, which cause electromagnetic compatibility issues.
- Depending on the logic family, the fall time is usually slightly shorter than the rise time.
- Signal-integrity problems are more likely to occur when switching from a high-to-low transition than from a low-to-high transition.
- The rise time is typically measured from the 10% point to the 90% point on the rising edge of the signal.
- The 3 dB point is where the signal is reduced down to 70% of the amplitude of the input signal.
- As rise times become faster, a more accurate approach is to use an upper knee frequency.
- The knee frequency is an estimate of the highest frequency content of the signal, which depends upon the rise time of the signal.
- When selecting the most appropriate dielectric materials for a design, one should consider the bandwidth up to the fifth harmonic. **DESIGN007**

Further Reading

• B. Olney, "Beyond Design: When Legacy Products No Longer Perform," *The PCB Design Magazine*, August 2017.

• D. Brooks, "Rise Time vs. Frequency: What's the Relationship?" I-Connect007, June 7, 2011.

• E. Bogatin, *Signal and Power Integrity: Simplified,* Prentice Hall, 2008.

• H. W. Johnson & M. Graham, *High-Speed Digital Design: A Handbook of Black Magic,* Prentice Hall, 1993.

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 icd.com.au. To read past columns or contact Olney, click here.

MilAero007 Highlights

What It Takes to Be a Milaero Supplier, Part 3 ▶

The decision to pursue military and aerospace (milaero) certification impacts every facet of the organization, and not every shop is prepared to make this transformation. In Part 3, Anaya Vardya explores what it takes to be a milaero supplier in the areas of purchasing and quality.

SAIC Awarded $655 Million ▶

U.S. Air Force Engineering Development Integration Sustainment Contract Engility Services, a subsidiary of Science Applications International Corp., was awarded a $655 million award to provide systems engineering, planning, integration, and sustainment services to the U.S. Air Force Space and Missile Systems Center (SMC) Innovation and Prototyping Directorate through the Engineering, Development, Integration, and Sustainment (EDIS) contract.

Lone Star Circuits Recognized by Raytheon With 4-Star Honors ▶

Lone Star Circuits Inc. was one of 86 companies recognized by Raytheon's Integrated Defense Systems business for 4-Star honors.

Defense Speak Interpreted: Why Is Defense Hyper Over Hypersonics? ▶

Perhaps you have noticed that the term "hypersonics" is now a buzz phrase in a big part of the Department of Defense research effort. What does hypersonic mean, and why is so much work needed in this weapons field? Dennis Fritz explains.

From the Hill: Sampling Plan Language in MIL-PRF-31032 ▶

Conformance to military standards is all about using the correct sample specimen and testing the proper quantity. However, military specifications have numerous tests with various sample specimen types that all require different quantities for test or inspection. Mike Hill provides an overview of the key parts of compliance.

BAE Systems Wins DARPA Contract to Develop Machine Learning Analytics as a Service for Constant Global Situational Awareness ▶

BAE Systems aims to develop machine learning analytics as a service—a first-of-its-kind, cloud-based model for the government—that can leverage commercial and open-source data to deliver constant worldwide situational awareness for a diverse range of challenges.

Draganfly's 'Pandemic Drone' Conducts Flights Near NYC ▶

Draganfly Inc. an award-winning, industry-leading manufacturer within the commercial Unmanned Aerial Vehicle, Remotely Piloted Aircraft Systems, and unmanned vehicle sector, announced the first-ever series of U.S. "pandemic drone" test flights in Westport, Connecticut, considered a COVID-19 "hotspot," to identify social distancing and detect symptoms presented by the virus, in an effort to keep the community safe.

DoD Contract for 60 N95 Critical Care Decontamination Units: $415M Contract, Each Unit Can Decontaminate 80K N95 Masks Per Day ▶

The Department of Defense's Defense Logistics Agency, on behalf of the Department of Health and Human Services (HHS), has awarded a $415 million contract for 60 Battelle Memorial Institute Critical Care Decontamination Systems (CCDS), that can decontaminate up to 80,000 used N95 respirators per system per day, enabling mask reuse up to 20 times.

Checking in With Todd MacFadden of Bose

Flex007 Feature Interview
by Andy Shaughnessy
I-CONNECT007

Bose's Todd MacFadden is one of the many designers and engineers around the world who found themselves moving from the world of rigid boards into flexible circuits. Once famous primarily for their cutting-edge speaker systems, Bose is now involved in everything from automotive electronics to wearables for the healthcare industry. I touched base with Todd recently to get his thoughts on the current state of flexible circuits.

Andy Shaughnessy: Hi, Todd. For those who may have missed your earlier articles, give us a brief background on yourself and your work at Bose.

Todd MacFadden: I am a PCB technology engineer at Bose, where I have worked for 15 years. Among my responsibilities, I develop internal electronics technology roadmaps to ensure the supply base can support the critical technologies needed for upcoming products. I have a master's degree in chemical engineering from UMass Lowell and bachelor's degrees in political science and music from the University of Colorado Boulder.

Shaughnessy: Everyone seems to have a different reason for using flex. How did Bose wind up in the flex world in the first place?

MacFadden: One word: miniaturization. Our latest generation of products requires us to incorporate ever-increasing functionality into ever-shrinking spaces, and the only way to do this is to work in three dimensions.

Shaughnessy: What do you see as the biggest challenge for flex designers today?

MacFadden: The impossibly small spaces allotted for our electronics means that we generally violate industry bend radius guidelines more than we adhere to them. We have learned—often the hard way—how to achieve overly-tight bends without fracturing traces, but it hasn't been easy.

Shaughnessy: Do you think the flex standards are keeping up with innovation, or is that even possible?

MacFadden: Considering the breakneck pace of technology development, industry standards and guidelines will always be a step or two behind. But they serve an invaluable role for us as the industry-accepted baseline reference and to ensure standardization of materials and performance among our suppliers.

Shaughnessy: What advice would you give someone who is moving from rigid board design into flex design?

MacFadden: Leverage the expertise of your suppliers. Engage them early and often in the upfront development phases for advice on material selection, bend configurations, and panelization to ensure the most reliable and cost-effective solution. We often build early dummy boards in form factor for simple continuity testing to make sure the conductors will survive the tight bends.

Shaughnessy: Thanks for your time, Todd.

MacFadden: Thank you for the opportunity. I appreciate it. FLEX007

Right Climate for Battery Testing

Electric vehicles are widely regarded as the future. But while bold targets are in place for the UK and the EU to transfer its transport fleet to electric propulsion by 2035 or 2040, there needs to be substantial evolution of the technology before we get there.

MAHLE Powertrain has recently opened a facility specifically for the testing and development of electrified powertrain battery packs and modules.

The aim of this new facility, which is based at the company's Northampton UK headquarters, is to expedite the validation and testing of new battery technologies, parts and assemblies and therefore reduce the time and cost of the development process.

One established method of accelerating the testing procedure is to be able to precisely control the testing environment, rapidly cycling the temperature and humidity within a test chamber. As these climate-controlled chambers are key to the success of the operation, MAHLE Powertrain turned to expert Unitemp, who is the exclusive UK distributor of Espec environmental test equipment. For this battery cell evaluation project the Espec Platinous Series PL-3J test chamber was selected.

Espec Platinous Series PL-3J Chamber Series Cabinet

This chamber, like all others from Espec, meets the (European Council for Automotive R&D) EUCAR level 6 standard. This means that it will withstand fire, gas venting and 'energetic release of materials', which is reassuring when dealing with new battery chemistries that have unknown and potentially dangerous behaviours when subjected to the real-world type testing simulated in the environmental chambers. Beyond meeting these stringent requirements, the Espec chambers were also selected for their energy-efficient operation and the large amount of test space available in relation to the footprint of the machine.

(Source: Unitemp)

Flex007 Highlights

American Standard Circuits Now Provides Long (30 Inches) Rigid-flex PCBs ▶

West Chicago-based circuit board fabricator American Standard Circuits announced that the company is now building 30-inch rigid-flex PCBs. President and CEO Anaya Vardya said, "Producing these longer boards is something we have been working on for quite some time. So many of our good customers told us that it would be extremely helpful to them if they could design rigid-flex boards that extend up to 30 inches."

Flexible Circuit Technologies Continues Operations as an 'Essential Supplier' ▶

Carey Burkett, vice president of business development, announced that Flexible Circuit Technologies (FCT) is fully operational as an "essential supplier" given the critical nature of the electronic products and services they supply to medical customers who are helping in the global fight against COVID-19.

DIS: It's All About Alignment ▶

Jesse Ziomek, VP of sales for DIS, updates Pete Starkey on the capability of the company to achieve ultimate accuracy in layer-to-layer registration, not just in rigid multilayers, but also in flex and rigid-flex builds. Jesse also comments on keeping technology exciting enough to attract young engineers into the industry.

Flex and Rigid-flex: Materials, DFM, and Real-world Advice ▶

Flex and rigid-flex are a significantly growing portion of the global PCB market. Whether you are starting with single- and double-sided flex, learning the ins and outs of rigid-flex technology, or just beginning to utilize additive processing for very fine-line rigid-flex, working with technology new to you is both exciting and challenging.

MFLEX Increases Growth ▶

Happy Holden spoke with Jay Desai of MFLEX about the latest flex work the company has been doing and its aim to transition toward more automation and a smart factory approach.

Innovative Circuits Promotes Chris LaCroix to Plant Manager ▶

Flex manufacturer Innovative Circuits has promoted Chris LaCroix to plant manager effective immediately. In his new role, LaCroix will oversee all plant operations and department managers will report directly to him. LaCroix will continue to report directly to Innovative Circuits (ICI) President George Schudy.

DuPont Teijin Films Introduces Clear, Flame-retardant Polyester Film ▶

To help product designers achieve improved safety in the industrial, transportation, construction, electronics, and label industries, DuPont Teijin Films is introducing clear, flame-retardant, PET (polyethylene terephthalate) polyester films.

Trackwise's 'Improved Harness Technology' Designed Into Circuits for Electric Vehicles ▶

Trackwise's IHT is a proprietary roll-to-roll manufacturing process, enabling the production of unlimited length, multilayer flexible printed circuits (FPCs). Trackwise's roll-to-roll technology is being used to manufacture FPCs for use at a module level in both HV and LV circuits in the vehicles' battery packs, reducing the part count, assembly time, and saving on space and weight.

When Expectations and Results Don't Line Up

Flexible Thinking

by Joe Fjelstad, VERDANT ELECTRONICS

Around 20 years ago, I had the good fortune of receiving a recommendation to read the book *The Four Agreements* by Don Miguel Ruiz and subsequently picking it up. It is a short and simple book that the author says is based on ancient Toltec wisdom. While the book amplifies the premises by examples, these are the four personal agreements:

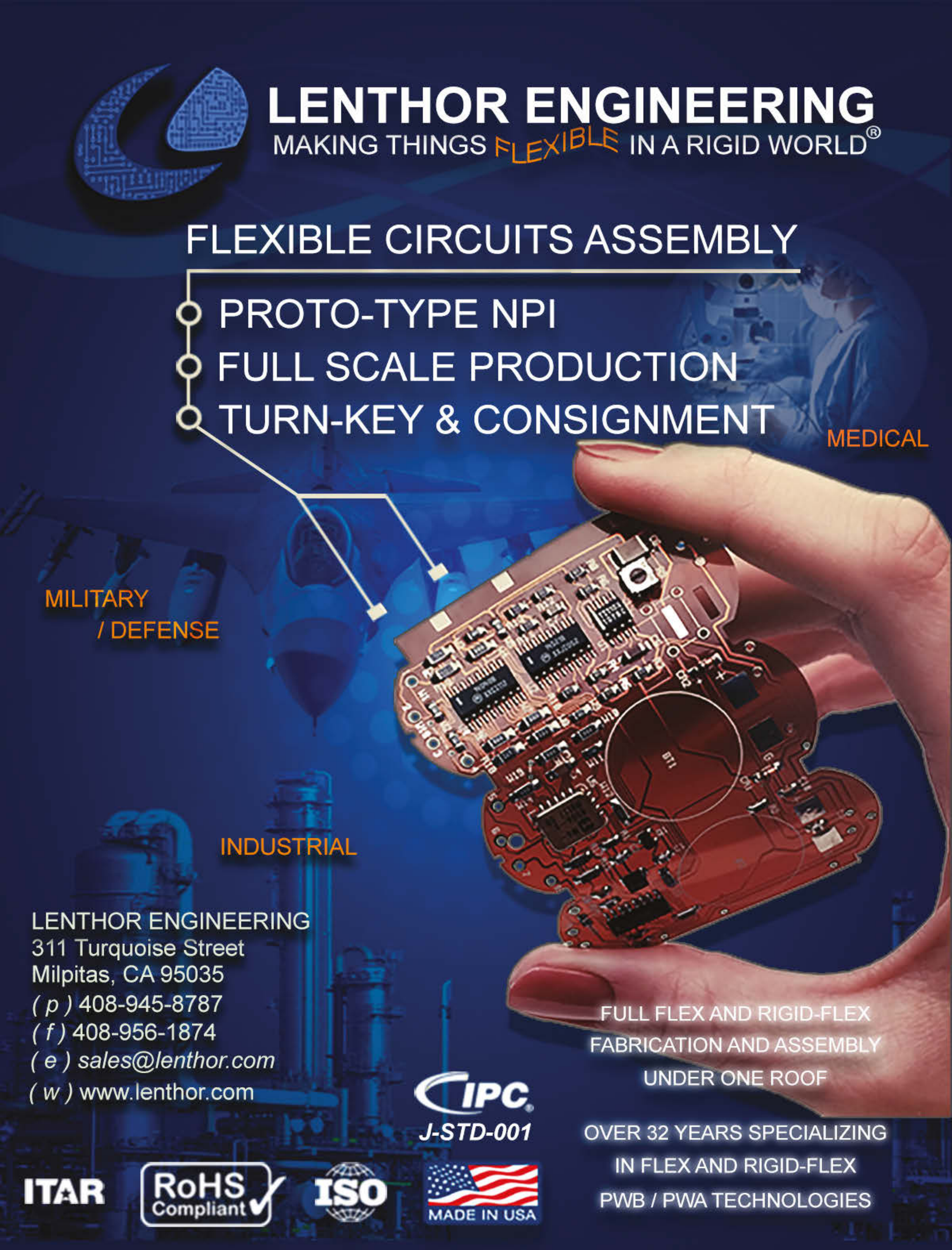
1. Be impeccable with your word.
2. Don't take anything personally.
3. Don't make assumptions.
4. Always do your best.

I do my best to keep these agreements with myself in mind on a daily basis as I navigate the world around me. I've found that they are often applicable in many unanticipated ways, from personal relationships to business and even into process development and invention, which have been central to my career since I first encountered them.

This brings us to this month's theme: "What does the electronics industry expect of its designers," or from the designer's perspective, "What do you expect from me?"

At this point, one might be inclined to say, "I want perfection, of course," but we all know it is unlikely that will ever happen. There are always things that don't quite go as planned. Mistakes are "as perennial as the grass" (to lift and perhaps misuse a line from Max Ehrmann's wonderfully inspirational poem "Desiderata").

While first-pass perfection may be unlikely, it is not beyond approach. In last month's column, I put forth the notion that rather than



LENTHOR ENGINEERING
MAKING THINGS FLEXIBLE IN A RIGID WORLD®
FLEXIBLE CIRCUITS ASSEMBLY
PROTO-TYPE NPI
FULL SCALE PRODUCTION
TURN-KEY & CONSIGNMENT
MEDICAL
MILITARY / DEFENSE
INDUSTRIAL
LENTHOR ENGINEERING
311 Turquoise Street
Milpitas, CA 95035
(p) 408-945-8787
(f) 408-956-1874
(e) sales@lenthor.com
(w) www.lenthor.com
FULL FLEX AND RIGID-FLEX FABRICATION AND ASSEMBLY UNDER ONE ROOF
IPC J-STD-001
OVER 32 YEARS SPECIALIZING IN FLEX AND RIGID-FLEX PWB / PWA TECHNOLOGIES
ITAR
RoHS Compliant
ISO
MADE IN USA

suggesting that designers adapt to designing *for* manufacturing, they should be designing *with* manufacturing. Designers, I argued, need to understand manufacturing before they can design effectively.

Agreement 1

This brings us to the first agreement: "Be impeccable with your word." Designers must be honest with themselves as to their knowledge and carry that knowledge shamelessly into any discussion with manufacturing. It is okay not to know something. The same requirements hold true for the manufacturer. There is a partnership between design and manufacturing, and open and honest communication is required—especially when something new is being added to the mix. Bluff and bluster have no place at the table.

If some design feature is beyond a designer's experience, set up expectations early on in the process. As stated earlier, mistakes are okay. That's how we learn and grow, but if the risks and costs are going to be high, they need to be known up front. Remember that you are a team, not adversaries.

Agreement 2

"Don't take anything personally" is the second agreement, and it is also important. An individual's feelings can be potentially injured by others whose manners are less sensitive. Sometimes, criticisms are made that are well intended but can sting on occasion if delivered carelessly. I trust that all designers take pride in their work, and when others make what are intended to be constructive comments, it can undercut our self-esteem, but it should not and cannot.

Don't react at the moment but ponder the event. If you step back for a moment, you may be able to better appreciate that the comments were not about the individual but about the product, which is separate from yourself. It also opens us to be more receptive to the message and potential improvement of the product. That said, one should also guard against being gaslighted by the occasional individual who does not fully appreciate or understand the idea and makes one question their own thinking and convictions. They are out there.

Agreement 3

Agreement three is, in some ways, akin to agreement 2. The advice not to make assumptions is not always easy to follow. If the designer assumes that what they have put together can be built because they have designed it, they may fall into a trap. Circling back to last month's comments, the reality of manufacturing's capabilities may not meet with assumptions designers make, and the project comes to a screeching halt.

One way to avoid this is, again, design with manufacturing. Don't forget your assumptions completely, however, because they could be a future springboard to something important yet to come—a proverbial "aha!" moment. But more likely, it will be something seemingly unmomentous. The great science fiction writer Isaac Asimov once sagely said, "The most exciting phrase to hear in science—the one that heralds new discoveries—is not 'Eureka!' but 'That's funny...'" It is the seemingly mundane that can often yield the greatest surprise and reward. Making assumptions can rob one of such moments.

Agreement 4

The final agreement is one that I think is perhaps the most important one: "Always do your best." No one should expect less than the best from themselves. I had a boss years ago that would flip out if he heard someone say something was "good enough." Good enough is never good enough; it is a capitulation to the mediocre. That said, the best that one can do may well vary from day to day. We all have on and off days. When the designer—or anyone, for that matter—stumbles and makes an error, they are human.

We strive for perfection; hopefully, we get close or even achieve it from time to time. However, we ought not to beat ourselves up when we fall short; we simply need to resolve to make our best better in the future. Design-

ers are the leaders of the electronics parade, as I have said before; they script the music, and the rest of the industry plays. If the industry hits the occasional discordant note, that's part of the process of making something special in the end—something all involved can celebrate when it is ultimately done right.

Summary

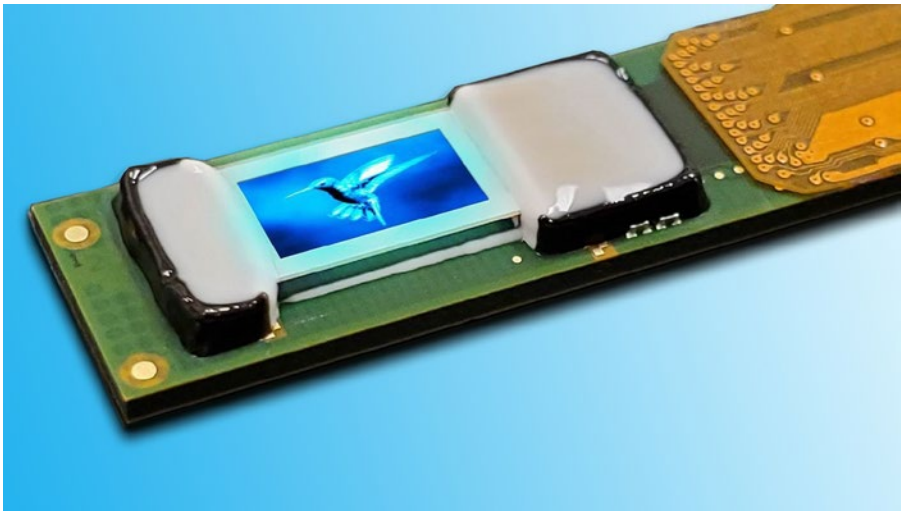
Things will not always go perfectly on the road from design to finished product. Expectations will not always be met on the first pass, but by making a few simple agreements with ourselves, we can hopefully navigate our way to the desired end with the least amount of angst and frustration.

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.

Compound Photonics Backplane Enables World's Smallest MicroLED AR Displays

Compound Photonics has announced the wide availability of its high-performance digital backplane to leading microLED developers worldwide for integration into complete microdisplay subsystems.

CP re-engineered its market-ready LCoS backplane technology into an innovative constant current drive configuration for microLED pixels based on its industry-leading 0.26" diagonal (~3 µm pixel) 1080p display format. MicroLED developers can accelerate their time to market by bonding their devices to a backplane driven by CP's field-proven NOVA display drive architecture to enable complete display subsystems meeting critical AR requirements for compactness, optical performance and brightness with high frame rate, low latency and low power consumption.



"Our custom, constant current pixel circuit design provides greater tolerance to forward voltage variation and IR drops in the microLED array resulting in a previously unattainable level of uniformity. It additionally features globally on-the-fly programmable pixel current control that greatly increases the system bandwidth, enabling higher frame rates while maintaining full bit depth," commented Ian Kyles, CP Vice President of Electrical/Software Engineering. "The backplane also has additional steering pixels beyond its native 2048x1080 resolution to enhance alignment/integration of the display within the optical system."

MicroLED developers using this backplane can access CP's monolithic integrated display module (IDM) (7.25 x 15.5 x 3.1 mm) with a low pin count interconnect and a direct MIPI input packaged into a compact subsystem amenable to smaller optical engine size. The IDM integrates CP's proprietary NOVA drive architecture's software defined platform to enable customizable frame-by-frame control of video frame rates (up to 240 Hz), bit depth, and other parameters to optimize for low latency, short persistence and low power while maintaining near 100% duty cycle according to type of image content and use case.

(Source: Business Wire)

1 This Month in Design007 Magazine: Design Economics With Kelly Dack ▶


Kelly Dack

Andy Shaughnessy recently spoke with Kelly Dack—CID, CID+, and a PCB designer and instructor who has worked in the design and manufacturing segments over the years. Thanks to his background, Kelly provides an intriguing viewpoint on cost-aware design and the philosophy of design economics in general.

2 Beyond Design: Transmission Line Termination ▶

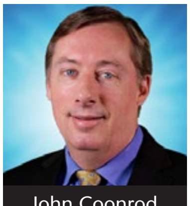

Barry Olney

Whenever a signal meets an impedance variation along a transmission line, there will be a reflection, which can seriously impact signal integrity. By understanding the causes of these reflections and eliminating the source of the mismatch, a design can be engineered with reliable performance. Barry Olney looks at how to effectively terminate transmission lines.

3 Mentor Video: Get To Know DDRx, SerDes, and PDN Tools ▶

This digital theater presentation demonstrates how HyperLynx contains full, automated pre- and post-route design flows for DDRx, SerDes channel, and power delivery network (PDN) design, with workshops that guide you through the process to get you up and running quickly.

4 Lightning Speed Laminates: Understanding Dk Data Key to Cost-aware Design ▶


John Coonrod

In the development stages of a circuit for a new PCB application, there are usually several iterations to the circuit. These many changes can be costly, and it is not uncommon for a project to have 4–8 changes before it can be released to the market. According to John Coonrod, one item that can substantially reduce the number of changes and the associated costs is the use of a good circuit simulation software.

5 Elementary, Mr. Watson: The Positive Side of COVID-19 ▸


John Watson

With the recent COVID-19 outbreak worldwide, most of us have been forced to reshuffle how we work, live, and play. Something like this has never happened before in our lifetimes, and it is scary and challenging, but difficult times develop resilient people. John Watson shares some of the positive things he has already noticed come out of this situation.

6 The Digital Layout: The Foundation of the PCEA Is Being Laid ▸


Stephen Chavez

Stephen Chavez highlights the Orange County Chapter's recent meeting and their transition from IPC to PCEA affiliation, recent PCEA activities, and the evolution of this column, including introducing Kelly Dack, CID+, PCEA's new communication officer.

7 Freedom CAD's Scott Miller: Taking Care of Customers and Staff ▸


Scott Miller

COO Scott Miller explains that Freedom CAD remains fully operational during the COVID-19 quarantine. Staff members have been telecommuting for years, so the company's day-to-day operations are relatively unchanged. He also discusses the company's plans to help employees and customers during this time, and Miller asks anyone with design questions—customers or not—to contact the company any time.

8 Cadence Helping Users to Save Time, Money With Automation ▸


Brad Griffin

During DesignCon, Andy Shaughnessy spoke with Brad Griffin, the group director for product management for the system analysis group at Cadence Design Systems. We discussed some of the areas where PCB designers can cut costs and how EDA companies can help these designers by automating certain time-consuming tasks. As Brad says, "The 'A' in EDA is for automation, right?"

9 Connect the Dots: The Seven-year Etch ▸


Stevenson and Tise

PCB etching seems like a simple task on the surface, but quite a few things can go wrong during this process. Adhering to best practice and continuous improvement is a must to help avoid issues with your finished board. Bob Tise and Matt Stevenson share their design tips for a better etching process.

10 The Shaughnessy Report: Design for Profitability Now Part of the Process ▸

It's easy to define profit, but it's much more difficult to define exactly what "design for profitability" (DFP) means to today's PCB designers and design engineers. How can technologists create profit in every design when the board's stakeholders are often spread out across several time zones and continents? It's a tough concept to get your arms around. Some of you work in giant OEMs; do you have any idea how much your last design cost—man-hours, components, laminates, etc.?

Career Opportunities

Career Opportunities

Image Department Operator

Alpha Circuit Corporation is a manufacturer of printed circuit boards located in Elmhurst, IL. We are currently seeking an operator in our Image department.

- All safety gear will be provided
- No experience required but a plus
- Full paid training provided
- Benefits: Health Insurance, 401(k), paid time off

Responsibilities:
- Expose dry film and liquid photo imageable ink
- Develop exposed photo imageable ink
- Develop exposed dry film
- Laminate dry film resist on inner layer and outer layer printed circuit panels
- Learn, understand, apply, and accept responsibility for in-process quality standards
- Be able to lift up to 15 lbs. shoulder high

If you are interested in this position, please contact Nita Buccino.
Email: nvb@alphacircuit.com,
cell: +1-847-489-2341.

apply now

Service Engineer
Schmoll Laser Drilling and Direct Imaging

Burkle North America seeks a full-time service engineer in the Northeastern U.S. This position will provide expert-level service on multiple laser drilling and direct imaging product lines. Install, commission, and maintain Schmoll products at multiple customer sites across the Northeast. The candidate will perform modifications and retrofits as needed. Maintain complete and detailed knowledge of Schmoll products and applications and handle a wide variety of problems, issues, and inquiries to provide the highest level of customer satisfaction. Assist customers with the potential optimization of their machine functions and work with clients on application improvements.

Qualifications

Required: Bachelor's degree from a technical college/university in an associated field. Three years directly related experience, or equivalent combination of education and experience. Must possess a valid driver's license and have a clean driving record.

Preferred: Experience in control systems and electronic troubleshooting, as well as in general electrical and mechanical service tasks. Experience and knowledge in the PCB manufacturing process, with a focus on laser drilling and/or direct imaging.

Send resume to hr@burkleamerica.com.

apply now

Career Opportunities

WHELEN® TRUSTED TO PERFORM™

Process Engineering Director

Whelen Engineering Co., Inc. seeks full-time process engineering director in Concord, NH, to develop, plan and execute GreenSource Fabrication, LLC Div.'s process technology business strategy; manage process engineering activities, staff and compliance; improve process design, cost, quality and resource utilization; interact w/ customers and incorporate feedback; develop financial capital and labor projections; travel internationally for conferences, supplier and customer visits (15-25% worktime); write white papers, IP applications and give talks re. Division's products/processes.

Min. req.: U.S. Bachelor's or foreign equivalency in environmental science or engineering; min. 10 yrs. work exp. in: PCB fabrication process engineering; comprehensive and current experience in PCB fabrication/substrate markets w/ SAP tech; developing chemical and mechanical processes, chemistries and equipment for PCB manufacturing demonstrated by international experience implementing complex processes; ability to direct and troubleshoot PCB manufacturing problems; min. 5 years exp. leading, managing and training process engineering teams, developing and executing process technology business strategies and plans in worldwide PCB markets, including Japan, Taiwan, China, Europe; min. 3 years exp. giving talks, writing and presenting white papers; ability to travel internationally (15-25% worktime).

Send CVs to: Corinne Tuthill,
ctuthill@greensourcefab.com
or GreenSource Fabrication, LLC,
99 Ceda Road, Charlestown, NH 03603.

apply now

FREEDOM CAD SERVICES

Sr. PCB Designer–Allegro

Freedom CAD is a premier PCB design service bureau with a talented team of 30+ dedicated designers providing complex layouts for our enviable list of high-tech customers. Tired of the commute? This is a work-from-home, full-time position with an opportunity for overtime at time and a half.

Key Qualifications

- EXPERT knowledge of Allegro 16.6/17.2
- Passionate about your PCB design career
- Skilled at HDI technology
- Extensive experience with high-speed digital, RF and flex and rigid-flex designs
- Experienced with signal integrity design constraints encompassing differential pairs, impedance control, high speed, EMI, and ESD
- Experience using SKILL script automation such as dalTools
- Excellent team player that can lead projects and mentor others
- Self-motivated, with ability to work from home with minimal supervision
- Strong communication, interpersonal, analytical, and problem solving skills
- Other design tool knowledge is considered a plus (Altium, PADS, Xpedition)

Primary Responsibilities

- Design project leader
- Lead highly complex layouts while ensuring quality, efficiency and manufacturability
- Handle multiple tasks and provide work leadership to other designers through the distribution, coordination, and management of the assigned work load
- Ability to create from engineering inputs: board mechanical profiles, board fabrication stack-ups, detailed board fabrication drawings and packages, assembly drawings, assembly notes, etc.

apply now

Career Opportunities

Career Opportunities

Career Opportunities

Career Opportunities

U.S. CIRCUIT

Sales Representatives (Specific Territories)

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

- Florida
- Denver
- Washington
- Los Angeles

Experience:
- Candidates must have previous PCB sales experience.

Compensation:
- 7% commission

Contact Mike Fariba for more information.

mfariba@uscircuit.com

apply now

ZENTECH

Zentech Manufacturing: Hiring Multiple Positions

Are you looking to excel in your career and grow professionally in a thriving business? Zentech, established in Baltimore, Maryland, in 1998, has proven to be one of the premier electronics contract manufacturers in the U.S.

Zentech is rapidly growing and seeking to add Manufacturing Engineers, Program Managers, and Sr. Test Technicians. Offering an excellent benefit package including health/dental insurance and an employer-matched 401k program, Zentech holds the ultimate set of certifications relating to the manufacture of mission-critical printed circuit card assemblies, including: ISO:9001, AS9100, DD2345, and ISO 13485.

Zentech is an IPC Trusted Source QML and ITAR registered. U.S. citizens only need apply.

Please email resume below.

apply now

Career Opportunities

BLACKFOX
Premier Training & Certification

IPC Master Instructor

This position is responsible for IPC and skill-based instruction and certification at the training center as well as training events as assigned by company's sales/operations VP. This position may be part-time, full-time, and/or an independent contractor, depending upon the demand and the individual's situation. Must have the ability to work with little or no supervision and make appropriate and professional decisions. Candidate must have the ability to collaborate with the client managers to continually enhance the training program. Position is responsible for validating the program value and its overall success. Candidate will be trained/certified and recognized by IPC as a Master Instructor. Position requires the input and management of the training records. Will require some travel to client's facilities and other training centers.

For more information, click below.

apply now

Professionals Seeking Employment

D.B. Management Group L.L.C. is currently working with many professionals who are seeking new positions. If any of these qualified professionals sounds like someone you would like to learn more about, contact **Dan Beaulieu** at **207-649-0879** or danbbeaulieu@aol.com. If you are a qualified professional looking for a new opportunity, contact Dan as well. Fees are 10% of candidates' first year's annual compensation. There is no fee for candidates.

Click here to learn more ▶

President, Company Leader, Business Builder
This professional has done it all. Built new businesses and turned around hurting businesses and made them successful. A proven record of success. This candidate is a game-changer for any company. He is seeking a full-time leadership position in a PCB or PCBA company.

General Manager PCB and PCBA
Senior manager with experience in operations and sales. He has overseen a number of successful operations in Canada. Very strong candidate and has experience in all aspects of PCB operations. He is looking for a new full-time position in Canada.

Regional Sales Manager/Business Development
Strong relationship management skills. Sales experience focused on defense-aerospace, medical, high-tech PCB sales. Specializes in technical sales. Also has experience in quality, engineering, and manufacturing of PCBs. He is looking for a fulltime position in the Southeastern U.S.

Field Application Engineer (FAE)
Has worked as a respected FAE in the U.S. for global companies. Specializes in working alongside sales teams. Large experience base within the interconnect industry. He is looking for a full-time position.

Business Development Manager
Understands all aspects of interconnect technical sales from PCB design and fabrication to assembly and all technologies from HDI microvias to flex and rigid-flex. Has also sold high-tech laminates and equipment. Proven record of sales success. He is looking for a full-time position.

CEO/President
Specializes in running multi-million dollar companies offering engineering, design, and manufacturing services. Proven leader. Supply chain manager. Expert at developing and implementing company strategy. Looking to lead a company into the future. He is looking for a full-time position.

PCB General Manager
Forty years of experience serving in all capacities, from GM to engineering manager to quality manager. Worked with both domestic and global companies. Available for turn-around or special engineering projects. He is looking for long-term project work.

Process Engineering Specialist
Strong history of new product introduction (NPI) manufacturing engineering experience: PCB/PCBA. Held numerous senior engineering management positions. Leads the industry in DFM/DFA and DFX (test) disciplines. He is looking for either a full-time position or project work.

VP Sales Global Printed Circuits
Worked with a very large, global company for a number of years. Built and managed international sales teams. Created sales strategies and communicated them to the team. One of the best sales leaders in our industry. He is looking for a full-time position.

Plant Manager
This professional has years of experience running PCBA companies. Led his companies with creative and innovative leaderships skills. Is a collaborative, hands-on leader. He is looking for a full-time position.

National Sales Manager
Seasoned professional has spent the past 20 years building and growing American sales teams for both global and domestic companies. Specializes in building and managing rep networks. He is looking for a full-time position.

Global Engineering Manager/Quality Manager
Has experience working with large, global PCB companies managing both engineering and quality staff. Very experienced in chemical controls. She is interested in working on a project-by-project basis.

CAM Operators and Front-end Engineers
These candidates want to work remotely from their home offices and are willing to do full-time or part-time projects.


Need better thermal management techniques?
THE PRINTED CIRCUIT DESIGNER'S GUIDE TO...™
Thermal Management with Insulated Metal Substrates
Didier Mauve and Ian Mayoh
Ventec International Group
I-007e Books
Download now.
FREE EBOOK
I-007e Books

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

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

SALES MANAGER: **BARB HOCKADAY**
(916) 608-0660; barb@iconnect007.com

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

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

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

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

ASSOCIATE EDITOR: **KIERSTEN ROHDE**
kiersten@iconnect007.com

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

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

TECHNOLOGY SPECIALIST: **BRYSON MATTIES**

COVER: **SHELLY STEIN**

COVER IMAGE: **ADOBE STOCK © Drobot Dean**

DESIGN007 MAGAZINE

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

May 2020, Volume 9, Number 5
DESIGN007 MAGAZINE is published monthly,
by BR Publishing, Inc.

ADVERTISER INDEX