I-Connect007

JANUARY 2019

DESIGN007 MAGAZINE

Designing for the Supply Chain Shortage

DESIGN007 MAGAZINE

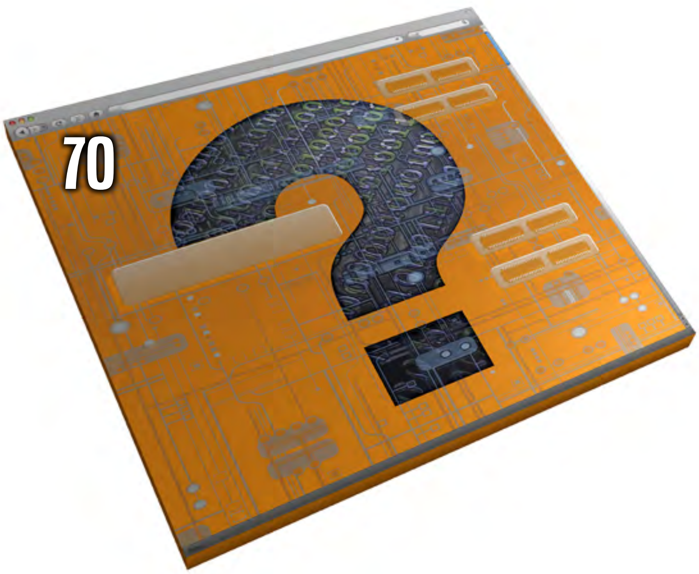
The Supply Chain Shortage

The component shortage is getting crazy. Some PCB designers are finding their favorite capacitors on 50-week and 80-week lead times, or worse. How do you design a board today when the components you need won't be available for a year or more?

But don't expect any sympathy from component suppliers; they've watched their profit margins continue to shrink over the years, and in their eyes, the OEMs have been reaping the rewards.

Where does this leave you and your next design? This month, we asked our expert contributors to explain the current component shortage, as well as some of the workarounds that can help you get your next design out the door sooner rather than later.

DESIGN007 MAGAZINE



Beating Supply Chain Blues

The Shaughnessy Report
by Andy Shaughnessy, I-CONNECT007

The waiting is the hardest part. –Tom Petty

When I got my driver's license in 1979, my dad rejoiced. No, not because he wanted me to enjoy my newfound freedom and get used to making decisions on my way to becoming a normal adult and valued member of the community. He just wanted someone else to wait in the gas lines.

Some of you won't remember this, but gas lines were an everyday occurrence for a year or so in the 1970s. The "energy crunch" was on, and President Jimmy Carter told us to put on a sweater instead of cranking up the heat in the winter. OPEC raised the cost of crude oil to the point that gas cost over $1. The old analog gas pumps could only register up to $0.99 per gallon, so gas stations had to charge by the half gallon until they could get new pumps.

But Tom Petty had it right; waiting was the worst part. It wasn't uncommon to wait two or three hours in a gas line, and sometimes the gas station would run out of gas before you got yours. Then, things got interesting. Tempers would flare as drivers tried to get out of this traffic jam and head to another gas station, and the process would start all over again.

My dad said that I was building character by waiting in gas lines. I don't know about that. I just remember trying to look cool in my mirrored sunglasses and listening to the radio for three hours. By Sunday afternoon, most gas stations had run dry, and they'd put up signs that read "out of gas." You might drive past a

"Gas by Appointment" sign in Portland, Oregon before odd-even gas plan was put into effect in 1979. (Source: Wikimedia)

Out of gas, but looking cool...Andy, circa 1979.

half-dozen gas stations before you found one with gas, and there might be a four-hour line.

I was reminded of the energy crunch of the '70s while researching this month's issue on the component shortage. Some PCB designers are finding their favorite capacitors on 50- and 80-week lead times, or worse. How do you design a board today when the components you need won't be available for a year or more? Waiting isn't an option if your product needs to be on store shelves in time for next Christmas. What options do designers have?

Things are getting crazy. In many software tools, the field for lead time only has two digits, so designers can only input "99 weeks" even if the wait is much longer. How did we get to this situation?

Component makers seem to be focusing on adding capacity to handle products of the future, and OEMs who manufacture mature legacy products are getting hit the hardest. Component distributors have responded by raising the prices on the available caps, resistors, and discretes. We've heard horror stories of components going from $0.05 a piece to over $1 apiece in the space of a week, and that's only if you buy one million at a time. Electronics companies will pay whatever they have to for the components they need, and you know that smaller OEMs are feeling the pinch right now.

But don't expect any sympathy from component suppliers; they've watched their profit margins continue to shrink over the years, and in their eyes, the OEMs have been reaping the rewards. Turnabout is fair play, after all.

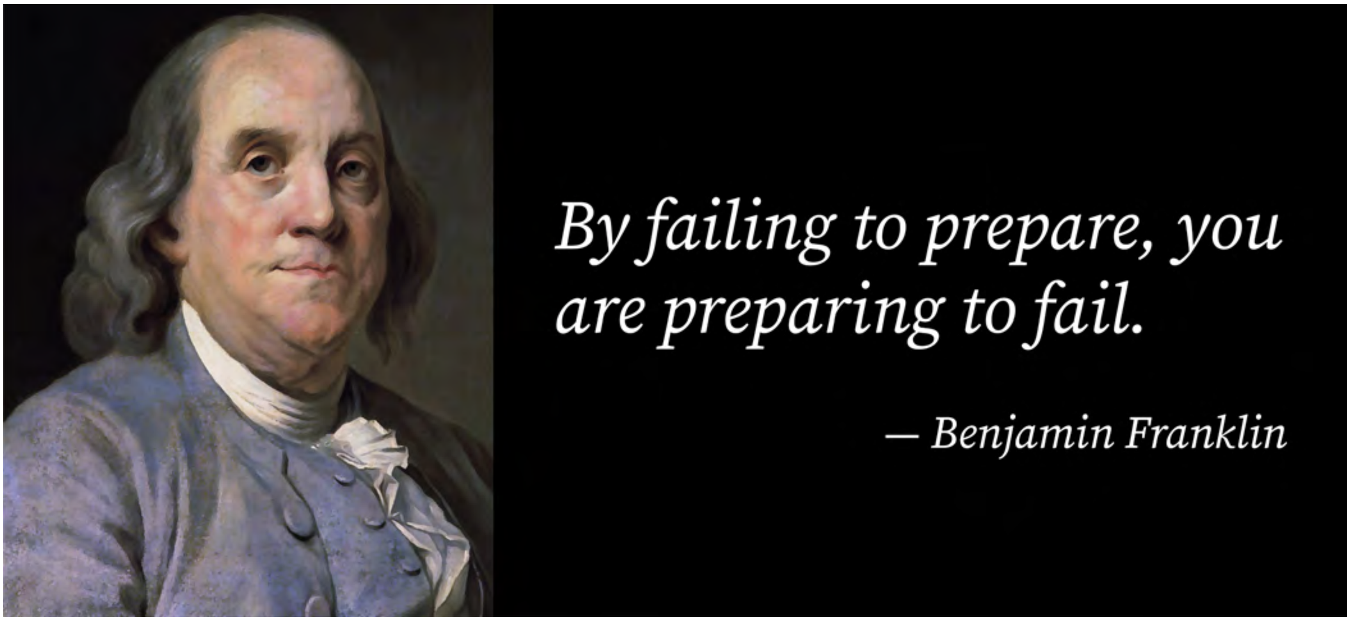
Where does this leave you and your next design? We asked our expert contributors to explain the current component shortage, as well as some of the workarounds that can help you get your next design out the door sooner rather than later. In our lead story, LeGrand's John Watson, CID, discusses how the industry arrived in this predicament, the major drivers behind this problem, and the need to ensure component availability well before beginning to design a PCB.

Next, we have an interview with SnapEDA CEO Natasha Baker. She explains how transparency in online libraries can help designers facing supply chain issues and offers strategies for meeting parts availability challenges head-on. Then, we bring you a conversation with Digi-Key COO Dave Doherty who discusses his efforts to keep customers supplied with components during this perfect storm, and the need for designers to look at other options if their component of choice is unavailable in the near term. Also, Carl Schattke, automotive PCB designer and CID+, shares some of his techniques for stackup design, along with processes that can help designers dealing with component shortages such as the need to make supply chain decisions as early in the design cycle as possible.

We also have columns from our regular contributors including Barry Olney, Stephen Chavez, Vern Solberg, Tim Haag, Bob Tise and Dave Baker, and Alistair Little. Next, we bring you a technical article by Yuriy Shlepnev, president of Simberian, on the effects of meshed reference planes on interconnect.

I know some of you are getting ready for DesignCon and IPC APEX EXPO. I-Connect007 will be covering both shows (which take place simultaneously this year, just to make things interesting), and I hope to see you on the road. **DESIGN007**

Andy Shaughnessy is managing editor of *Design007 Magazine*. He has been covering PCB design for 18 years. He can be reached by clicking here.



The Electronic Component Shortage Crisis: A Veteran Engineer's Perspective

Feature by John Watson, CID
BUILDING CONTROL SYSTEMS, LEGRAND INC.

Most of us are familiar with the defense readiness condition (DEFCON) levels. If we could apply such a DEFCON level to the electronics industry and how it relates to the issue of component shortages, we would be sitting at a DEFCON 1, meaning nuclear war is imminent.

This issue has a huge effect on everyone who works in the electronics field, impacting schedules and how things are done. The problem began to hit in earnest about a year ago in 2017. It first appeared as longer lead times on multilayer ceramic capacitors (MLCCs) and tantalum capacitors. With longer lead times, the available stock began to disappear, and before we knew it, we had a perfect storm and an international crisis.

From where we stand now, at the beginning of 2019, we see lead times for some components in the short range of up to 16 weeks; medium-to-high is 32 weeks, and long lead times are as far out as 80 weeks. In other words, if we ordered a component today, it would arrive in over a year a half from now (maybe).

This all started with the capacitors (we will see why later), but we now see other component series being sucked into this problem (Figure 1).

With such volatility in the market, it has brought those who were not prepared for it to a standstill.

- Board Mount Temperature Sensors
  (~ 5 week lead time)
- Power Switch ICs
  (~6 to 10 week lead time)
- MOSFETs
  (~26+ week lead time)
- Suppressors/Diodes
  (~49 week lead time)
- Operational Amplifiers & Relays
  (~52 week lead time)
- Thick/Thin Film Resistors
  (~6 to 80 week lead time)

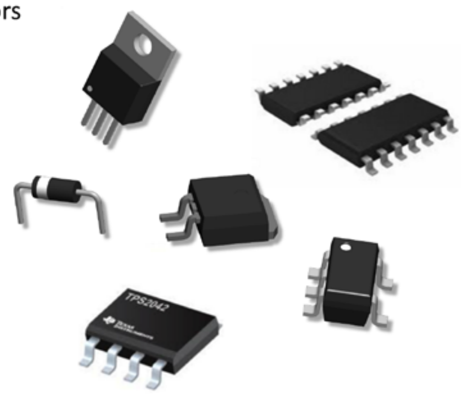
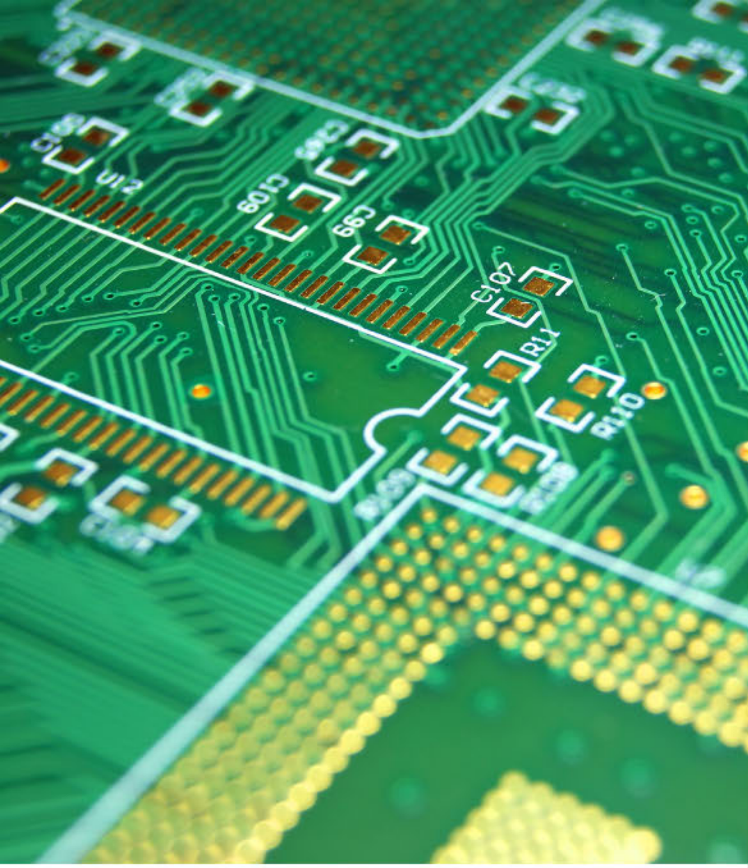


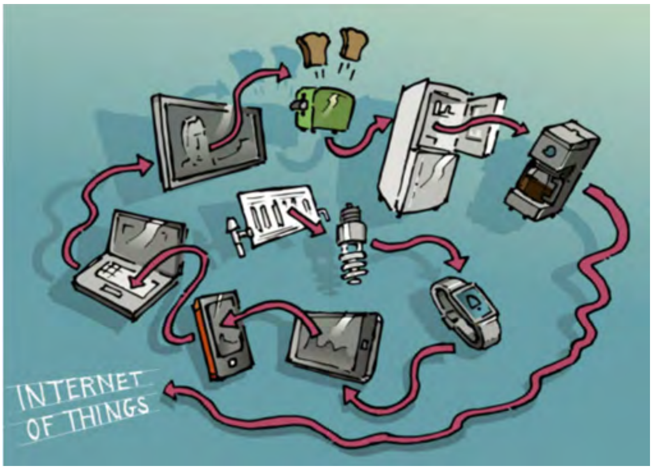
Figure 1: Common parts in short supply with typical lead times.



What Is Driving This Crisis?

It has come down to a simple principle of economics supply and demand. While the supply has gone way down, the demand for many components has exploded. The component manufacturers have not kept pace with the demand for several reasons.

There are three main sectors of the electronic industry driving these component shortages.



1. Internet of Things (IoT)

Demand for smart devices has exploded, ranging from TVs to Bluetooth speaker systems; home devices from Amazon, Google, Apple, and others; renewable energy products; solar panels; and cloud computing. It seems like we now have a "smart" everything!

On the more humorous side, what happens when IoT goes bad? For example, if you go to the refrigerator to add a dozen donuts to your shopping list, and the fridge has been connected to and talking with the scale, does it decide to place you on a diet instead?

According to Gartner, there will be more than 20 billion new IoT devices deployed by 2020, which will be a 100% growth rate in the number of these devices in the next couple of years. That's a lot of hardware to be placed into the market. It is estimated that there will not be an industry or area of our lives that will not be touched by IoT in some way.

2. Mobile Phone Industry

Personal phone usage has seen a huge increase; it was recently reported to have doubled since 2015. If you want evidence of this, go to your local mall, sit in the food court, and watch people—everyone is completely engrossed in their mobile device. It is estimated that approximately 1.5 billion smartphones will be manufactured in the upcoming year, and each flagship model contains roughly 1,000 capacitors. The current estimate is that there is a worldwide production capacity of 3 trillion MLCC capacitors. By those numbers, nearly 50% of the MLCC capacitors produced are already designated and used strictly in the mobile cellphone sector.

3. Automotive

The hybrid and full electric vehicle industry currently has double-digit growth. However, technical advancements have spilled over into traditional gasoline vehicles with the addition of new technologies in automated driving systems (ADS) including all the new automated gadgets such as parking sensors, auto windscreen wipers, etc.

A standard combustion engine car requires somewhere between 2,000–3,000 capacitors. An electric vehicle has up to 22,000 capacitors required in a single car. Furthermore, the higher temperatures inside the control circuits

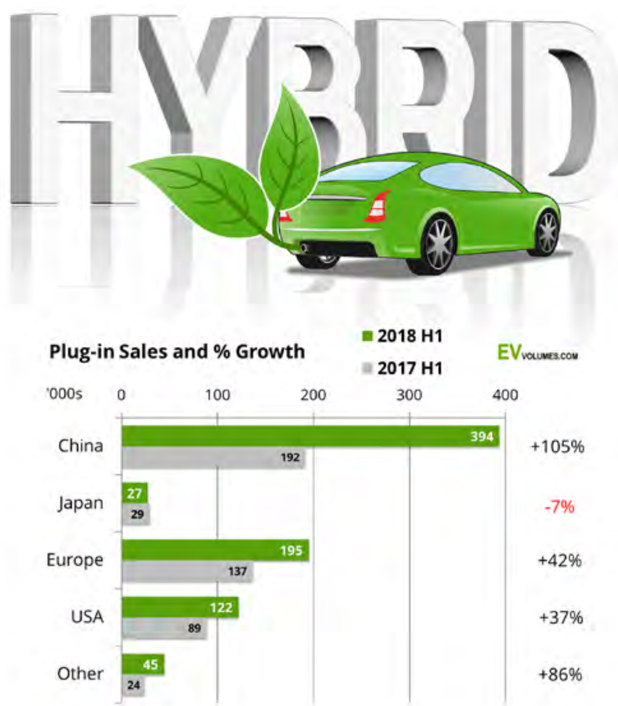


Figure 2: Global hybrid car sales. (Source: ev-volumes.com)

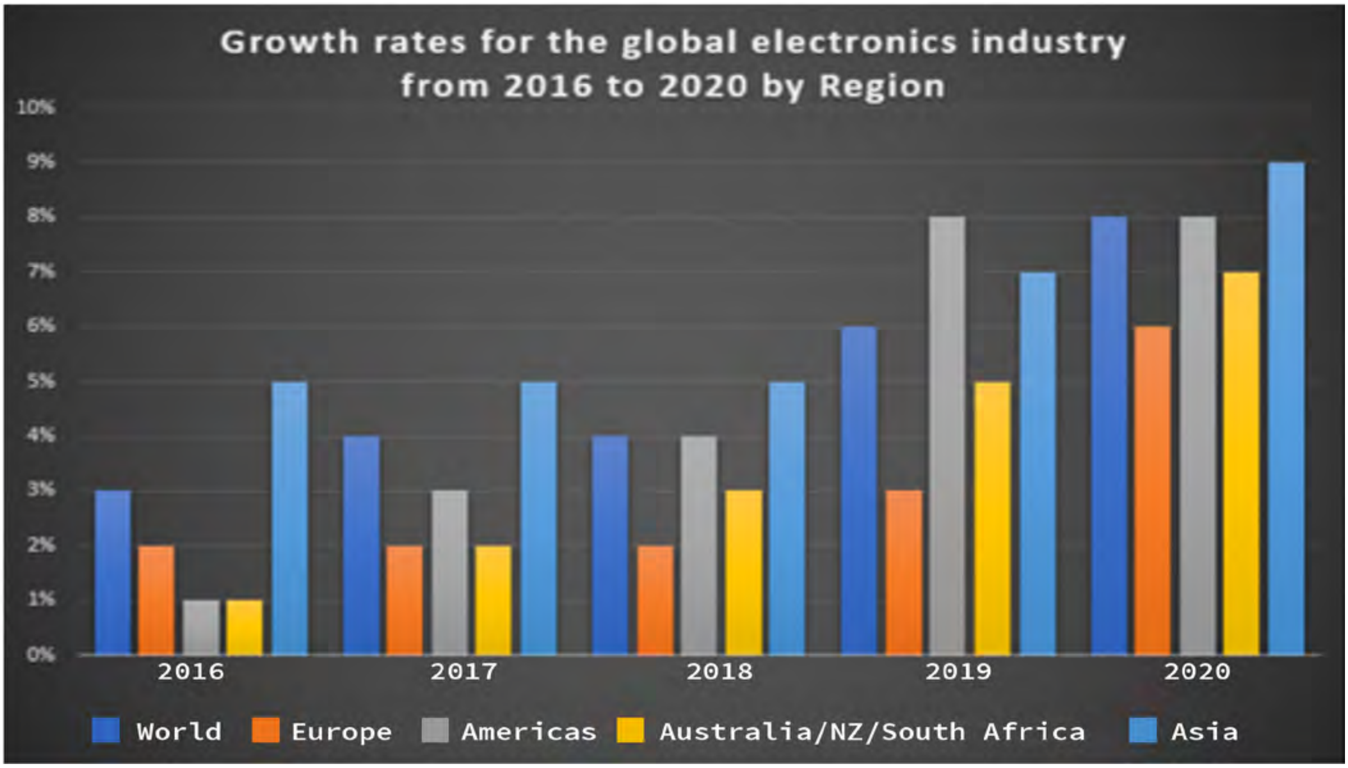


Figure 3: Growth rates for the global electronics industry.

of electric vehicles mean that traditional plastic film capacitors are no longer suitable, so ceramic MLCCs are increasingly being used. This requirement for MLCCs has brought about a new regulatory agency within the Automotive Electronics Council (AEC) whose mission is to promote the standardization of reliability and qualification for automotive electronic components including high-temperature and high-humidity resistance, thermal-shock resistance, and durability.

As a new agency, AEC needed standards and requirements such as:

- AEC-Q100: Integrated circuits (ICs)
- AEC-Q101: Discrete semiconductor components (transistors, diodes, etc.)
- AEC-Q200: Passive components (capacitors, inductors, etc.)

With these new standards, it is estimated that nearly 50% of the components tested have failed, which has resulted in a five-fold increase in the demand for these specific electronic components.

How Long Will This Crisis Last?

The million-dollar question is, "How long do experts think that this will last?" I am not exactly sure, but I do not see this ending anytime soon due to several main reasons:

1. Expected Growth in the Electronic Industry

According to Statista, the electronics industry is estimated to grow 6% in 2019 and 8% in 2020. That is great news for our entire industry, but it comes with some major problems. First, will there be enough coffee to support this massive engineering effort? And second, all of this growth will require new hardware (Figure 4).

2. Part Manufacturers to End Entire Lines of Less-profitable Components

The parts on the chopping block are some of the larger package or case sizes such as anything above 0603 for discrete components. Manufacturers are closing those lines to convert them over to the higher demand components. This narrows the component selection and reduces the supply.

3. Self-inflicted Problems

Many of the part vendors have now switched over to Allocation. In Allocation, manufacturers divide the available inventory, so only a percentage of the stock is given to specific manufacturers. The suppliers, of course, want to work with those companies that place the biggest orders.

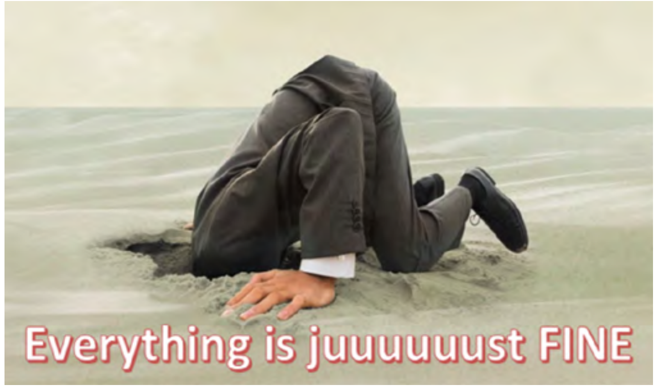
On the company side, Allocation has caused an absolute panic. To make sure to have the available components, the new common practice is to double and triple order quantities. Then, the parts consumers stockpile these components for future use. This only puts a further strain on an already fragile system of supply and demand.

Being on the front line with a manufacturing company working to get products off of the drawing board and into a sales catalog, this parts availability issue has had a huge impact on every aspect of our business. From the very first day of a new design, we are forced to question what components we use. However, the problem is that we don't know what the specific conditions are from moment to moment. We have seen the problem from both sides in that components we originally thought would not be a problem have become a problem halfway through a design (and vice versa). It has gotten so bad that on some occasions, once we finally did locate components, the parts would be already gone before we could fill out a purchase order minutes later.

How Do We Get Through This Crisis?

What I recommend to all the PCB designers I work with is to follow a simple rule: This is not business as usual. Do not assume anything in such an environment.

The attitude in a dog-eat-dog world by some is not allow a crisis to go to waste. Many black market and counterfeit components have started to pop up. We have seen parts in our supply chain that were believed to be counterfeit because they did not match the specifications on the datasheet. Thus, the components we can find and get into designs cannot be trusted. My only response to that is, "Ouch."



This crisis will end. It may take a while for supply to catch up with demand, but it will end. One option—although not a good one—is to bury your head in the sand and hope it all goes away. But this problem will not go away any time soon. The sooner that we realize that as an industry, the better.

Here are some suggestions to get through this crisis:

1. Design Outside of the Norm

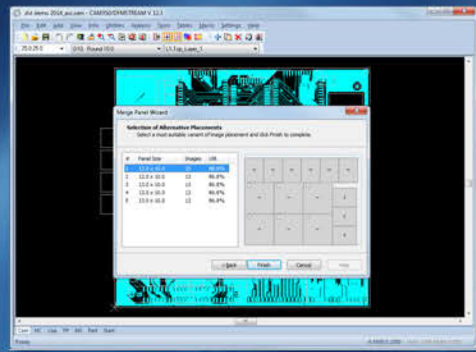
Since we know that there is a narrowing of the supply and a reduced supply for some components, we need to be flexible. Look and see if any of the specific parameters of a component can be adjusted out of what would be considered "the norm." For example, "Do we need to use a 0.1-uF 1% cap for our bypass caps?" If you have any ICs on your design at all, the answer to that question is, "Probably." Since this is the most common bypass cap value, everyone is scrambling for that specific component. You will be able to open more stock of components by changing the parameters especially by taking it up to a 5% tolerance. Many times, the components used are overkill for the class of design. Making that simple change has resulted in a whole new available stock of components.

2. Let the PCB Designer Drive Other Areas of the Process

Now, we are giving over to procurement the list of probable components that may be a

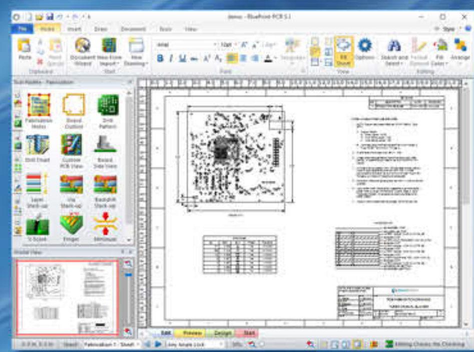
Solutions to Ensure PCB Manufacturing Success!

CAM350®



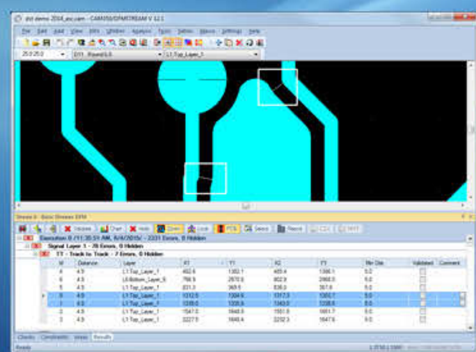
Verify and Optimize PCB Designs for Successful Manufacturing.

BluePrint·PCB



Create comprehensive documentation to drive PCB fabrication, assembly and inspection.

DFMStream™



Verify Design and Manufacturing Rules Any Time During the PCB Design Cycle.

At DownStream Technologies we are committed to delivering comprehensive solutions and workflow process improvements for all your PCB design post processing needs.

For more information please visit downstreamtech.com or call 800-535-3226 or stop by booth #1716 APEX EXPO IPC 2019

DownStream Technologies

problem early in the design process. This has been facilitated by using tools in Altium 18 such as ActiveBOM. Before, it was the other way around; often, engineering and procurement wouldn't even speak to one another. Now, we are making sure that the information available to us through ActiveBOM is being filtered throughout the company, so the decisions that affect the outcome of a design are known and decided on early in the process not once it is too late. Nobody likes getting those calls from the assembly house telling us that a part has been placed on a design that is no longer available; most likely, a design must be redone because of it. Further, no one likes making that long walk to the manager's office to explain why the deadline was missed because of failing to plan.

3. Use Multi-footprint Components

Benjamin Franklin once said, "By failing to prepare, you are preparing to fail." To that end, we have started to use multi-footprint components, laying multiple footprints on the design for alternative components that we might use if there is a shortage of our first choice. This has resulted in some interesting layouts to accommodate everything.

In addition, there are the three "Ps" to get through this crisis: plan, prepare, and be proactive. Stay ahead of the problem by not starting with a design that is already in trouble. Many times, engineers will use a previous design thinking that those components are available when many of them may be obsolete or in a deprecated state already. We cannot assume that everything is okay. If you see that parts are already having stock problems or are not recommended for new designs, that situation will not get any better over time—it will probably get worse. Also, realize the further someone gets into a design, the harder it is to make changes without impacting your wallet or time schedule.

A Helpful Tool

ActiveBOM has become one of the most important tools for us. Since knowledge is power, being able to see "live" component stock availability information and appropriate AVLs is crucial. ActiveBOM gives us the ability to rank vendors and set up multiple sources

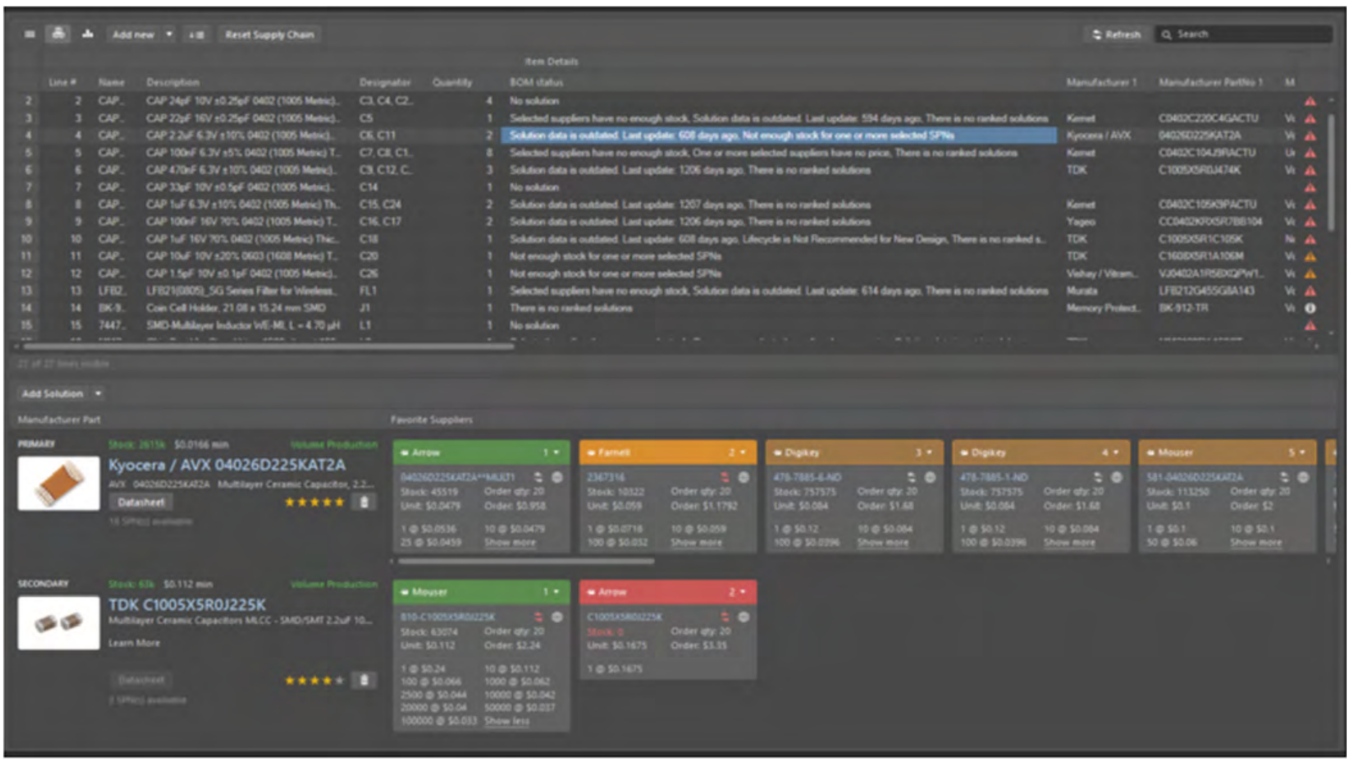


Figure 4: Screenshot of Altium's ActiveBOM tool.

and vendors for each component. We can find this information readily at our fingertips; we do not need to wait until the back end of a design to find out that we just created the company's latest doorstop and stock of bad PCBs. Once we get that schematic done, we can run it into ActiveBOM and find out where our problems are. If there was a second rule after "it is not business as usual," it would be "check component availability often throughout the design process."

Conclusion

In summary, I would recommend that you stay informed. Many of the component vendors publish their component forecasts. Stay aware of trends in our industry. The sooner you know of the problem or the direction, the faster you can make a sound decision on needed changes. This will require reading electronic journals and news. Stay up to date with what is being reported by some of the great PCB industry leaders such as I-Connect007. Sooner or later, the industry will back away from DEFCON 1. **DESIGN007**

John Watson's career has spanned over 20 years in PCB design. His experience includes various manufacturing companies and PCB design service bureaus with diverse projects such as high-density digital, DDR, analog, power supply, and high-frequency RF. Now, as a senior PCB engineer at Building Control Systems of Legrand Inc., Watson leads innovative PCB design teams of 50 designers based in several divisions that span the globe where he emphasizes training and mentoring. He has become very proficient in the PCB design process flows and standardization. In addition, Watson is a highly sought out consultant, writer, and conference speaker.

How Game Theory Can Bring Humans and Robots Closer Together

Researchers at the University of Sussex, Imperial College London, and Nanyang Technological University in Singapore have used game theory to enable robots to assist humans in a safe and versatile manner.

The research team used adaptive control and Nash equilibrium game theory to program a robot that can understand its human user's behavior to better anticipate their movements and respond to them. The researchers believe the breakthrough could help robots complementing humans for sport training, physical rehabilitation or shared driving.

Lead author Dr. Yanan Li, lecturer in control engineering at the University of Sussex, said, "It is still very early days in the development of robots and at present, those that are used in a working capacity are not intuitive enough to work closely and safely with human users. By enabling the robot to identify human users' behavior and exploiting game theory to let the robot optimally react to them, we have developed a system where robots can work along humans as humans do."

Professor Etienne Burdet, Chair in Human Robotics in the Department of Bioengineering at Imperial College London and senior author of the paper, added, "Game theory has had important impacts in economics during the last century and lead to several Nobel prizes such as Nash's one. To apply it for human-robot interaction, it was necessary to understand how the robot can identify the human user's control goals simultaneously to smoothly interacting with them."

(Source: University of Sussex)

Natasha Baker: Supply Chain Transparency Inside the CAD Tool

Feature Interview by Nolan Johnson
I-CONNECT007

Natasha Baker, CEO and founder of SnapEDA, an online parts library, discusses the benefits of transparency in online libraries to designers, and discusses strategies on how to solve supply chain challenges, and more.

Nolan Johnson: Natasha, can you give us an overview of what your company does?

Natasha Baker: My company is SnapEDA. We launched in 2013 and built the internet's first parts library for circuit board design. SnapEDA is a place for engineers to access the libraries they need to build PCBs. Users can drag and drop symbols, footprints, and 3D models right from our website. There are many challenges to face when designing a circuit board in both the design and manufacturing process. Our mission is to remove the barriers and make the process more seamless when bringing a product to life.

Johnson: This is an emerging space in the entire PCB marketplace. How do you handle conversations around supply chain issues?

Baker: What we do is give engineers transparency into different areas that they've never had transparency in before, including where to find the part, how to select parts that have availability in the marketplace, how to substitute a part if it goes out of stock, and how can you find a part with particular specifications. The biggest thing we're giving engineers is ready-to-use data, transparency into the quality of that data, and a view into the supply chain. We address questions such as, "Is this component available? Who has it in stock? Would this part be wise to put into my design, or is the component obsolete and I should not use it?"

Johnson: What are some ways that transparency might help your users and what are the bottlenecks your users are talking to you about in this area?

Baker: The most direct example is that engineers can see right away whether an item is in stock or not, and how much it costs at various distributors. If it's not in stock, we'll recommend one they can use with similar specifications and functionality, and even give them the CAD files.

Johnson: The traditional way to specify parts is by performance characteristics, but now you have supply chain availability to consider. It seems like your customers can now shop for the functionality of a part appropriate for a particular design. Are these the dynamics you're seeing from your customers?

Baker: Definitely. Since we have built up a large database of vendor specifications, we can make recommendations for similar parts. If we see that the one that the engineer wanted to use (or maybe even currently is being used in a product that's being updated) is obsolete, we'll recommend one that is in stock that has specifications. This is a win-win, because no one likes to rework their design and have to scramble to find a substitute part.

This is a win-win, because no one likes to rework their design and have to scramble to find a substitute part.

Johnson: As a broker between the designers and the manufacturers, does your community have discussions about parts availability or counterfeit parts? If so, how do you communicate availability issues to your users and are you actively doing anything to identify counterfeit parts?

Baker: Yes, it's definitely a concern for our community. Since we promote distributors to our community, it's important that, we only work with authorized distributors. Since this is such a huge problem for engineers, we made the conscious decision to only work with trusted, authorized distributors.

In the supply chain, we find ourselves in an interesting position because we're engaging at this pivotal point where engineers are making their design decisions. Engineers look to us to make sure the components they choose are going to be right for their designs and that we'll only link them to trusted sources. We now sell components directly from their websites. Getting components directly from the manufacturers is another way that engineers can guarantee they are ordering are legitimate.

Johnson: You're involved in the conversation between the designers and the distributors who sell the parts in a way that hasn't been done before. It seems like distributors could positively respond to you or see you as a disruptor. How would you describe the relationship between you and the distributor?

Baker: We have a very collaborative and complementary relationship with distributors. Our website is a place to discover and design-in components, but ultimately, we're referring designers to distributors to buy. If you visit one of these distributors' warehouses, it's absolutely incredible the logistics that go into what they do. It would be very hard for any company, except maybe Amazon, to disrupt these companies.

Overall, we want to help engineers discover components and design boards. We also want to connect people to the various places in the supply chain and help break down barriers without replacing anything that already exists. We're not looking to replace anyone's business. We're a new type of business model that hasn't existed before. We've invented our own niche that provides value to the vendors we work with and the distributors, which is why we're seeing traction.

Johnson: Does it seem like the majority of your users are active in accessing your database?

Baker: Yes, we have a very active user base.

Johnson: Once they join and see the value, do they keep coming back to what you have to offer?

Baker: Absolutely. A lot of people don't get what we do at first. But once they try it, they keep coming back because it's a bit of a magical experience.

Johnson: Is one body of your audience easier to integrate with than the other?

Baker: In the beginning, it was easier to work with distributors because they instantly see the value since people buy components. The vendor portion took longer, but now our value is clear to them. We're working with them more. I'm excited about this because I feel like we can help them a lot.

Johnson: Where do you see your product going in the next three to five years?

Baker: There are so many things I want to do and that I'm so excited about, but I think I'm going to keep them up my sleeve for now.

Johnson: One thing we can take away from this conversation is that you have a pretty extensive roadmap for the next three to five years.

Baker: Yes. I feel like we haven't even scratched the surface of where I want to go. Businesses can be hard because you always want to do so many things, but you can only do a portion of them. It's a constant frustration.

Johnson: That makes sense, but it seems like you have a good strategy that's disciplined and will keep you focused. Before we finish, is there anything else you'd like to discuss?

Baker: We're working something really special for electronics engineers and PCB designers over here at SnapEDA, which is so much more than just a symbol and footprint library. Yes, we're a company, but this is just as much a passion project for our team members, built out of our own experiences designing PCBs. We've made an incredible free product that saves engineers so much time, and we're constantly improving it based on the feedback we get from our community. I'd definitely recommend that readers create a free account and give it a try!

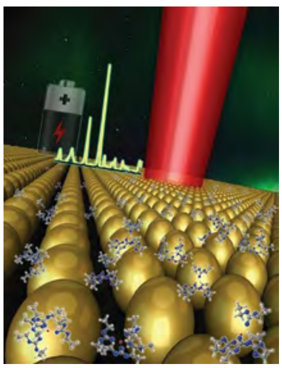
Johnson: Sounds awesome. Thank you for your time, Natasha.

Baker: Thank you so much for the opportunity to be involved in this. **DESIGN007**

Batteries—Solid-liquid Interface

By studying the inner workings of lithium-ion batteries, Oak Ridge National Laboratory researchers have developed a highly sensitive technique to characterize and measure at the electrolyte and electrode interface.

Their finding, published in *ACS Nano,* could help in understanding the fundamental factors that determine the composition and stability of solid electrolyte interphase, or SEI.



"A robust SEI is key to the performance and safety of Li-ion batteries used to power electric vehicles," said Jagjit Nanda, senior staff scientist and team lead, spectroscopy and interfaces, at ORNL.

Li-ion batteries comprise positive and negative electrodes, each containing an electrolyte, or salt, solution, separated by a membrane. The researchers used surface enhanced Raman spectroscopy to evaluate how the lithium-salt interacts between the liquid electrolyte and electrode.

"We found that the ion-solvation at the interface differs from what we observed in the bulk liquid electrolyte," Nanda said. Understanding this phenomenon could lead to improved electrolytes resulting in batteries with higher performance and better stability.

(Source: Oak Ridge National Laboratory)

Digi-Key's Dave Doherty: Tweaking the Supply Chain

Feature Interview by Nolan Johnson and Andy Shaughnessy
I-CONNECT007

Digi-Key COO Dave Doherty discusses supply chain disruptions and shortages and shares a number of ways in which Digi-Key is helping to smooth out the delivery of components even with the current turmoil.

Nolan Johnson: What we want to talk about today is the supply chain.

Andy Shaughnessy: This is a big thing for designers. They're wondering, "How do you design something for next Christmas when you can't get the capacitor for 80 weeks?"

Aerial view of Digi-Key headquarters in Thief River Falls, Minnesota.

David Doherty: That's the question.

Johnson: Right, which is exactly where we're going on this, Dave. From Digi-Key's spot in the supply chain, you have a great view of what's going on. What do you see as some of the major current drivers right now, plus any challenges?

Doherty: It's really two-fold. People ask if our businesses are up a considerable amount this year; we're going to close this year up about 40% after being up 26% last year. So, they say, "What market's up?" the real answer is, "What market isn't up?" If you look across industrial, medical, telecommunications, military, aerospace, and automotive, we're in a very robust part of the cycle where there are more electronics being put into more applications. However, I would say what's putting the strain on the infrastructure as much as anything is automotive.

We visited a number of Japanese discrete suppliers, and they showed us some charts where in the past, automobiles consumed up to 3,000 MLCCs. There are more than 10,000 now, and you know as well as I do, they're going into all aspects of automobiles. There are

more components in hybrids, and even more in the fully electric cars from the autonomous features, infotainment, and all of the driver assist stuff that we know and love now. The sensors, cameras, monitors, and consumption of electronics are incredible, and even if the automotive market was flat to down (which they aren't), the consumption rates are up marginally, or they've been up. On top of that is increased content per automobile.

So, while that's not a particularly large end segment, we don't ship a ton to tier-one automotive. It puts a strain on our suppliers, which has a ripple through effect to the rest of the channel.

Johnson: Right. When your suppliers are selling into their very large customers, even Digi-Key starts to look like a middle tier.

Doherty: No doubt. And you get into questions such as, "What is the cause and effect, and is this different than in the past?" It certainly feels that that way in both the severity and the longevity of it. I would say that what I find is if supply was instantaneous, you'd never have these disruptions. And even though it's not instantaneous, there's always the ability, as long as there's capital available, to add more foundry and capacity. But there's a little bit more reluctance we're seeing from suppliers to add capacity.

The areas that come to the forefront where there are some of the most commodity of areas, like chip resistors, are the MLCCs that we just mentioned. If you look at those industries and talk to those suppliers, you know the usage is measured in the number of units, so that's been increasing for the last 20 years or so. But for almost that same time cycle, their overall resales have decreased, so demand for cost reduction in these areas have made it less desirable for the suppliers to continue to invest ahead of the demand and try to keep with it. That's different than what I've seen in the past with memory crunches or other areas of semiconductors.

Johnson: We're hearing a lot of feedback about product teams running into issues where, surprisingly, even standard commodity parts are being obsoleted way sooner than one would've thought.

Doherty: Yes, I would say I back that. From my perspective, I think we've seen that as well. I could validate that observation is accurate, and for a lot of these suppliers, it's coming down to economics. If you have a certain amount of capacity and you're not going to expand it, where do you make your margins? That's going to dictate where you're going to put your wafer starts these days. And some of it is the most standard of commodities that have been impacted the most. The prices have been driven down into the mud, and they're asked to produce more and more product each year at less and less total resale. It's a model that's not sustainable.

Johnson: Some of the feedback we've heard, as we've been researching this topic, included strategies to start going to the smallest, newest parts that they can use.

Doherty: Clearly, suppliers have been putting their focus on smaller case sizes where they can get more units per wafer and then get a little bit more economies of scale. The other is looking at options, especially if you talk about resistors and capacitors; there are a number of different flavors. I know most of the readers are savvy enough to know you can buy something with a higher tolerance that's going to be a better-than part.

Put all of those options on your AVL because when you turn those Bills of Materials over to your purchasing folks, more often than not, they're trying to buy to a part number. More and more today, there are different part numbers. Some are just packaging, some might be a 5000-piece reel versus a 10,000-piece reel. Again, others are better-than or equivalent parts. The more the engineer can specify a broader selection of products on the front end, the better chance the purchasing agent is going to match up to something that's going to fit that need. That's just taking the exact need that you have. The more you can work in some flexibility into your design, the better off you're going to be as well if you can accept a little bit wider range on tolerances in some of these parameters.

Johnson: I know that a lot of designers work with their design on one monitor and the Digi-Key website up on another as they search for the parts that have the right parameters for their circuit. How do you see Digi-Key improving that decision-making process?

Doherty: It's a great question. Part of it I want to make sure that we share too, is we've been fortunate. We think that our suppliers have been as generous as they can be with us because our primary model is to support NPI. Our suppliers appreciate that while they have to support today's demand, they can't cut off tomorrow's next-generation products because that's their future livelihood.

Johnson: You're right.

Doherty: Our primary model is to support that NPI and design activity. Now, suppliers' MOQs (minimum order quantities) are usually high enough that when we buy the necessary product, we need to support that. There is some additional inventory that supports spot shortages or unanticipated demand from some of our customers, but that's opportunistic. That's not strategic for us, and so our commitment and desire is working with suppliers to have sufficient quantities on the shelf to support those opportunities early in the design stage so that they can prototype and get to preproduction. They can validate the design, and we keep the innovation flowing while their counterparts in purchasing focus on keeping their existing supply chain for their production needs to stay up and running.

We're bringing in more inventory. I would challenge anybody to do so, whether they've been able to bring in the amount of inventory in the last two years at the same rate we've brought in. It's astronomical amounts. Being private, we have the ability to do so; we're not measured on terms or financial performance. We don't have Wall Street looking over our shoulders, so we saw some signs of this uptick a little over two years ago and started to aggressively purchase more product. You have physically just as much product as possible available on the shelf. Then ask, "Can we do a better job with our parametrics so that we can compare functions and our users can look and see their BOM, critical parameters, and widen their search to see what parts meet that capability or add more parts to the AVL on the front end?"

We value and hope designers continue to trust and have our website up in conjunction as they work on their design. We want those tools, parametrics, and additional content to be there as well so that they can get the flexibility we just alluded to, which makes it easier for the purchasing folks.

Johnson: I think you've also answered our next question as well, as to whether this is affecting your major customers versus the smaller volume customers. Do you see Digi-Key taking any uniquely different steps for the smaller, more innovative design teams?

Doherty: When I encountered that question, I chuckled to myself a little bit because we really don't distinguish between large and small customers. We tend to play in small- and medium-size quantities with all customers, so every customer is equally important. You can be the largest automotive or handheld devices company, and they typically use us for innovations or short-run productions just like the company where their entire production line is similar or maybe even less volume.

Everything about our model, whether it's tech support or the pipeline of inventory, services that space regardless of customer size. One of the first things we did was cut back on some of the high-volume column breaks. We would prefer to have a large number of customers come to us to buy smaller numbers of quantity each versus some large OEM look for us because our model will not make up for the shortage constraints across the industry, or frankly, even across any single OEM or large CM. We want to continue to support that long tail of customers, typically more on the front end of their design or their unforeseen spot shortage that fits our model.

We look at our in-stock rate and do everything we can to continue to keep that up. Again, the suppliers have been very supportive because they see two things. One, they want to support that design activity for their next-generation products. Two, they find it's an easy place if they have any additional upside. Let's say they were planning to produce 100,000 units and they get 102,000. For any of that upside capacity, they can put into one place and very easily channel it to Digi-Key. Then, we can expose it globally for them and try to reach that wide net of customers so it doesn't just get consumed by that one large OEM that's screaming for more product.

We think we've been in a little better situation than most, and inside of that, we do some of the things we alluded to around content and parametrics to work with designers. There's no doubt this isn't a fun time for some of these individuals in trying to source and find products. However, we want to be that trusted partner still, and the one thing we do see is regardless of the constraints, our customers still want to go to authorized sources.

Authenticity matters. They don't want to risk having a counterfeit product, and we'd like to think that's the other reason they come to Digi-Key first is they know and trust any and all of the product we have. That model will never change. It's all coming directly and only from manufacturers that we authorized.

Johnson: What's the perspective at Digi-Key? What are the long-term parts of this shortage that we're going to be dealing with, and what are the blips?

Doherty: For anything that's been more than six or seven quarters, I would say there's light at the end of the tunnel. I think the worst is behind us, although some would say there are

still going to be constraints. I'm going to be a little bit of a contrarian here because I find in our industry, when things are good, people never think there will be another constraint, and when there are constraints, they think we'll never come out of it.

Right now, it would be irresponsible for me not to share with folks that lead times continue to be long in a number of product areas like passives, so stay close to us or any of your sources of supply. But I also tend to find that suppliers are adding capacity and tend to normalize fairly quickly.

Perhaps a large OEM has double-ordered in their supply chain, and once they start to normalize, they trim back their backlog. The next thing you know, that trimmed-back backlog frees up some capacity for somebody else who may have over-forecasted. The correction tends to come more quickly, so I think we'll still be in some level of constraints Q1 and Q2. However, I think the uptick of the supplier activity, as well as the fact that we're seeing some slowing on the general production demand, could help. If you know what to attribute it to, whether it's some of the tariffs or just general economic conditions seem to be a little less robust, I think the combination of the two will lead us to more normal conditions by the second half of 2019.

Shaughnessy: I've heard a couple of fairly intelligent people say that this is actually a sign that things are going well. Maybe not the 80-week lead time in itself, but it's a good problem to have because when we come out of this, we'll be stronger than we were. What do you think about that?

Doherty: I think it's a good thing that it doesn't feel good right away, but our suppliers are taking a more measured approach to capacity. I mentioned earlier they've been reluctant and careful not to add too much capacity too quickly. When the systems are in balance, that's optimal. For our suppliers—us as consumers or customers—I'm sure we'd love to get things for free, but we realize it's not a sustainable model. So, if our suppliers can match the capacity with the true demand where they can have an economically sustainable model, that's good for the industry.

Again, some of the problems are that we've gotten ourselves into is a situation where we had overcapacity for an extended period of time, and responded to our own boards and shareholders as they had to. We cut back or froze capacity, and demand kept inching up to the point that it flipped on us. Having some level of lead times isn't the worst thing in the world.

Predictable lead times aren't an issue for us. We plug them into our system, give the information to our suppliers, and buy ahead. It's the quickly changing lead times where your system is gearing towards nine or 18 weeks, and within four weeks, it flips out to 60 and you can't react. That's the unpredictability that causes strife in our supply chain. I think this is going to pull itself back a little more slowly. It's taking longer but in a more organized fashion. To your point, Andy, I think it's going to be a healthy return.

We talk about soft landings in the economy. The worst thing that could happen is they all overproduce capacity and flip this back into the situation where none of them are making money or can sustain their model, and then we're back into maintaining this unstable environment of production and availability. Does that make sense?

Shaughnessy: Sure. There are so many different takes on this. I just wanted to hear your side of it.

Doherty: I would guess there isn't anybody who could say it's a great thing for suppliers to have extended lead times. They are all working their tails off. They realize that when lead times have been pushed out, that means there's going to have dissatisfied customers and will work hard to ensure that they can they expand capacity and yields.

This is where I think the users can help themselves. The other area that we didn't talk about is looking at different technologies. There are a lot of applications for polymers or tantalum in the capacitor world where, again, the more flexible you can make your design and the more options you can accommodate, the better position you're putting your company to be able to source product.

Johnson: That explanation makes a lot of sense to me. Everything starts to tie together—part availability, pricing volatility, rapidly changing lead times, and premature parts obsolescence all come together as being symptoms of a measured response to a dramatic change in demand.

Doherty: I think that's more than fair, Nolan. I think the solution is communication. The partners that you work with—specifically for those who work with Digi-Key—reach out and ask, "How can we help you with information? Can we show you the number of customers?" We have more and more API relationships with customers where we can provide information and help them. We have 100+ engineers on staff. We can review their BOM and give them a risk assessment, such as, "This product has much fewer customers than these other variations."

We, and frankly our suppliers, want customers to design good, safe, long-life products. No one likes to see a customer in dire straits where there's been an obsolescence, and typically, those decisions aren't made at the spur of the moment. There's some thought that goes into it, so use your partners early in your supply chain, and save yourself that grief where the first time you pick up the phone isn't when you have a part shortage.

Johnson: Dave, who do you want to talk to in the design chain? Do you want to talk to the original specifying engineers or the buyers? Where should they be plugging in for that conversation with Digi-Key?

Doherty: From a title perspective, that varies company to company, but the definition is the person that has an influence on that BOM. It's really that AVL. For some companies, it's one and the same; the buyer is the engineer. But in many of the larger companies, the buyers will say, "My hands are tied. Once the BOM is handed over to me, I'm constrained. We've already gone through our testing and validation." Then, we need to be further upstream if we're going to provide any significant value.

Johnson: That's a thought-provoking answer. How does Digi-Key plan to increase that connection and make that known to the designer, so they know to check in with you?

Doherty: We get about 1,400 contacts—technical support questions or inquiries—from our customers a day. A lot of them already know to use us. I think so many people have come to

Digi-Key's planned expansion in Thief River Falls, Minnesota.

expect such low service levels that they don't realize there are still some partners out there who believe that it's their responsibility to invest in this whole design chain.

Our engineering staff is available to help consult. We have customers physically send us their BOM and ask us to score it. There are third-party software tools out there that will do that. We do it with some of the information, and we have visibility for free. When they're in the purchasing mode down the road, we want that to be an easy, hassle-free experience, and as we've stated a couple of times now, you can't impact it at the time of procurement. It has to be upstream of that, so start with a phone call, webchat, or an email to us, and if not Digi-Key, there are other services out there to subscribe to that I would say are worth their weight in gold in times like this if you can design that little extra flexibility.

I continue to be amazed at the proliferation of part numbers, the uniqueness, and again, in some cases, it's ironic where it's just the case of a small versus a large reel. One of the purchasing people will say, "No, I have to buy this particular part. Don't you have any?" We say, "No, but you can get two small reels to meet your needs. It will fit your same pick-and-place equipment, but it's going to have a different suffix so the manufacturer can designate the small versus large reel." They'll say, "Well, it doesn't match my BOM." In some cases, it's as simple as that.

Johnson: Dave, is there anything else you want to mention?

Doherty: This is an important topic for readers. Our whole livelihood is helping those folks find hard-to-get parts, and we have resources on staff to provide that support. So many times, you get stuck, call a company for help, and hear, "Dial three for this, dial six for this, etc." Next thing you know, you're back at the beginning. When you call Digi-Key or use our webchat function, you're going to get somebody ready to help.

Finally, we're just in the process of closing in our expansion, which will almost quadruple the size footprint of the building that we're in and this warehouse is across the street. It's just a further commitment to show that we're excited about where this industry's going, and we need more space to inventory more product.

Johnson: Thank you for your time.

Shaughnessy: Thanks, Dave.

Doherty: I appreciate it. DESIGN007

Carl Schattke on Stackup Design and Managing the Component Shortage

Feature Interview by Andy Shaughnessy
I-CONNECT007

At AltiumLive, I met Carl Schattke, CID+, a lead PCB designer with an American automaker. Carl and TTM's Julie Ellis taught a packed class on good stackup practices complete with plenty of slides showing examples of all kinds of stackups. After the class, Carl explained why the stackup is often the root of manufacturing problems downstream, and why today's discrete component shortages are likely to be around for quite some time.

Andy Shaughnessy: Carl, I saw part of your class with Julie Ellis. It was pretty interesting and entertaining stuff. How did you get the idea to put the class together?

Carl Schattke: We were thinking about different presentations, and Julie thought that it might be useful to co-present on PCB layer stackups because I bring a design perspective and she brings the manufacturer perspective. We decided it would be useful to put a presentation together that would address how to communicate between a designer and the manufacturer, what information to communicate, and why that's important.

When I was planning the talk, initially, I thought, "Wow, this is a big subject." There's a lot of different areas to pull in because you have the physics of the board technology, and then you have the communication with your vendor. Then, there are all of the parameters that are involved with PCB layer stackup. It impacts your cost and lead times, and impacts who you can make the board with. As I said in my talk, it's basically how an artist uses a canvas. Our artwork requires a board layer stackup. When we put all of our geometry on there, it starts with the layer stackup, and our paintbrushes are the components and traces we use; that's how we end up with beautiful art work.

It's technical, unlike somebody with an easel, but it's the same kind of thing. We are looking to create artwork that is going to be manufacturable, cost efficient, and effective in providing the electrical connectivity that


U.S.C.

we are looking for. I wanted to start the presentation with some of the basics, then discuss what could go wrong, so we covered a variety of examples of the kinds of problems people can encounter with their PCBs and their stackup. Next, we covered the physics of PCB layer stackup and how you solve those problems. We talked about some of the areas when communicating to get a certain type of board built. Then, we went in to some of the costs and different problems.

Shaughnessy: What was your main objective with this class?

Schattke: Julie and I wanted to cover, in a broad range, the common issues that designers face, and we wanted to represent how you would solve those problems and avoid them or mitigate them. We partnered with Altium to have their field application engineer (FAE) explain how that translates into the tool. We also discussed some of the enhancements that they have with Altium Designer 19 coming out, and how that is going to support impedance driven trace widths, etc. We felt that it should be a strong, well-rounded presentation to help people with their PCB stackup problems. Several people spoke with us after the talk and said how valuable it was already; they really enjoyed it and learned a lot.

For me, I wanted to give a lot of value to people who were in the presentation so that they learn about PCB stackups and how to ensure success. PCB stackup problems are generally the most expensive ones we see in industry. If you get that wrong, you either have to redesign—which could be months of effort—you spend more money than you need to, or you don't spend enough money. It's usually one of those issues. We can mitigate a lot of problems by changing the layer stackup or getting it right the first time.

Shaughnessy: The DFM horror stories were really good. I think the designers like to see the horror stories because then they feel like they're not the only ones having problems like that. You feel bad for the one designer Julie mentioned who didn't know his board was going to be built in Asia, and then the Asian shops couldn't get the spaces and traces he needed.

Schattke: The local shops here in the U.S. are geared up for good registration. They have prototyping tools that allow them to reach these hard targets, but as soon as you go to a lower-cost geographic region to get the boards built, you're stuck. You really have to think about that before you kick off your design. A problem like this is not simple to solve because it's every space and trace, and the board has to be tweaked to get it ready for higher-volume manufacturing. I like to design the boards that I do so they can be built as cost-effectively as possible. I never want to add complexity to a board that doesn't need it. I want to use the simplest technology that will fulfill the engineering requirements. Ultimately, this will make it easier to manufacture.

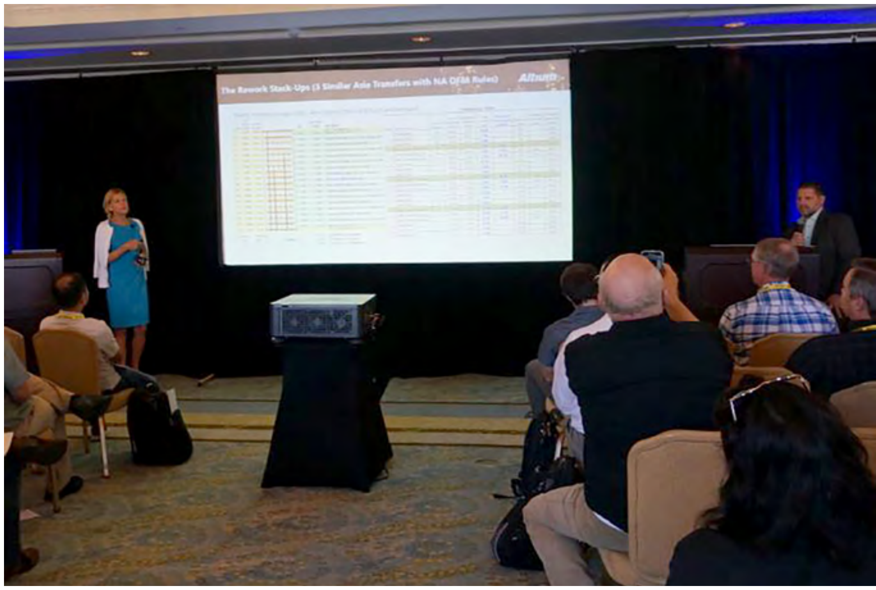
Shaughnessy: I think designers are finally starting to see that they can control the cost of the whole product. They have the power in their hands. Now, they're starting to take control.

Schattke: Part of our role as a designer is to educate the engineers that we work with, and also to discuss the cost and ramifications of different issues. As

Schattke with Rick Hartley.

designers, it's imperative for us to control the risk of the products we design, and also to send up warning flags if electrical considerations won't be met with a certain stackup. We work very closely with the electrical engineers and the product designers to make sure that we design for manufacturing. It's huge.

Shaughnessy: Now, you are on the automotive side of things. Many people are blaming automotive and cellphone companies for taking all of the components. We're seeing 50-week lead times for some capacitors. Are you grabbing all the caps?



Schattke and Julie Ellis team teaching at AltiumLive.

Schattke: I think that part manufacturers make more money on active parts than they do on passive parts. We're seeing some big players drop off their production, and we don't see new actors coming online for that. A lot of it is the capital expense of building those kinds of plants because it's just not as profitable as building something for active components.

It's critical to work with your suppliers—which includes your passive vendor suppliers—to make sure that you've ensured your supply chain. You want to forecast what you're going to need and then work with them to make sure they can meet that forecast. Yes, the problem has to be addressed much earlier in the supply chain, and your engineers have to work with it. Basically, you want to drive supply chain decisions as early as possible so that you ensure a robust supply for the product you're building.

Shaughnessy: That's interesting. We didn't hear much too much about parts shortages until a couple of months ago. Now some components are 80 weeks out. How can you design a board if the parts aren't available for a year and a half?

Schattke: We had this exact problem in 1999 with Y2K. All the same parts were under allocation or shortage, which is the same problem we see today. Business is humming right now. I've heard that we have the highest job participation rate in the United States in 49 years or something like that. I guess it's just crazy good right now for a lot of people. Business is booming everywhere. To have supply chain problems where you can't get enough of something is a fantastic problem to have from a business perspective. For the person who has to go buy that thing to make their product, it's a bad problem, but these are good problems to be having.

Shaughnessy: It's definitely a great time to be in this industry. Everyone is working, and companies are innovating nonstop.

Schattke: We're seeing miniaturization that's enabling all kinds of products we haven't seen before—some really, really cool stuff. This is an exciting time to be alive. There are so many people working on so many different, innovative products. It has been fun and exciting to talk with people at AltiumLive about all the neat stuff that people are trying and doing.

Shaughnessy: Great meeting you, Carl. Thanks again.

Schattke: Thank you, Andy. DESIGN007

PCB007 Highlights

Ventec's Marketing Strategy and Their Newly Appointed Technology Ambassador ▶

At electronica 2018, Mark Goodwin, chief operating officer at Ventec International Group, discusses the company's marketing strategy along with their newly appointed technology ambassador, Alun Morgan, and how he sees the world.

Unimicron Says ABF Substrate Demand Remains Robust ▶

Demand for ABF (ajinomoto build-up film) substrates continues to be strong while demand for other IC substrate and PCB products is being affected by seasonal factors, according to Unimicron Technology.

Flexible Thinking: Achieving Continuous Flexible Circuit Innovation ▶

Since their introduction, flexible circuits have continued a steady climb from relative obscurity to center stage in the world of electronic interconnections. Today, they are among the most popular choice for solving challenging electronic interconnection problems. Those who use this technology on a regular basis are familiar with the many reasons for the popularity of flex.

Upcoming 2019 Trade Show Season in the USA ▶

A look back at Dan Feinberg's predictions made a year ago, and a preview of what we might see at CES 2019 along with some new projections. CES—the first, largest, and most influential and predictive of global electronics trends of this group of trade shows.

Flex Talk: The Myth About Rigid-flex Costs ▶

Do you cringe when you think of the option of rigid-flex? It is not an uncommon reaction when talking with designers and engineering managers about using rigid-flex to solve a packaging problem. Why? The most frequent answer is, "They are so expensive." While it is true that a rigid-flex PCB is typically more expensive on the surface when compared to rigid-board solutions with cables and connectors, a lot is being missed with that mindset.

It's Only Common Sense: The "It" Factor—Are You a Dedicated Salesperson? ▶

As 2018 ends and the new year is almost upon us, we should all do a little soul searching and self-evaluation; a tune-up to make sure that we still have "it." We should strive to find the "it factor" that makes us great salespeople, causes us to wake up every morning ready to go and make those sales, and drives us to be successful even when we don't feel like it.

The Shape of Things to Come: Curved, Flexible, Stretchable, and Three-Dimensional Electronics ▶

The seamless integration of electronics into flexible, curved, and even stretchable surfaces is being requested for several markets, such as automotive (dashboards, lighting, sensors), smart buildings (lighting facades, air quality, solar panels), medical (health patches, X-ray, analysis), and smart clothing (position tracking, sports).

FLEX 2019 and MSTC to Highlight SMART MedTech, Transpo and IoT ▶

Flexible and printed electronics innovations and autonomous mobility sensors will take center stage as more than 700 attendees gather for 120 market and technical presentations, 70 exhibits and four short courses at the co-located FLEX 2019 and MEMS & Sensors Technical Congress (MSTC) in Monterey, California, February 18-21, 2019.

10 Fundamental Rules of High-speed PCB Design, Part 5

Beyond Design

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

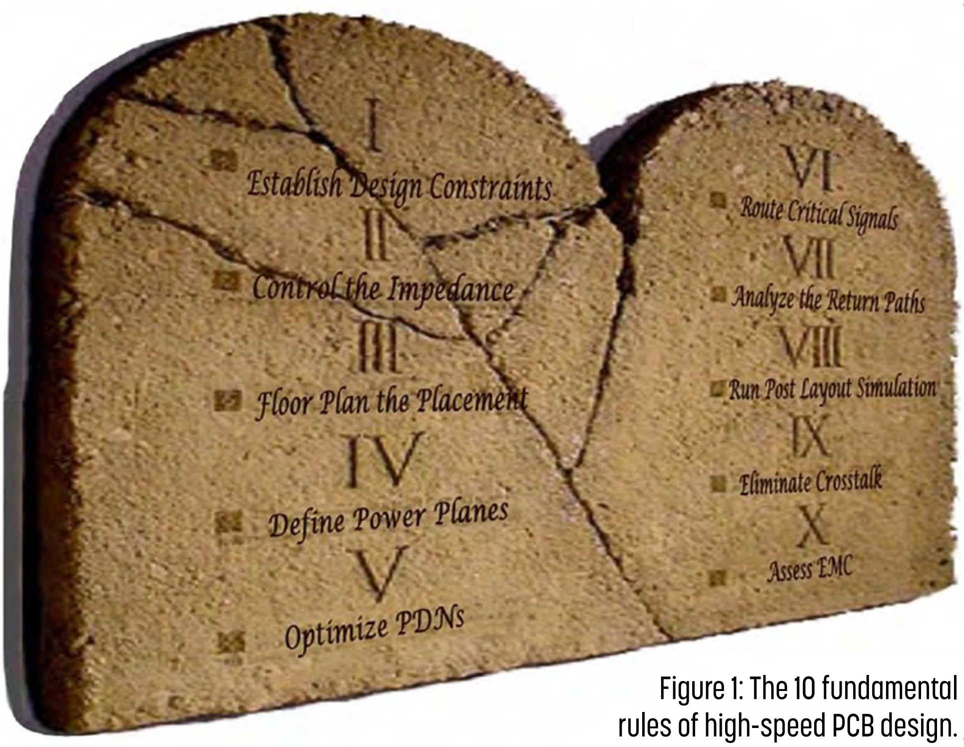


Figure 1: The 10 fundamental rules of high-speed PCB design.

The final part of the 10 fundamental rules of high-speed PCB design (Figure 1) focuses on board-level simulation encompassing signal integrity, crosstalk, and electromagnetic compliancy. Typically, a high-speed digital design takes three iterations to develop a working product. However, today, the product life cycle is very short, and therefore, time to market is of the essence. The cost per iteration should not only include engineering time but also consider the cost of delaying the products market launch. This missed opportunity could cost millions. Also, if an issue is not caught in the design phase and slips to through production and into the field, it could possibly damage a company's reputation.

Unfortunately, simulation is often engaged towards the end of the design cycle. Ideally, the simulation should be done during the design process as part of standard practice. However, post-layout simulation is still necessary to validate the final signal and power integrity.

Board-level simulation cuts costs and a pre-layout simulation identifies issues in the conceptual stage so that they can easily be avoided. Post-layout simulation catches the issues during the design process, eliminating the potentially disastrous final stage changes.

VIII. Run the Post-layout Simulation

Simulate critical signals and match signal propagation and timing. Check for signal ringing and eye jitter.

The eye diagram (Figure 2) is a common indicator of the quality of a signal in high-speed digital transmission lines. In an ideal world, eye diagrams would look like rectangular boxes. In

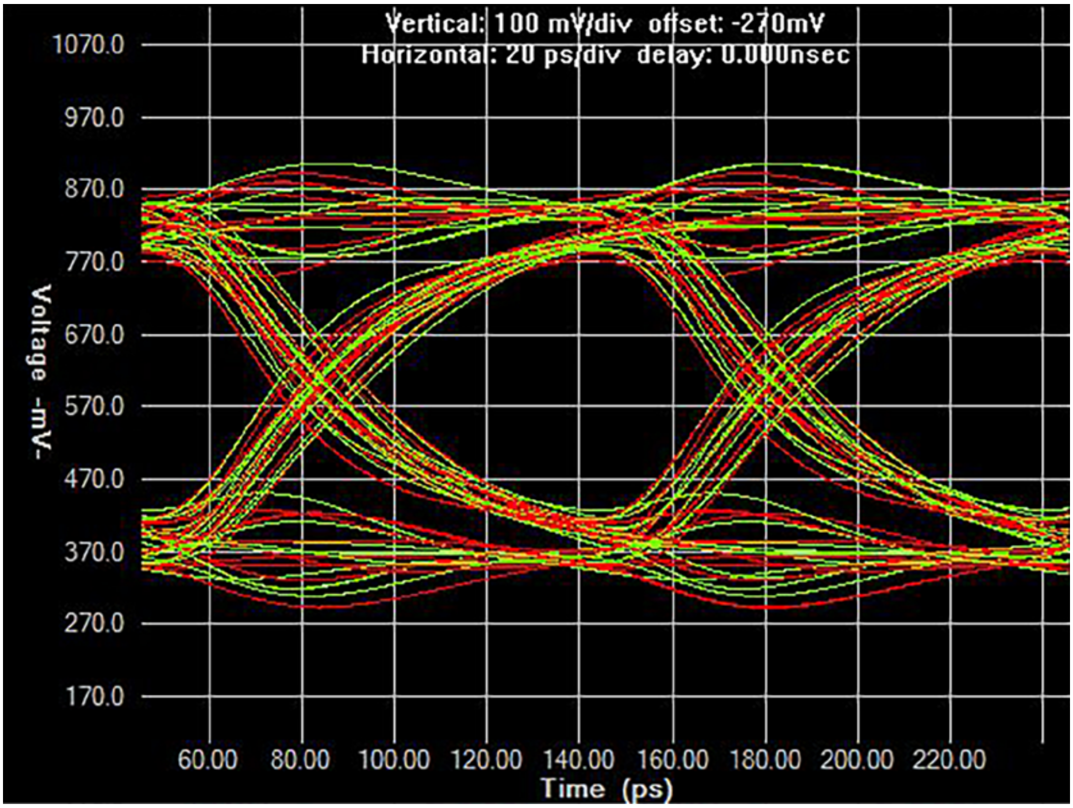


Figure 2: Eye diagram indicating signal quality (Figures 2-6 simulated in HyperLynx).

reality, data transmission is imperfect, so the transitions of the bit pattern do not perfectly align on top of each other, and an eye-shaped pattern results. An open-eye pattern with little jitter (horizontal disparity) and noise (vertical deviation) is the objective.

To attain this objective one needs to consider all the previous fundamental rules particularly:

- Control the impedance.
- Match the driver to the transmission line impedance.
- Tightly couple all signal traces to a contiguous reference plane and have a clearly defined minimum loop inductance return current path.
- Maintain constant impedance along the entire length of differential pairs.
- Synchronous data and address buses, plus associated clocks and strobes, should have matched propagation within their timing margin.

A preliminary batch mode simulation should initially be completed on the design. Default IC characteristics, crosstalk of 150mV maximum and EMC to FCC or CISPR Class A and B, are set up in the simulator. The batch mode simulation automatically scans large numbers of nets on an entire PCB, flagging signal integrity, crosstalk, and EMC hot spots.

The post-layout simulation analysis can then be prepared using supplied specifications. This is an extensive interactive board level simulation which takes the analysis to the next level–simulating trouble spots identified by the batch analysis to further resolve the issues with greater accuracy. Keep in mind that any signal integrity concern will create issues downstream with ringing causing excessive crosstalk leading to electromagnetic radiation. Each issue should be dealt with one-by-one until they are all resolved.

Flight times of the critical signals should be examined thoroughly. One could compare the matched lengths of each signal, but the delay will vary depending on the meander pattern and the signal layer in the stackup. Also, the trace either side of a series terminator needs to be added to obtain the total delay. So instead,

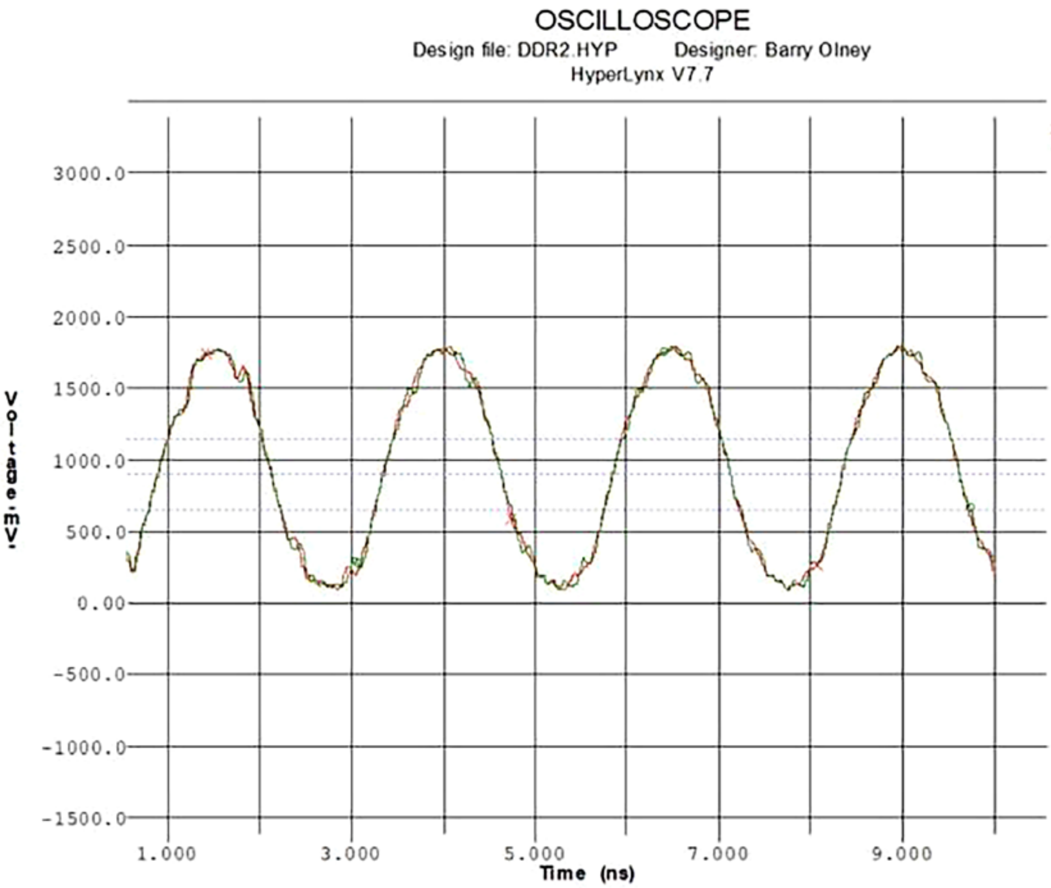


Figure 3: Flight times of data compared to strobe.

it is best to compare the skew between, for example in Figure 3, the MDQ0 (data) and MDQS0 (strobe) at the receiver to ensure the flight times are correct. In this case, there is 10pS difference, so this is well within the timing specification.

IX. Eliminate Crosstalk

Scan the board for possible crosstalk. Crosstalk can be coupled trace-to-trace on the same layer or broadside coupled by traces on adjacent layers.

Crosstalk is caused by the coupling of the electromagnetic fields. Electric fields cause signal voltages to capacitively couple into nearby traces. Capacitive coupling draws a surge of drive current which causes reflections on the transmission lines. Whereas, magnetic fields cause signal currents to be induced into nearby traces. Inductive coupling produces ground bounce and power supply noise. Crosstalk falls off rapidly with the square of the distance, and the degree of impact is related to the aggressor signal voltage, available board real estate and thus the proximity of signal traces.

Crosstalk can be coupled trace-to-trace on the same layer (Figure 4) or can be broadside coupled by traces on adjacent layers (Figure 5). The coupling is three dimensional. Broadside coupling is difficult to spot as generally we look for trace clearances when evaluating crosstalk, but a simulator will pick this up. Traces routed in parallel and broadside cause greater amounts of crosstalk than those routed side-by-side due to the greater coupling area. Therefore, it is good practice to route adjacent signal layers in the stackup orthogonally to each other to minimize the coupling region. A better solution is to only have one signal layer between two planes to avoid broadside coupling altogether.

When interactive routing, one tends to group signals for aesthetic reasons–this is the artistic side of the PCB Designer showing through.

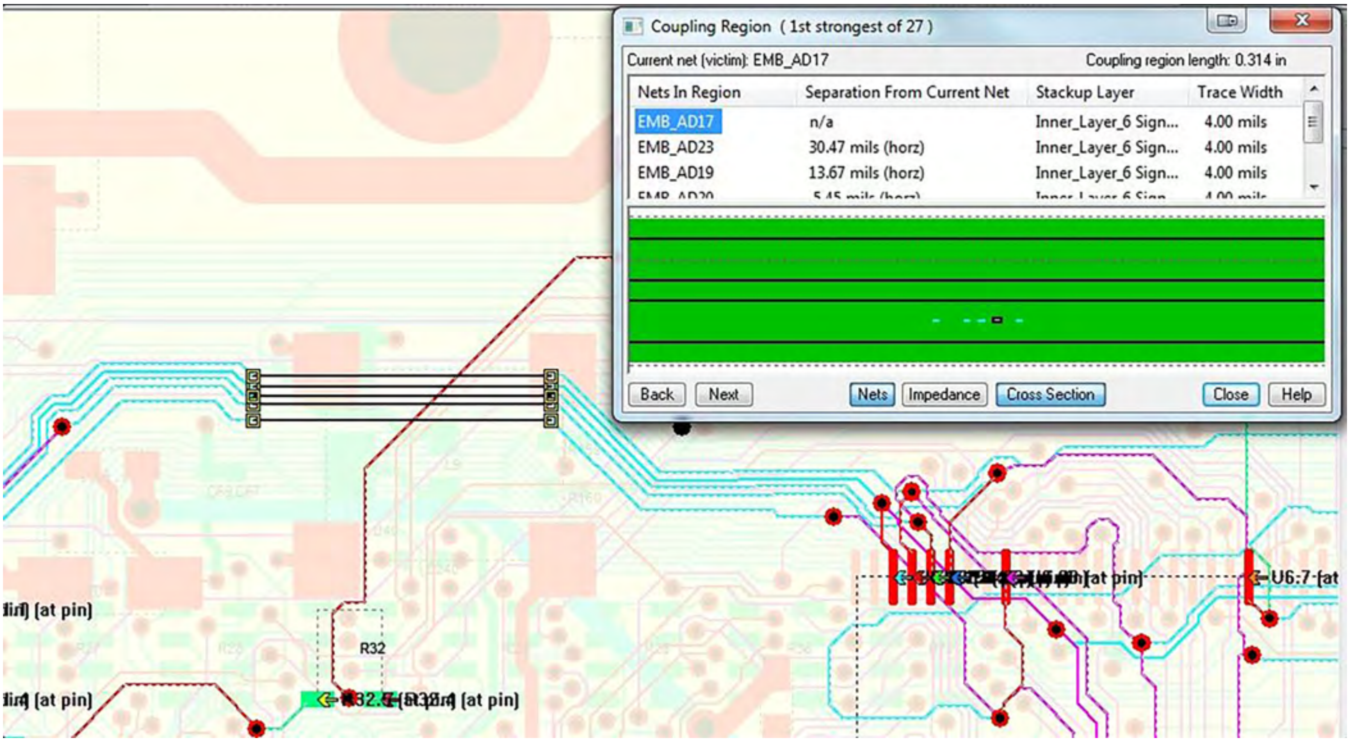


Figure 4: A walk through the coupling regions.

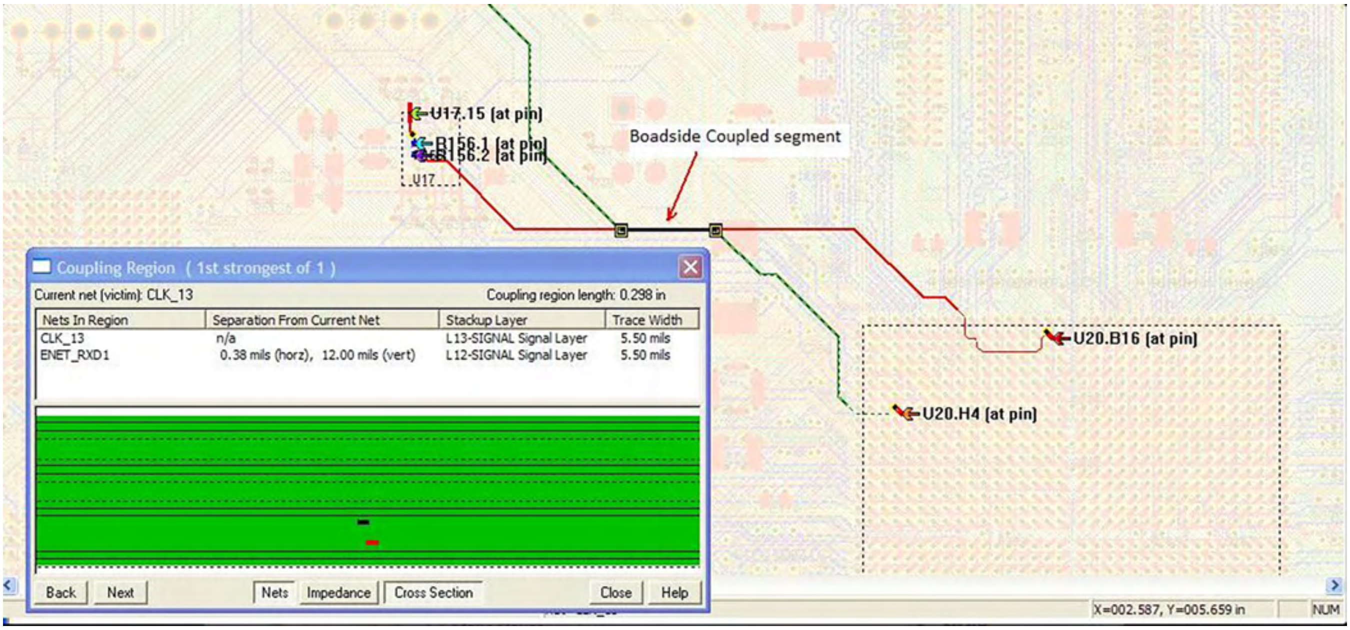


Figure 5: Broadside-coupled crosstalk.

But, although it looks nice and neat, it may not perform so well.

It is recommended that critical trace segments should be spaced by three times the trace thickness, where possible; alternatively, the trace to plane height can be reduced in the stackup if real estate is limited.

X. Assess Electromagnetic Compliancy (EMC)

Control EM radiation at the source. Ensure that differential mode signals do not convert to common mode and eliminate any possible antennae.

To be prudent, it is best to design the PCB layout with electromagnetic compatibility in mind rather than to be faced with excessive emissions at the prototype stage, or worse still, just before production. Unfortunately, too many of the problems are uncovered at the testing stage. Shielding and absorbing materials are then often used to reduce the emissions.

The preferred approach is to identify the problems at the board level and to rectify them there. Even then, if particular offending frequencies are identified, engineers are still faced with the nightmare of locating which net segments are the cause of these particular emissions.

Once excessive emissions are detected, the goal is to quench these emissions. Crude techniques involve the design of sealed enclosures, chokes and ferrites to restrain the launching of common-modes along cables and a variety of other methods that are too little and too late. Alternatively, and more proactively, designers recognize that the problem should be addressed at the board level where the radiation emanates.

Since all products must comply with strict EMC regulations, all critical high-speed signals should be simulated to determine the amount of expected radiation. The EMC standards for FCC Class B limit the radiation to a 54-dB average with a peak of 74 dB (at a 3-m distance) above 1 GHz. However, harmonics can still cause unforeseeable problems (Figure 6). Generally, these issues are caused by routing traces on the outer (microstrip) layers where radiation is much higher than that of the inner (stripline) layers. Routing critical signals between the planes can reduce the EMI by more than 10 dB.

Electromagnetic radiation from differential pairs can occur as either differential or common mode. Differential mode is typically equal and opposite, and therefore, any radiating fields will

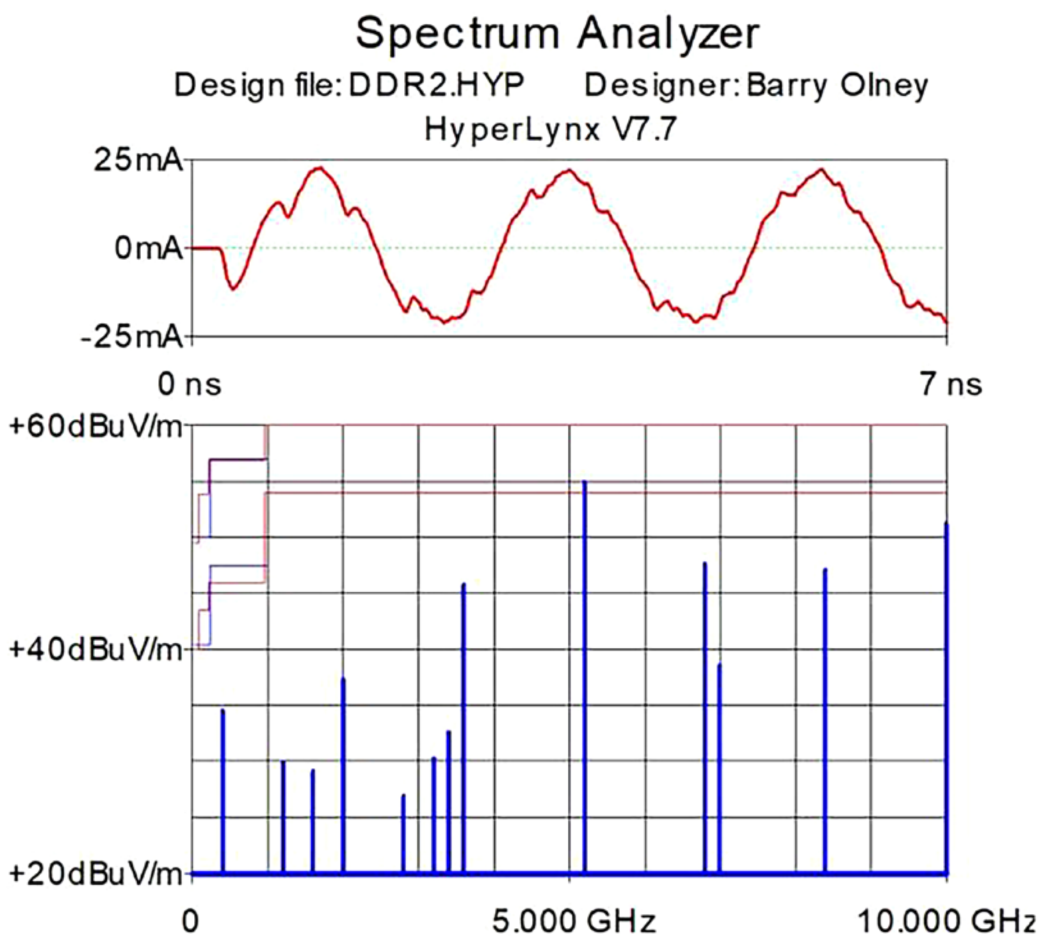


Figure 6: Electromagnetic radiation from MDQS0 strobe.

cancel. Conversely, common-mode radiation from two coupled conductors is identical; it does not cancel, but reinforces. Unfortunately, differential-mode propagation can be converted to common mode by parasitic capacitance or any imbalance caused by signal skew, rise-fall time mismatch, or asymmetry in the channel.

Also, dangling via stubs distort signals passing through an interconnect and also decrease the usable bandwidth of the signal. This is due to the via stub acting as an antenna, which has a resonant frequency determined by the quarter wavelength of the structure. The conventional solution to this problem is to back-drill (or control depth drill) the vias to bore out the via stub barrel, so that the via stubs are reduced in length if not completely removed.

The Zeroth Rule

In Part 1 of this series, I mentioned that there is always room for one more rule at the top, and this rule may be the foundation of earlier rules.

The most important rule is to scrutinize the design proactively as it unfolds, so you have the ultimate control over the outcome. Do not totally rely on the tools. High-speed PCB design is not just a process. As we all know, things can and do go horribly wrong at times. You cannot teach people to be good PCB designers. I have presented many training courses over the years, and unfortunately, individuals, although very capable, just don't have what it takes.

A successful high-speed PCB designer must:

- Communicate well with others within the team and the industry
- Appreciate the challenge and have good problem-solving skills
- Be creative and thorough, and pride yourself in the quality of your work
- Understand and implement IPC design standards and project-specific requirements
- Have a thorough knowledge of your EDA tools and PCB fabrication, assembly, and testing processes
- Understand transmission-line signal propagation, controlled impedance, and recognize where signal currents flow and how coupling occurs at high frequencies
- Keep an eye on the ball during the entire design process, catching any small issues before they become a major problem

But what really makes an exceptional PCB designer? Ability must align with enthusiasm!

Key Points

- Simulation should be done during the design process as part of standard practice
- The eye diagram is a common indicator of the quality of a signal in high-speed digital transmission lines
- Any signal integrity concern will create issues downstream with ringing causing excessive crosstalk leading to electromagnetic radiation
- It is best to compare the skew between signals at the receiver to ensure the flight times are correct
- Crosstalk can be coupled trace-to-trace on the same layer or broadside coupled by traces on adjacent layers
- Traces routed in parallel and broadside cause greater amounts of crosstalk than those routed side-by-side due to the greater coupling area
- To reduce crosstalk, critical trace segments should be spaced by three times the trace thickness where possible; alternatively, the trace to plane height can be reduced in the stackup if real estate is limited
- It is best to design the PCB layout with electromagnetic compatibility in mind
- Proactive designers recognize that an EMC problem should be addressed at the board level where the radiation emanates
- Routing critical signals between the planes can reduce the EMI by more than 10 dB
- Differential-mode propagation can be converted to common mode by parasitic capacitance or any imbalance caused by signal skew, rise-fall time mismatch, or asymmetry in the channel
- Dangling via stubs distort signals passing through an interconnect and also decrease the usable bandwidth of the signal

Further Reading

- Beyond Design: Crosstalk Margins by Barry Olney, *Design007 Magazine,* July 2018.
- Beyond Design: How to Handle the Dreaded Danglers, Part 2 by Barry Olney, *The PCB Design Magazine,* September 2016.
- Beyond Design: Introduction to Board-level Simulation and the PCB Design Process by Barry Olney, *The PCB Magazine,* March 2012.
- Beyond Design: Board-level Simulation and the Design Process, Plan B–Post-layout Simulation by Barry Olney, *The PCB Magazine,* February 2012.
- Beyond Design: Controlling the Beast by Barry Olney, *The PCB Magazine,* December 2011.
- Beyond Design: Common Symptoms of Common-mode Radiation by Barry Olney, *Design007 Magazine,* May 2018.
- "Board-level Simulation" by Barry Olney, Electronics News (Australia), January 2013.
- "EMC Simulation Techniques for Printed Circuit Boards" by Dr. Al Wexler, Quantic EMC Inc. **DESIGN007**

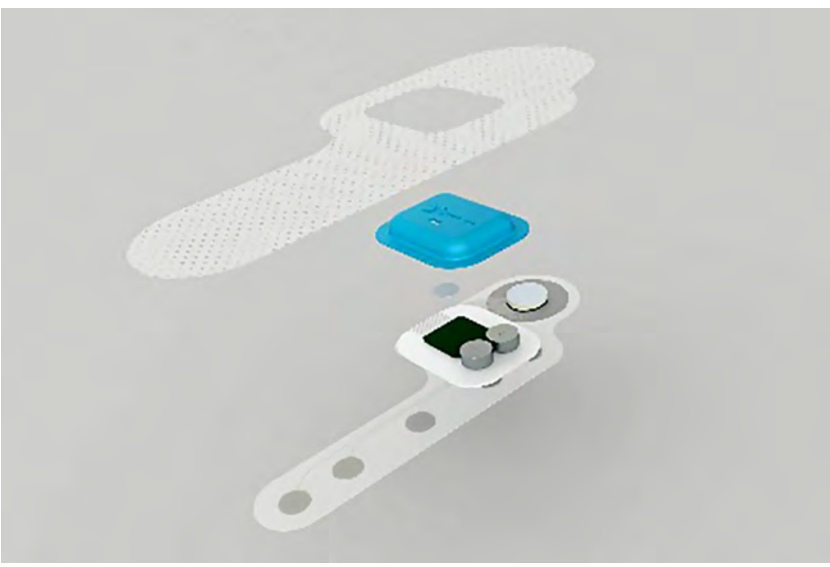
Barry Olney is managing director of In-Circuit Design Pty Ltd (iCD), Australia, a PCB design service bureau that specializes in board-level simulation. The company developed the iCD Design Integrity software incorporating the iCD Stackup, PDN, and CPW Planner. The software can be downloaded from www.icd.com.au.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

Disposable Health Patch to Measure Vital Signs

At CES 2019 in Las Vegas, imec and TNO are presenting the latest version of their health patch. Developed in the framework of the Holst Centre in Eindhoven, the new health patch offers unprecedented comfort and a long battery life, previously unseen in this type of device. It can also be manufactured at a fraction of the cost of previous generations.

The patch uses a mix of skin friendly and biocompatible materials. A comfortable silicone adhesive is used to provide long-term adhesion at high comfort. "Completely watertight, the new health patch is built for maximum user comfort and can be worn for up to seven days before needing to be replaced," explains Prof. Jeroen van den Brand, program director of printed electronics, TNO.

For patients, a single, disposable patch that can be worn for several days is more convenient and can reduce hospital visits as it no longer needs to be returned after use. This is particularly important for chronically ill patients as it provides an affordable, single-use device that can be easily used to monitor their vital signs and physical activity at home. Easy to apply and comfortable to wear for long periods of time, it enables patients to perform their normal daily activities with minimal impact while providing valuable information to optimize their treatment and medication. The single-use concept can also reduce time spent in the hospital as it allows patients to be sent home earlier while still remotely monitoring their vital signs.



(Source: Holst Centre)

Cascade Chapter and 2019 Activities

The Digital Layout

by Steph Chavez, CIT, CID+, IPC DESIGNERS COUNCIL

Happy New Year! This month's column highlights the Cascade chapter in the Seattle, Washington, area. Also, after ringing in 2019, mark your calendars for two upcoming January shows in California (IPC APEX EXPO and DesignCon) as well as this year's CID and CID+ training and certification schedule.

Chapter Spotlight

by Cherie Litson, MIT, CID/CID+

The IPC Designers Council is an international network of designers. Its mission is to promote printed circuit board design as a profession and to encourage, facilitate, and promote the exchange of information and integration of new design concepts through communications, seminars, workshops, and professional certi-fication through a network of local chapters.

This month's spotlight is the Cascade chapter, which recently held a CID+ class at Lake Washington Institute of Technology (LWIT) in Kirkland from December 4–7 (Figure 1). LWIT has partnered with EPTAC Corporation and Mentor, a Siemens Company, for a training facility focused on IPC certification classes. One of the full-time LWIT instructors, Joe Gryniuk, is an IPC-600 and IPC-610 instructor as well as one of the principal instructors for electronic technician classes for the past 20+ years. Thanks to Joe's efforts, we have been able to present our CID and CID+ classes, host our Designers Council meetings, and support other IPC certification courses at LWIT.

Our CID + class had four people enrolled who came from around the area. All who attended learned about how board materials affected

Figure 1: Cascade chapter's latest IPC DC meeting.

Figure 2: Board members (L-R) include Paul Berndt, Tim Mullin, Cory Grunwald, Aubrey Moore, and Jerome Larez.

their designs including EMI, EMC, impedance control, power distribution techniques, board stackups, and placement strategies in more detail than what was in the CID. Further, participants took the exam first thing Friday morning. This allowed us time to attend the Designers Council "lunch-and-learn" meeting down the hall, and the Altium user group met afterward in the same location.

Starting our Altium user group was a huge success. There were many requests from our members who do attend the meetings for such a group. David Haboud from Altium came and did a short presentation on the new Altium 19 features. Then, we discussed how these features will affect how we use Altium going forward. We were very excited about the 3D functionality and hope that it expands to be able to control more of the board-to-board signals. There was some skepticism about the new library functionality and how that will affect existing libraries. Most of us want to explore this in more detail at a future meeting. We'll probably have another one around March.

History

The Cascade Designers Council started in 2001 with a small group of designers and Prototron Inc. who were interested in networking and learning more about PCB design. Many of us had just taken our CID and were hungry for more information. We meet at various places including Prototron, Microsoft, Monsoon Solutions, SonoSite, and Medtronics—anywhere we could find a meeting room available. Some meetings were held at lunch and others in the evenings.

The chapter flourished for about 6–7 years, getting up to 80 members at one time, and then the crash of '08 came. Many people moved to other areas and jobs or just retired completely. Some of us tried to get the chapter going again, but it just didn't take until 2016. A new group of individuals stepped up and took on the task of making the chapter viable again. These individuals still run the chapter today: Tim Mullin, president; Paul Berndt, VP; Aubrey Moore, secretary; Jerome Larez, treasurer; and Cory Grunwald, webmaster (Figure 2).

They put in the time and effort to get the chapter registered in the state, re-opened a bank account (more difficult than starting over), and reached out to the designer community again. Let's admit it—designers are "tough nuts" to open! It is always difficult to get us to move away from the computer and actually socialize, even when you wave pizza or sandwiches in front of our noses.

However, this group started to get that shell to open up a little bit. First, they tried evening meetings trying (traffic sucks here, which is difficult), and then lunch meetings, which both proved to be challenging. They have given away wonderful door prizes such as IPC standards, Prototron's "Solder Mask Sampler," books from Dock Brown and Douglas G. Brooks, and field solvers from Polar Instruments. What will it take for you to step up and own your time and continued education as a designer?

Presentations

There have been some amazing topics covered these last two years (including some by myself) [1]:

- **"Flex Printed Circuit Fabrication"**
  Jerome Larez, MicroConnex (December 2018)
- **"Achieving Cleanliness for High-reliability Boards"**
  Dock Brown, CRE (September 2018)
- **"DFM and DFT, Part 2—The Risk-Cost Factor Defined and Quantified!"**
  Cherie Litson (June 2018)
- **"DFM and DFT—How to Reduce Your Costs!"**
  Cherie Litson (March 2018)
- **"Copper Roughness and How Roughness-related Loss is Calculated in Their Si9000e Field Solvers"**
  Geoffrey Hazelett, Polar Instruments (November 2017)
- **"Understanding the Variables of Dielectric Constant for PCB Materials Used at Microwave Frequencies"**
  Dale Doyle, Rogers Corporation (July 2017)
- **"Component Library Roundtable Discussion"**
  Group discussion (May 2017)
- **"Common Fabrication Concerns and Dealing with Today's Controlled Impedances"**
  Mark Thompson, Prototron Circuits (January 2017)
- **"The History of Signal Integrity Issues on PCBs—Four Stages of Problems and Solutions"**
  Douglas G. Brooks, Ph.D. (November 2016)
- **"Design for Reliability"**
  Dock Brown, CRE (September 2016)
- **"Trace and Via Currents and Temperatures—The Story that Grew and Grew!"**
  Douglas G. Brooks, Ph.D. (May 2016)

The Future

More great subjects and presentations are being planned right now. Some of our officers are getting busy with work and would like to have others step up this year and take on some of the responsibilities. We're looking for a new webmaster and anyone else with ideas. We'd

2019 Training and Certification Schedule

IPC Certified Interconnect Designer (CID)		IPC Advanced Certified Interconnect Designer CID+
Jan. 31-Feb. 3: San Diego, CA	Aug. 6-9: Baltimore, MD	Jan. 31-Feb. 3: San Diego, CA
Feb. 26-Mar. 1: Manchester, NH	Aug. 26-29: Markham, ON	April 16-19: Markham, ON
March 5-8: Santa Clara, CA	Sept. 6-9: Santa Clara, CA	Sept. 6-9: Santa Clara, CA
March 19-22: Kirkland, WA	Sept. 19-22: Schaumburg, IL	Sept. 10-13: Kirkland, WA
April 30-May 3: Schaumburg, IL	Oct. 29-Nov. 1: Anaheim, CA	Sept. 19-22: Schaumburg, IL
May 21-24: Pittsburgh, PA	Nov. 2-5: Raleigh, NC	Oct. 29-November 1: Anaheim, CA
June 18-21: Kirkland, WA		Nov. 2-5: Raleigh, NC
		Nov. 26-29: Manchester, NH

also love to have Rick Hartley come and do a two-day class for us in the future.

There are many other opportunities for learning and networking that will be coming up in 2019. We really hope to have more people—current designers, engineers, manufacturers, software vendors, and those interested in PCB design—come join us. This area has so much potential! Visit cascade-ipcdc.org for more information on our chapter and upcoming events.

Note: Dates and locations are subject to change. Contact EPTAC Corporation to check current dates and availability. A minimum enrollment of seven students is required for a class to be held.

Upcoming Events

IPC APEX EXPO 2019
- January 26–31: Meetings and courses
- January 29–31: Conference and exhibition
- January 31–February 2: IPC Designer Certification CID/CID+
  San Diego, California
  ipcapexexpo.org

DesignCon 2019
- January 29–31
  Santa Clara, California
  designcon.com

References

1. IPC Designers Council Cascade Chapter, "Previous Meetings," 2016–2018.

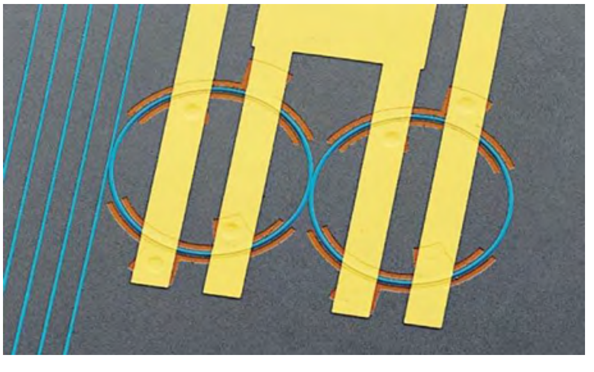
Stephen Chavez is a member of the IPC Designers Council Executive Board and chairman of the communications subcommittee. To read past columns or contact Chavez, click here.

Programming Light on a Chip

Researchers from the Harvard John A. Paulson School of Engineering and Applied Sciences (SEAS) have developed a new integrated photonics platform that can store light and electrically control its frequency (or color) in an integrated circuit.

Microwave signals are ubiquitous in wireless communications, but researchers thought they interact too weakly with photons. That was before SEAS researchers, led by Marko Loncar, the Tiantsai Lin Professor of Electrical Engineering, developed a technique to fabricate high-performance optical microstructures using lithium niobate, a material with powerful electro-optic properties.

Loncar and his team previously demonstrated that they can propagate light through lithium niobate nanowaveguides with very little loss and control light intensity with on-chip lithium niobate modulators. In the latest research, they combined and further developed these technologies to build a molecule-like system and used this new platform to precisely control the frequency and phase of light on a chip.



Next, the researchers aim to develop even lower-loss optical waveguides and microwave circuits using the same architecture to enable even higher efficiencies and, ultimately, achieve a quantum link between microwave and optical photons.

"The energies of microwave and optical photons differ by five orders of magnitude, but our system could possibly bridge this gap with almost 100 percent efficiency, one photon at a time," said Loncar, senior author of the paper. "This would enable the realization of a quantum cloud—a distributed network of quantum computers connected via secure optical communication channels."

(Source: Harvard John A. Paulson School of Engineering and Applied Sciences)

Embedding Components, Part 7: Semiconductor Placement and Termination Methodologies

Designers Notebook

by Vern Solberg, CONSULTANT

Progress in developing high-density embedded-component substrate capability has accelerated through the cooperation and joint development programs between many government and industry organizations and technical universities. In addition to these joint development programs, several independent laboratories and package assembly service providers have developed a number of proprietary processes for embedding the uncased semiconductor elements.

Developers have found that embedding the semiconductors on an inner layer of the PCB or package substrate directly in line with active and passive components mounted on the outer surface ensures that the conductor interface between related components will be minimized.

There are a number of methods used for interconnecting uncased semiconductor components. Semiconductor elements can be mounted onto the core substrate in the faceup orientation or facedown. When placing the die with the active surface of the die facing up, termination will likely adopt copper-plated microvia methodology. Meanwhile, facedown placement will enable the direct interface to land patterns provided on the designated layer of the circuit structure

As noted in Part 6 of this series, the semiconductor fabrication process initially furnishes the die with aluminum bond pads on its perimeter for the traditional wire-bond interface process.

Faceup Semiconductor Termination

Both gold wire-bond and ribbon-bond processes may be applied for completing the die-to-substrate interface. In preparation for this process, a cavity is provided in the substrate (typical of that described in Part 4) to provide clearance for both the die attach and terminal interface. The faceup attachment method traditionally adopts an adhesive material (liquid epoxy or film) for initially attaching the die to the substrate's surface. Termination lands are positioned on the upper layer of the cavity in line with the wire-bond termination sites on the die element (Figure 1).

When alternative interface methods are required, the semiconductor terminal sites must be furnished with an alloy that will be compatible with the specific attachment material or interface method selected. For

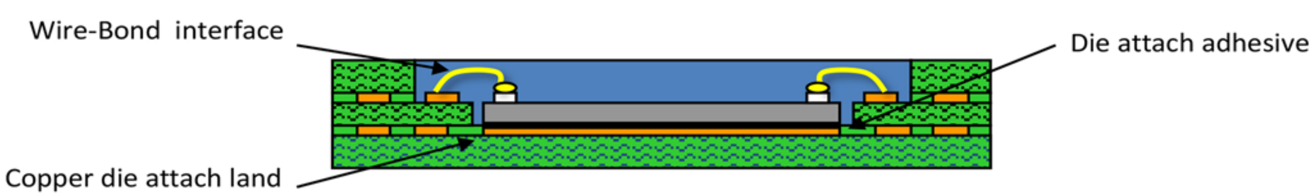


Figure 1: Cavity-mounted die for wire-bond termination.

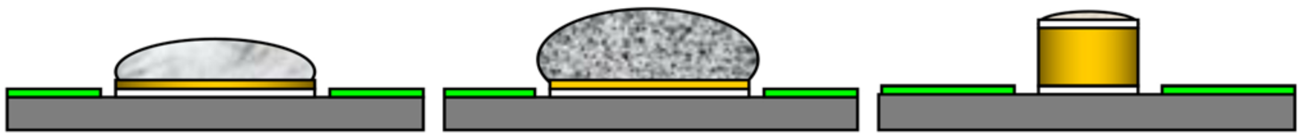


Figure 2: Comparing semiconductor terminal sites for facedown attachment.

example, companies using solder, a conductive polymer, or a plated-microvia interface will require an additional metallization process to provide a copper termination at each bond site.

Facedown Semiconductor Termination

Several processes have evolved for joining the semiconductor die to the embedded substrate surface. Smaller outline die with wire-bond sites greater than 400 microns may be furnished with solder bump or conductive-polymer bump features for termination. However, when the terminal spacing on the die is less than 400 microns, companies may consider implementing a selective plating process that furnishes miniature copper pillar-like terminals. Examples of common termination features are illustrated in Figure 2.

More complex, high I/O semiconductors are commonly processed further to redistribute the wire-bond terminals to a uniform array contact pattern as described in Part 6. The array format enables larger termination features for interconnect and affords greater flexibility for circuit routing. The most common materials utilized for flip-chip termination are tin-alloy based solders and conductive polymers.

Flip-chip Assembly Process

Reflow solder processing is a common technique for flip-chip assembly and a substantial infrastructure already exists for supplying systems for both high- and low-volume manufacturing. To ensure long-term reliability, the solder alloy composition selected must be compatible with the contact alloy and surface finish supplied on the mounting structure. As expanded in Part 5, tin-silver-copper (SAC) alloy compositions become liquidus and enable flux activation at a temperature range between 220°C and 230°C. The temperature required to complete the joining of the component to the land patterns, however, will reach 245–260°C.

A question one may ask is, "What will be the impact of the solder interface on the buried components during when the same solder processing temperature required to attach components on the outer surface(s) of the finished multilayer substrate or PCB?"

To avoid the potential for disturbing the buried termination sites assembly process, specialists may consider adopting one of the lower temperature alloy composition for attaching the surface-mounted components. Solder alloys are available requiring no more than 185°C peak temperature to complete the joining process (compared to 245°C+ for SAC alloys).

Alternative Conductive Polymer Assembly

Polymer-based joining materials containing fine particles of a silver alloy are commonly dispensed directly onto the land-pattern sites provided for the semiconductor die prepared for facedown mounting.

In preparation for conductive polymer termination, the land patterns on the mounting surface must be clean and free of oxidation to ensure conductivity with the fine silver particles embedded in the polymer compound. The cure temperatures required may range from room temperature for two-part epoxy systems to a range of 80–175 °C for other single-component materials (far less than solder processing). Although several of the materials referenced in Part 5 require elevated temperatures to complete the joining process, a number of anisotropic materials are available for room temperature joining using a single component UV-curable polymer.

Gold-on-gold Termination

Raised gold bump terminals are formed on each bond site utilizing the same basic system developed for wire-bond assembly. The actual die attachment technique utilizes heat, pressure, and ultrasonic or thermosonic energy to form a welded interconnection from the gold-bumped die to the substrate's gold-on-copper land pattern. Furthermore, the GGI process eliminates the need for applying flux and cleaning before dispensing underfill and applying the prepreg laminate or RCC material.

Microvia Semiconductor Termination

When the die is mounted with the active surface facing away from the substrate surface, interconnect between die and substrate can rely on copper-plated microvia termination. In preparation for microvia termination, the termination sites on the semiconductor must be furnished with a copper surface.

Following the placement and lamination encapsulating the die element, laser systems are employed to ablate material at each terminal site followed by a sequence of plating, imaging, and etching processes (Figure 3) to complete the interface from the die to the buildup circuit layer.

Microvia interface methodology is favored by a growing number of companies because they can eliminate several process steps: material dispensing, thermal curing or reflow processing, cleaning, and underfill application. Although the core mounted-semiconductor element is a popular technique for embedding active die, a number of innovative coreless embedding methodologies have evolved as well.

Die-first Coreless Process

Developed in Europe with a consortium of the industry and academia, a die-first assembly method adopts a variation of the Occam process for embedding and interconnecting semiconductor die elements. The process begins with laser marking fiducial targets on the surface of an ultra-thin copper foil base layer (Figure 4). A pattern of adhesive material that is slightly larger than the semiconductor outline is printed onto the copper surface and partially cured to furnish a stable surface for device attachment.

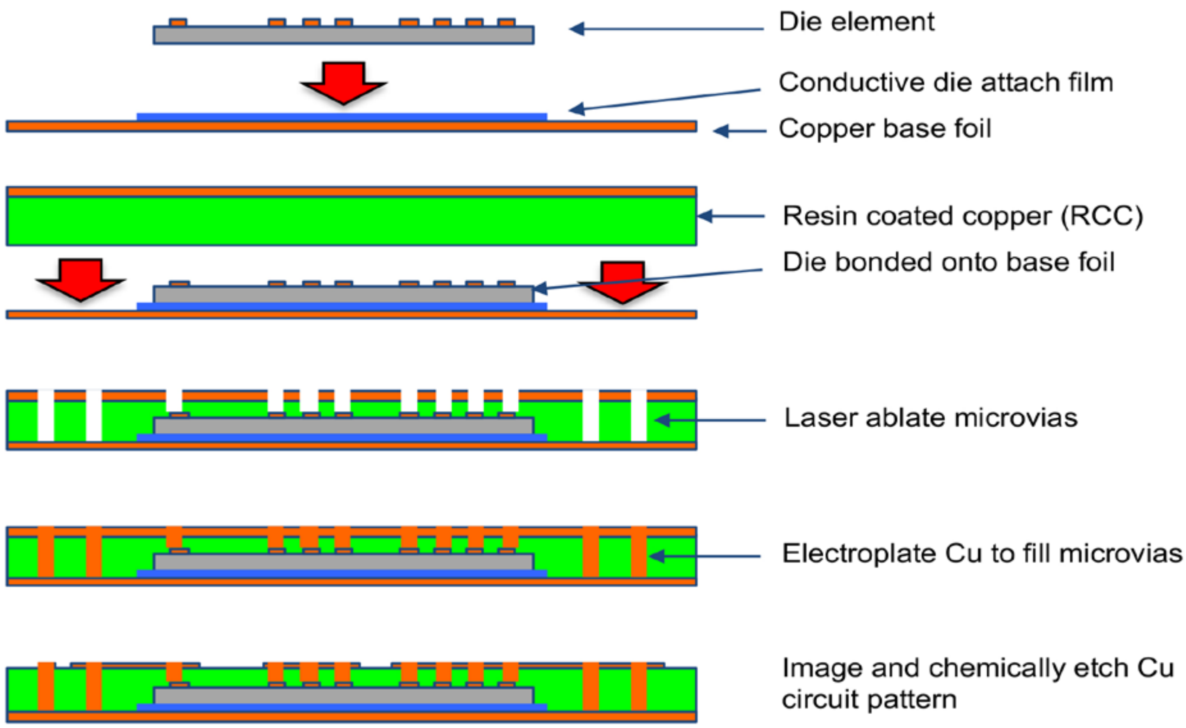


Figure 3: Microvia-to-faceup-semiconductor interface process sequence.

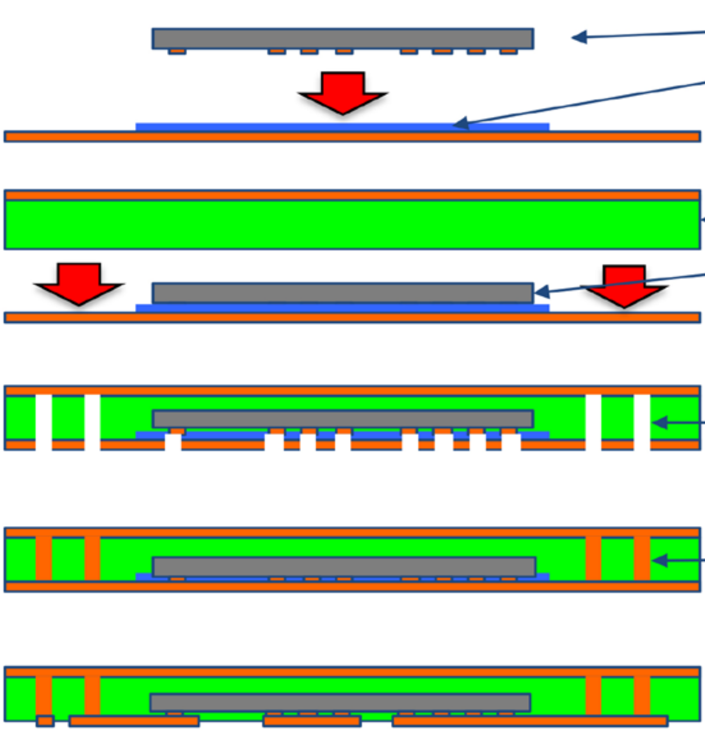


Figure 4: Die-first (facedown) coreless assembly process sequence.

Using the fiducial target features on the copper surface as a guide, the components are placed into the partially cured adhesive pattern with the copper-plated terminal features of the die facing down. A thermal curing process follows to complete the bonding of the semiconductor element to the copper foil base. Resin-coated copper (RCC) foil is next laminated over the backside of die encapsulating the element. The embedded structure then follows a microvia ablation and plating procedure similar to that previously outlined. Additional circuit layers are sequentially added to complete interconnects to components mounted on the outer surface(s) of the substrate or PCB.

Summary

The decision to embed components within the PCB or substrate structure must be reinforced with a compelling need to improve product performance, minimize product size, or meet stringent operating environment concerns. Embedding components is not a trivial endeavor and should include a strong and reliable support group of suppliers. Key issues to be addressed include:

- Procuring semiconductors in a wafer format
- Outsourcing metallization and thinning
- Testing embedded mixed function assemblies

The PCB fabricator will need to perform a full, functional electrical test at the substrate level. One question is, "How do we test, what do we test, and what features are needed to enable test?" Ideally, the originating companies will bring together the two primary suppliers: the circuit board fabrication specialist and assembly service provider. These partnerships must be willing to adjust their portion of the generated revenue against the overall process yield, which includes the sharing of losses from fabrication process defects and losses due to damaged components. DESIGN007

Vern Solberg is an independent technical consultant specializing in surface mount technology, micro-electronics design, and manufacturing technology. To read past columns or contact Solberg, click here.

A Job Worth Doing

Tim's Takeaways

by Tim Haag, CONSULTANT

My dad used to say, "A job worth doing is worth doing right." I had no idea what that meant at the time; I just knew that I was sick and tired of hearing his constant anecdotal phrases. This was only one of many that I heard while growing up, and after a while, I would turn a deaf ear to them. But as I got older, all of these phrases—especially this one—began to make sense. As much as I am loathe to admit it, I may have even repeated them a time or two to my children as time went by. As they say, what goes around...

The point is that I learned not only from my father's stories but also from the example he set in life. If you are going to do something, then make sure you do it right. Spend the time, be thorough, and complete the task to the best of your abilities. That was the lesson that my dad tried so hard to teach me, and it has proved itself to be very solid advice over the years.

The question then becomes, "How do I do this?" You might be thinking that you already are working as hard as possible to get the job done and you can't work much harder. Rest assured that I know exactly what you are saying. We who design PCBs for a living tend to be the type of people who will work themselves into an early grave if given the opportunity.

As an example, several years ago, I came in late one night to my company's office after swimming at a friend's house to check on the progress of a layout that I had in the autorouter. Back then, you would put the design in overnight and check the results in the morning. But since I was in the area, I thought that I would stop in and check it out. A tweak here, a few corrections here, and before I knew it, I was doing all kinds of rerouting on my board so that it would work in the router the way I wanted it to. Then, I noticed that it was getting brighter outside and people were beginning to come in. I had worked through the entire night without realizing it and was still wearing just a swimsuit and flip-flops.

So, yes, I get it. We PCB designers are made of the kind of tough stuff where we will work ourselves to death if given the chance. But in our all of our efforts, are we really doing it right, or could we somehow be doing it better? Let's take a moment to consider some other ways that we might help ourselves to improve.

Think Outside of the Box

I realize that I started this column whining about anecdotes, and now I'm throwing another one at you. Sorry about that, but let's take a look at it anyway.

Many years ago, I managed both a design group and the PCB design tools that the company used. Because of this, I often had people in my office that were panicky, frustrated, and just plain mad about the problems they were having with the tools. I did my best to try to help them or at least talk them off the ledge of their angst, but it didn't always work the way that I hoped. Some of these people would get really angry over the difficulties they faced. Then, I had an idea.

I brought in a recliner and set it up in my office. The test of my idea came sooner than I expected when almost immediately an engineer came into my office extremely upset because of some failure with the design tools. He was red-faced and spitting nails, so I invited him to sit down in the recliner and relax, which he did. Shortly after that, his anger melted away, and we were able to work through his problem together for a quick and easy resolution. I'm not sure that my upper management ever really bought into the idea of "the consulting chair," but from my perspective, it really helped smooth over some of the problems that I had to deal with. This out-of-the-box solution allowed me to calm people down so that I could more quickly get to the heart of the matter and find a way to help them.

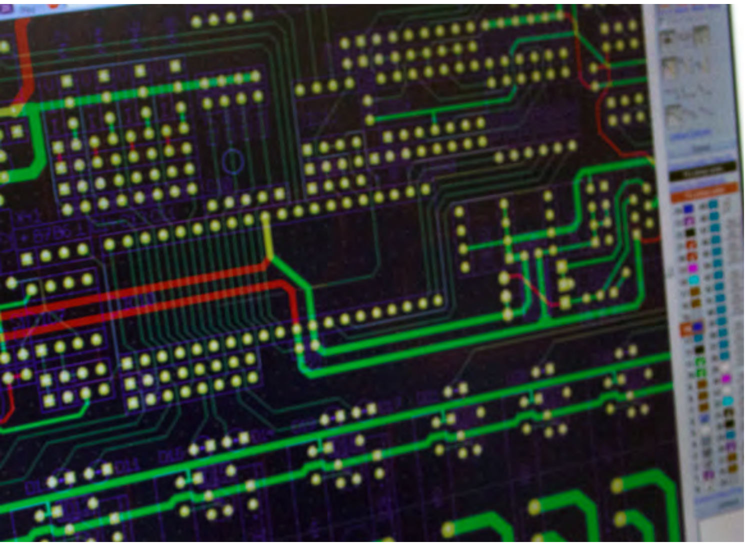
What kind of creative alternatives and solutions can you think of to help you through some of the problems that you face?

Take a Break Now and Then

You would think that I would have a grasp on the importance of this point more than anyone. After all, I've written about it before, and I've been dealing with it all my life. However, I still get snagged by it just as much as anyone else.

Last night, I was writing a piece on a topic that required a lot of research. I felt like I wasn't getting anywhere with it, so I kept working harder and harder and harder to MAKE IT HAPPEN. Finally, my dear wife dragged me by the scruff of the neck out of my office to eat dinner. After a while, my dark and stormy countenance began to wear off as my mind was refreshed and I entered into normal human-like conversation with those around me at the dinner table. Afterward, I returned to the office, and all of those talking points that didn't make any sense 45 minutes earlier were now coalescing into something that I could write about. I just needed a break.

One of the best ways that you can ensure that you are doing the job right is to stop doing the job now and then. Take a break, refresh your mind, and you will be stronger than before and have more creative energy to invest in getting the job done right.



Use All of the Resources Available to You

This may seem puzzling at first. After all, you are probably already working with a great set of PCB design tools that give you an incredible amount of versatility and power. But are you using the tools that you have to their fullest extent? Are you using all of the features and functionality that the tools have to offer you?

There have been many times in my career where after doing something the same old way for a long time, I found out that there was an easier way to do it. Perhaps there was a simpler menu pick or keystroke shortcut to use, or a way to accomplish the task using only three steps instead of five. The point is to keep your eyes open for better and more efficient ways of doing things in your design tools. There are probably a lot of them; you just have to find them. Try taking a look at some of the user chat channels and tooltips from your CAD vendor. You may find something that will save you a lot of time and effort that you never knew existed before.

In the same vein, there are often advanced features in our design tools that we simply have never used. It could be that our design process hasn't required their use, or perhaps they seemed too difficult to learn. In some cases, we don't want to take the time to add a new process to our already overloaded schedules. But these tools are designed to help you, and you may be missing out on some functionality that could solve some of your design problems.

Supply Chain Management

This issue of *Design007 Magazine* is dealing with the challenges of supply chain management. The entire design community is being confronted with the problem of parts becoming more expensive and not as easily available as they used to be. Finding parts when you need them and for the price that you can afford has become more important than ever before. This search can cost a lot of your time and effort to find what you need. You may even be spending extra time and money on additional staff and resources to help you.

Did you know though that many CAD tools can connect you online with your preferred component vendors for pricing, availability, and the latest data sheets? Without knowing which tools you use, I can't make a blanket statement that your specific tools can do this, but I do know that many tools out there have these capabilities. You can use these tools to help find the parts you need for the price that you want to pay that will be available when you need them. You can then pull those parts into your library and directly onto your schematic while consulting their online data sheets and other information resources.

There are now so many other features and functions in design tools than the simple place-and-route tools that we are used to using. Do yourself a big favor and take a look to see if you may already have some of these useful features that can help while saving you time and effort.

It's taken a lot of years, but I've come full circle and am ready to admit that my dad was right all along. A job worth doing is worth doing right; we just need to find those better ways of doing it. **DESIGN007**

Tim Haag is a PCB design consultant based in Portland, Oregon. To read past columns or contact Haag, click here.

Selecting the Right Board Thickness—A PCB Designer's Balancing Act

Connect the Dots

by Bob Tise and Dave Baker, SUNSTONE CIRCUITS

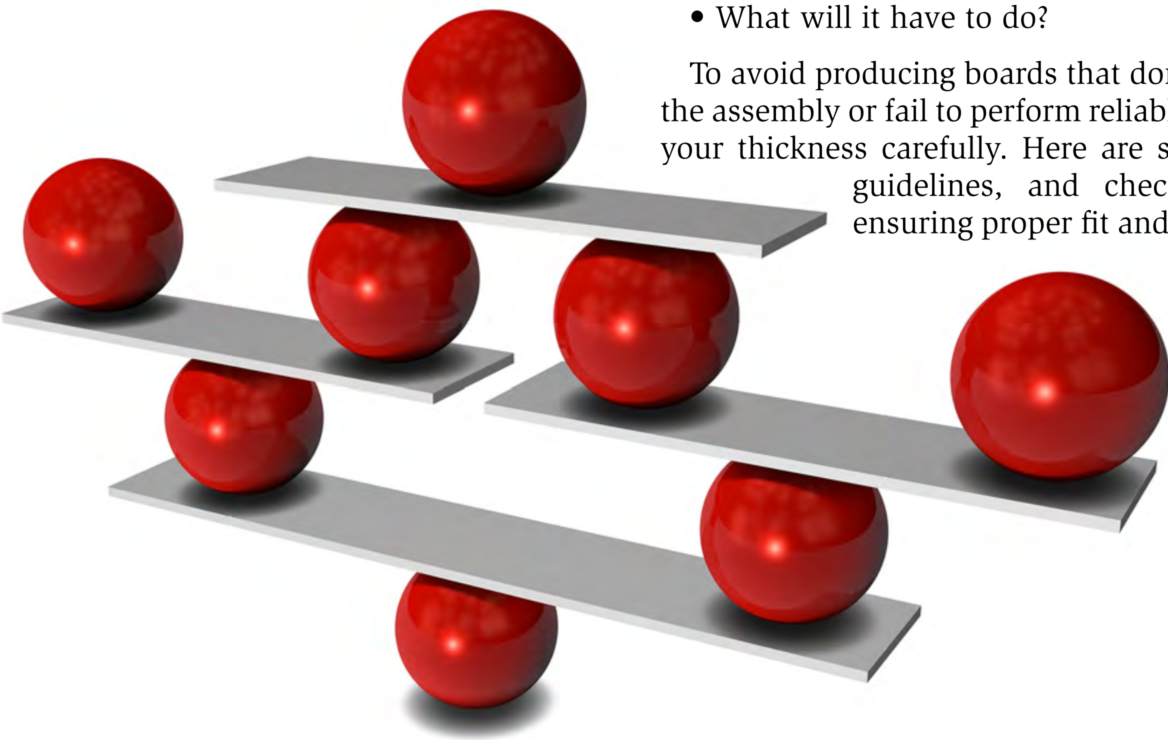
Choosing wisely is critical for PCB quality and performance, but it can be tricky depending on size constraints, functional requirements, and environmental factors. While we sometimes have a general idea about assembly requirements or how the board will be used, there can still be a lot of unanswered questions as we begin the manufacturing process. After all, there's a big difference between a PCB going into a drone and a PCB that will be part of a submersible drone and needs to be the size of a tennis ball, withstand intense heat or cold, and function forty fathoms below the surface.

When we receive a design, there's a lot we can evaluate before production to ensure its manufacturability and functionality. However, we are limited in our ability to judge whether a board is too thick or too thin as designed. In general, you might think thicker is better because the board will be less brittle and won't break as easily. But thicker is also heavier, hotter, and not appropriate for many applications or assemblies.

As you design your PCB, you should ask yourself:

- How thick does it need to be to work?
- How thin must it be to fit?
- What will it have to do?

To avoid producing boards that don't fit into the assembly or fail to perform reliably, choose your thickness carefully. Here are some tips, guidelines, and checklists for ensuring proper fit and function.



IT-170GRA1

IT-170GRA1 Value Proposition

- Halogen Free
- Best in class performance in its space
- Best in class CAF performance
- Very attractive price to performance ratio
- Pass MRT-6 requirements
  - -CAF > 1000 hours
  - -0.65mm pitch equivalent, no thermal issue
- Available with RTF and 2μ copper which provides additional performance on a very solid platform
- Df = 0.0075 @10GHz

Intel Purley Mid-Loss Solution – IT-170GRA1

Futures

Halogen free mid loss, Df < 0.01
Tg 180ºC by DSC, low CTE

Applications

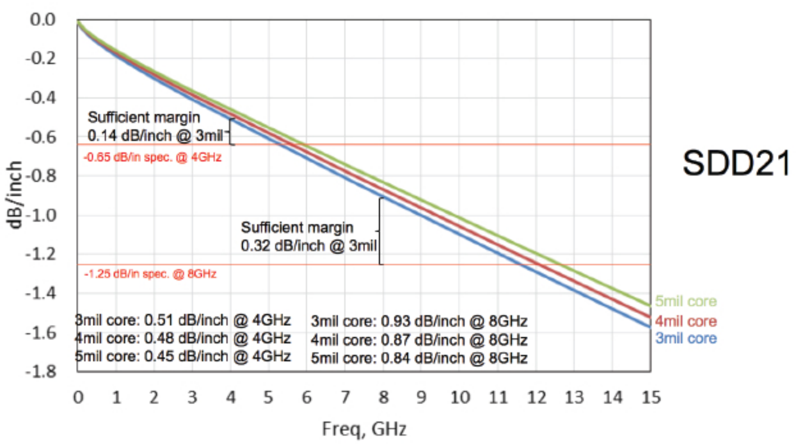
Server/Storage/Switch (10G, 40G)
Backplane, Telecom
Base station

Feature — *Purley Platform Mid-Loss Solution – Hi Tg*

General Properties

Items	Methods	IT-170GRA1
Tg (℃)	DSC	180
T-288 (w/ 1 Oz Cu, min)	TMA	60+
Td-5%(℃)	TGA 5% loss	380
CTE (%), 50-260℃	TMA	2.4
Peel strength (lb/inch)	1 oz	7.0
Water absorption	D-24/23	0.1
Dk: 2-10 GHz	Bereskin	3.96 – 3.99
Df: 2-10 GHz	Bereskin	0.0073 - 0.0075

IT-170GRA1 Insertion Loss



ITEQ

Test Your Design Assumptions

Keep in mind, PCB design is often as much an art form as it is a science. It's not as straightforward as plugging a bunch of numbers into an equation and receiving an answer.

So, before you lean towards a thicker or thinner board design or accept your design tool's default thickness, consider as many relevant factors as possible:

- Do you need to prioritize thermal transfer or insulation?
- Are there weight restrictions?
- How flexible or stiff does the board need to be?
- Will it be well protected inside the assembly?
- How much space do you have to work with?
- Are component lengths long enough for thicker boards?
- Is drill aspect ratio acceptable?
- Can your design be manufactured, and does it fall within manufacturer tolerances for very thick or thin boards?

Guidelines for Choosing PCB Thickness

Determining the answers to those questions will go a long way toward establishing peace of mind during the transition from design to prototyping. At a high level, thicker boards make sense when:

- There is a high layer count
- The board needs to be stiff because of its size or lack of protection in the assembly
- Durability is a priority

Thinner boards are preferable or necessary when:

- The board must be light
- In rare instances, flexibility is required to facilitate installation or solve interconnection issues; in general, you don't want to bend your boards
- The design calls for tiny vias; if the board is too thick when drilling through holes, drill bits will break, so consider keeping aspect ratios below 10:1 (preferably 8:1)
- The assembly is smaller with support for a board sandwiched into a tight space
- The device runs hot; a thinner board can help dissipate heat because thermal vias are shorter and pass through a smaller volume of insulating material

We recognize cost can influence decision-making. Your fabricator will have a standard board thickness, probably 0.062". Optional thicknesses may be more expensive, but savings evaporate quickly if the PCBs aren't durable enough for their intended use. Thinner boards can be damaged and may warp during solder processes.

Checklist for Determining Thickness

As you consider thickness in your design, we encourage an easily replicated checklist approach. Here are some factors to consider as you design your board:

- How big is your device? Thinner PCBs take up less space, so if you're building a device that will fit in your hand, go as thin as necessary to accommodate the space.
- It ain't heavy, it's my board: PCB weight can be an issue more often than you might think. Is your board part of a cellphone, Bluetooth headset, or tablet device? Thinner boards are lighter and can help give your device that trim figure we all appreciate.
- Know your connections: PCB edge or trace-based connectors, for example, require a board with thickness matched to fit in the mating portion of the connector. Some through-hole parts have pin lengths that limit board thickness.
- What's your frequency? Layer-to-layer thickness will affect signal integrity, crosstalk, impedance calculation, and signal loss. For high-frequency

signals, impedance control is a factor in determining the right thickness of your board.
- Be aware of the side effects of board flexibility: Will your product be subjected to excessive shock and vibration? Thin boards that flex can cause broken connections or components, so be sure it's properly supported in your assembly.

When it matters, the rest of your design can be absolutely perfect, but if it's the wrong thickness, it will fail electrically or mechanically. Before you send your design to be manufactured, take a few moments to be sure your board is the right thickness for your needs. **DESIGN007**

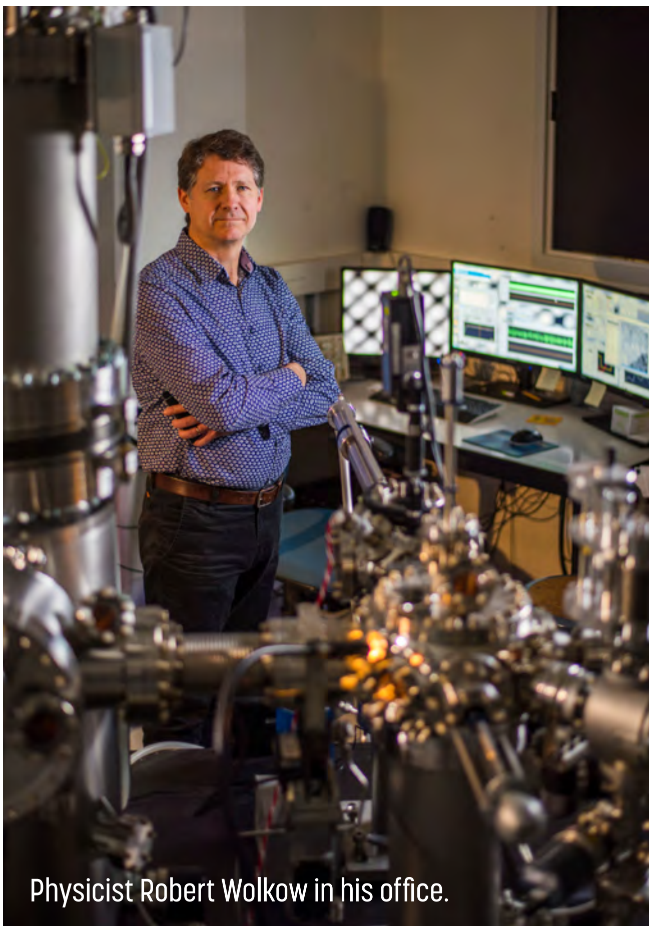

Bob Tise


Dave Baker

Bob Tise and **Dave Baker** are engineers at Sunstone Circuits. To read past columns or contact Tise and Baker, click here.

Atomic-scale Binary Logic Could Power Faster, More Energy-efficient Electronics

Researchers at the University of Alberta have designed atomic-scale versions of the binary logic components that allow computer processors to perform complex operations—the latest in a series of advances that lay the groundwork for faster electronics that use far less power.


Physicist Robert Wolkow in his office.

"The atom-scale devices we are developing create a new basis for computer electronics that will be able to run at least 100 times faster or operate at the same speed as today but using 100 times less energy," said atomic physicist Robert Wolkow.

The proof-of-concept devices Wolkow's team designed work much like logic gates in current microprocessors. But rather than using metal-oxide transistors mounted on silicon wafers, the atomic-scale logic gates use individual electrons confined in "quantum dots" directly on the silicon surface, dramatically reducing the space needed to pack millions of them into a microprocessor and the electricity needed to run them.

This year alone, Wolkow and his team—supported by his graduate students and research associates at the U of A, the National Research Council of Canada and the spinoff company Quantum Silicon—published research demonstrating an atomic-scale simulated computer circuit, a technique to boost the storage capacity of solid-state memory by 1,000 times, and a method for automated manufacturing of atomic-scale circuitry.

"Today's electronics have reached a point of maturation and can't be made any better," said Wolkow. "We have to stop using so much electricity to run our computers, and that means we need a drastic change in the kind of computers we use."

The research was published in *Nature Electronics.* Wolkow's work was supported by Western Economic Diversification Canada.

(Source: University of Alberta)

Getting the Best Performance from Encapsulation Resins

Sensible Design

by Alistair Little, ELECTROLUBE

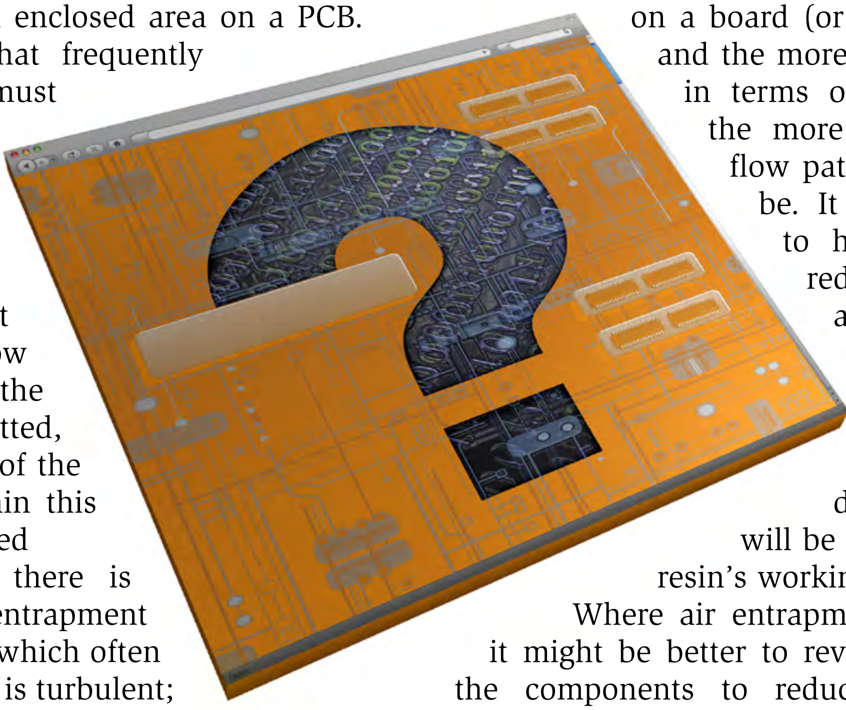
When I last broached the subject of potting and encapsulation resins, I went into some depth on the subject, explaining their chemistries and physical properties, how they behave when being mixed, applied and cured. For this column, I'm going to return to our tried-and-trusted Q&A format to offer four commonly asked questions about resins and their application, together with my responses, which I hope will help you achieve the best outcomes for all your potting and encapsulation jobs. So, setting material choice aside for the moment, let's start with a key aspect of potting: getting the resin in place.

Q: What common problems are associated with the physical mechanics of potting?

A: There are several problems that you will encounter when applying a resin to pot a component or an enclosed area on a PCB. Two problems that frequently crop up and must be addressed if you are to avoid some basic problems include:

Achievement of an even flow of resin within the volume to be potted, ensuring that all of the components within this volume are covered

Ensuring that there is minimal air entrapment within the resin, which often occurs if the flow is turbulent; air bubbles formed in close proximity to the components are most undesirable as the heat transfer performance of the resin is compromised, and component overheating is likely to occur, which could also create an area for chemicals to attack the component



Q: How do I avoid these problems?

A: This will depend upon the size and geometry of the unit to be potted. For example, where medium to large size units are concerned, it is better to apply the resin in two or more locations to allow control of the flow of resin. If you are using a mobile dispensing head, I would suggest a layer of resin across the whole unit rather than just depositing the required amount of resin in one location and allowing it to flow naturally throughout the unit.

The more components there are on a board (or in an assembly), and the more variable they are in terms of their geometry, the more complicated the flow patterns are likely to be. It may be possible to heat the resin to reduce its viscosity and improve the flow around a complicated component layout, but the downside of this will be a reduction in the resin's working and cure times.

Where air entrapment is a concern, it might be better to review the layout of the components to reduce the incidence


Mentor®
A Siemens Business
Speed up your
PCB design verification
by up to 90%
Get HyperLynx DRC for
FREE for 12 months
COMPATIBLE WITH ANY PCB LAYOUT TOOL!
mentor.com/get-drc
Visit us in booth #1900
APEX EXPO IPC 2019

of turbulent flows and the potential for air entrapment. Alternatively, you might want to consider the use of vacuum potting, which will ensure that the risk of air entrapment is as low as possible and that the resin will penetrate all of the available spaces.

Q: How do I select the most appropriate resin for the job?

A: Beyond ensuring that your electronic assembly will meet its design criteria in use, choosing the correct resin for encapsulation purposes is possibly the second most critical aspect of the entire design process. It is important to understand where and how the finished unit is going to be utilised and what performance criteria are expected of it. It is best practice to draw up a list of the standard operating conditions that the unit will be exposed to in order of importance, and then a list of what the extremes of those operating conditions are likely to be.

It is best practice to draw up a list of the standard operating conditions that the unit will be exposed to in order of importance, and then a list of what the extremes of those operating conditions are likely to be.

Also, the duration of time that the unit is exposed to the extremes of the expected operating conditions is critical. There is a world of difference between specifying a chemically resistant resin that can withstand fully permanent immersion in antifreeze, for example, and one that only requires resistance to occasional splashes of antifreeze that are wiped clear after short periods of exposure.

In a similar vein, if an application reaches a maximum temperature of 150°C, but this only occurs once a day for a couple of minutes, and the rest of the time the normal operating maximum is 90°C, then it is quite satisfactory to specify a resin with a normal operating temperature of 120°C, for instance. Most modern materials will tolerate quite wide excursions for short periods.

Other application requirements to be considered might include flame retardancy, should UL certification be necessary; optical clarity and UV resistance, particularly in the case of LED lighting assemblies; opacity, which is often desired when protecting circuit designs from potential IP theft; and RF signal compatibility. It is always recommended to undertake some testing to confirm the suitability of the selected resin as every application is unique in terms of the operating parameters, conditions, and the geometry of the unit.

Q: What resin chemistries are available and how do they differ?

A: There are three major classes of resin chemistry: epoxy, polyurethane, and silicone. Epoxy is the strongest and most chemically resistant of the three, but it is brittle, difficult (if not impossible) to remove for rework and repair, and is typically limited to operating temperatures between -40°C and +150°C. However, epoxy resins offer excellent adhesion to a wide range of substrates.

Polyurethanes are both tough and flexible and are suitable for applications operating at lower temperatures. However, due to the limitations of polyurethane chemistry, this type of resin is only suitable for applications reaching a maximum temperature of 110°C (though some can go to 130°C). The chemical resistance of a polyurethane resin is generally lower than that of epoxy, but polyurethanes—depending on the chemical backbone—outperform epoxies in water splash or immersion and high-humidity environments.

Polyurethane resins are typically used in marine applications where water penetration resistance is critical, and for applications subject to a high level of physical or flexural stress, such as the potting of accelerometers or sensors embedded in road surfaces. If large temperature swings are expected over a short timeframe or prolonged temperature cycles, then the flexible nature of the resin is advantageous, as there is a low probability of stress cracking occurring.

Silicones have the widest possible operating temperature range (-50°C to +220°C), and when cured, are the most flexible of the three resin chemistries. Their adhesion to certain substrates is poor—as is their chemical resistance—particularly with reference to acids, aromatics, and ketones. However, due to their aliphatic backbone, they offer excellent UV and colour stability, particularly for direct exterior applications. While they tend to differ widely in terms of performance characteristics, all resin types have excellent electrical insulation properties across their respective temperature ranges.

Conclusion

Readers should bear in mind that the foregoing offers only general guidelines. Resin chemistries have advanced considerably in recent years, and there are now resins from leading suppliers—such as Electrolube—that exhibit properties and performance criteria often well beyond the normal boundaries expected for that resin type. **DESIGN007**

Alistair Little is global business/technical director for resins at Electrolube. To read past columns from Electrolube, click here. To download your copy of Electrolube's micro eBook, *The Printed Circuit Assembler's Guide to... Conformal Coatings for Harsh Environments,* click here.

Design Tool Helps Inspire Flexible Electronics for Smart Everyday Objects

Researchers in the Sustainable Electronic Technologies Group at the University of Southampton are working with industrial partner PragmatIC to devise a process design kit (PDK) that will expand the potential for its mass market uses of flexible electronics.

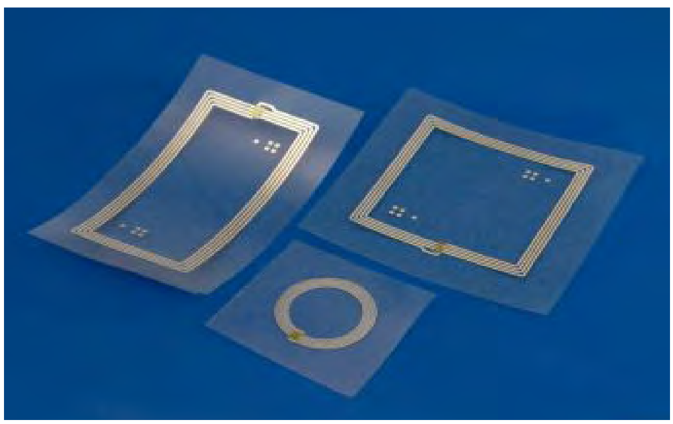
PragmatIC created ultra-low cost flexible integrated circuits (FlexICs) thinner than a human hair that can be embedded into everyday objects. The technology enables concepts like smart packaging, which can personalize product information and promotional offers, and interactive toys that can track moving pieces and dynamically change rules during play.

The new PDK at Southampton is being tailored to be used with industry-standard tools, making the new technology available to a wider community of designers for future applications.

Project lead Professor Mark Zwolinski says, "We have been researching, developing and using electronic design automation (EDA) tools for many years in the School of Electronics and Computer Science (ECS) and this project benefits from over 30 years of expertise that includes student projects and design exercises. We are excited to be working with PragmatIC on this promising technology and look forward to seeing creative uses of its FlexICs in smart everyday objects in homes around the world."

The collaborative project, known as a knowledge transfer partnership (KTP) and which is supported by InnovateUK, will make use of design tools that are already used in teaching and research in IC design within ECS.

(Source: University of Southampton)



MilAero007 Highlights

Standard of Excellence: Preparing Your Vendors for the Future ▶

Now, it's time to talk about the future with your PCB vendors. Referring to our previous columns, if you've done everything right so far, you will now have a strong working relationship with your PCB supplier. They understand all they need to know to fabricate perfect boards for you at this time. With trust and respect between you, you've truly formed a strong partnership going into the future.

Ventec's Marketing Strategy and Their Newly Appointed Technology Ambassador ▶

At electronica 2018, Mark Goodwin, chief operating officer at Ventec International Group, discusses the company's marketing strategy along with their newly appointed technology ambassador, Alun Morgan, and how he sees the world.

Artificial Intelligence Meets Materials Science ▶

A Texas A&M University College of Engineering research team is harnessing the power of machine learning, data science and the domain knowledge of experts to autonomously discover new materials.

NATO Members Drive Fastest Increase in Global Defense Spending for a Decade ▶

Global defense expenditure grew 4.9% in 2018, the fastest growth rate since 2008, according to the annual Jane's Defense Budget report, released today by business information provider IHS Markit.

Defense Speak Interpreted: PERM–Pb-free Electronics Risk Management ▶

In this column, we explore PERM—the Pb-free Electronics Risk Management Consortium. No, the group members do not all have curly hair! The name was chosen around 2008 by a group of engineers from aerospace, defense, and harsh environment (ADHE) organizations.

Sales of Military Antennas to Reach $1 Billion in 2018 ▶

The report forecasts a steady decadal growth rate of 4.9% for the military antennas market, with revenue slated to reach $986.8 million by the end of 2018.

Drones Can Now Flap Wings, Ride Wind Currents Like Birds ▶

The next generation of unmanned drones will act more like birds than machines, thanks to new research.

First Lockheed Martin-built Next-generation GPS III Satellite Responding to Commands ▶

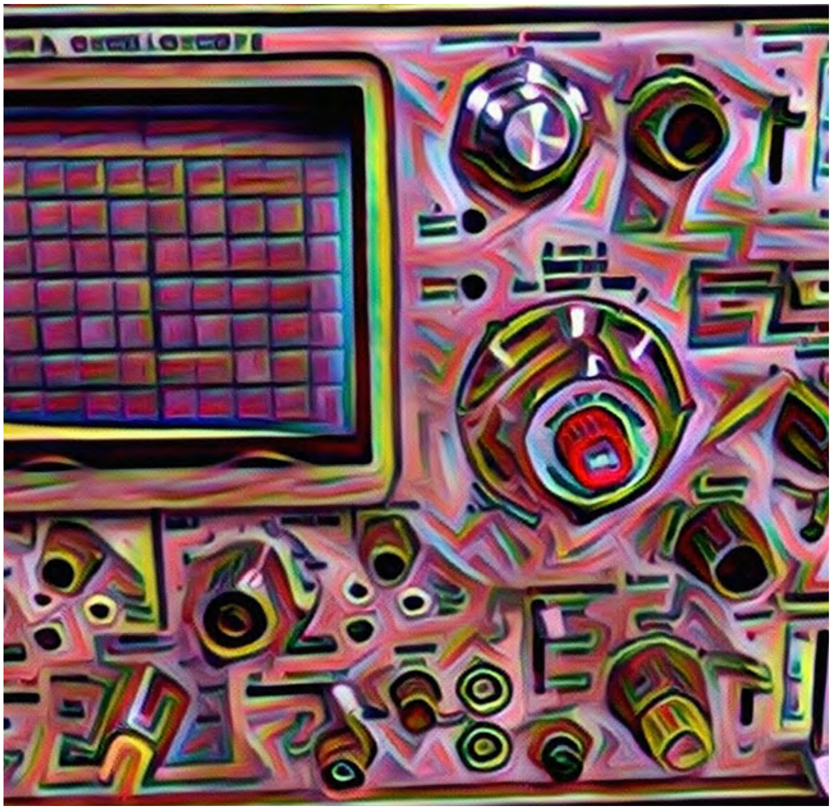
A major milestone in the U.S. Air Force's plan to bring new technology and capabilities to the GPS constellation, the first Lockheed Martin-built GPS III satellite began "talking" with engineers and operators from ground control, as planned, following its successful launch this morning.

Marsquake Sensor Lands Safely on Red Planet ▶

A Marsquake detector designed in part by Imperial engineers is set to help reveal Mars' inner structure.

NASA Looks to the Future, Seeks Next Level Visionary Aerospace Concepts ▶

NASA is looking for trailblazing ideas that could one day change what's possible in space. The NASA Innovative Advanced Concepts (NIAC) program is seeking Phase II proposals for the continuation of Phase I research studies.



The Effects of Meshed Reference Planes on Interconnects

Article by Yuriy Shlepnev
SIMBERIAN INC.

Interconnects in rigid and flexible boards can be formalized and simulated as transmission lines: strip, microstrip, coplanar, single-ended, or differential. The number of conductors in transmission line models is two for single-ended and three for differential lines. The signal conductors in the electrical model correspond to the interconnect traces, and the reference conductors correspond to one or two reference planes. Static or quasi-static field solvers are usually used to extract modal (impedance, attenuation, and phase delay) and per-unit-length (RLGC) parameters for analyzing data links. These models [1] may be accurate up to very high frequencies depending on the geometry and material models. Reference planes in such models are assumed to be solid. That is usually correct assumption for most of the rigid-board interconnects (except BGA fields in some cases).

But most of the flexible interconnects have meshed or hatched planes; the reference conductors have periodic cutouts. The question is how do you build accurate models for such structures? It turns out that the traces over conductors with periodic cutouts can be effectively simulated as periodic structures; that requires a 3D electromagnetic analysis of a small segment instead of the analysis of a cross-section or complete link. Similar to the regular transmission lines, periodic structures can be characterized with per-unit-length impedance, admittance, and the modal parameters: attenuation, phase delay, characteristic impedance [2 & 3]. That is very convenient for building models for interconnects with arbitrary length. This article shows what one can learn from this analysis using a particular example. Simbeor software was used for the analysis of interconnects with meshed planes.

Electromagnetic Waves in Traces Over Planes With Periodic Cutouts

In general, a periodic structure is made up of repetitions of a unit cell in one, two, or three dimensions. For instance, Figure 1 provides a simple example of the periodic repetition of a rectangular cutout in transmission line reference plane [4]. The cutouts are repeated in

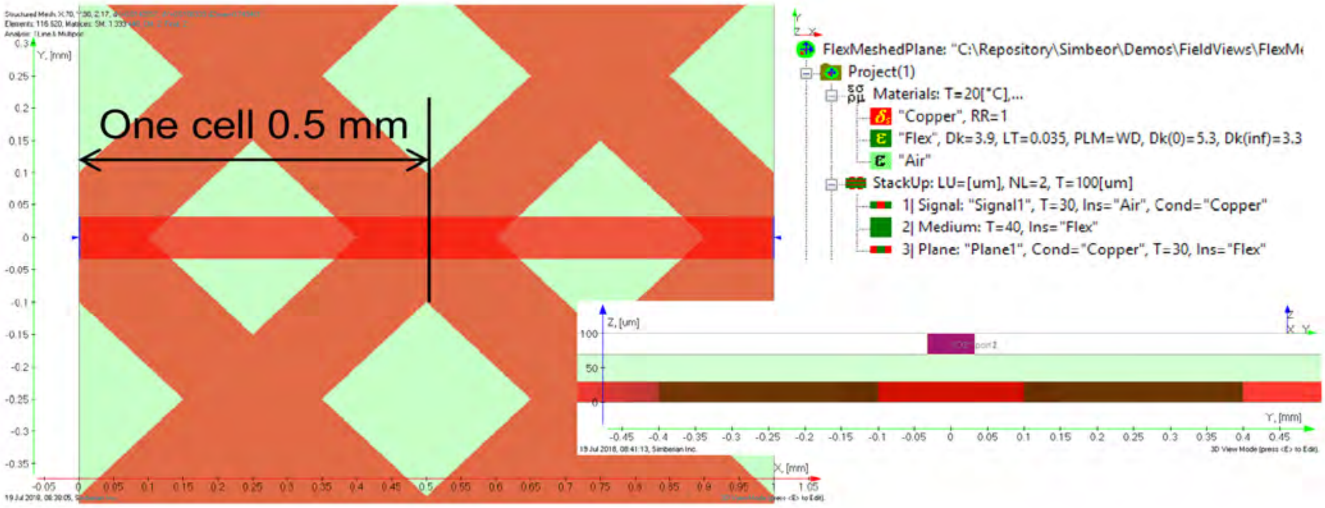


Figure 1: A trace over a meshed plane as a periodic structure. The geometry of the two cells is required to extract modal and per-unit-length parameters.

two dimensions, but for the wave propagating along the trace, it is essentially a one-dimensional periodic structure (it becomes 2D for leaky-mode investigation). To extract the per-unit-length and modal parameters of such a structure, 3D electromagnetic analysis is needed due to the non-TEM structure of the waves propagating along the traces.

Most of the transmission lines in PCB or packaging interconnects have so-called quasi-TEM waves with the electric and magnetic fields mostly perpendicular or transverse to the propagation direction. Parameters of such transmission lines can be accurately approximated with the analysis of a single cross-section in a 2D field solver. With the cutouts in the reference plane, the waves become non-TEM and not even quasi-TEM due to the presence of the longitudinal components in the electric and magnetic fields. The electric field above the cutouts is shown in Figure 2. You can clearly see the non-transverse nature of the electric field in the areas of angles in the reference conductors.

Each cutout is a discontinuity that distorts the electric and magnetic fields. As a result,

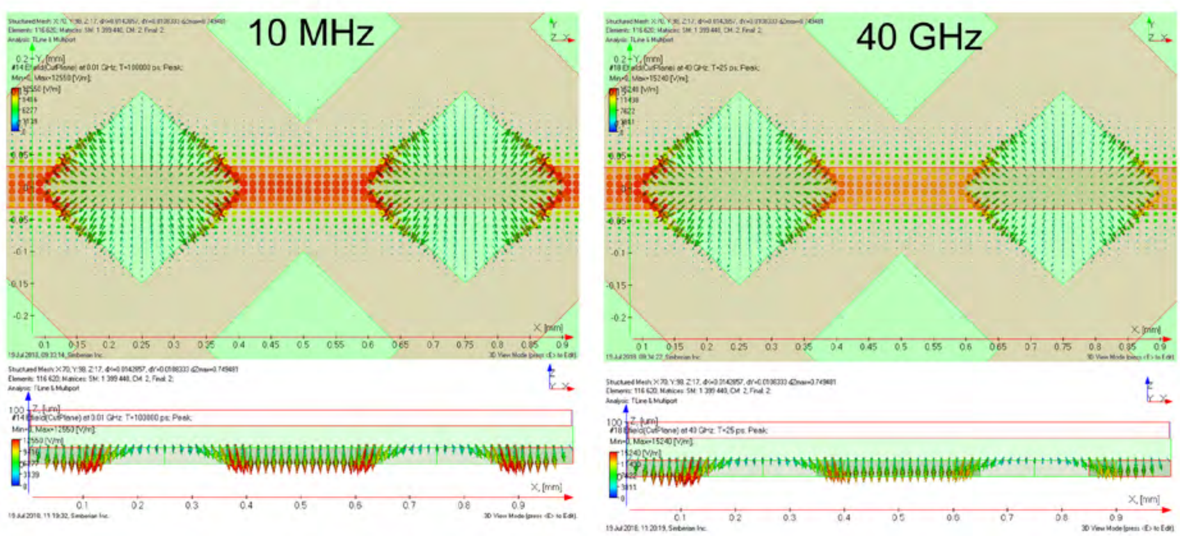


Figure 2: Peak values of electric field intensity in the plane with a Z-coordinate of 35 μm (right above the cutouts).

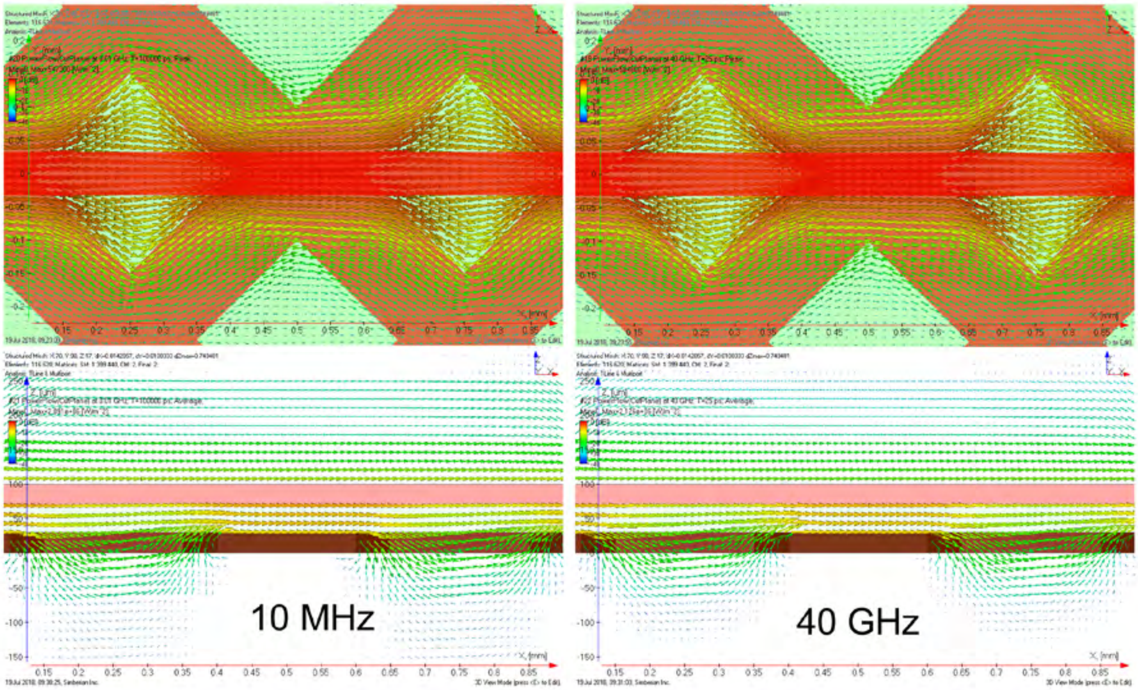


Figure 3: Peak values of power flow density in plane with the Z-coordinate at 35 μm (top) and along the trace.

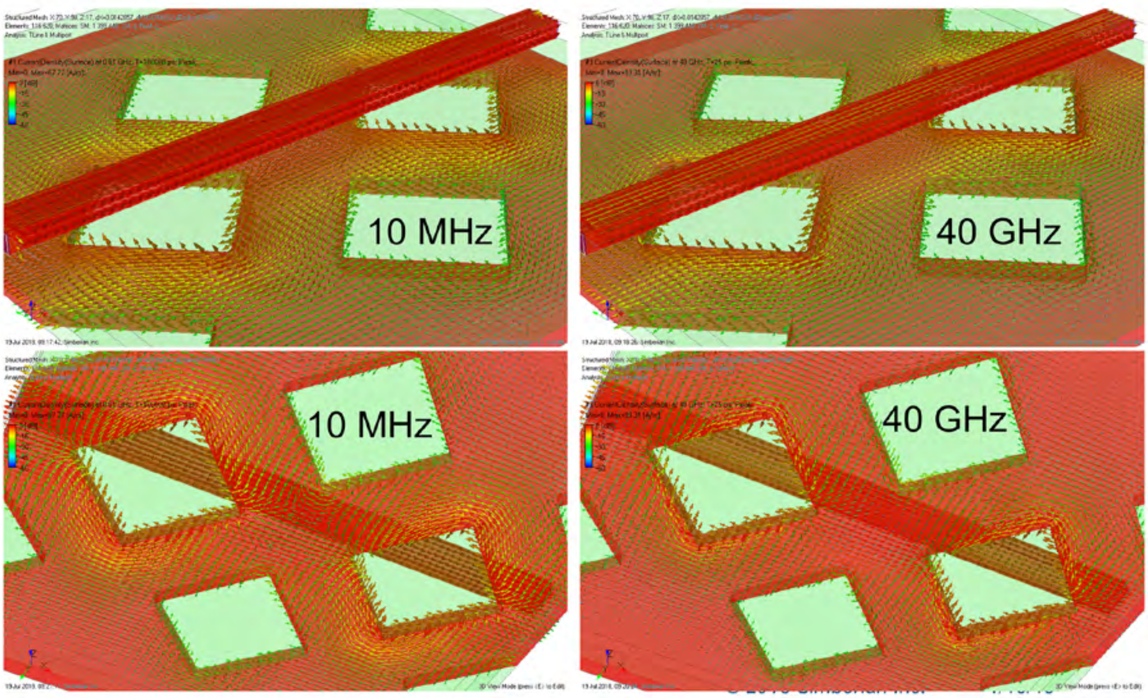


Figure 4: Peak values of the surface current flow density on strips below (top) and the opposite side (bottom).

power flows not along the trace—as in the cases or transmission lines with the solid planes—but deviates in the horizontal as well as in the vertical directions (Figure 3). The power flow density depicts where the energy of the signal propagate; it is a very useful tool to investigate the field localization [5]. The cutouts below or close to the trace also prevent the return currents from flowing straight along the traces (Figure 4).

As you can see, the currents in the strip are pretty much the same as expected in cases with the solid planes. However, the cutouts destruct the return current flow; it has to bypass the cutouts. Also, the currents flow on the opposite side of the meshed plane. As you will see, that may cause unwanted coupling to the traces shielded by the meshed planes. All of those effects must be simulated to design predictable interconnects.

Characteristic Impedance

Now, let's get back to the basics and take a look at the characteristic impedance of the dominant mode in the periodic structure formed by the trace over a meshed plane. Any deviation from the link target impedance can cause the increase of the reflection losses in interconnect link. With the stackup defined in Figure 1, a 65-µm wide trace over the solid reference plane gives about a 51-ohm transmission line impedance at 1 GHz (green lines in Figure 5). Relatively large cutouts in the reference plane right below the strips increase the impedance to about 62 ohms (red lines in Figure 5). This may cause the design failure due to excessive reflection losses or possible resonances in the system with non-uniform impedance links. This is the worst-case scenario for this size of cutouts.

The trace may be shifted to have more metal below the trace to provide a better path for the return current. This shift reduces the impedance down to 55.5 ohms (blue lines in Figure 5). The deficiency of the reference conductor area reduces the capacitance and increases the inductance of the periodic structure. In reality, without control of the trace position over the cutouts, one should expect the impedance variations from 55.5 to 62 ohms in this case. Can you get it back down to 50 ohms? Yes, but, unfortunately, only for a particular position of the trace. For instance, if you increase the trace width to 100 µm, it is going to be about 50 ohms if it goes directly over the cutouts in the reference planes (black lines in Figure 5). The impedance will decrease substantially if the trace is shifted. In general, all possible scenarios must be simulated.

Attenuation and Delay

Other important interconnect design parameters are signal attenuation and delay. Those two parameters are derived from the complex propagation constant of a transmission line or periodic structure mode. Attenuation is the energy loss to heat up dielectric (polarization losses) and conductor (conduction losses). The minimal phase delay corresponds to the signal front delay in general. Attenuation in dB/mm and phase delay in ps/mm for different configurations are compared in Figure 6.

Note that attenuation, delay, and characteristic impedance depend on the material

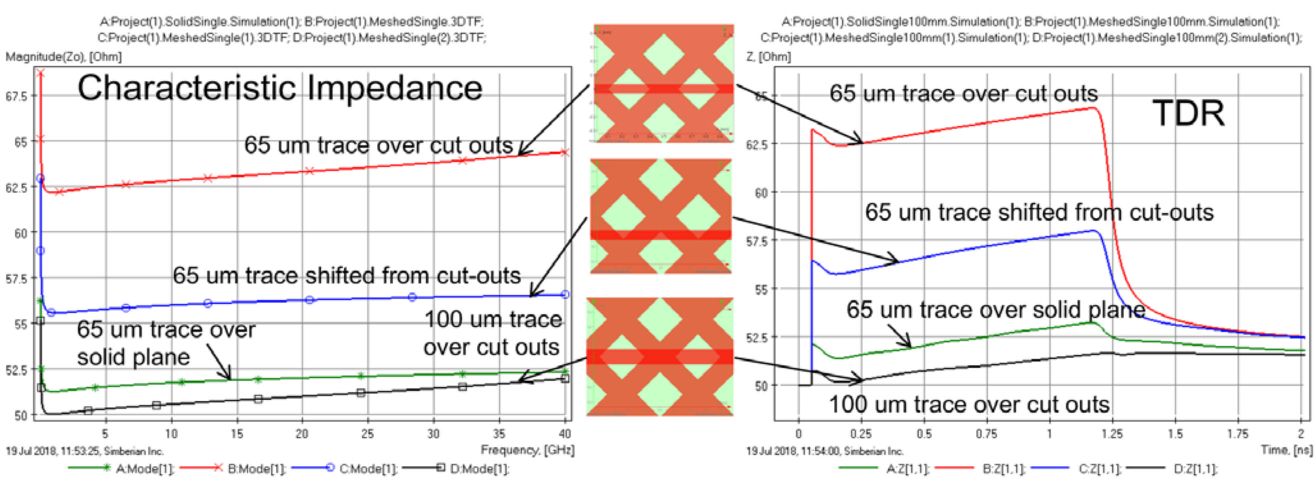


Figure 5: Characteristic impedance for traces over solid and meshed plane (left) and corresponding TDR of a 10-cm trace segment.

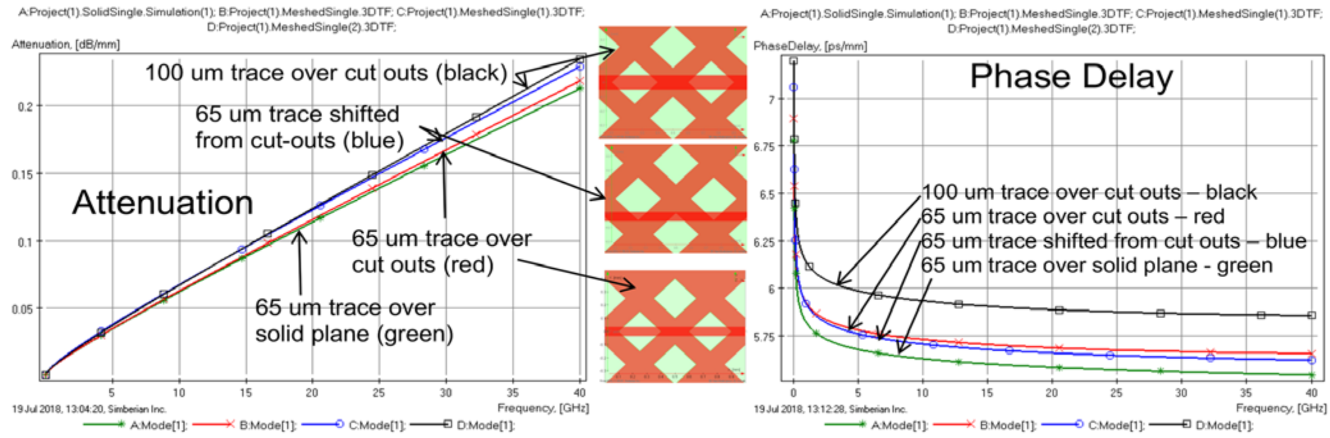


Figure 6: Attenuation (left) and phase delay (right) for traces over a solid and meshed plane.

models used in the analysis. To have accurate interconnect models, the material models must be either confirmed or identified [1]. Thus, the things observed on the graphs here cannot be used to draw any design guidelines for any other cases.

In this particular case, one can observe approximately 0.01-dB/mm differences in attenuation and a0.4 ps/mm in phase delay for the investigated structures. This is the case of a high-loss dielectric. The conductor roughness was neglected too, and this assumption alone can ruin your design [1]. The conductor roughness can be easily accounted for in the simulations, but there were no numbers to define the model for this case. The outcome of such investigation may be substantially different if the dielectric is very low loss and conductor roughness is taken into account. The point is that all of that should be simulated and taken into account.

S-parameters

Scattering or S-parameters and compliance metrics derived from them are becoming popular in designing interconnects for digital systems. S-parameters of a trace segment with the meshed reference plane can be easily computed as soon as the modal and per-unit-length parameters of the corresponding periodic structure are extracted. Note that this type of analysis is not an approximation; it follows from the physics of the periodic structures [2 & 3]. An alternative to this analysis is a simulation of a complete link in a 3D EM solver, which could be a very time- and resource-consuming process. Note that most of the flexible interconnects have micron-sized conductors that bring the onset of skin effect into the multi-GHz bandwidth. Thus, simulation of such links requires meshing of the conductor interior, as it is done with Trefftz finite elements in all examples in this article (Simbeor 3DTF solver).

As an example, insertion and reflection losses of a 10-cm segment of the traces over a meshed plane were computed and compared with the trace over the solid reference plane (Figure 7). The insertion loss differences are within 1 dB at 20 GHz for the investigated configurations. However, reflection losses vary dramatically. This is the result of differences in the characteristic impedances.

Coupling and Mode Transformations

Finally, what if the meshed plane is used to shield traces on the opposite sides? You can expect almost ideal isolation in case of the solid plane; in other words, the traces can be considered as not coupled with the solid plane. The fields do not penetrate through the solid planes at the frequencies with well-developed skin effect. Coupling at lower frequencies below the skin effect does not matter for the signal integrity analysis (interconnects become electrically short). When you cut the holes, the

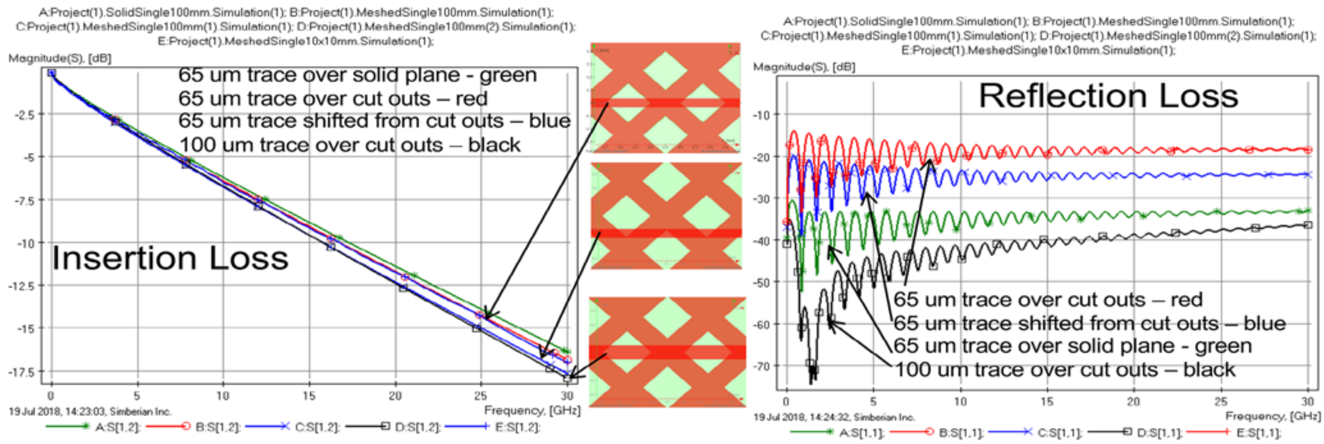


Figure 7: Insertion loss (left) and reflection loss (right) for 10-cm traces over a solid and meshed plane.

power flow through the holes and currents flow on the opposite side of the plane (Figures 3 and 4). It can cause crosstalk if another trace gets into that area; this is yet another example of a possible violation of electromagnetic field localization [5]. However, unlike with the via holes, this loss of localization effect can be easily simulated.

For example, two traces are simulated with the results shown in Figure 8. The traces are located exactly on top of each other. The worst-case scenario is with the traces right over the cutouts. It gives the maximal exposure through the holes, and as a result, maximal coupling. One can observe very significant far-end crosstalk (FEXT) (blue line on the left graph in Figure 8). This crosstalk can kill the signal with the main spectral harmonic around 8–10 GHz in this particular case. The insertion loss goes sharply down as shown on the right graph in Figure 8 (red curve), which means that the signal harmonics will not get through 10 cm of this interconnect!

Shifting the traces off of the cutouts helps, but not much. With the losses and geometry of interconnects used in this example, the insertion loss null is shifted to about 17 GHz and became wider too. The near-end crosstalk (NEXT) is not significant in this case. This example shows how important to model all possible signal degradation factors. Looking at just attenuation (Figure 6) or the insertion loss in a line segment (Figure 7) may be way too optimistic in this case if the meshed reference plane is used as the common reference for traces on opposite sides. That scenario would be a disaster for signals with 8 Gbps and higher data rates!

Conclusion

In this article, a 3D electromagnetic analysis of traces over a meshed plane is used to illustrate the following:

- Traces with meshed reference planes are periodic structures and have to be simulated as the periodic structure with a 3D EM solver
- Meshed reference planes have a significant effect on the electromagnetic fields, power flow, and return current distribution
- Meshed planes significantly change the characteristic impedance as well as the reflection and insertion losses
- Cutouts may cause significant crosstalk as well as EMI/EMC issues

Similar observations are valid for a differential case [4]. In addition to the effects observed in the single-ended case, one can expect differential-to-common-mode conversion if two traces of a differential pair are not aligned with the cutout in exactly the same way (can be considered as a type of crosstalk). All of that must be simulated to design predictable interconnects.

The Simbeor 3DTF solver based on Trefftz

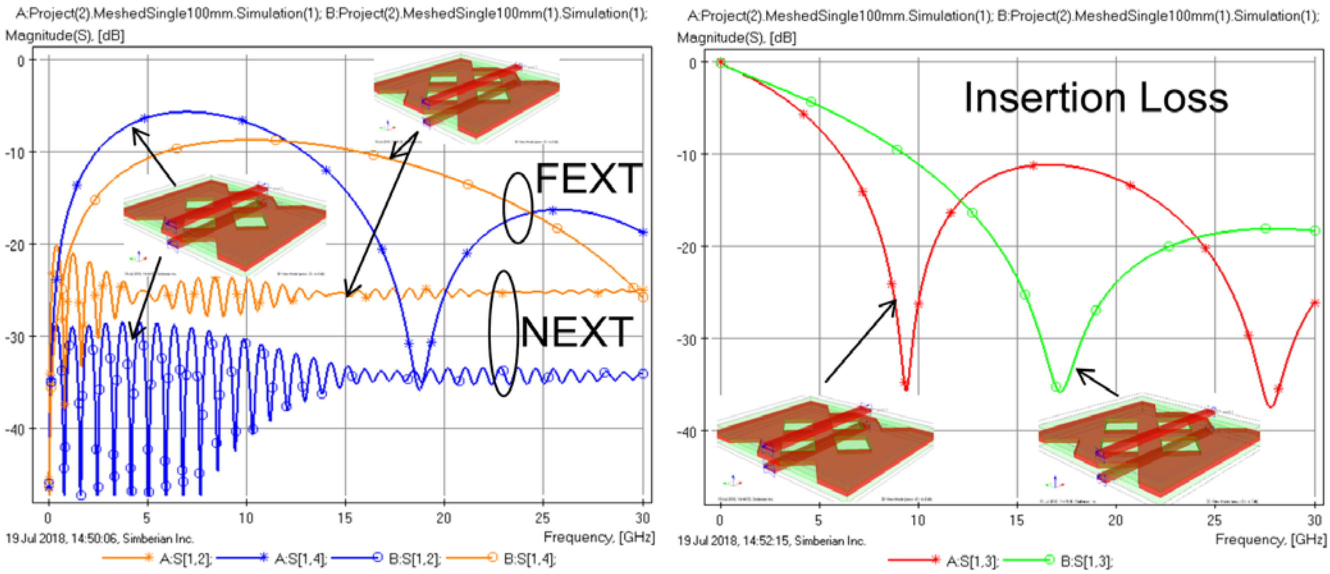


Figure 8: Near-end crosstalk (NEXT) and far-end (FEXT) crosstalk (left) and corresponding insertion loss (right) for 10-cm traces over a meshed plane for traces over the cutouts and mostly over the conductor.

finite elements was used for all simulations in this article. Step-by-step instructions on how to set up analysis of single and coupled traces over meshed planes [6]. Note that traces in BGA fields, with close via fences, and some other structures can also be formalized and simulated as the periodic structures. This is an elegant, accurate alternative to the brute-force approach of an exhaustive 3D EM analysis of a complete link. **DESIGN007**

References

1. Y. Shlepnev, "Advanced stackup planning with impedance, delay and loss validation," June 28, 2018.
2. R. E. Collin, Field Theory of Guided Waves, Wiley-IEEE Press, 1991.
3. D. Pozar, Microwave Engineering, Wiley, 2012.
4. Simbeor Demo Videos
5. Y. Shlepnev, "Life beyond 10 Gbps: Localize or Fail!" April 13, 2018.
6. Simbeor Application Notes

Yuriy Shlepnev is founder and president of Simberian Inc.

Powering Up Fuel Cells

Oak Ridge National Laboratory scientists studying fuel cells as a potential alternative to internal combustion engines used sophisticated electron microscopy to investigate the benefits of replacing high-cost platinum with a lower cost, carbon-nitrogen-manganese-based catalyst.

"We used electron microscopy to demonstrate that atomically dispersed manganese can act as an oxygen reduction reaction catalyst while also increasing durability," said David Cullen, R&D Staff at ORNL.

Fuel cell technologies hold promise for use in vehicles because of their high-power density, low operating temperature and carbon-free emissions. Yet, the high cost associated with platinum-based catalysts and insufficient durability of alternative platinum-free catalysts remains a market barrier.

"Our team's finding could open up the potential for widespread use in transportation and other energy conversion applications," said Cullen.

ORNL researchers were part of a team that produced the results published in Nature Catalysis.

(Source: Oak Ridge National Laboratory)

WHAT DO ALL OF THESE LEADING COMPANIES HAVE IN COMMON?

ATOTECH

Cerambus Technology Group

CIMS Creativity in Motion

COMETGROUP Technology with Passion

DownStream Technologies

esi Designed for Brilliance. Engineered for Production.

KIC Making Ovens Smarter™

KYZEN

MacDermid Alpha ELECTRONICS SOLUTIONS

Mentor A Siemens Business

MIRTEC

orbotech The Language of Electronics

PULSONIX

Rehm THERMAL SYSTEMS

schmoll asia pacific

ventec INTERNATIONAL GROUP 騰輝電子

VJ ELECTRONIX Process Control Solutions CONFIDENCE DELIVERED

環球集團 World Wide Group

They all benefit by advertising with us–PCB007 China Magazine.

Should your name be on the leaders' list?

GET STARTED NOW!

I-Connect007

GOOD FOR THE INDUSTRY

pcb007china.com

Editor Picks from PCBDesign007

1 Beyond Design: 10 Fundamental Rules of High-speed PCB Design, Part 3 ▶


Barry Olney

Planes are essential in today's high-speed multilayer PCBs. Unfortunately, the number of power supplies required is increasing dramatically with IC complexity. Now, accounting for them all has become a real challenge. The high number of supplies generally leads to higher layer count substrates.

2 Mentor Discusses New DFT, DFM, and Design Verification Tools ▶

At electronica 2018, John McMillan, digital marketing program manager-Electronic Board Systems, and Mark Laing, business development manager-Valor Division of Mentor, a Siemens business, discuss new tools for PCB design verification, as well as design for manufacturing (DFM) and design for testing (DFT).

3 Top 10 Most-read Design007 News of 2018 ▶

Every year, we like to take a look back at the most popular PCB design news and articles. These are the top 10 most-read PCB design news items from the past year. Check them out.

4 Quiet Power: Measurement-to-simulation Correlation on Thin Laminate Test Boards ▶

A year ago, I introduced causal and frequency-dependent simulation program with integrated circuit emphasis (SPICE) grid models for simulating power-ground plane impedance. The idea behind the solution was to calculate the actual R, L, G, and C parameters for each of the plane segments separately at every frequency point, run a single-point AC simulation, and then stitch the data together to get the frequency-dependent AC response. This month, I will demonstrate how that simple model correlates to measured data and simulation results from other tools.

5 Martyn Gaudion on Signal Integrity Modelling and Stackup Tools ▶


Martyn Gaudion

The accuracy of signal integrity modelling continues to improve, and stackup tools are becoming widely used, which now include material suppliers' datasheet information. During the recent electronica show in Munich, Germany, Martyn Gaudion, managing director at Polar Instruments, explained how Polar often serves as a bridge between PCB design and fabrication, and why educating his customers is so critical.

6 Connect the Dots: Six Tips to Ensure Parts Fit on Your Board ▶

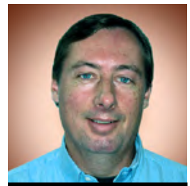
Alternative through-hole parts occur when the land pattern matches, but the pin size is off. Pins may not fit through the holes, or if they do go into the holes, they may not solder well. Solder will need to flow through the gap between the pin and the hole barrel. If there is not enough space to allow enough solder mass to flow through the hole, the circuit board will absorb heat from the molten solder and cause the solder to solidify partway up the hole. This is called a cold solder joint and can result in a premature failure of your circuit.

7 Rick Almeida Discusses DownStream's Latest News ▶


Pete Starkey and Rick Almeida

At the 2018 electronica exhibition in Munich, Rick Almeida, founder of DownStream Technologies, brings Editor Pete Starkey up to speed with the company's latest news.

8 Lightning Speed Laminates: Skin Depth and Its Interaction With Final Plated Finishes ▶


John Coonrod

Skin depth is the depth within a conductor where the majority of the radio-frequency (RF) current resides. Imagine looking at a cross-sectional view of a circular wire and being able to see how much current is within that cross-section. If the current is supplied by a battery and is direct current (DC), the amount of current is distributed evenly across the cross-sectional area of the wire. The current density is the same everywhere in the area of that wire.

9 Martin Cotton's Parting Shot ▶


Martin Cotton

Martin Cotton is a unique personality in the PCB industry. As Cotton says, "I'm a designer—look at my haircut!" Cotton gave the keynote at the Institute of Circuit Technology's 2018 Harrogate Seminar, challenging his audience to consider laminate dielectric properties in the context of power and cost in a presentation entitled 'The Effect of the Dk of a PCB Laminate on the Cost-effectiveness of Office Rental Space. Intrigued?' Read on!

10 Altium Releases Designer 19 ▶

Altium Designer 19 introduces new features that make designing complex, high-quality projects easier, faster, and more accurate from concept to production.

Career Opportunities

Pssst!
Are You Looking for Someone?

Place your notice in our Help Wanted section.

For just $500, your 200 word, full-column—or, for $250, your 100 word, half-column—ad will appear in the Help Wanted section of all three of our monthly magazines, reaching circuit board designers, fabricators, assemblers, OEMs and suppliers.

Potential candidates can click on your ad and submit a résumé directly to the email address you've provided. If you wish to continue beyond the first month, the price is the same per month. No contract required. We even include your logo in the ad, which is great branding!

To get your ad into the next issue, contact:
Barb Hockaday at barb@iconnect007.com or +1.916.608.0660 (-7 GMT)

I-Connect007
GOOD FOR THE INDUSTRY

smt007 pcb007 pcb design007

Career Opportunities

Career Opportunities

Manncorp™

SMT Field Technician
Huntingdon Valley, PA

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

Duties and Responsibilities:

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

Requirements and Qualifications:

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

We Offer:

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

apply now

ventec INTERNATIONAL GROUP 騰輝電子

Technical Support Engineer, Germany

We are looking for a technical support engineer to join our team at our German facility in Kirchheimbolanden. The successful candidate will assist potential and current customers in appreciating the benefits of using and optimizing the use of Ventec materials in their PCB manufacturing processes, enhance customer loyalty and satisfaction, spread the use of Ventec materials, and grow sales. The technical support engineer will provide a two-way channel of technical communication between Ventec's production facilities and U.K./European customers.

Skills and abilities required for the role:

- Scientific and technical educational background
- Experience in the PCB industry in engineering and/or manufacturing
- Strong communications skills (German and English) with the ability to write full technical reports for group or customer distribution
- Ability to work in an organized, proactive, and enthusiastic way
- Ability to work well both in a team as well as an individual
- Good user knowledge of common Microsoft Office programs
- A full driving license is essential
- Willingness to travel regularly throughout Europe and occasionally to Asia

We offer:

- Excellent salary and benefits commensurate with experience

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

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

apply now

Career Opportunities

Career Opportunities

Gardien

Sales Personnel, Japan

The Gardien Group is looking to expand the sales team in Tokyo, Japan, and seeking highly motivated team players with a positive attitude. Prior experience in the PCB industry is an advantage but not necessary for the right candidate.

The role involves working closely with the customer to identify their needs and deliver the right solution. The candidate should be able to offer a high level of customer satisfaction to ensure ongoing sales.

Training will be provided along with a competitive benefits package, excellent growth opportunities, and periodic bonuses.

Interested candidates, please contact us at careers.jp@gardien.com with your resume.

Kindly note only shortlisted candidates will be notified.

apply now

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

Career Opportunities

Career Opportunities

Career Opportunities

Mentor®
A Siemens Business

PCB Manufacturing, Marketing Engineer

Use your knowledge of PCB assembly and process engineering to promote Mentor's Valor digital manufacturing solutions via industry articles, industry events, blogs, and relevant social networking sites. The Valor division is seeking a seasoned professional who has operated within the PCB manufacturing industry to be a leading voice in advocating our solutions through a variety of marketing platforms including digital, media, trade show, conferences, and forums.

The successful candidate is expected to have solid experience within the PCB assembly industry and the ability to represent the Valor solutions with authority and credibility. A solid background in PCB Process Engineering or Quality management to leverage in day-to-day activities is preferred. The candidate should be a good "storyteller" who can develop relatable content in an interesting and compelling manner, and who is comfortable in presenting in public as well as engaging in on-line forums; should have solid experience with professional social platforms such as LinkedIn.

Success will be measured quantitatively in terms of number of interactions, increase in digital engagements, measurement of sentiment, article placements, presentations delivered. Qualitatively, success will be measured by feedback from colleagues and relevant industry players.

This is an excellent opportunity for an industry professional who has a passion for marketing and public presentation.

Location flexible: Israel, UK or US

apply now

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

Events Calendar

48th NEPCON JAPAN ▶
January 16–18, 2019
Tokyo, Japan

IPC APEX EXPO 2019 Conference and Exhibition ▶
January 26–31, 2019
San Diego, California, U.S.

DesignCon 2019 ▶
January 29–31, 2019
Santa Clara, California, U.S.

SMTA Pan Pacific Microelectronics Symposium ▶
February 11–14, 2019
Kauai, Hawaii, U.S.

EIPC 2019 Winter Conference ▶
February 14–15, 2019
Milan, Italy

China International PCB & Assembly Show (CPCA Show 2019) ▶
March 19–21, 2019
Shanghai, China

MicroTech 2019 ▶
April 4, 2019
Cambridge, U.K.

Medical Electronics Symposium 2019 ▶
May 21–22, 2019
Elyria, Ohio, U.S.

PCB Pavilion @ LCD EXPO Thailand ▶
June 27–29, 2019
Bangkok, Thailand

Additional Event Calendars

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

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

TECHNICAL EDITOR: **PETE STARKEY**
+44 (0) 1455 293333; pete@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 © SINISA BOTAS**

DESIGN007 MAGAZINE

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

January 2019, Volume 8, Number 1
DESIGN007 MAGAZINE is published monthly, by BR Publishing, Inc.

ADVERTISER INDEX