SEPTEMBER 2022

I-Connect007

DESIGN007 MAGAZINE

THE SUPPLY PAIN SCALE

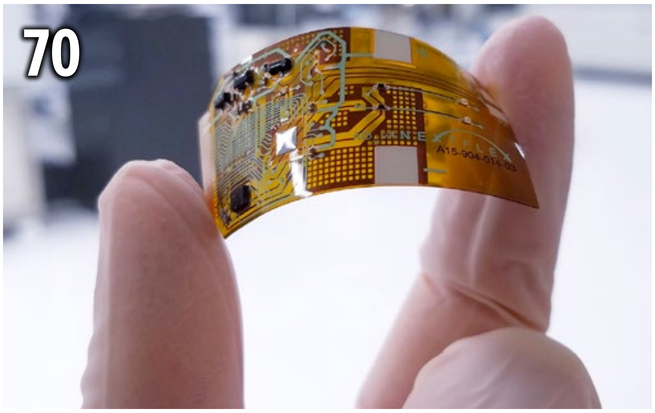
DESIGN007
MAGAZINE

Supply Chain Management

Supply chain management is a dynamic situation, with a number of global forces that can create pain for PCB designers. In this issue, we speak with a variety of experts and bring you up-to-the-minute insight about designing PCBs in this ever-changing environment.

10

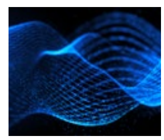
DESIGN007 MAGAZINE



SHORTS

DEPARTMENTS

COLUMNS



ARTICLES & INTERVIEWS

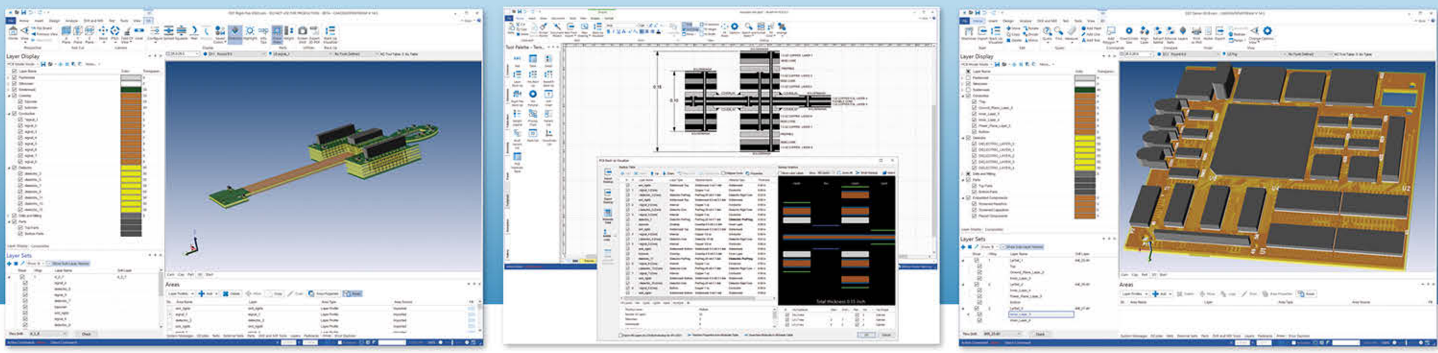
HIGHLIGHTS

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

BluePrint-PCB®

CAM350®

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



DownStream Technologies

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

Working Through the Design Pain

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

In the August issue of *SMT007 Magazine,* Managing Editor Nolan Johnson discussed what he termed "supply pain management." This reminded me of the question that doctors often ask: "What's your pain level on a scale of 1–10?"

You all really deserve a lot of credit. For years, you've been working through your supply chain pain, like Rip Wheeler after he got shot on "Yellowstone." It hurts, but we're short on cowboys, so get back to work. Designers and design engineers have learned to navigate this supply chain craziness, snatching up components in short supply or making do with lower-tech parts because they're available.

When was the supply chain's last normal month? It's hard to put a date on it, because even in the years prior to the pandemic, we went through a period of 50-week lead times. We had barely recovered from that when COVID hit. Changes of this magnitude have created pain up and down the supply chain.

Designers have always been the "hub of the wagon wheel," as Mary Sugden once said, but now you've become much more. You've become supply pain managers. You do the best you can to ameliorate the pain felt by you and the other stakeholders. Today's designers try to avoid the pain with a little extra early plan-

ning, but sometimes the pain gets bad enough that you might want to take a painkiller.

Supply chain management is a constantly changing dynamic situation, and our systems were not set up for this level of "component awareness." Until recently, many PCBs were designed with single-sourced components because it just wasn't an issue for most parts. Now, many OEMs are demanding that all components have multiple sources.

An upside to the supply chain misery is that everyone else in the electronics industry is going through the same thing. You're all in this together, and I know you'll come out of this stronger than ever.

Many of these new design tricks—technical and otherwise—are likely to become part of the typical design cycle moving forward. Now that designers and engineers are accustomed to having second and third sources for everything on the BOM, do you think we'll ever go back to the old way? I doubt it.

While gathering content for this issue, we spoke with designers and design engineers of every stripe. We heard horror stories and tales of succeeding against all odds, as well as a variety of clever workarounds. We learned that communication—always a slippery topic—has become vital in this environment.

Our experts bring you up-to-the-minute insight about designing PCBs in this ever-changing environment. First, we have a conversation with our columnist Kelly Dack, who shares several new techniques for designing PCBs in today's atmosphere. Chintan Sutaria of CalcuQuote explains how software tools can help you keep on top of ever-changing inventories. Next, Altium's Dan Schoenfelder discusses their Electronic Digital to Design Index (EDDI), which provides a current snapshot as well as a historical record of each part's availability back to January 2020. And columnist Chris Young offers a set of guidelines for "designing through" the supply chain pain.

We have an article by Malcolm Thompson of NextFlex, who discusses some of the innovation that has come out of the chip shortage. We also have another installment of Anaya Vardya's DFM101 series.

And we have columns from our regular contributors Barry Olney, Matt Stevenson, Tim Haag, Dave Wiens, Joe Fjelstad, and Beth Massey and Beth Turner. And let's welcome Trina Taylor, guest columnist for Sunstone Circuits this month.

Hard to believe that summer is almost over, and that means trade show season is just around the corner. I hope to see you on the road. **DESIGN007**

Andy Shaughnessy is managing editor of *Design007 Magazine.* He has been covering PCB design for 20 years. To read past columns, click here.

A New Sourcing Paradigm

Feature Interview by the I-Connect007 Editorial Team

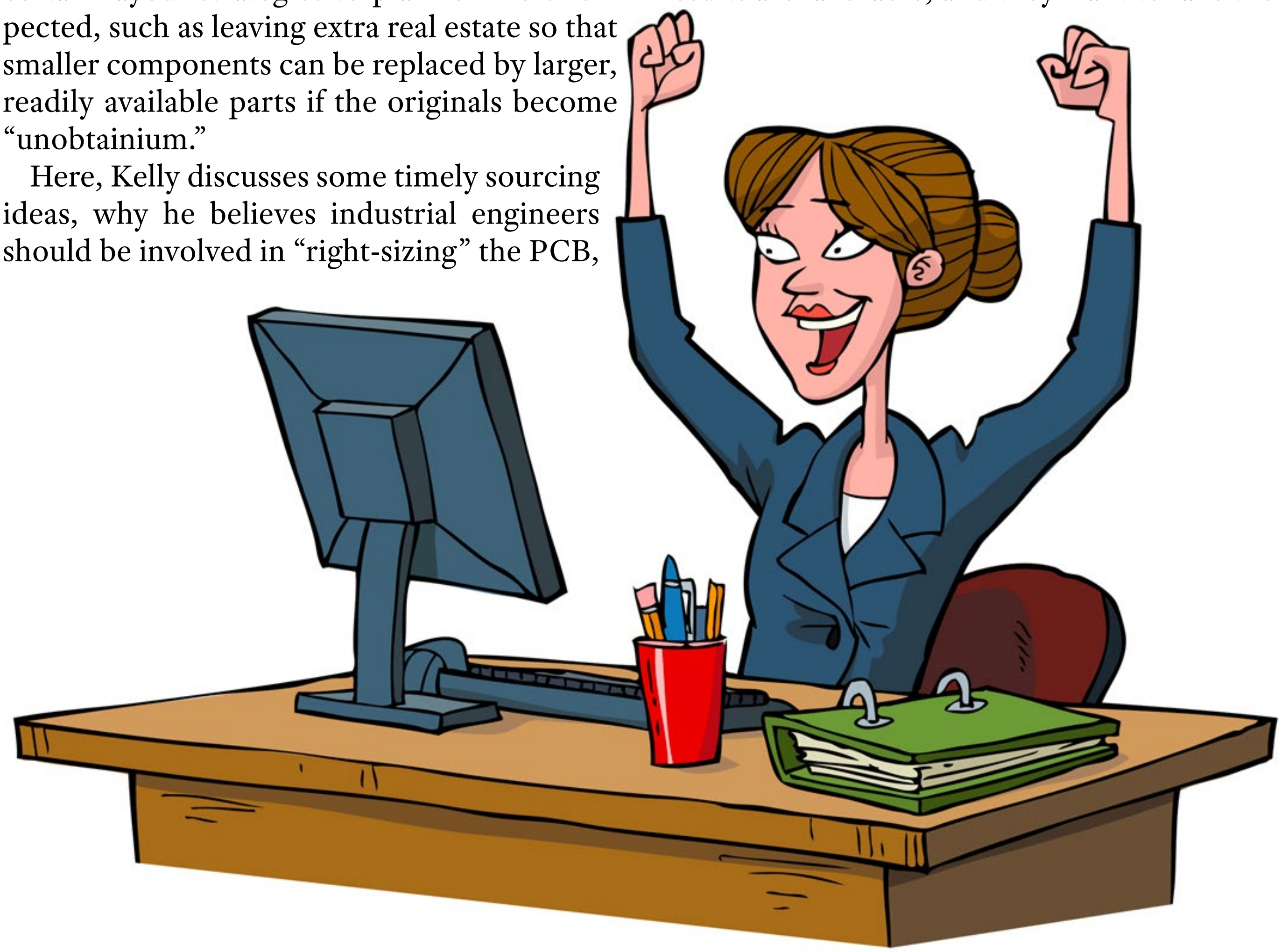
We've seen many changes over the past few years, and nowhere are they more evident than in the world of sourcing components. Sourcing has become one of the biggest challenges facing PCB designers and design engineers today. Gone are the days of procuring parts from a single source, and communication between stakeholders and distributors is critical.

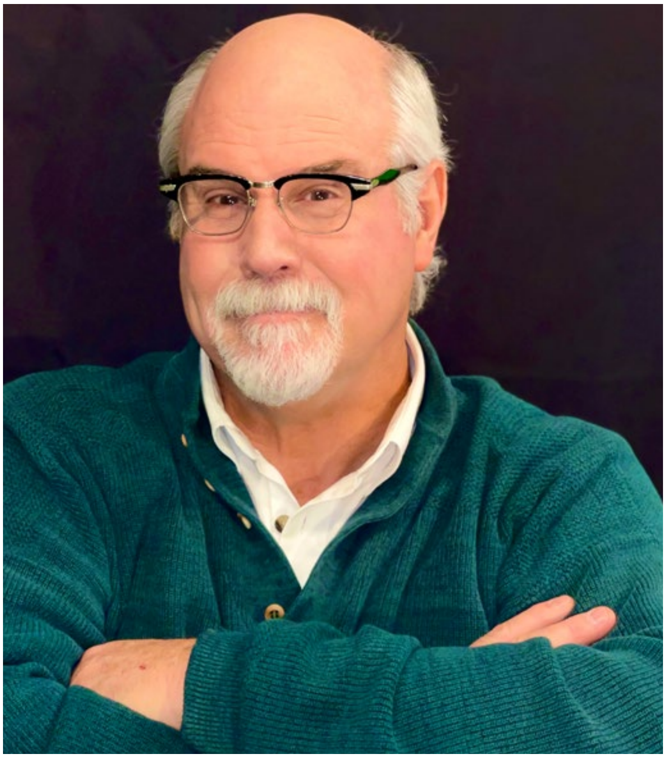
But as we learned in a conversation with I-Connect007 columnist Kelly Dack, an EPTAC design instructor and PCB design team leader at an assembly provider in the Pacific Northwest, PCB designers can use certain layout strategies to plan for the unexpected, such as leaving extra real estate so that smaller components can be replaced by larger, readily available parts if the originals become "unobtainium."

Here, Kelly discusses some timely sourcing ideas, why he believes industrial engineers should be involved in "right-sizing" the PCB, and why sourcing efforts must also focus on recruiting—or creating—PCB designers.

Andy Shaughnessy: Kelly, you've been involved with DFA issues for years. Why don't you talk about some of the challenges you see with sourcing?

Kelly Dack: Sure. The problems we see are projects that begin as traditional projects. We have customers who come to us much like on "Shark Tank." There's an idea, some seed research and development has been done, the results are favorable, and they want to take the





Kelly Dack

product to production. At that point, we evaluate the product to determine its feasibility for meeting the capabilities of our assembly lines. Everything is fine, just like it always is—until it's not. That's because with today's rusty, knotted supply chain we are not able to source certain components. That one little thing can slow down or stop the entire project. We limp along until we can either change the design, modify it so that we can find an alternative component, or redesign the circuit so that we can use alternate components that will do the same thing.

Happy Holden: You understand design for manufacturing. Now we're not talking about process capability; we're talking about part availability, but that's still manufacturing.

Dack: That is still manufacturing and sourcing, and that plays into it, Happy. A product can be totally 100% designed and look very good from a production standpoint and from a DFM standpoint. But now, when a part suddenly becomes unavailable, it could require another part to be designed in, which may not be form-fit and functionally the same, but it could still work. But now designing in this part has to be done properly so that design for manufacturability is considered. How many parts do you have to nudge? What's the domino effect if we do manage to fit this part in?

Holden: So, we have to make a change in design so that the information is known up front and it's not a do-over?

Dack: The problem we're finding is that everybody is using the smaller parts, and the large parts are left over. So, if you want to find an equivalent functional part, chances are all the tiny ones have all been gobbled up, and you must be willing to buy the larger-scale parts. But how do you fit the larger scale parts into your layout?

Shaughnessy: There are those dominoes, right?

Dack: Right. Let's take a step back. During the shrinkage era in the '90s, when we were working on iterating boards down as small as they could be, I coined the term "practical packaging density plus." The idea is that we determine an appropriate packaging density, and then once you get that dialed in, what's wrong with adding just a little more for future design purposes or manufacturing purposes? Why do we constantly design to the minimum, tightest density?

Shaughnessy: Are you talking about adding more real estate?

Dack: It can be real estate. Packaging density has to do with the amount of real estate on a given PCB vs. the components and everything else, and all the geometries involved with the components.

Barry Matties: But you're bound by the enclosure that it's going to live in. You have those constraints on you. How do you adapt to that?

Dack: Practical packaging density plus starts at the front-end of the projects, with the industrial engineers. This has been part of the problem—there's a sweet spot for product size. That sweet spot used to be, "Just shrink it, and make it as small as possible. The smaller our product, the more we're going to sell."

But then it went in reverse when it became ergonomically challenging. For instance, the smart wristwatch products began to swell in size and utilize more of the larger OLED screens. That gave us more room to design in more manufacturability to the product.

Matties: Are the industrial designers more aware of this now and taking that into consideration because they understand the shortages?

Dack: I'm not sure. I don't think it comes in at that point. The realization that a PCB design is too dense happens when mechanical engineering forwards the PCB outline to the electrical engineering department, and a feasibility study is done with the given amount of board area. At that point, we typically find that the parts we selected won't fit on the board and we have to go back through the process again.

Shaughnessy: Is somebody buying up all the components during the course of the design? I've heard of OEMs blaming Tesla and NVIDIA for hoarding capacitors, but isn't it smart to be prepared?

Dack: Funny you mention that. I just talked to one of our component engineers, and he's seeing a lot of parts being bought up. I asked him if the term "hoarding" was appropriate. Well, it might not be explained like that, but if the parts are found, they're being snatched up and reserved.

Matties: Yes. I expect that some stockpiling is going on.

Dack: This is a really dynamic problem. It would be "project suicidal" to spec in a part that's not available. That's not what we do. The problem is that months in advance, during the concept feasibility stage, a customer has prototyped with parts that are available in prototype quantities. When it goes to volume, now those parts in volume quantities aren't available.

Matties: When do you think this starts to resolve, Kelly? Are we talking a year or two down the road, or more?

Dack: The numbers that I've seen show that 2023 still looks very bad. There's still a logjam in the ports. But even when the logjam is alleviated and the parts become available in mass quantities, it will be too late for a lot of projects. These projects will either have been canceled or some other technology will have replaced the first-choice technology. There was a story I heard recently: People are finally getting their parts delivered and then saying, "We don't need them anymore." I understand that Amazon is telling them, "Just keep the parts. Don't even ship them back to us," because they're having challenges restocking right now.

The numbers that I've seen show that 2023 still looks very bad. There's still a logjam in the ports.

Shaughnessy: Right. They'll even refund your money, but they don't want the parts back.

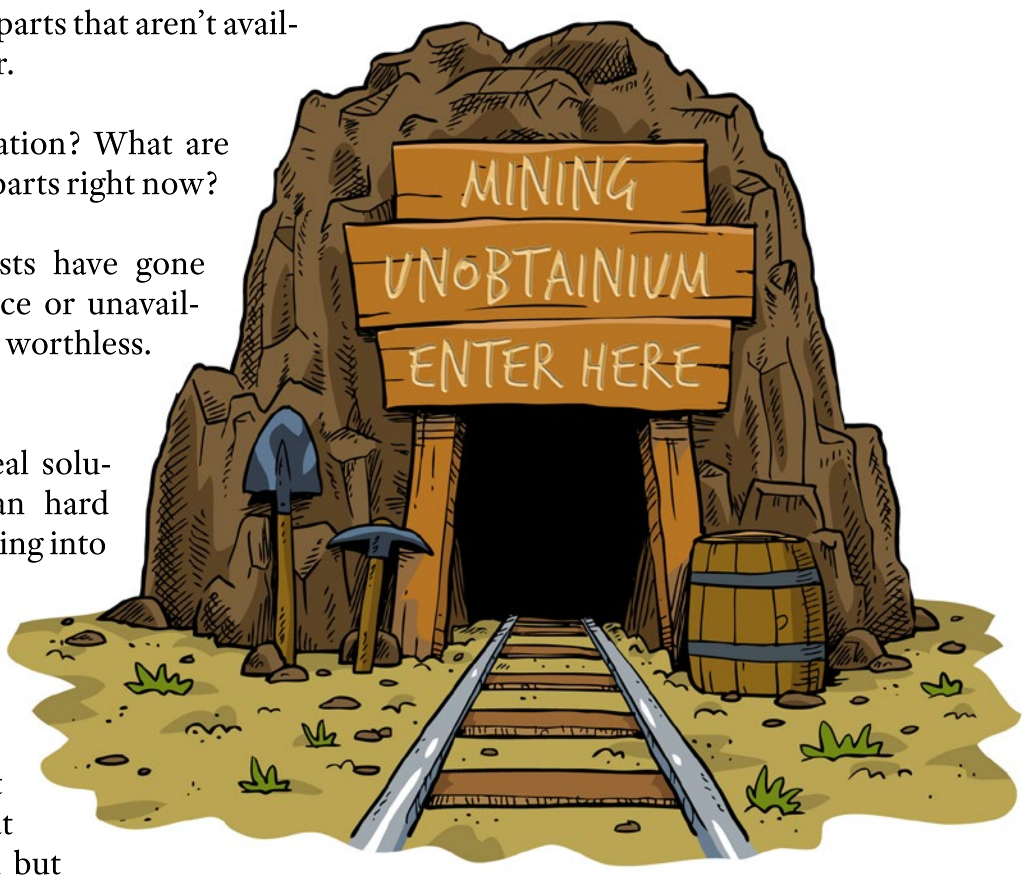
Matties: This is becoming common practice. But there's going to be a good surplus market for that stuff.

Dack: Yes, there will. But right now, it's about having awareness up front, being tied in with the sources, and that's huge. Our component engineers and purchasing are definitely tied in, but we still know of cited parts that aren't available. It's a project-stopper.

Matties: What about inflation? What are the pricing dynamics for parts right now?

Dack: Transportation costs have gone sky-high, but obsolescence or unavailability can render a part worthless. It's a mix.

Matties: So, there's no real solution out here other than hard work and a lot of effort going into sourcing.



Dack: I use the term "unobtainium" from time to time—trying to create something out of nothing. I know that people are working hard but working hard on mining unobtainium only gets you so far. I think we're frustrated in the industry because there are no answers for certain questions.

Shaughnessy: When you're working with the customer, how has that changed? It must be really frustrating for you and the customer.

Dack: Well, if the customer can get it redesigned and send us the specifications, then we provide the internal documentation and manage the manufacturing of the product. It's pretty seamless. However, we're also experiencing a "supply chain" issue with personnel. We're not immune to the shortage of mid-level to senior-level designers in our group. Talk about unobtainium because there is a shortage of experienced PCB design engineers. We've determined that if we can't find personnel with experience, we will create that experience. We'll hire inexperienced candidates with potential and desire, then train and teach them what they need to know.

Shaughnessy: Where are you finding these potential designers?

Dack: We're using the term "PCB design engineer" now; that's who we want to create. We will take a candidate with a technical aptitude or an electrical engineer who wants to learn more about printed circuit design and turn them into a PCB design engineer.

Holden: Is this partially due to the gray wave and the pandemic upscaling?

Dack: Absolutely. We talked about the gray tsunami, Happy. The first time I heard that term was a few years ago. It's real; we're losing the experienced designers. How will they be replaced? We see that schools are pumping out engineering personnel, they're very quick to learn printed circuit design software and tools, but they're running with scissors. It's evident that they're not being trained in DFM or DFX principles. We must still teach that.

Holden: Many of these experts know that they can move up the ladder to a higher-paying job that doesn't include layout. But there are others, like your job, that involve managing and mentoring a PCB design engineering team.

Dack: It's amazing. Everyone knows it's not uncommon for someone with mid-level experience in printed circuit design to be making six figures.

Matties: Are you adding more resources to the search team?

Dack: I'm bad at metaphors, but it's like you're mining unobtainium. Will more miners help you find something that's not there? That sounds so negative, but it's valid. However, we are well connected with a dynamic search team we found through I-Connect007. Thanks for featuring their services in your publications, by the way.

> I'm bad at metaphors, but it's like you're mining unobtainium. Will more miners help you find something that's not there? That sounds so negative, but it's valid.

Matties: People feel like they must actively do something to find new designers as well as electronic components, right?

Dack: Sure. If you've got a customer and stockholders, it's not a good sign to raise your hands in a meeting and say, "There's nothing I can do." I think you've got to show that you're trying to respond. That's just good work ethics.

Shaughnessy: Kelly, we were talking recently about sourcing strategies, and how some designers are providing two or three different sources per component. You said, "Well, we probably should have been doing that anyway, but it just didn't matter until now." Do you think this is going to be standard practice from now on?

Dack: Yes, multi-sourcing is a good practice. Typically, in a company that's procuring parts, a requirement is that the part cannot be sole-sourced, or you need many signoffs if a part will be sole-sourced. That's typically avoided.

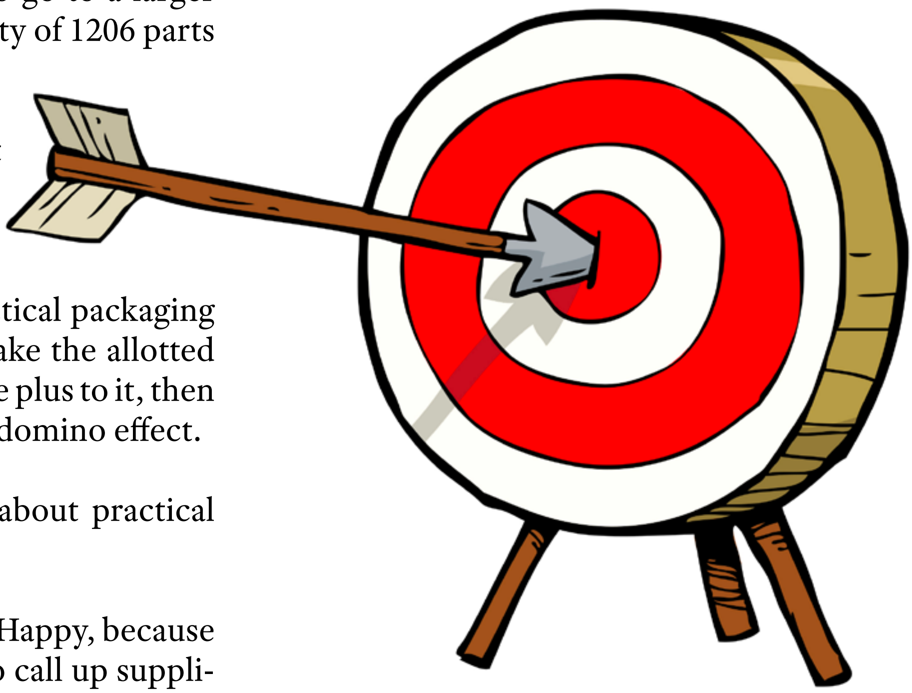
Holden: We have not lost any wafer fabs and no component manufacturer has gone out of business. As far as I know, we have the same capacity for components that we had before the pandemic. Why is there a shortage? I understand why the automobile guys got shorted, because they canceled orders, somebody else took their place, and then demand exceeded supply.

Dack: Right. Once the part becomes available, it's too late. Even with great component search engines, there's no guarantee that we could meet the requirements if the parts involved are unobtainium.

Holden: Now, on another issue, can't you set up access, testing, and rework keep-outs on your library parts so that when that part is placed, you have the room to do a rework or get a printed circuit test on it? Wouldn't that be handled by the libraries?

Dack: Sure, but let me provide some context. Say, for instance, a simple chip capacitor or diode becomes unavailable. The diode or the chip package might be an 0402, so the library part is created as an 0402 with all the proper spacing and courtyards so that it's pure DFM. But the problem is that once there's a part shortage and everybody bought all these up, the only option now may be to go to a larger part because there may be plenty of 1206 parts that are functionally the same.

It's just that now you need a space for a 1206 for that part that used to be an 0402. How do you incorporate that larger part into the area it was allotted for? This is where practical packaging density plus comes in. If you take the allotted space for the part and add a little plus to it, then you can potentially reduce the domino effect.



Holden: I'd like to read more about practical packaging density plus.

Dack: Let's collaborate on that, Happy, because the tendency for designers is to call up suppliers or sources and find out, "How small can I get this thing?" For some reason, we gravitate to thinking, "I don't want to waste any material."

You suck all the dimensions around everything so that we're not wasting all this material. But that has ramifications, and it robs some of the stakeholders. When you make things to the minimums, it causes problems for a majority of the stakeholders. While it may meet the requirements of the industrial design stakeholder, because now it fits into their package and all the front-end parameters are met, the people who have to deal with rework and repair and redesign can be affected adversely. So, you can't discount the advantage of including the plus; adding a little extra space in there is good.

Holden: Well, I hear you. My soapbox is particularly with engineering. Can we put this into a model? Before you ever start, you would do the practical packaging density plus, plug in the numbers and you say, "Uh-oh, I've got a problem, because it's larger than the available area they gave me," rather than putting the design out and realizing that you have to start all over. Predictive engineering lets us predict the bottlenecks before we fall into the bottlenecks.

Dack: That sounds very interesting. It's like right-sizing, and it sounds good, but it's subjective. An effort like this, to come up with a formula or analysis, would require buy-in from all the stakeholders. What is the target condition of each individual stakeholder? How do we compromise? What's the best way to compromise to right-size the product design? We teach in the IPC CID program that this critical buy-in opportunity should be facilitated for all project stakeholders at the start of a project so we can effectively "measure twice and cut once."

Matties: But that may be what you have to do. You don't want to go up to the minimum just because you can, especially with this supply chain.

Shaughnessy: Well, Kelly, it's been great, as usual. Thanks for your time.

Dack: Thank you guys. I'm looking forward to seeing you all at trade shows. DESIGN007

Some Relief, But Hold Off on the Party

Q&A with Chintan Sutaria

To help PCB designers and design engineers get a clearer picture of the stress points in the industry, particularly from a company that deals directly with EMS providers, we reached out to CalcuQuote CEO Chintan Sutaria with a list of questions. The following Q&A explores trends in the PCB supply chain and heady advice for dealing with long lead times and counterfeit parts.

Q What trends are you seeing in the PCB supply chain now? There seems to be some relief in the last month or so, but is a celebration premature?

A The supply and demand imbalance is naturally evolving to correct itself, but I would not start celebrating yet. We still see a non-stock ratio of X% on customers' BOMs, which is well below normal levels. With each non-stock component representing an incomplete assembly, this continues to be a top concern for the electronics supply chain. But there is still some relief happening. Part of that is coming from customers who have better adapted to the new normal by identifying and evaluating the viability of alternatives and developing resilience into their BOMs. There is also some optimism from the industry at large with a renewed interest in investing in domestic manufacturing capabilities. That will surely help in the long run and is a reason for celebration, but it does not address the immediate needs of manufacturers who want to ship finished assemblies to their customers. While celebration may be premature, there is still an

Chintan Sutaria

exciting opportunity as the industry moves toward more innovative ways to solve these challenges.

Q We've heard of companies expanding their approved vendor lists (AVLs) or lowering the requirements for getting onto the list. Are you hearing stories like this?

A Having multisourcing options on a BOM is a best practice that is showing its value during this time. It is natural to take shortcuts on a BOM because of the rapid design cycles of products, but when the supply chain undergoes a disruption, it has a disproportionate impact on those who were unprepared. One exciting opportunity that has come out of this supply chain disruption has been innovation through digitalization. With the focus on digital connectedness, more data is at our fingertips, and this allows everyone to have more information around AVLs, ultimately making the approval process easier. This isn't a cure-all because there are still cases where a part cannot be crossed based on the necessarily unique specs it has.

Q What advice would you give PCB designers and engineers who are trying to source components that have lead times of 40 or 50 weeks?

A First, although many parts have long lead times, it does not mean that the lead time should be taken on face value. Designers should work with their sourcing teams to continue checking on stock availability because it is possible that the component may turn up. We recently released WatchCQ, which allows customers to create a watchlist to be alerted when a part becomes available. Second, consider altering your requirements. See if other AVLs will work or if you can rev your product to use a different component. This may mean making tough choices about the features of your product. Finally, change your expectations. It used to be normal to expect that a brilliant design could be easily sourced and built within weeks. The current supply chain climate has made this very challenging. It is important to manage your own expectations and plan accordingly for longer product cycles and maybe fewer iterations.

Q Some OEMs are seeking components from alternative suppliers, but they're worried about risky and counterfeit parts. What kind of due diligence should they undertake?

A Due diligence should be proportional to the use case of the product. If you are building a low-cost prototype for internal testing in a non-critical application, then take the chance and continue your development. If your product needs to be high reliability or has the potential to cause significant damages, then you should exercise more caution. It is always preferable to go through sources you trust: suppliers you have worked with before and have stood by their own quality processes.

Digital tools continue to create a more transparent and connected supply chain. This allows more visibility into the inventory that exists in the supply chain, better planning, and less avenues for illegitimate products to enter. Hopefully the need for sourcing from potentially questionable sources will continue reducing.

Q Do you think there will be any "lessons learned" that we can benefit from after this whole supply chain debacle is over?

A The frequency, duration, and magnitude of supply chain "debacles" seems to be growing. Building resilience into the supply chain will be important to addressing the impact these disruptions have. But building agility is just as important. Successful supply chain organizations will learn to respond earlier and with more agility to micro and macro events. This also shows the need for digitalization and innovation. Through better technology and data sharing, we could have done a better job of predicting the shortages and responding to them. Even with a limited view into supply chain transactions, CalcuQuote's data analysts were able to foresee problems fairly early on. The CHIPS Act is an example of a lesson learned and applied. While it won't produce instant results, or soften the pains of today's market, it will help to avoid major concerns in future market shifts.

Q Any final thoughts on this topic?

A Although the last couple of years have been challenging, they have accelerated a wave of innovation in the electronics supply chain. It is my hope that the next few years will maintain this pace of innovation (without the hardships of having to chase after parts). **DESIGN007**

NASA Engineer Develops Tiny, High-Powered Laser to Find Water on the Moon

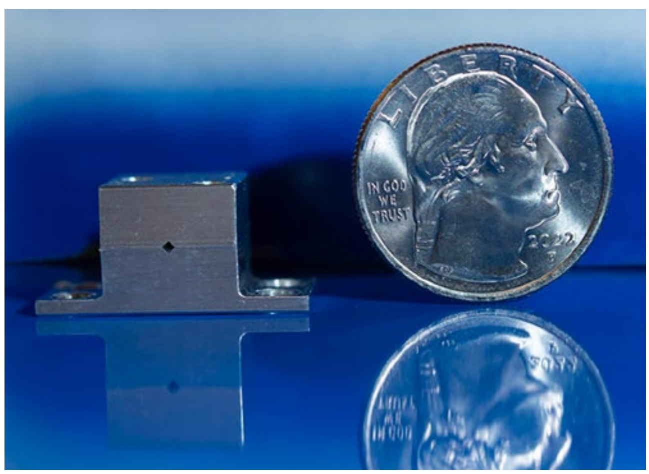
Finding water on the moon could be easier with a Goddard technology that uses quantum tunneling to generate a high-powered terahertz laser, filling a gap in existing laser technology.

Goddard engineer Dr. Berhanu Bulcha said a heterodyne spectrometer could zoom in on particular frequencies to identify water sources on the Moon. It would need a stable, high-powered, terahertz laser. Dr. Bulcha's team is developing quantum cascade lasers that produce photons from each electron transition event by taking advantage of some unique quantum-scale physics of materials layered just a few atoms thick.

In these materials, a laser emits photons in a specific frequency determined by the thickness of alternating layers of semiconductors rather than the elements in the material. In quantum physics, the thin layers increase the chance that a photon can then tunnel through to the next layer instead of bouncing off the barrier. Once there, it excites additional photons. Using a generator material with 80 to 100 layers, totaling less than 10 to 15 microns thick, the team's source creates a cascade of terahertz-energy photons.

The integrated laser and waveguide unit reduces dissipation by 50% in a package smaller than a quarter. Bulcha hopes to continue the work to make a flight-ready laser for NASA's Artemis program.

(Source: NASA)



Utilizing a Field Solver for Stackup Planning

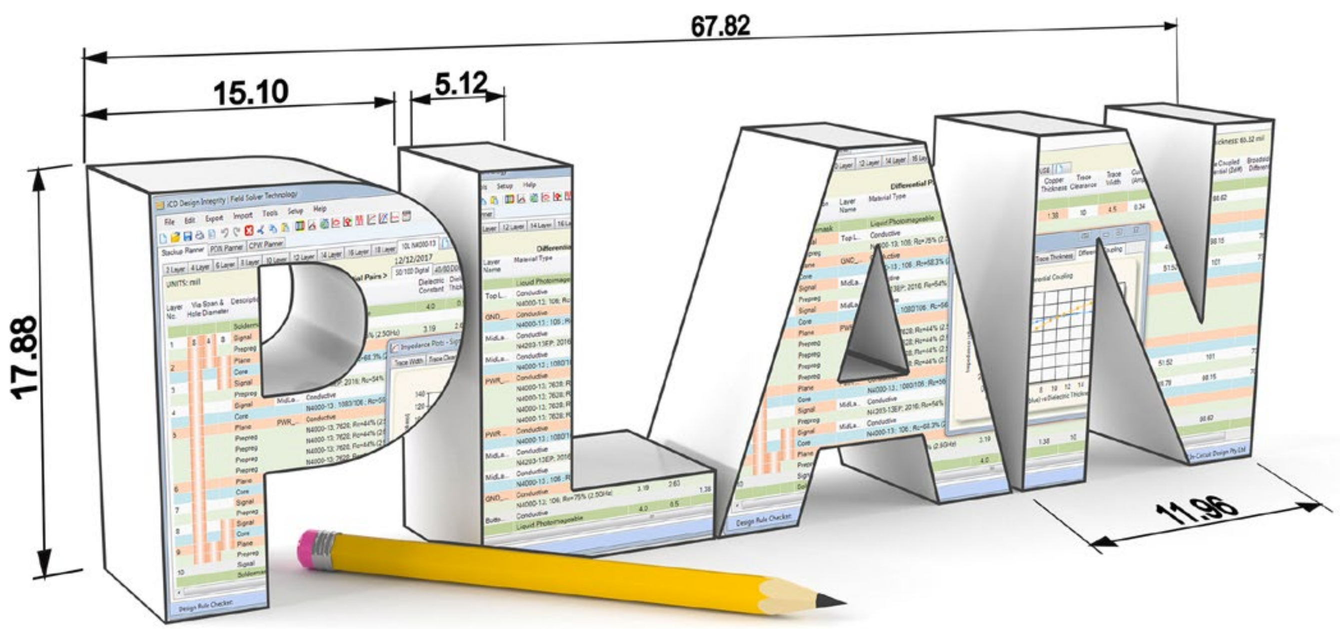
Beyond Design

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

In a previous column, I deliberated on why the 2D field solver is an essential tool for all high-speed PCB designers. But like all tools, one needs to know how best to apply its unique features to enhance your design process. Obviously, calculating transmission line impedance, in its various forms, is the prime function but field solvers can also provide additional information to ensure good design practice long before the layout begins.

Many PCB designers believe that it is the fabrication shop's role to calculate all the required impedances at the end of the process to best suit the materials that they stock. But by then, it's too late; the horse has already bolted. It is difficult and time-consuming to go back and change the constraints once the board is complete.

So, where do we start to build the perfect stackup for our project? Initially, virtual materials are used to get the rough numbers. Then the impedance is fine-tuned once the actual materials are substituted. Every digital board will require 50-ohm impedance and generally a 100-ohm differential pair. This is our target impedance. However, multiple technologies are often used on complex designs which need to be accommodated on the same layer.



Keep these tips in mind when planning the board stackup:

- All signal layers should be adjacent to and closely coupled to an uninterrupted reference plane, which creates a clear return path and eliminates broadside crosstalk.
- There is good planar capacitance to reduce AC impedance at high frequencies. Closely coupled planes reduce AC impedance at the top end and dramatically reduce electromagnetic radiation.
- High-speed signals should be routed between the planes to reduce radiation.
- Reducing the dielectric height will result in a large reduction in your crosstalk without having a negative impact on available space on your board.
- The substrate should accommodate several different technologies. For example, 50/100 ohm digital, 40/80 ohm DDR3/4 and 90 ohm USB.

Unfortunately, not all these rules can be accommodated on a four- or six-layer board simply because we have to use a buffer core in the center to realize the total board thickness of 62 mils. However, as the layer count increases, these rules become more critical and should be adhered to.

Given the luxury of more layers:

- Electromagnetic compliance (EMC) can be improved or more routing layers can be added.
- Power and ground planes can be closely coupled to add planar capacitance, which is essential for GHz plus design.
- The power distribution networks (PDNs) can be improved by substituting embedded capacitance material (ECM) for the planes.
- Multiple power planes/pours can be defined to accommodate the high number of supplies required by today's processors and FPGAs.
- Multiple ground planes can be inserted to reduce the plane impedance and loop area.

Although power planes can be used as reference planes, the ground is more effective, as local stitching vias can be used for the return current transitions rather than stitching decoupling capacitors, which add inductance. This keeps the loop area small and reduces radiation. As the stackup layer count increases, so does the number of possible combinations of the structure. But if one sticks to the basic rules then the best performing configurations are obvious.

So, the big question everyone asks is about determining layer count. A good place to start is looking at a reference design with similar characteristics and then adding two layers. I say to add two additional layers because reference designs are typically squeezed onto as few layers as possible. But if one wants to avoid routing signals on the outer microstrip layers and reduce signal propagation skew and electromagnetic emissions, the extra layers will be needed.

Experienced PCB designers get a feel for it after a while, but a good way to check whether you have enough layers is to autoroute the board. With no tweaking, the autorouter needs to complete at least 85% of the routes to indicate the selected stackup is routable. The performance of the autorouter also impacts the completion rate. You may have to re-evaluate the placement a couple of times to get the best results. In general, eight layers is a good starting point for DDR type designs. Remember, it is much easier to increase the number of layers than to reduce them, so start with the minimum.

A field solver can be used to determine the unknown variables for an established impedance goal (Figure 1). Impedance plots use multiple passes of the field solver to plot the curve of impedance vs. dielectric thickness. In this case, a 3-mil prepreg is required to achieve

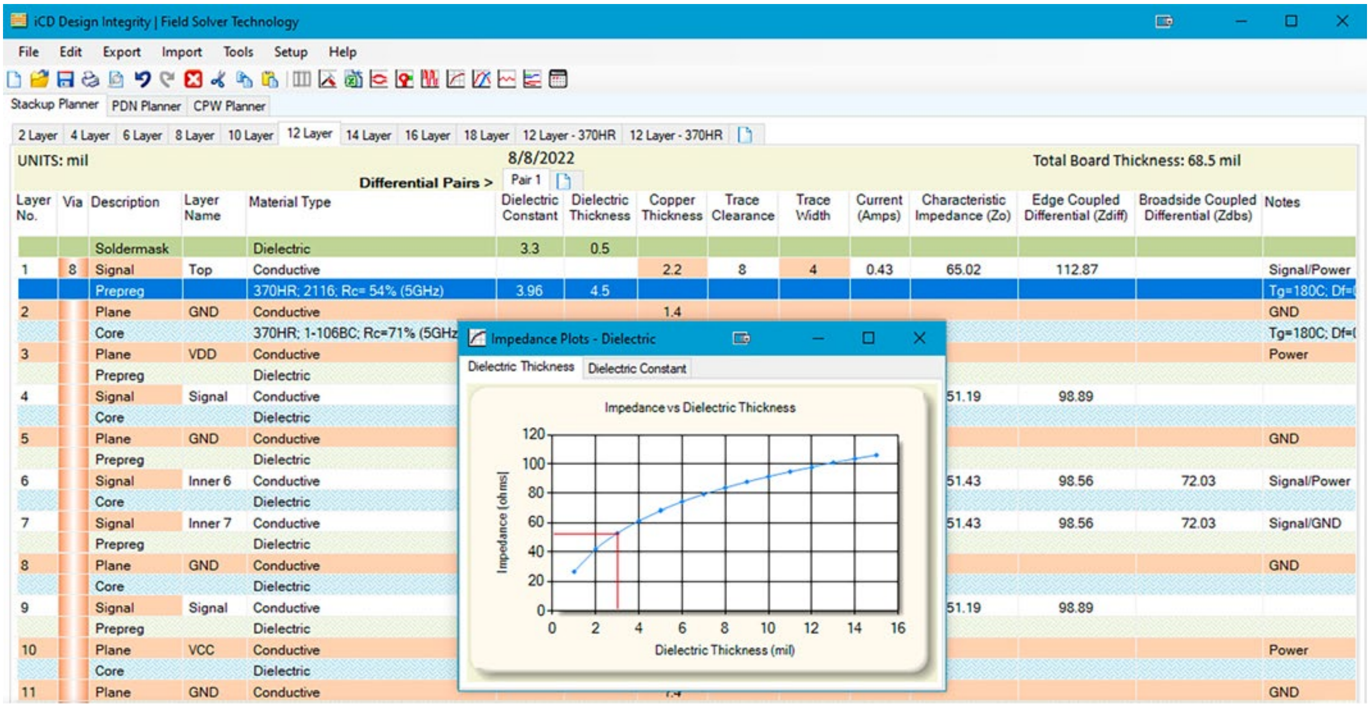


Figure 1: A field solver can be used to determine the required dielectric thickness (simulated in the iCD Stackup Planner).

50 ohms impedance. Dielectric constant, trace thickness, width and clearance can also be established in this way.

For DDR type designs, I like to start with closely coupled plane pairs on the second and third layers from the top and bottom (Figure 2). Unfortunately, standard decoupling capacitors have little effect over 1 GHz and the only way to reduce the impedance of the PDN above this frequency is to use tightly coupled power/ground plane pairs or, alternatively, on-die capacitance. These plane pairs should be positioned

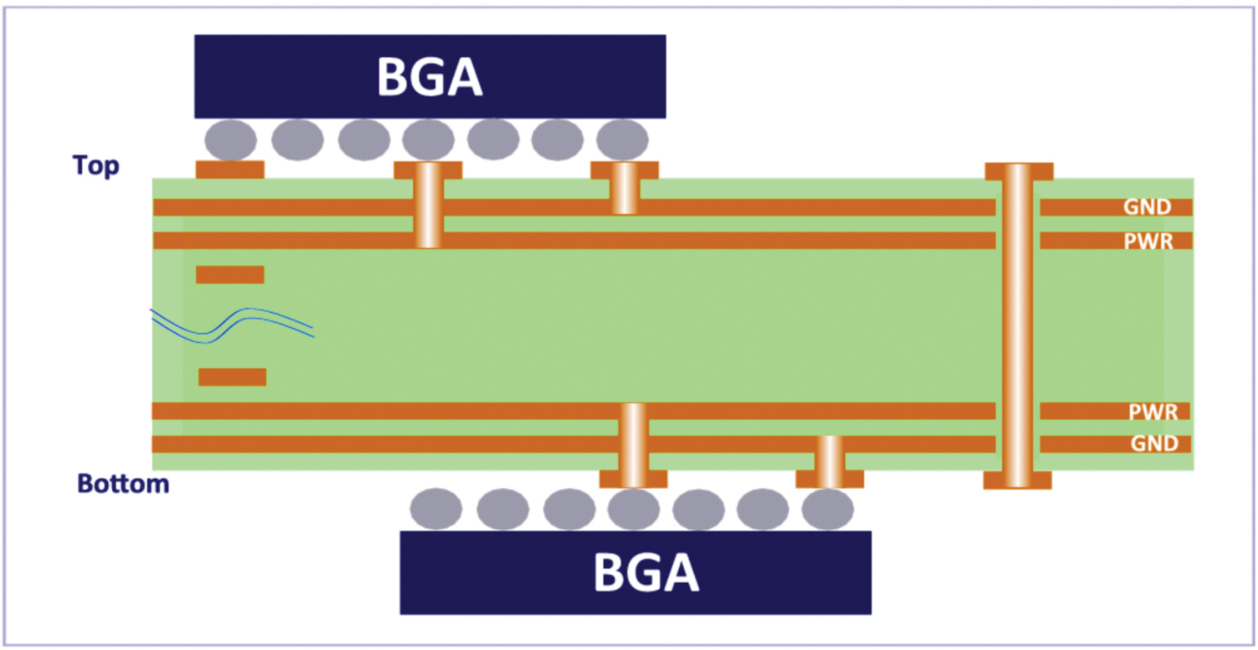


Figure 2: Tightly coupled plane pairs reduce loop inductance.

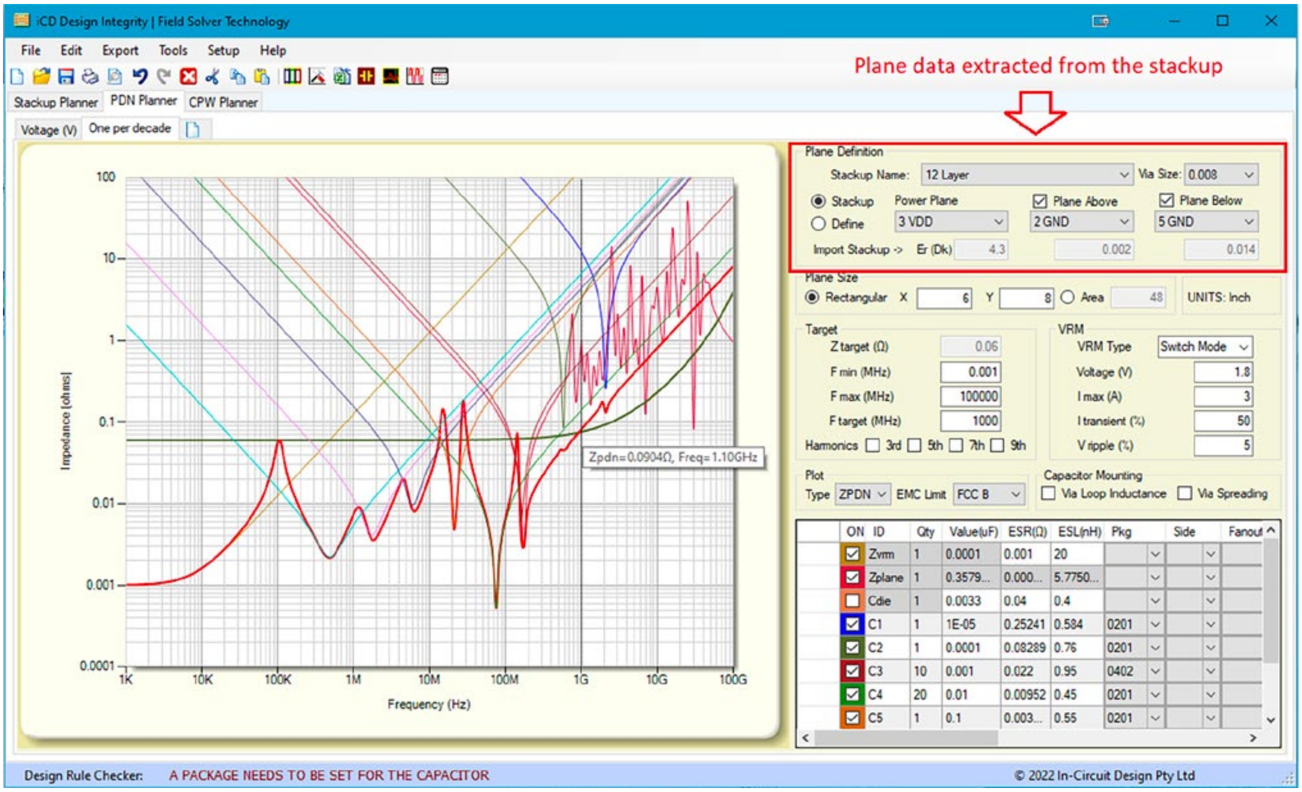


Figure 3: Plane pair data and via size is extracted from the stackup to the PDN (iCD Design Integrity).

in the stackup close to and directly below/above the IC. This reduces the loop inductance dramatically (Figure 3).

The ability to extract the plane data and via definition from the stackup to a power distribution network (PDN) planner is also an advantage as it allows one to see the plane resonance and the impact it has on the AC impedance of the PDN (Figure 3). The AC impedance of the PDN should be kept below the target impedance up to the fifth harmonics of the fundamental. Plane resonance can be dampened further by adjustments to the dielectric constant and thickness as well as plane size to avoid anti-resonance peaks.

Setting up the via span is also important at the stackup level (Figure 4). This gives you a clear definition of all the plated through-hole and blind and buried vias used in the design, and can be conveyed to the fabricator.

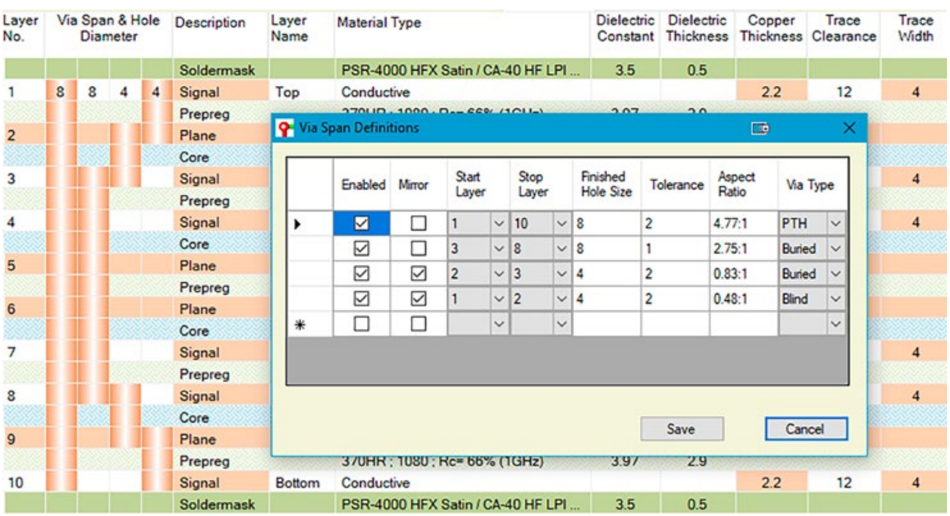


Figure 4: Via span definition is defined in the stackup (Source: iCD Stackup Planner).

If one has no choice but to route critical signals on the outer microstrip layers then there will be a large discrepancy in timing due to the variance in flight time through different materials of the outer microstrip and inner stripline layers.

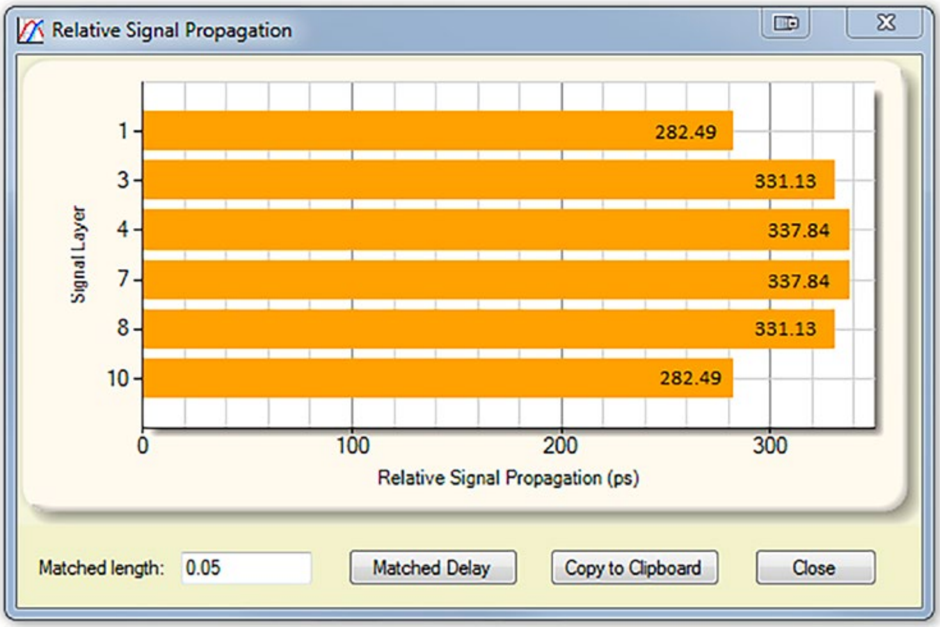


Figure 5: Relative signal propagation of microstrip and stripline, simulated in iCD Design Integrity.

The electric fields surrounding the microstrip exist partially within the dielectric material(s) and partially within the surrounding air. Since air has a dielectric constant of one, which is always lower than that of FR-4 (typically 4.3), mixing a little air into the equation will speed up the signal propagation. Even if the trace widths are adjusted on each layer, so that the impedance is identical, the propagation speed of microstrip is always faster than stripline, typically 13–17% (Figure 5). The speed of propagation of digital signals is independent of trace geometry and impedance.

If you are aware of this issue, then the trace delays can be matched to compensate for the varying flight time, so that at the nominal temperature, all signals running on either microstrip or stripline will arrive at the receiver simultaneously.

Unfortunately, drivers do not have the same impedance as the transmission line (typically 10–35 ohms) so terminations are used to balance the impedance, match the line, and minimize reflections. Reflections occur whenever the impedance of the transmission line changes along its length. This can be caused by unmatched drivers/loads, layer transitions, different dielectric materials, stubs, vias, connectors and IC packages. By understanding the causes of these reflections and eliminating the source of the mismatch, a design can be engineered with reliable performance.

As shown in Figure 6, using a 12 mA LVCMOS 1.8V driver of a Spartan 6 FPGA, an 18.7-ohm series resistor is required to match the driver to the 51.67 ohm trace on the outer layer. This is automatically derived from the IV curves of the Spartan 6, IBIS model by the iCD Termination Planner (Figure 6). Correct termination is extremely important as it reduces transmission line reflections that can cause multiple issues for the design integrity.

So, apart from the accurate determination of single-ended, differential and broadside couple impedance, a field solver can offer further insight to enhance the signal and power integrity of a design.

Key Points

- Although power planes can be used as reference planes, the ground is more effective.
- To calculate the layer count, look at a reference design with similar characteristics and then add two layers.
- With no tweaking, the autorouter needs to complete at least 85% of the routes to indicate the selected stackup is routable.
- It is much easier to increase the number of layers than to reduce them.
- A field solver can be used to determine the unknown variables for an established impedance goal.

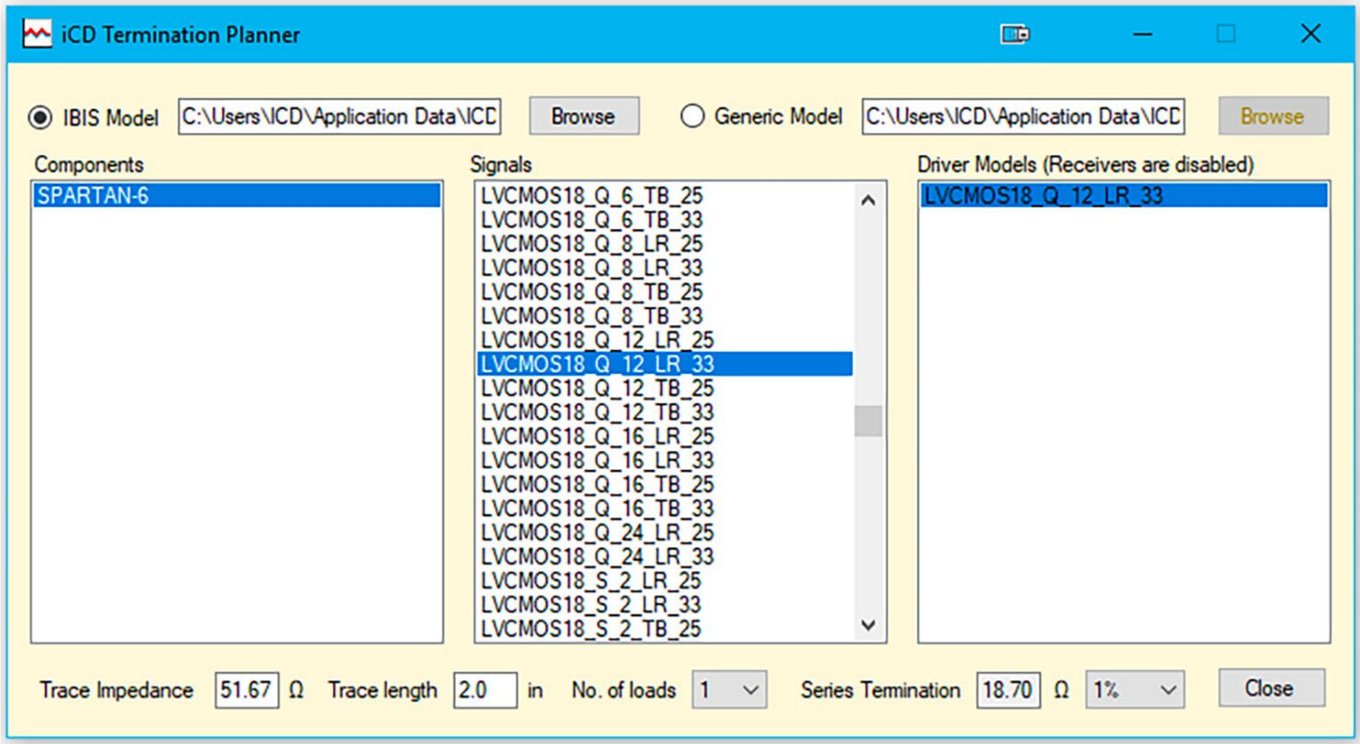


Figure 6: Matching the Spartan 6 driver to the transmission line. (Source: iCD Termination Planner)

- For DDR type designs, start with closely coupled plane pairs on the second and third layers from the top and bottom. These plane pairs should be positioned in the stackup close to and directly below/above the IC.
- The ability to extract the plane pair data and via size from the stackup to a PDN planner allows one to see the plane resonance and the impact it has on the AC impedance of the PDN.
- The AC impedance of the PDN should be kept below the target impedance up to the fifth harmonics of the fundamental.
- Plane resonance can be dampened by adjustments to the dielectric constant and thickness as well as plane size to avoid anti-resonance peaks.
- If critical signals are routed on the outer microstrip layers, then there will be a large discrepancy in timing due to the variance in flight time through different materials of the outer microstrip and inner stripline layers.
- The propagation speed of microstrip is always faster than stripline—typically by 13-17%.
- The speed of propagation of digital signals is independent of trace geometry and impedance.
- Drivers do not have the same impedance as the transmission line, so terminations are used to balance the impedance. **DESIGN007**

Resources

1. Beyond Design by Barry Olney: Signal Flight Time Variance in Multilayer PCBs; Stackup Planning Parts 1-6; The Fundamental Rules of High-Speed PCB Design Part 2; The Impact of PDN Impedance on EMI.

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

MilAero007 Highlights

IPC Lauds Passage of 'CHIPS and Science' Act; Electronics Industry Calls for a Holistic Approach to Reviving Domestic Electronics Capabilities ▶

IPC's John Mitchell issues a statement to comment on President Biden's signature on the "CHIPS and Science Act" in Washington, D.C.

Ventec Expands Flex-rigid Material Range for Critical Military, Aerospace, and Ultra-high Reliability Applications ▶

Ventec International Group Co., Ltd. has added to its flex-rigid No Flow/Low Flow prepreg range with the introduction of tec-speed 4.0 (VT-462(L) PP NF/LF), a next-generation no & low flow FR 4.0 prepreg material that offers high-Tg, low Dk, low loss, and excellent thermal reliability.

IPC Advanced Packaging Symposium to Draw Industry, Government Representatives ▶

Nolan Johnson speaks with IPC's Chris Mitchell, vice president of global government relations, and Matt Kelly, chief technologist, about the inaugural IPC Advanced Packaging Symposium, scheduled for October 11-12, 2022, in Washington, D.C.

Space Perspective Unveils Patented Capsule Design ▶

Space Perspective, Planet Earth's leading luxury space travel company, unveils the patent-pending Spaceship Neptune capsule design now in production at the company's state-of-the-art campus, near its Operations Center at NASA's Kennedy Space Center, Florida.

Cicor Awarded with a Major Multi-year Aerospace and Defence Business ▶

Cicor Group has been awarded with a major multi-year follow-up business for a total value of around CHF 30 million by a market-leading European manufacturer of specialized aircraft solutions.

Eltek Receives $1.7M Purchase Order from an Existing Defense Customer ▶

Eltek Ltd., a global manufacturer and supplier of technologically advanced solutions in the field of printed circuit boards, announced that the company has received a purchase order in the amount of $1.7 million from an existing customer in the defense sector.

REGENT, Siemens Collaborate for Revolutionary Zero-emission Seaglider ▶

Siemens Digital Industries Software announced that REGENT has adopted the Siemens Xcelerator portfolio of cloud-based software and services to help pioneer a new category of vehicle called the seaglider.

Sparton Received Vendor Status for U.S. Navy Sonobuoy Orders ▶

Sparton Corporation, an Elbit Systems of America company, announced that it had received a vendor status under the United States Navy's sonobuoy Multiple Award Delivery Order Contract (MADOC). With MADOC vendor status, Sparton qualifies to compete for annual delivery orders for each sonobuoy variant that may be needed by the service.

Examining the Benefits of Laser Direct Imaging

Connect the Dots

by Trina Taylor, SUNSTONE CIRCUITS

One of the most amazing advances in PCB manufacturing technology has been the advent and usage of laser direct imaging (LDI) technology. For PCB manufacturers, this technology has been a game changer, helping to reduce costs, speed production, and improve quality.

Though the LDI revolution began more than 20 years ago and usage of film for image transfer has reduced by half in that time, there's still room for more PCB manufacturers to invest in this powerful tool. As an image department team leader and ISO process owner, I have seen firsthand the benefits of laser direct imaging for both my team and our customers.

Imaging is a key production step where the PCB design is transferred from its original digital format onto the physical manufacturing panel. This process ultimately defines all the copper features on a given layer, everything from pads to traces and plane areas.

As the name implies, LDI transfers the digital design data directly to the manufacturing panel without the use of an intermediary. The result is sharper and more precise features, with better registration and the ability to place smaller features more effectively.

We make higher quality, more reproducible boards with LDI. The higher resolution transfer makes the final board closer to the digital design. Figure 1 compares the LDI imaging process with that of a traditional approach. As we can see, the LDI process uses a photo-based method to transfer the image

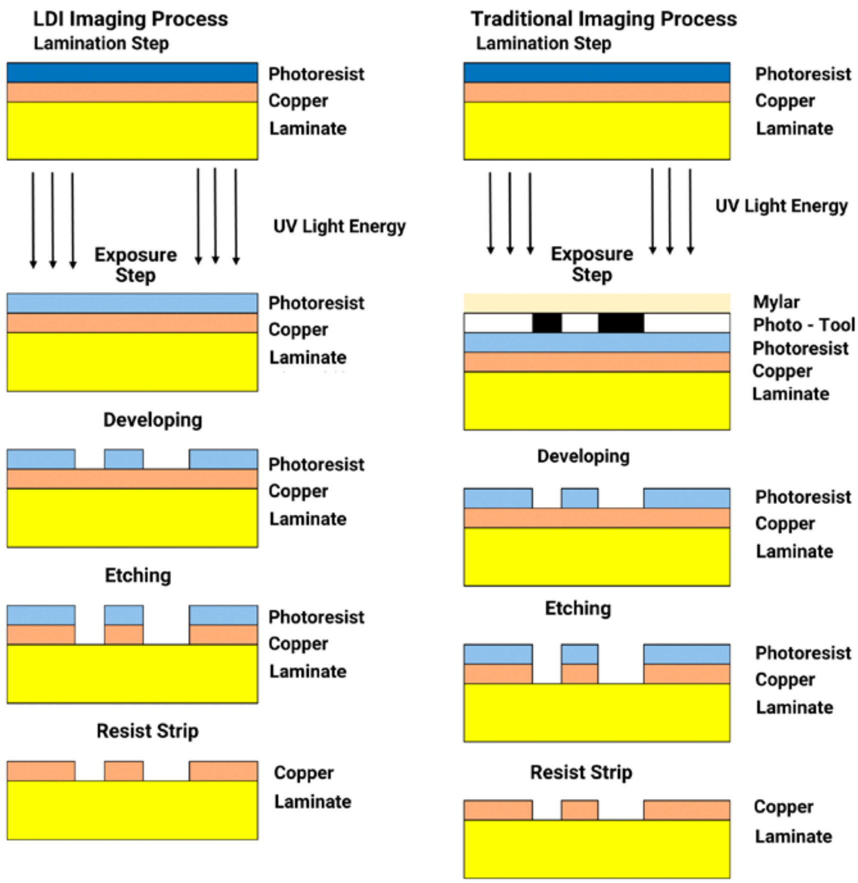


Figure 1: Unlike the traditional imaging process, the LDI imaging process, does not require use of a photo tool as an intermediary during the exposure step.

to the panel much like the traditional process. The LDI imaging process does not require a phototool as an intermediary during the exposure step.

Eliminating the phototool and mylar or glass layer from the exposure process allows the light source to expose the photoresist more directly. This reduces interference and limits opportunity for the creation of air gaps in either of those two extra layers. By eliminating the mylar layer the quality of the image process improves. The thickness of the clear mylar layer being present does two things: scatters the light source as it transfers between multiple types of layers, and it increases the effective distance between the light source and its ultimate destination—the photoresist. Eliminating this layer provides a more direct path from the light to create a sharp and defined image.

Air gaps cause issues with the quality of the exposure and can cause defects by allowing exposure in areas where it is unwanted. LDI eliminates the part of the process that can cause air gaps, generating a sharper and truer representation of the data being transferred.

In traditional photo-based methods, the alignment of the image with the panel can be a very tedious, and in some cases, impossible task. The LDI process better aligns the image with the panel than with traditional methods. The LDI process uses aligning targets called fiducials to calculate any movement that has occurred in the panel and correct for it. We can also calculate and adjust for any stretch, shrinkage, or skew that has occurred during the previous manufacturing steps. The copper image is, for practical purposes, perfectly aligned with the drilled holes.

The LDI imaging process makes my team better at our jobs. We have realized improvement in numerous areas because we adopted this method for imaging. Most importantly, we have improved the quality of our output and that reverberates throughout the PCB manufacturing process. The result is better boards.

The boards are not just higher resolution, they take less time to produce. We don't spend

Trina Taylor and Matt Stevenson reviewing a project at Sunstone.

time plotting, cleaning, and punching image films. What was once a two-hour process now takes 30 minutes, and we're not going through 150 films a day, which results in cost savings.

LDI eliminates much of the error potential present using traditional methods. We're not handling film, moving it from place to place and potentially damaging it, so there is less scrap. Inspection and quality assurance take less time because there are fewer defects to find.

Since less production time is spent on imaging, other production teams are less apt to be rushed and they have more time to focus on quality and process improvement. This has made LDI a critical component of our continuous improvement efforts and Lean manufacturing adoption.

Improving just one of the hundreds of processes in our PCB manufacturing facility pays dividends. Yes, my team can produce better results more efficiently, and this is great for our customers. The process improvements created by LDI also offer everyone on the team more opportunities for professional development. With less time spent on tedious traditional imaging efforts, we can focus on learning new skills that can lead to career advancement.

Technology advancements like the ones created by LDI have opened doors and helped me grow professionally. I started as a part-time chip coordinator and now I'm the image department team leader. When measuring the value of tools like LDI, we tend to focus on improving the quality of the boards produced and increasing customer satisfaction.

These tools also improve the quality of our working lives and increase satisfaction with our professional development. **DESIGN007**

Trina Taylor is team lead, image department, at Sunstone. To read past Connect the Dots columns, click here.

Tech Employment Increases as Companies Keep Up Pace of Hiring

Tech sector companies added 25,500 net new workers in August, with growth in five major occupation categories, CompTIA's analysis of the U.S. Bureau of Labor Statistics (BLS) "Employment Situation" report reveals. Companies throughout the economy added an estimated 21,000 tech workers for the month. The unemployment rate for tech occupations edged up to 2.3% paralleling the directional change of the national unemployment rate (3.7%).

"Stability in tech hiring continues to be an overarching theme this year," said Tim Herbert, chief research officer at CompTIA. "Despite all the economic noise and pockets of layoffs, aggregate tech hiring remains consistently positive."

Employer job postings for tech positions eased back in August, coming in at just under 320,000. Among industries, the largest numbers of job postings occurred in professional, scientific and technical services, finance and insurance, manufacturing, information and retail trade. Employment opportunities in tech are available across the country, in markets large, medium and small. The New York City, Washington, Dallas, Chicago and Los Angeles metro areas had the most job postings for tech positions, while Allentown, PA, Raleigh, NC, and Columbus, OH recorded the largest month-over-month increases in tech job postings.

(Source: PRNewswire)

Altium's EDDI Report Tracks Component Availability—Today and Historically

Feature Interview by Andy Shaughnessy
I-CONNECT007

The following interview originally appeared in the September 2022 issue of SMT007 Magazine.

There's one lesson that all designers have learned over the past few years: Components might be here today and gone tomorrow, so tracking your parts is more important than ever. Any resources that help you keep tabs on your required parts are invaluable in these days of 40-week lead times.

Earlier this year, Altium released one such resource: the Electronic Design to Delivery Index (EDDI) report. Assembled from millions of bytes of data gleaned from the Octopart search engineer and the Nexar platform, the monthly EDDI report provides part availability histories going back years, as well as a real-time snapshot of global inventories. It's free to download.

We asked Dan Schoenfelder, head of Nexar sales, to discuss the EDDI document and why PCB designers should take advantage of this handy report.

Andy Shaughnessy: Dan, welcome. You all have been talking about this EDDI report. I had the chance to review it, and it's pretty interesting. Would you walk me through it?

Dan Schoenfelder: Yes. Altium has a really interesting place in the market, where we have user experiences that span design, supply chain, and manufacturing workflows. Because of that, we have a lot of interesting data that we collect and which we mine to provide trends back to the electronics industry. Any stakeholder in the electronics space can benefit from this information. One of these products is the Electronic Design to Delivery Index, affectionately referred to as the EDDI.

We have two major signals: One is an industry supply signal and the other is an industry


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

demand signal. They're both intended to give stakeholders a view of where the things stand today relative to history for availability of components and how challenging it might be to source components.

Shaughnessy: It was interesting to me that it had a running historical ledger of where parts were. How far back does it go? How granular can you get? Give us some details.

Schoenfelder: The report itself, and the indices that are a part of it, are pegged to a baseline of January 2020. We intentionally did that because it's pre-pandemic and probably the closest thing that any of us can remember to what was normal. The reports that we generate monthly show two years of history compared to January 2020.

In the EDDI, you'll find that we look at an aggregate signal for both demand and supply, but then we break it down further into key categories. These nine categories include integrated circuits, passives, and discrete semiconductors, among others.

Shaughnessy: That's really good. I'm curious where this idea came from.

We had this idea to give back to the industry some of the analytics that we're able to capture.

Schoenfelder: We had this idea to give back to the industry some of the analytics that we're able to capture. Under the Altium umbrella, we have design tools, a powerful API, and Octopart, the component search engine.

All these different user experiences have user interactions with data. The EDDI takes those interactions and signals of intent that data exhaust, and aggregates and normalizes that into a product that shows trends in demand and supply relevant to stakeholders of the electronic component space.

Shaughnessy: I imagine a lot of this comes through Octopart, right?

Schoenfelder: Most definitely; portions of the EDDI are fortified by Octopart, such as inventory trends and search activity.

Shaughnessy: And Nexar also includes other search engines, so you've got a wide universe to cull this data from.

Schoenfelder: That's a good point. We like to talk about our signals as having both breadth and depth. Octopart itself sees several million unique visitors per month. The Nexar API receives about 15 million calls per week. So there's significant activity that provides us a broad but granular signal of what's happening in the industry. There's no question that market conditions have driven a lot of activity over the last 18 months or so.

Shaughnessy: And you're offering this free of charge?

Schoenfelder: Yes, the EDDI itself is a free report. You can subscribe at Nexar.com. It comes out monthly, usually around the middle of the month. We also have about 300 subcategories that go beyond what's in the free report. So, if users would like to see something that's not represented in the reports, we offer some bespoke reporting.

Shaughnessy: Now, you look at this information all the time. What trends do you see?

Schoenfelder: There are some distinct trends that you can see in the latest editions of the EDDI. Component availability is returning to normal for many product categories. We're


Hmm, what is the recommended **minimum solder mask** width to be able to get a solder mask bridge **between two copper pads?**

seeing that across all passives, and we're seeing some semiconductor products that are returning to normal as well.

But we also see some that are not. Microcontrollers is a great example of a subcategory that continues to have low availability in the market, with extremely long lead times and extremely high demand as a result. What we glean from this is that we're not in the same market conditions that we were in, say, nine months ago, but there are definitely categories that are normal with others showing signs of recovery.


Dan Schoenfelder

Shaughnessy: Right. What we're hearing is that by Christmas, we may not be back to 100%, but the supply chain should be in much better shape by the end of the year.

Schoenfelder: Yes, I think the trend is that most categories that are not in the semiconductor space are operating at something that's close to normal lead time, and normal demand and supply. But semiconductors are just lagging now.

Shaughnessy: So, the EDDI is free to download, and you just have to register and provide your email, which seems like a fair trade.

Schoenfelder: Yes. I don't think I'm talking out of school here to say that sometime soon we will launch this as a self-service tool on our portal. We wanted to get to market quickly with this reporting because we think it's really valuable. For now, users can register with an email, and we will send you a PDF on a monthly basis, but in the future we'll make it self-service so users can access EDDI directly from our site.

Shaughnessy: Well, I signed up for it, and it has a lot of timely information. Even though we're seeing improvements, the supply chain is still a little wacky right now.

Schoenfelder: It's interesting, Andy, that things have become so challenging in some aspects of managing the supply chain that individuals who may not have had to pay attention to these market conditions, such as engineers and designers, now need to take these considerations into their day-to-day work. The EDDI is free and accessible industry insights, with very low barriers, and it can be influential in a designer's decision-making.

Shaughnessy: A designer recently told me, "We probably should have been double- and triple-sourcing components all along, but we never had to." That's one trick that's not going away, right?

Schoenfelder: Yeah, every company should have an alternate source strategy. Having your finger on the pulse of availability in the market, particularly relative to your demand requirements, is critical. I think the EDDI is a tool that can help assess where there is opportunity and where there is risk present.

Shaughnessy: Very good. Is there anything else you want to add, Dan?

Schoenfelder: That's it for now. I appreciate the time, Andy.

Shaughnessy: Thanks for taking the time to talk to us, Dan. DESIGN007

PCB007 Highlights

PCB Finance Class With Jeff De Serrano ▶

Class is in session! PCB Technologies President Jeff De Serrano takes us through some of the struggles of the PCB industry over the past 30 years, how U.S. legislation seeks to level the playing field, and whether a "monopoly" of fabricators is healthy for our industry. In his chat with Nolan Johnson, Jeff gets out his investment playbook and talks strategy. Don't miss this one.

IPC's I-Connect007 Acquisition Update With John Mitchell ▶

Editor Nolan Johnson speaks with I-Connect007 Publisher Barry Matties and IPC President and CEO John Mitchell about IPC's acquisition of the publishing company, and what this means to I-Connect007's readers.

The New Chapter: My Time on the IPC Board of Directors—Standing on the Shoulders of Giants ▶

At Joe O'Neil's Hall of Fame ceremony in January, he talked about his first IPC APEX EXPO. He said he felt he was sitting at a table with the "giants of industry." That analogy perfectly describes how I felt during my tenure on IPC's Board of Directors.

Punching Out: Concerns About ESG Issues in PCB M&A ▶

Recently, ESG issues have become more prominent in business as well as in M&A deals. The PCB industry has dealt with environmental issues before, so it's not really anything new. Water and waste treatment regulations have been in place for many years and the standards continue to increase.

Q4 Concerns: Hold on to Your Hats ▶

IPC Chief Economist Shawn DuBravac has plenty to share about the state of the U.S. economy and how the electronics manufacturing industry might weather the storms of high inflation, rising interest rates, and low unemployment.

What's Going On in Congress? Your Handy Guide to PCB Legislation Headlines ▶

We know you have so many questions about what the legislation means for you. Will there be funds to expand or upgrade my facilities? What about tax breaks? How will my specific needs be known? What will the current legislation mean for U.S. vs. China relations?

Happy's Tech Talk #10: Optical Alignment/Coupon Welding for Stackups ▶

In this month's column, I will discuss optical alignment for pinless lamination stackup, a topic that complements the induction lamination in my November 2021 column. Pin tooling plates have been used for lamination since it first started sometime in the 1960s.

Dan's Biz Bookshelf: Leading With Gratitude ▶

I like this book because gratitude is something in which I have always believed. So, it was fun for me to find a book, a very good book I might add, that said the same thing.

Schweizer Announces Preliminary Results for 1H2022 ▶

According to preliminary figures, the SCHWEIZER Group achieved consolidated sales of EUR 64.6 million in the first half of 2022 (first half of 2021: EUR 59.4 million).

Optimizing Co-design Across Multiple Domains

Digital Transformation

by David Wiens, SIEMENS EDA

This series of columns has explored multiple approaches to achieving a digital transformation within the electronics design process. This time I'd like to look at design and collaboration across multiple domains (e.g., electronics, electrical, and mechanical) and how a digital thread enables that.

Higher system speeds have necessitated consideration of signal propagation delay and quality not just within electronics, but through wire harnesses. Tighter form factors have minimized the typical "board in a box," where a simple rectangular board had enough clearance that there were rarely problems. Rigid-flex circuits often stress cross-domain ECAD/MCAD design with their multiple stackups and bendability. Even if form/fit is achieved, high-performance systems with significant heat dissipation need to ensure adequate air/fluid flow through the structure.

Emerging technologies like additive manufacturing will drive further alignment between domains; for example, there is the potential to print interconnects in automotive body panels instead of using cabling and wire harnesses or electronics molded to the contours of instrumentation panels.

The worlds of the electronics, electrical, and mechanical domains are similar and unique at the same time. Electronics design (hardware systems) and electrical design (the wiring between hardware to create larger electronic systems) naturally work together and have often been confused since they're both called "ECAD," and both employ electrical engineers who start a design with a schematic. Both

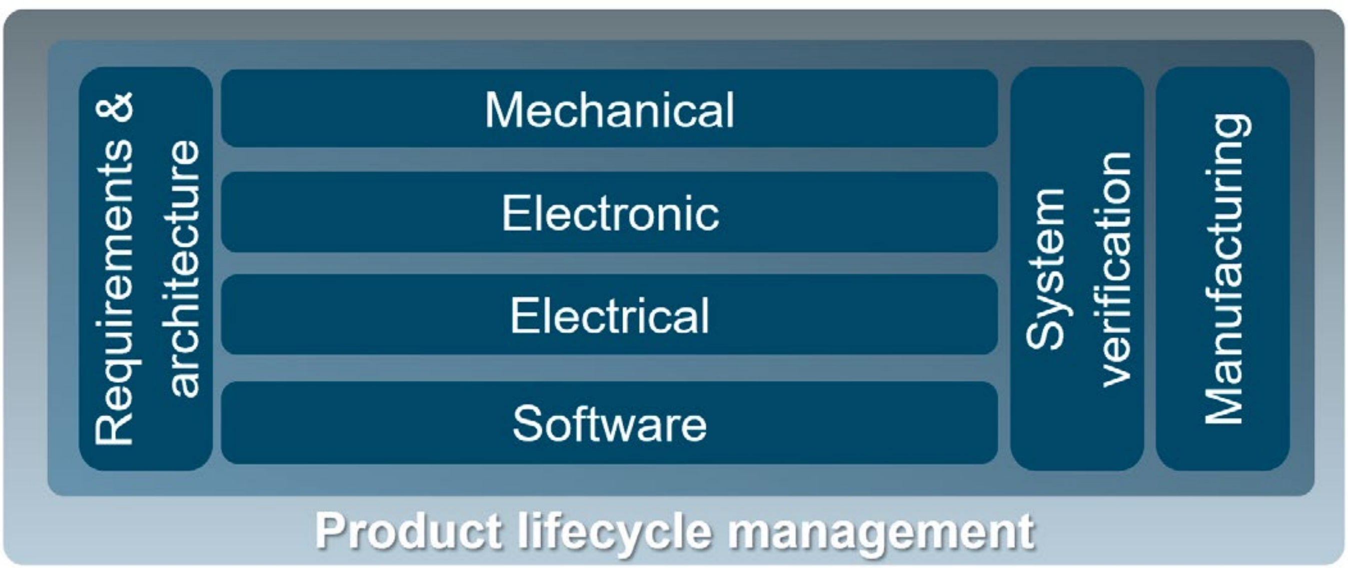


Figure 1: Multi-domain design flow.

electrical and electronics design share common challenges when integrating with mechanical design.

Historically, cross-domain collaboration was inhibited by a number of issues. The different domains had completely different tool chains, user specialties, languages/terms for communication, and databases. This made it difficult to communicate changes while both domains proceeded in parallel. The best that could be expected was email, drawings on post-it notes, or "voice-driven mouse" (assuming they're in the same facility). Design complexity has also driven use of multi-discipline simulations of the digital twin (e.g., signal/power integrity, electromagnetics, thermal, structural, and acoustics) to minimize iterations and optimize products, but data transition into those tools is inhibited by the same problems. The net result was infrequent communication, with limited or no digital verification. It wasn't unheard of to wait until system integration to realize things didn't fit. Aberdeen studies have shown that 59% of complex products will require at least two additional design iterations to address electro-mechanical problems. Sixty-eight percent of companies cite electro-mechanical data synchronization as a significant challenge.

The industry has evolved, providing a digital thread in the form of industry standard formats. Decades ago, DXFs became a common way to pass graphical data, but the information was very limited (often 2D, no intelligence about objects), thus requiring interpretation, so it was typically used only in a one-way path from MCAD to ECAD. STEP enabled more 3D intelligence, including enclosures. IDF was constructed for bi-directional collaboration, but it transferred the entire database without any tracking to identify changes. IDX gave us the ability to send incremental changes and traceability. To optimize collaboration even further, some tool pairs provide data integration beyond even these standard formats. This is a positive step but requires synchronization of all tool releases in the chain. Some companies have created tools that merge domains but lack sufficient depth, so these are limited to generalist use on simpler designs.

Today, most tools support multiple formats, yet adoption of the most advanced approaches is limited. Some of this is due to organizational inertia—it's difficult to change processes, particularly when custom code may have been created to peanut-butter over inefficiencies. It's even harder to change if multiple domains are involved, with their siloed teams and unique tool chains.

Yet the promise of optimized co-design across multiple domains enabled by a digital thread is significant. Studies show that best-practice processes can reduce physical prototypes and respins through consistent, iterative communication to avoid rework late in the design process. Engineering efficiency can be significantly increased, reducing development cost and time. First-pass success is much more likely. More robust designs can be created through extensive collaboration and verification.

Let's review the benefits of optimized cross-domain co-design:

Increased productivity
- Enables "what if" scenarios to avoid time-consuming design iterations
- Allows ECAD and MCAD designers to co-design in their own environments without learning new tools
- Provides more time for new projects due to fewer design iterations

Improved design robustness
- Facilitates the optimization of today's complex compact form factors
- Ensures higher quality, reliability, and performance with early verification of the digital twin
- Inherently less error prone and therefore reduces risk

Increased collaboration and efficiency
- Provides consistent, iterative communication throughout the development process
- Accelerates decision making to mutually agreed changes
- Left-shifts 3D clearance and collision checking into the ECAD domain

Achievable first-pass success
- Provides an integrated process to avoid rework due to electro-mechanical issues
- Reduces design iterations by verifying design intent throughout the development process
- Increases the probability of meeting the product launch target

So, what does a best-practice deployment look like? Here are a few recommendations:

1. Obviously, the tighter the connection between tools in each domain, the cleaner the data transition and the higher the collaboration efficiency. The industry standard for electro-mechanical integration is IDX, facilitating frequent, incremental collaboration. But as noted, even tighter integrations are possible.

2. A model-driven approach, with both ECAD and MCAD sharing the same component library, can save significant library creation time and ensure that all of engineering is on the same page.

3. 3D ECAD design tools enable layout designers to view what the mechanical engineer is working on without leaving their native environment and verify their design in that context. Likewise, PCB data down to the trace and via levels helps the mechanical engineer accurately model and simulate the board in their environment. On that point, as noted earlier, there are a number of analysis tools available to validate the digital twin before prototype. Ideally, these work directly off the ECAD or MCAD authoring database (a high-fidelity digital twin) to minimize rework. While adopting these tools sounds like extra work, it's been proven that "virtual prototyping" (analysis and verification during design) saves significant time and cost through respin reduction.

Take a look at your current cross-domain design process with an eye for inefficiencies. It may be "working" today (i.e., nobody's complaining), but a few steps can dramatically improve your engineering team's performance. DESIGN007

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

Additional content from Siemens Digital Industries Software:

- *The Electronics Industry's Guide to... The Evolving PCB NPI Process* by Mark Laing and Jeremy Schitter
- *The Printed Circuit Designer's Guide to... Stackups: The Design within the Design* by Bill Hargin
- *The Printed Circuit Assembler's Guide to... Smart Data: Using Data to Improve Manufacturing* by Sagi Reuven and Zac Elliott
- *The Printed Circuit Assembler's Guide to... Advanced Manufacturing in the Digital Age* by Oren Manor
- *The Printed Circuit Designer's Guide to... Power Integrity by Example* by Fadi Deek
- Siemens' 12-part, on-demand webinar series "*Implementing Digital Twin Best Practices From Design Through Manufacturing*"
- Rountable: RealTime with... *Siemens and Computrol: Achieving Operational Excellence in Electronics Manufacturing*
- You can also view other titles in our full I-007eBooks library.

Designing Through Supply Chain Pain

Feature Article by Chris Young
YOUNG ENGINEERING SERVICES

Engineers are accustomed to the demanding challenges of designing for miniaturization, cost reduction, cross platform compatibility, and harsh environments. What has proven to be the most painful experience of my career (and for many of my colleagues) is the sheer lack of components from which to build our designs. Development cycles—commercial, industrial, medical, avionics—have been severely impacted, from large enterprise corporations to small design/integration companies. Even as consultants, we have had a rough time. The good news is that design engineers tend to be tenacious in nature and this aids in finding ways to manage/mitigate difficult problems.

Awareness of the situation is the first step to understanding the underlying problems faced by today's design engineers industry wide. Here are a few of the situations I have faced in the last year alone.

Long Lead Times

I experienced a product launch failure due to component unavailability. The client had a great idea, practical experience with the market, and identified a need which is not currently being served. What was needed was a compact, robust design that would fit the application—my bread and butter. I could not bring this product to fruition because of the lack of simple microcontrollers and peripheral devices. I was upfront with the customer that component availability was an issue and we needed to procure prototype and initial production materials early on, with the expectation that not all purchases may be used in the design.

Several months into the development, I was notified that my orders were not only delayed but cancelled. Lead times for components previously on order now extended anywhere from 50 to 100 weeks, well beyond the needed date.

PCBs already in hand, I then proceeded to delay/cancel my assembly orders and inform the customer that I could not produce a prototype, let alone a production-ready device to meet their launch window; the current status is in limbo.

Cost Increases

Massive product component cost increases due to the use of brokers to procure components have been a standard process of late as well. I have designs in production that doubled or tripled in cost due to a few hard-to-find ICs that were once considered plain, run-of-the-mill components. I typically avoid using brokers due to the increased level of scrutiny (cost) needed to ensure the components are indeed authentic, not reused/reclaimed parts, and undamaged. Typically, I would initiate a redesign to mitigate the unavailable component(s). In today's barren component landscape, redesign has become second place to using a broker because of the risk of not being able to procure the replacement components.

Board Respins

The increased number of board respins I have performed lately has been troublesome and expensive. I have not been able to produce a design within the last year without having to perform a respin based on component unavailability. I now commonly hear from customers and colleagues that non-recurring engineering (NRE) cost is becoming more like recurring engineering (RE) cost. The sheer number of delivery cancellations and delays has caused me and many others to respin PCBs before they even hit the assembly line. In an attempt to stay positive, I will jokingly say, "These respins spin me right 'round, like a record, right 'round, 'round, 'round."

Where to Take Action

Now that we see unmet delivery promises stalling product launches, material costs are increasing at an alarming rate, and product development is taking much longer. What can we do about it? A "one solution fits all" typically does not work. I have, therefore, broken the down the types of actions that could be taken into three categories: enterprise, small business, and entrepreneur/consultant.

Enterprise

Enterprise or large corporations can affect supply chain behavior in significant ways. A corporation's policies carry the weight to affect the adoption of procedures and practices their vendors use. Large-scale adoption of Lean manufacturing and just-in-time manufacturing are well known and documented examples of corporate policy driving supply chain behavior. What I see as two beneficial opportunities for large scale supply chain behavior modification are integrated product development and adaptive material procurement. Truly integrating supply chain logistics into the product development process will give design engineers insight into component availability at the time of selection; conversely, this should allow component manufacturers early insight into what components are being considered for use, which should, in turn, yield more accurate silicon wafer allocation.

The good news is that steps toward this end appear to have been taken with Siemens' acquisition of Supplyframe and its Design-to-Source Intelligence (DSI) ecosystem. The ability to systematically procure materials is crucial to many enterprise organizations, in particular aviation, space, and defense, where the supply chain processes are centered around AS 9100, a standard with the intent to focus on the supply of reliable products. To this end, I propose giving component brokers a formalized place within the procurement process. There has been a large amount of procurement from component brokers due to widespread supply chain breakdowns, most likely through non-conforming material review boards or something similar.

This is an opportunity to take data on the brokerage transactions to develop standard processes in which to engage with component brokers in a manner where trust and component pedigree can be established without a need for a lengthy, expensive, out-of-the-ordinary method. An adaptive material procurement process should allow for direct transactions between the component manufacturers, approved distributors, and component brokers without triggering a non-conforming material review.

The design engineer's role in this corporate culture shift is to promote enterprise-level tools and processes that promote integrated product development, and to be more engaged with the procurement process. If a lack of component source information or tedious component sourcing processes are creating product development delays, write a corrective action against the process along with a suggested researched solution. Do not stay silent and quietly quit the problem.

Small- to Medium-sized Business

Small- to medium-sized businesses do not typically have the push behind them to directly affect supply chain behavior, which is why a focus on better management of the supply chain situation may be more appropriate. Small/medium companies are not often a top tier customer and are most often delegated to interacting with approved suppliers such as Digi-Key, Mouser, Newark, and Arrow. My suggestion is designers should become familiar with services like Octopart and Findchips.com to get a feel for what's available on the market. Both services are free, show distributor information, and offer some amount of sourcing intelligence. Another strategy: Instead of build to print (seemingly impossible these days), use build to function. Communicate with your customers that you can build to function, in addition to build to print. This requires the design engineers to develop a product/solution that is modular and able to be built in multiple configurations. Yes, this takes more time up front. However, it is better than not being able to build anything.

Entrepreneurs and Consultants

Entrepreneurs and consultants often find themselves beholden to the whims of the electronics supply chain as it ebbs and flows. Our tools of trade are flexibility and experience coupled with creativity and tenacity. Most of our problems and solutions are centered around the here and now. Extended lead times mean that unviable, cancelled orders become a crisis situation, and delayed/missed revenue is directly felt. These are some of the practices that have helped me and some of my colleagues overcome the current supply chain pain. One person's leftover stock is another's prototype.

Develop your network and get to know others in your area of expertise. This opens everyone's door to opportunities, solutions, and sharing. Stay focused on meeting your customer's needs vs. trying to sell them on some exciting new IC from Vendor X. Over the last year I have seen a lot of innovative, miniaturized power supply designs get replaced with 555 timer designs, due solely to component unavailability. Leverage advanced PCB fabrication capabilities (blind/buried vias, laser routing) to ease the burden of using large footprint technologies or use an all-flex design to reduce/eliminate interface connectors.

In closing, designers are capable of designing through the current supply chain pain by leveraging corporate policy/procedures to modify supply chain behavior, designing build-to-function product capability, and staying focused on meeting customer needs. **DESIGN007**

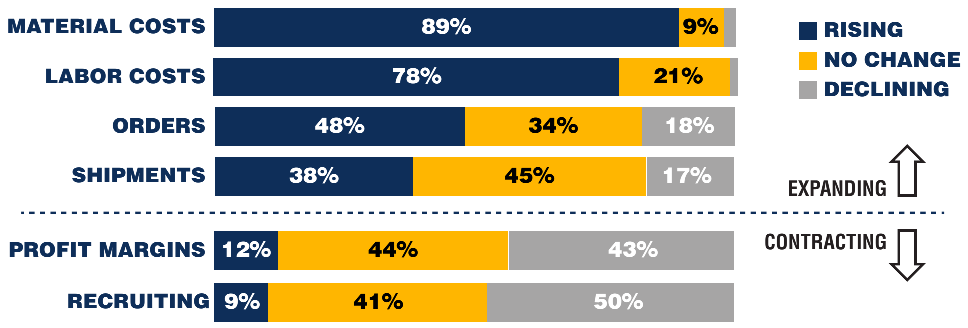
Chris Young is owner/lead engineer at Young Engineering Services LLC, and an I-Connect007 columnist. To read past columns, click here.

PCB FABRICATORS ARE FEELING THE SQUEEZE

CURRENT DIRECTION OF KEY BUSINESS INDICATORS

90% of electronics manufacturers are currently experiencing rising material costs, while 80% are experiencing rising labor costs.

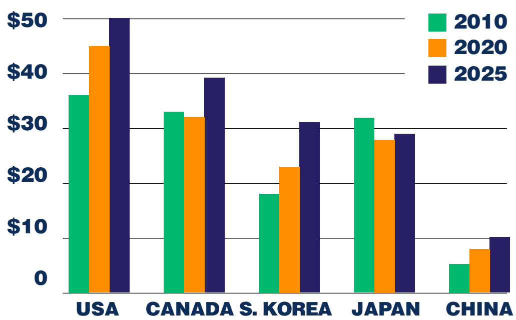
At the same time, ease of recruitment, profit margins, and inventories are presently declining.



Source: IPC Industry Survey, July 2022

RISING LABOR COSTS

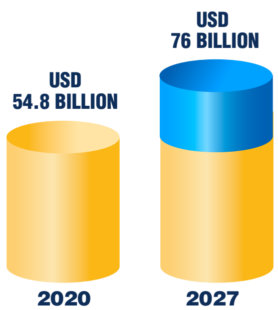
Average Hourly Wage (USD)



Source: The Economist Intelligence Unit

GROWING MARKET

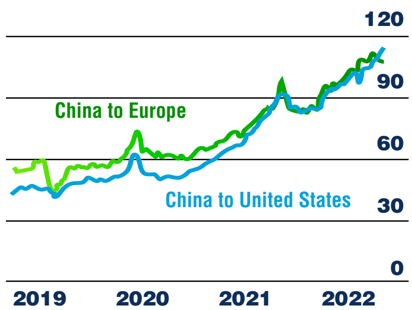
Growth will continue



Source: Research and Markets

SLOWER SHIPMENTS

Shipping time (days)



Source: Flexport Research

I-007e FAST FACTS

Success Begins With a Little Confidence

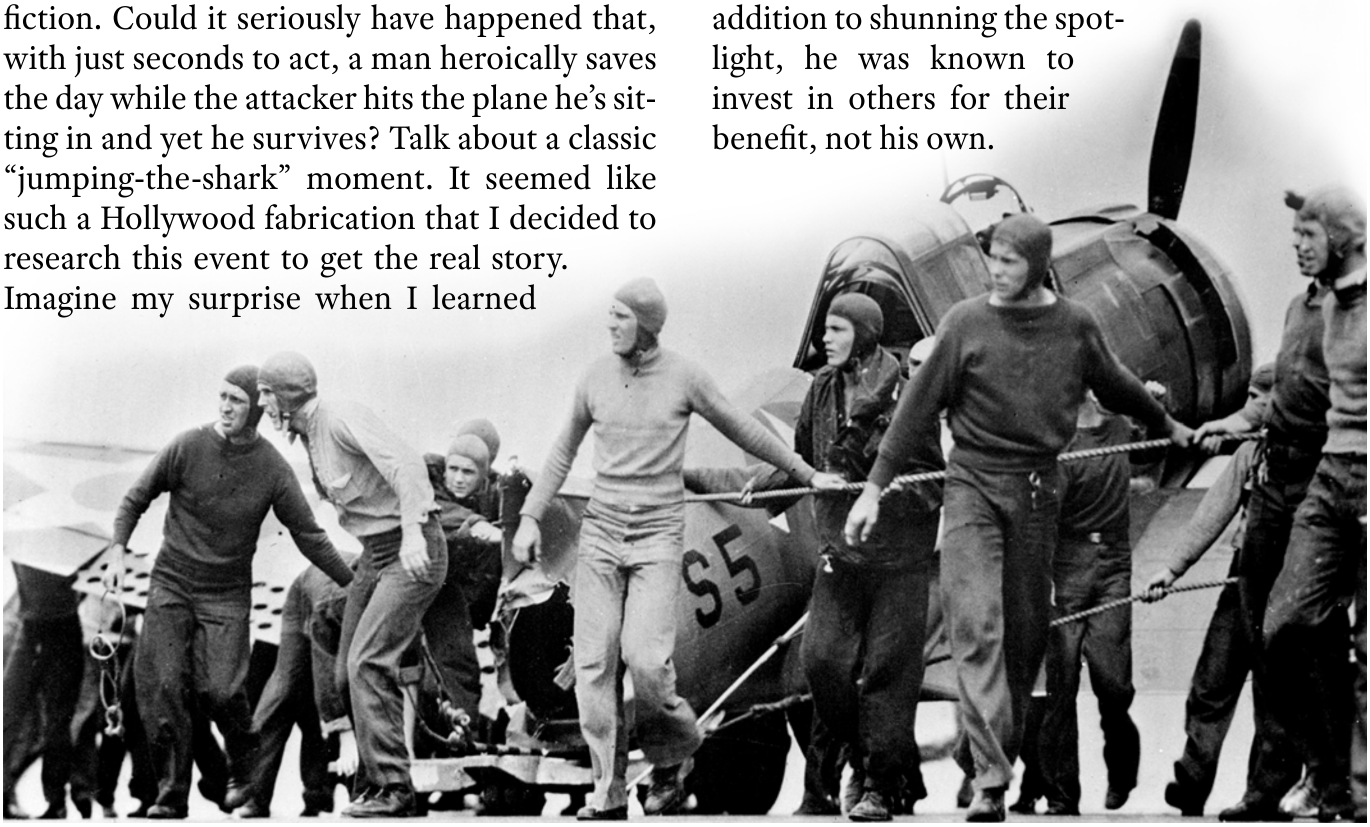
Tim's Takeaways

by **Tim Haag**, FIRST PAGE SAGE

Bruno Gaido was a young radioman-gunner portrayed by Nick Jonas in the 2019 movie "Midway." An early scene shows a Japanese bomber trying to sink the USS Enterprise by crashing into it with his plane. The scene shows the bravery of Bruno as he ran across the deck of the ship and jumped into the rear seat of a parked airplane, using its guns to shoot back. His shooting damaged the bomber just enough to force it off course, thereby saving the ship just as the bomber crashed into Bruno's plane, cutting the plane in half and spinning it around.

This spectacle was amazing, but I quickly dismissed it as "Star Wars" action-adventure fiction. Could it seriously have happened that, with just seconds to act, a man heroically saves the day while the attacker hits the plane he's sitting in and yet he survives? Talk about a classic "jumping-the-shark" moment. It seemed like such a Hollywood fabrication that I decided to research this event to get the real story. Imagine my surprise when I learned that not only is it a true story, but the movie was based on eyewitness accounts and ship records, so that is exactly how it happened, including Admiral William Halsey spot-promoting Bruno to Aircraft Machinist Mate First Class for saving the ship.

As impressive as that was, I learned some other interesting facts about Bruno not shown in the movie[1]. For instance, Bruno had a reputation of being a tough customer and was known as someone who got the job done without a lot of self-promotion. In fact, when Admiral Halsey promoted Bruno, a search party was formed because Bruno was trying to avoid attention. In addition to shunning the spotlight, he was known to invest in others for their benefit, not his own.



Here's one example. In June 1941, Lt. Junior Grade Norman "Dusty" Kleiss, a pilot new to the Enterprise, set out to make his first aircraft carrier landing. Learning to set a plane down on land is hard enough but landing on the narrow strip of a moving ship in the middle of the ocean can be incredibly challenging for even the most experienced pilots. Remember, this was 80 years ago when computer navigation and automated systems were concepts that hadn't even been imagined yet let alone incorporated into an airplane. The only thing that carrier pilots could rely on to get them safely back on the deck was their skill, experience, and confidence—characteristics that were in short supply for a greenhorn like Kleiss. However, when he got into his plane, Kleiss found Bruno Gaido sitting in the rear seat instead of the sandbags normally used to simulate a crewman's weight on a qualification flight. Because of his inexperience, Kleiss tried to talk him out of coming with him, but Bruno simply responded; "You got wings, don't ya?" Kleiss went on that day to be qualified with several perfect landings thanks to the confidence instilled in him by Bruno Gaido.


Bruno Gaido, Family Photo

Sometimes the difference between our failure and success is determined by the faith and confidence that others have in us.

I'm sure we've all benefited from the confidence that others have shown in us over the years. When it came to my first solo flight as a student pilot I was just as anxious as Lt. Kleiss. To be completely honest, I wasn't just nervous, I was terrified. But my flight instructor calmly looked me in the eye and said, "You can do this." And so I did. One time while I was searching for work, I had stopped believing in myself. In that instance, a hiring manager gave me the shot of confidence I needed when she said, "My company needs what you can do," and she hired me. But perhaps the best example, and one that still gives me confidence whenever I think about it, is many years ago when my design manager assigned me to lay out a new generation of motherboard for a large computer manufacturer. This would be a high-speed design using new CAD software and design tactics that I was unfamiliar with. I would also be required to work onsite at this company for several months away from my regular team. It was an intimidating prospect, but my boss gave me the assurance I needed by telling me that he believed in my abilities to get the job done.

As we all know, the current demographics of the printed circuit board layout industry is changing. Many designers are approaching retirement, and engineering groups are looking for new designers not only to continue their work but take it to the next level. Circuit board technology is also changing. The next generation of PCBAs will require new design methodologies to support their evolving specifications along with new materials and processes to manufacture them. But no matter how much the software and hardware in our industry will change, the simple fact is that printed circuit boards of one type or another will still be needed for a very long time. To keep the electronics innovation process flowing, it is essential that we all participate in the growth of the PCB design industry as much as possible.

How can we help to develop a pool of new layout designers that are ready to take on the challenges of the next-generation of electronic development? As we have seen from the example of Bruno Gaido, one of the most effective methods for helping a greenhorn designer is to build up their confidence and give them the stable foundation they need for success. Here are some ideas that I use when working with new people on the job:

- **Look for the diamond in the rough:** Not all designers will follow a traditional career

path. If you find someone with interest and aptitude, try giving them a little encouragement and see where it takes them. A young high school graduate named Mark Eaton was working as an auto mechanic in 1978 when a college coach encouraged him to enroll at his school and play basketball. Eventually, Mark went on to rank second in the NBA for career blocks behind Kareem Abdul-Jabbar, spending his entire professional career playing for the Utah Jazz. Just imagine if that college coach had seen only a mechanic instead of a potential NBA superstar. May we all learn to look beyond what we simply see on the surface.

- **Be accepting of new personnel:** Change can be tough, and it is understandably difficult to see a trusted co-worker move on and replaced with someone new. But face it, this scenario is going to repeat itself many times over; we must accept and make the most of it. That leaves us with two choices: we can either bury our heads in the sand (which I don't recommend because the view isn't all that interesting from that perspective), or we can welcome our new co-workers with open arms. And as I have learned, more often than not these new people bring with them fresh ideas and new ways to get things done that help in ways I would have never thought possible. It's a winning combination for everyone.
- **Help them to succeed:** When working with a new designer, make sure to start them with projects they can grow with and learn from. Otherwise, you may risk discouraging them before they have a chance to reach their full potential. You will also want to ensure they have adequate training in your processes and procedures, and be prepared for their questions. The goal should be to create an environment that encourages their success as a designer of your products, and not merely how to punch a time-clock and collect a paycheck.
- **Hold a crown over their heads, and help them to grow into it:** As these new designers come up to speed, don't be afraid to increase their responsibilities according to their abilities. In this way you will further help build their confidence. However, it is essential to keep an eye on their progress to ensure your new designers don't overcommit. I've seen far too many new employees burn themselves out because they took on too much or were not managed correctly and got in over their heads.
- **Encourage learning:** Explore different paths of continuing education to help your new designers grow in their careers. There are many options available out there that range from simple online seminars to large-scale design conferences. Not only will your designers learn new skills at these venues, but investing in their future like this will also pay huge dividends in building up their confidence.

Probably the best advice is to treat new designers and employees the same way that we would want to be treated if we were in their shoes. Yes, they can get the job done by hiding in the corner and doing only what they are told to do. But to truly excel in our industry requires taking a few risks, and that takes confidence. It's up to us to help build that confidence in those we work with so they can reach their full potential. If you are questioning your own ability to build confidence in others, let me be the first to say, "I know you can do this, I believe in you." Until next time everyone, keep on designing. DESIGN007

References
1. H-004-5: Toughness—Aviation Machinest Mate 1st Class Bruno Peter Gaido, by Samuel J. Cox, March 29, 2017, Naval History and Heritage Command.

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

DFM 101: Final Finishes—ENEPIG and IAg

Article by Anaya Vardya
AMERICAN STANDARD CIRCUITS

Introduction

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

Final Finishes

Final finishes provide a surface for the component assembler to either solder, wire bond, or conductively attach a component pad or lead to a pad, hole, or area of a PCB. The other use for a final finish is to provide a known contact resistance and life cycle for connectors, keys, or switches. The primary purpose of a final finish is to create electrical and thermal continuity with a surface of the PCB.

There are a number of final finishes in use in the industry today, including:

- **ENIG** (electroless nickel, immersion gold)
- **ENIPIG** (electroless nickel, immersion palladium, immersion gold)
- **ENEPIG** (electroless nickel, electroless palladium, immersion gold)
- **IAg** (immersion silver)
- **ISn** (immersion tin)
- **Sulfamate nickel/hard or soft gold** (Electrolytic nickel/gold)
- **HASL** (hot air solder leveling)
  - › **SnPb** (63/37 tin/lead)
  - › **LF** (lead free)
- **OSP** (organic solderability preservative)

Final finishes are primarily application driven, so there are a number of considerations

that should be part of any decision to choose a final finish:

- Lead-tolerant or lead-free (LF) process
- Shelf life
- Flatness
- Lead or ball pitch
- Wire bondability
- Lead insertion
- Solder joint integrity
- Corrosion resistance
- Potential problems
- Cost

Lead-free Finishes

Lead-free finishes are considered RoHS-compliant (< 0.1% by weight of finish, for Pb, Hg or Cd) with the single exception of tin/lead HASL. The RoHS compliant finishes include the following:

- ENIG
- ENIPIG
- ENEPIG
- ImmAg
- ImmSn
- Electrolytic nickel/gold
- LF HASL
- OSP

Lead-free PCBs require that the standard HASL surface finish cannot be used. There is still a significant amount of discussion on what the long-term surface finishes will be. Currently, the immersion silver and OSP surface finishes are the most prevalently specified surface finishes for solderable PCBs. Immersion tin is the prevalent surface finish for press-fit backplanes. Please contact your PCB fabricator for current information on where industry specifications are heading.

Final Finishes

- **ENEPIG** (electroless nickel, electroless palladium, immersion gold)

ENEPIG is a three-layer metallic coating of 1.97 µin (0.05 µm) minimum, 2-5 µin (0.05-0.012 µm) typical, immersion-deposited Au over 2 to 12 µin (0.05-0.30 µm) electroless-deposited Pd over 118.1-236.2 µin (3-to 6 µm) electroless-deposited nickel.[1]

This process plates a thin coat of nickel covered by a thin layer of palladium, then gold. The gold provides a very good solderable surface. When components are soldered onto these pads, the palladium and gold diffuse into the solder joint. The gold layer is very thin so it won't reduce the solder joint strength. This process is generally not utilized for high reliability, long lifetime, or high vibration applications. This is used for wire-bonding applications.

ENEPIG is typically used for flat surface/fine pitch devices. The benefits of ENIPIG are a long shelf life and environmental resistance, as well as low contact resistance and excellent solder joint with SAC solders.

- **Gold Wire Bondable**
  - › Expensive
  - › Cannot be reworked
  - › Lossy RF (phosphorus nickel)
  - › Palladium intermetallic bond to Pb in Sn/Pb is not ideal

- **IAg or ImmAg** (immersion silver)

ImmAg is a single layer metallic coating of 5 µin (0.12 µm) minimum, 5-20 (0.12 to 0.50 µm) typical, of immersion-deposited silver.[2]

This process plates a thin layer of silver directly on top of the copper surface. As with the other immersion surface finishes, the finished product produces a very flat surface, which is ideal for fine pitch SMD arrays. This surface finish has the ability to maintain high solderability after multiple heat cycles. This can also be used as an aluminum wire-bondable surface. It is compatible with no-clean assembly processes. This is becoming popular as a HASL replacement for lead-free soldering applications. This surface finish will yield a dull tarnished looking surface. There is significant industry data showing that the dullness does not affect solderability or reliability.

ImmAg is inexpensive. It is typically used for flat surface/fine pitch devices and is aluminum wire bondable.

ImmAg has good RF performance if the following guidelines are followed:

- Packaging and storage are critical
- Careful handling required
- High insertion friction—not well suited for press fit pins

Understanding the cost drivers in PCB fabrication and early engagement between the designer and the fabricator are crucial elements that lead to cost-effective design success. Following your fabricator's DFM guidelines is the first place to start. **DESIGN007**

References

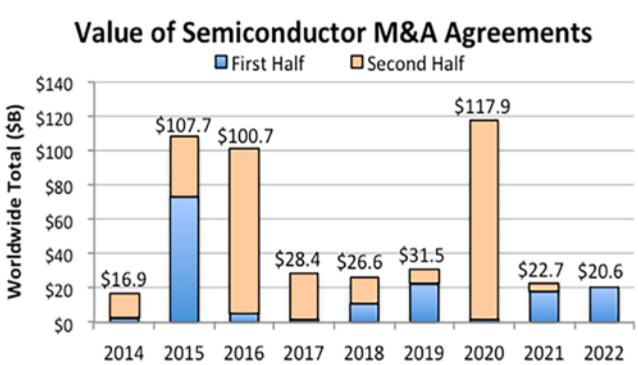
1. Per IPC-4556 Nominal pad size of 0.060" x 0.060" (1.5 mm x 1.5 mm).
2. Per IPC-4553A Nominal pad size of 0.060" x 0.060" (1.5 mm x 1.5 mm).

Anaya Vardya is president and CEO of American Standard Circuits; co-author of *The Printed Circuit Designer's Guide to... Fundamentals of RF/Microwave PCBs* and *Flex and Rigid-Flex Fundamentals;* and author of *Thermal Management: A Fabricator's Perspective*. Visit I-007eBooks.com to download these and other educational titles. He also co-authored "Fundamentals of Printed Circuit Board Technologies" and provides a discussion of flex and rigid flex PCBs at RealTime with... American Standard Circuits.

Four Large Agreements Prop Up 2022 Semiconductor M&A Total

The four largest semiconductor acquisition agreements in 1H22 accounted for essentially all of the $20.6 billion in M&A value during the first half of this year, according to M&A data contained in IC Insights' August 3Q Update of The McClean Report service.

Topping the M&A deals in 1H22 was an agreement by a Chinese investment consortium to take over bankrupt semiconductor conglomerate, Tsinghua Unigroup. In April 2022, government-backed Beijing Jianguang Asset Management Co. (JAC Capital) led a group of investment firms to complete a $9.4 billion injection of payments into Tsinghua Unigroup, which runs Yangtze Memory Technologies.



The second largest chip acquisition agreement in 1H22 was Intel's $5.4 billion deal to buy wafer foundry Tower Semiconductor in Israel. The cash purchase, announced in February 2022, is expected to be completed by early 2023 and is part of Intel's push into the silicon foundry business.

In May 2022, California-based MaxLinear announced an agreement to buy Silicon Motion in Taiwan for $3.8 billion in cash and stock. This acquisition is expected to be completed in 1H23 and will add controller ICs for NAND flash storage to MaxLinear's lineup of RF and mixed-signal products.

AMD, in April 2022, announced a $1.9 billion agreement to buy packet-processor and cloud-computing software startup Pensando Systems in Milpitas, California, to boost its presence in the data center market and increase competition with Intel. The acquisition of Pensando was finished in May 2022, three months after AMD completed its purchase of Xilinx in February 2022.

(Source: IC Insights)

Comparing Traditional and Bio-based Resins

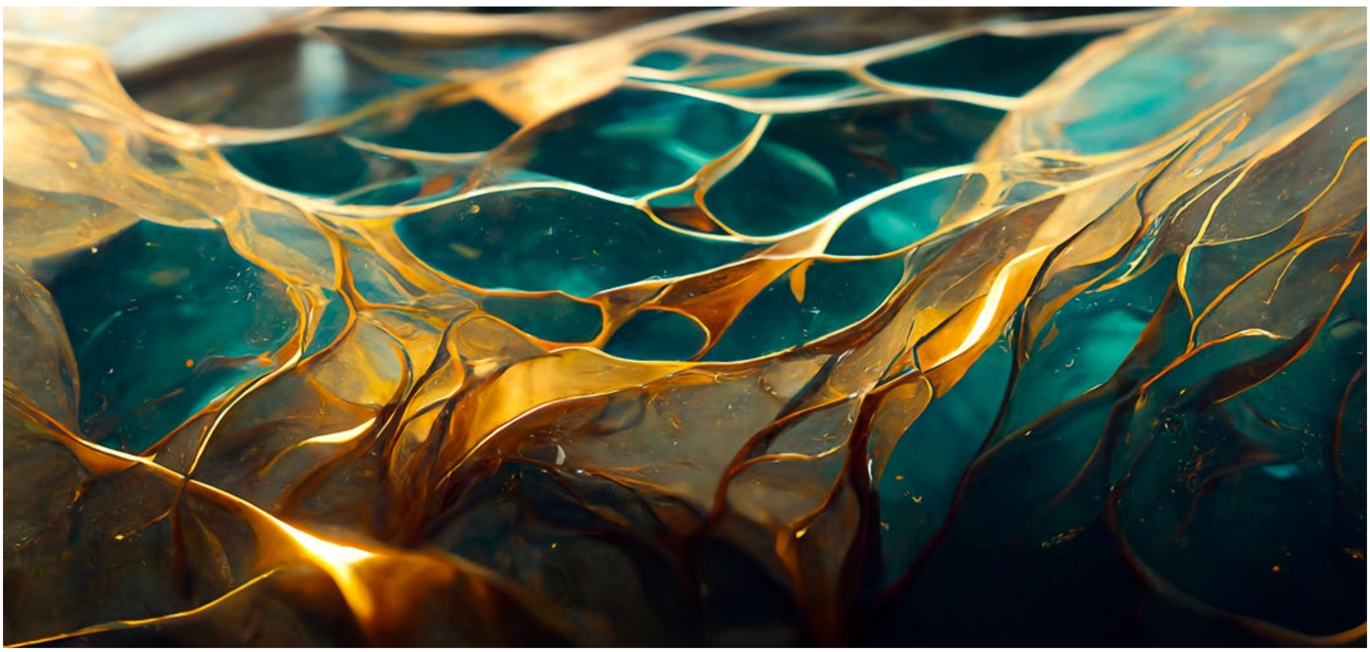
Sensible Design

by Beth Massey and Beth Turner, ELECTROLUBE

In this month's column, two of Electrolube's leading resin specialists have collaborated to give you the clearest focus of how introducing bio-based resin systems will impact your production process and the benefits they bring in terms of performance. Beth Turner is Electrolube's senior technical specialist and Beth Massey is Electrolube's global product manager for resins. Having two Beths working on resins may be a little confusing, but this column should help demonstrate the unique input each has and alleviate any confusion. Here we dive into how bio-based resins differ from their traditional counterparts, examining what they can bring to the table, aside from their obvious sustainable and environmentally-friendly benefits.

What percentage of the overall content can reasonably be bio-based without causing a detrimental effect to the performance of the product, and is the bio content added to part A or B?

The honest answer to this question is quite simple: It depends. Realistically, up to 70% of content can be bio-based, and both A and B can contain bio-based materials. The 70% limit has more to do with availability of bio-based alternatives than high levels having a detrimental effect. It's likely that some components of the formulation will never have a bio-based alternative, but the benefits we are seeing from changing to bio-based raw materials, where possible, does suggest we should



push to include content as high as possible. It goes without saying that bio-coatings have significantly less impact on the environment and meet the ethical needs of manufacturers and end users. However, the most surprising benefit we have observed is the improvement in performance. Even additives such as powders can be bio-based.

Does a particular chemistry, i.e., epoxy or polyurethane, lend itself to removing some of the chemicals present and substituting them for bio-based content. If yes, why is this?

Both are suited. If we refer to Beth Turner's award-winning white paper on next generation bio-resins, she states, "Nature offers an abundance of macromolecules and smaller molecular weight compounds that provide renewable sources for polymers, as opposed to crude oil. These renewable resources make ideal structural backbones for the synthesis of biopolymers, renewable-based monomers, fillers and additives, the key ingredients for polyurethane and epoxy thermoset materials. Research and development to promote innovative solutions that lead to a sustainable economy shows that bio-based materials can deliver a viable alternative to materials derived from crude oil, even in electronic encapsulation applications."

Do bio-based resins behave in a different way when being poured or when curing compared to their chemical versions, and are the tech datasheets almost identical in terms of the qualities exhibited?

Interestingly, bio-based resins show the same dispensing behaviour and cure profiles as conventional resins, so there is no need to invest in adaptive equipment. There is no negative effect on processability or curing when using bio-based materials and performance is equivalent. However, we do see significant benefits in cured properties when select bio-based raw materials are used. Combining a bio-based polymer matrix and hardener with a biogenic filler offers a significant improvement in thermal conductivity compared to a fully synthetic polyurethane using a synthetic polymer matrix derived from crude oil and mineral rock filler. The bio-based resin also offers better electrical insulation and thermal stability while also having a higher resistance to hydrolysis vs. a synthetic polyurethane resin.

Are bio-based products gaining popularity purely due to their association with sustainability or do they exhibit qualities that are superior to their chemical alternatives?

Initial interest is often sparked because of the increasing focus on sustainability but this is extended by research showing significant performance benefits for bio-based resins. The best way to succeed with bio-based products is to ensure consumers do not have to compromise on quality in order to make the more sustainable choice. As was documented in Beth Turner's white paper, use of certain bio-based raw materials offers substantial benefits in terms of the cured resin's electrical and thermal performance vs. synthetic-based raw materials. The fact that these high-performance, bio-based materials are a more sustainable solution is really an added bonus. It's especially exciting that some of the raw materials tested were from biogenic waste, adding another layer of sustainability by repurposing waste material from other industries.

In your experience, is sustainability something that genuinely affects manufacturers and their reputation in the market as doing their absolute best to reduce their carbon footprint, or is this more of a box-ticking exercise?

I think we are seeing a sea change in attitude at the moment, from sustainability seen as a niche focus to now being a big concern for society as a whole. In the past, the focus has been on sustainability almost as a marketing tool, leading to "greenwashing," but more manufacturers are realising that sustainability needs to be at the core of what they do, both to meet increasing consumer demands for sustainable products and to future-proof their business. We've seen supply shortages and price volatility making synthetic materials, such as those derived from crude oil, a much less attractive choice, which is an additional commercial incentive toward using more sustainable bio-based materials. The fact that our research shows using certain bio-based materials in the formulation improves resin properties also means a bio-based resin doesn't need to be sold on sustainability credentials alone. Its performance speaks for itself, as can be seen with one of our products that is a high temperature, chemically-resistant polyurethane resin.

Electrolube's extensive research and findings conclude that the future looks extremely promising for bio-based resins, which bring distinct performance advantages in harsh environments, including underwater applications as well as hot and humid operating environments. Bio-based chemicals are highly suited to a wide variety of applications within the electronics industry and even lend themselves to RF applications. One Electrolube product has a bio-sourced content of over 60% and has been used in many sensor applications. It is likely that the bio-based chemicals share within the electrochemical production segment will increase, and electrochemical manufacturers and associated OEMs will become an important part of the bio-economy of the future. **DESIGN007**


Beth Massey


Beth Turner

To read past columns from Electrolube, click here.

Download your free copy of Electrolube's book, *The Printed Circuit Assembler's Guide to… Conformal Coatings for Harsh Environments*, and watch the micro webinar series "Coatings Uncoated!"

Flex007 Highlights

Ventec Expands Flex-rigid Material Range for Critical Military, Aerospace, and Ultra-high Reliability Applications ▶

Ventec International Group Co., Ltd. has added to its flex-rigid No Flow/Low Flow prepreg range with the introduction of tec-speed 4.0 (VT-462(L) PP NF/LF), a next-generation no & low flow FR 4.0 prepreg material that offers high-Tg, low Dk, low loss, and excellent thermal reliability.

Rogers Corporation Reports Second Quarter 2022 Results ▶

Net sales of $252.0 million increased 1.5% versus the prior quarter resulting from commercial actions and higher demand in the EV/HEV, portable electronics, and defense markets.

American Standard Circuits Adds Industry Expert John Johnson as Director of Business Development ▶

Anaya Vardya, president and CEO of American Standard Circuits, has announced the appointment of industry expert John Johnson to the position of director of business development. Mr. Johnson's main effort will be focused on high tech business development, with special focus on the growth of Averatek's A-SAP technology.

Flexium 1H2022 Revenue Up 18.5% YoY ▶

Taiwan-based Flexium Interconnect Inc., a manufacturer of flexible printed circuits (FPC), has posted sales of NT$2.84 billion ($94.88 million at $1=NT$29.94) in June, up by 25.7% year-on-year (YoY), and by 3.4% from the previous month.

DuPont Reports Second Quarter 2022 Results ▶

DuPont announced financial results for the second quarter of 2022.

Trackwise Designs Awarded Cyber Essentials Plus Accreditation ▶

Trackwise is delighted to have been awarded the Cyber Essentials Plus accreditation, the Government's enhanced cyber security recognition, for the seventh consecutive year.

Compeq Posts 29% Revenue Growth in July ▶

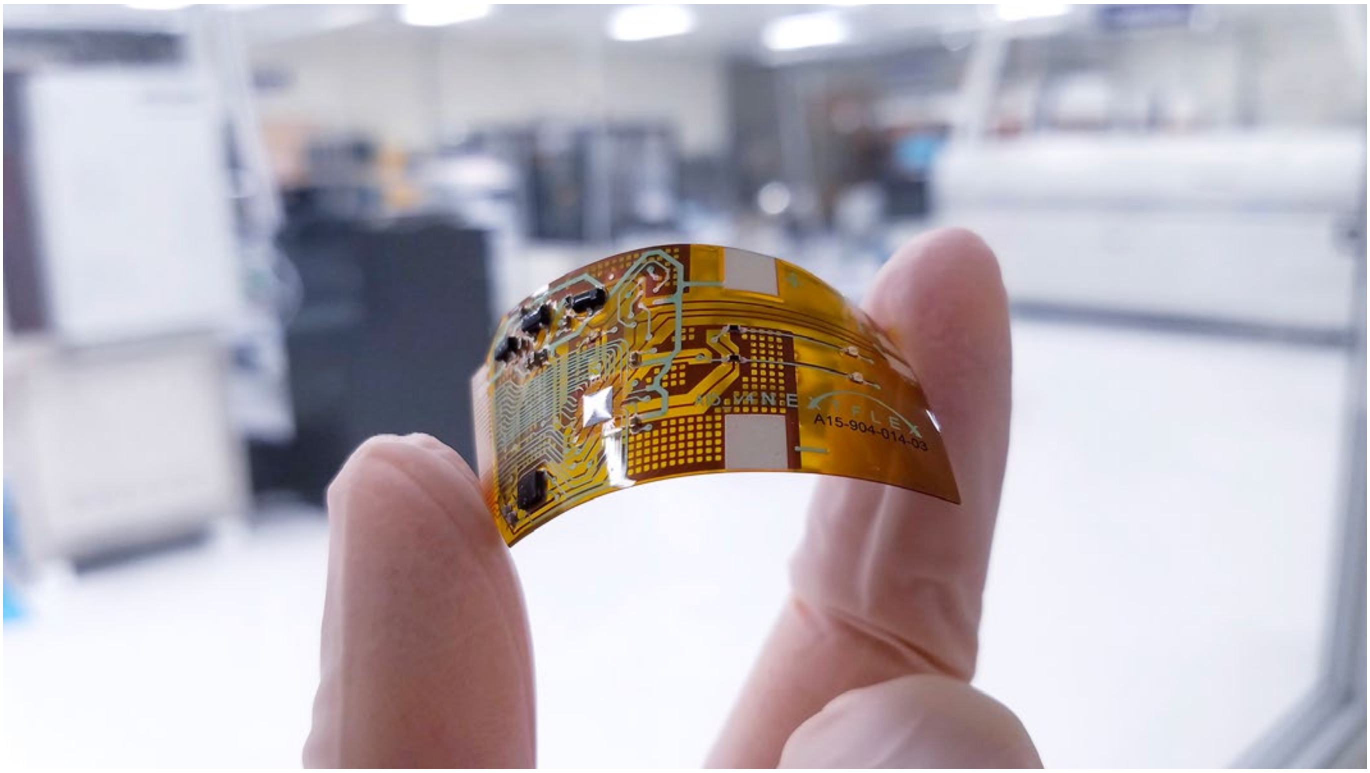
Taiwan-based Compeq Manufacturing Co. Ltd, a manufacturer of HDI, rigid-flex PCBs, and flex PCBs, has posted unaudited net sales of NT$6.89 billion ($229.66 million at $1:NT$29.99) for July 2022, up by 28.56% year-on-year and by 15.4% from the previous month.

Keysight Solutions Selected by Flexium to Deliver New Liquid Crystal Polymer Transceiver Applications ▶

Keysight Technologies, Inc. and Flexium Interconnect, Inc. are collaborating to improve millimeter wave modules and material test accuracy and efficiency.

AT&S Starts New Financial Year 2022/23 with Record Quarter ▶

AT&S continued its growth course unabated in the first quarter of the financial year 2022-23. Thanks to the successful implementation of its strategy, the company achieved record revenue and earnings levels.



The Chip Shortage Leads to Innovation

Feature Article by Malcolm Thompson
NEXTFLEX

The chip shortage is by no means over, with estimates expecting it will last into 2023. Some could see it taking even longer, such as Intel CEO Pat Gelsinger, who expects it to see shortages into 2024 due to those now impacting electronics production equipment[1]. But if there's any bright spot to be had, it's that a crisis often leads to long-term solutions. In this case, it's the increase in government funding for semiconductor production in the United States. Once the CHIPS Act proceeds, we can significantly accelerate building semiconductor fabs in the United States and work toward preventing future chip shortages that would put us back into our current situation.

In Every Challenge, There Is Opportunity

The impact of the chip shortage on automotive production and consumer electronics has highlighted the importance that electronics now play in our lives, from lawmakers to everyday consumers. The ongoing disruptions to chip supply and production have rippled through broad parts of medical, industrial, defense, and aerospace. Everstream data[2] now says the average lead time for advanced chips is 52 weeks. This is a problem that affects us all. Thankfully, because of this broad impact, lawmakers and industry members have recognized that the chip shortage goes beyond just silicon foundries.

To solve the overall problem, investment in wafer foundries must be accompanied by funding for advanced packaging of the chips that will come from the new foundries being supported by the CHIPS Act. Without that additional capability, many of these new chips will be sent overseas for packaging, which would defeat the bill's purpose of boosting domestic chip production overall. Thankfully, this is communicated in the CHIPS Act, and is getting more focus from players across the electronics industry.

While there is concern that the big players in advanced packaging will suck up all the funding, there's hope that additional complementary and novel approaches such as additive manufacturing will also see meaningful investment. Even a small portion of the overall budget planned for advanced packaging can go a long way in the hands of nimble, innovative companies that, when working together, can raise the bar for everyone in the electronics industry.

We Must Innovate Now, Or We'll Be Here Again

Collectively, we need to get serious about forming teams, writing proposals, and expanding the electronics space to encompass emerging technologies like additive electronics and flexible hybrid electronics (FHE), because we know other countries are pursuing new ways to create new classes of electronics, and have already made significant headway. Today there is limited semiconductor packaging in the U.S., with early funding intended to go into heterogeneous integration methodologies at large companies. The inclusion of investment in additive electronic interconnect packaging would be complementary and could have an impact on a broad set of businesses in the supply chain.

If we don't invest in these forward-thinking methodologies, we are likely to be back here in five years: discussing a U.S. electronics deficiency and how to get out of it. In fact, it will be even harder to achieve leadership in the future, because as stated above, we know that others around the globe are investing in additive processing for electronics packaging and interconnect technology.

China, Taiwan, and European nations have had investment strategies for basic R&D for decades, so the U.S. will quickly fall behind in an accelerating race if we do not make significant investments in this area.

The CHIPS Act changed this conversation overnight, and people are having discussions that were not previously thought of: We must put money into the whole solution and the supply chain, not just the front end of the line. Every interconnect layer, substrate, and package must be considered, including new and efficient ways to produce these layers and integrate them with the chips we rely on today, as well as ways to make chips thin, flexible, and capable of several different forms so they can be applied to even more uses.

Our Immediate Next Step is Building Out Our Chip-building Workforce

Another key focus of the CHIPS Act must be workforce development. The fact is, we don't have a ready workforce to run all the proposed chip fabs and advanced packaging factories. Innovation will follow the people, and the geographic locations of innovation hubs—such as Ohio, where Intel is planning to build its fab facility—will need to appeal to people for work and to live. For every job in manufacturing, many more supporting companies and staff are needed, so these manufacturing jobs have a multiplicative impact on a location's employment. Hundreds, if not thousands, of jobs are at stake, so workforce development becomes part of the essential infrastructure of not just the chip production facilities, but the local economies and national economy they will support.

These are high-wage, high-skill jobs at stake, and we need universities and community colleges with the facilities and curricula to educate and provide hands-on experience so that we can be ready to staff the jobs of the future. We're already seeing this happen across the country at schools such as Lorain County Community College in Ohio, Evergreen Valley College in California, and Drake State Community and Technical College in Alabama introducing learning programs for both professionals seeking more advanced manufacturing experience and early learning programs designed to generate interest in the field among K-12 students.

We Have a Chance to Build Something That Goes Beyond Chips

The ultimate takeaway is there will be additional infrastructure opportunities outside of today's supply chain. This will include opportunities for equipment manufacturers, materials companies, infrastructure providers, and more who recognize that the challenge the chip shortage presents represents an opportunity to solve it. Foundries together with advanced packaging solutions are clearly needed as a part of resurgent U.S. infrastructure, but also forward-thinking capabilities such as additive and hybrid electronics to prevent another shortage down the line, ensuring instead that this new electronic paradigm we're building is one that lasts. **DESIGN007**

References

1. "Intel CEO now expects chip shortage to last into 2024," by Kevin Stankiewicz, CNBC, April 29, 2022.
2. "The Chip Shortage is Easing—But Only For Some," by Will Knight, Wired, July 25, 2022.

Malcolm Thompson is executive director of NextFlex.

All Systems Go

Time Traveling to 2030 for ML-Augmented PCB Design

By Jorge Gonzalez and Luke Roberto

In our previous column, "Accelerate Your PCB Designs with Machine Learning," we explained how artificial intelligence (AI) is an umbrella term embracing technologies that empower machines to simulate human behavior, while machine learning (ML) is a subset of AI that allows machines to automatically learn from past data and events without explicitly being programmed to do so. As ML systems become increasingly complex and capable, the distinction between AI and ML is becoming increasingly blurred.

We also discussed how we are currently in the early years of the second era of AI, and how ML has started to appear in PCB layout applications. Remembering that we are still in the early days of ML deployment in the PCB space, we talked about the types of tasks ML can help with today, such as detecting when we start to perform repetitive low-level activities and assuming the responsibility of implementing these tedious, time-consuming, and error-prone tasks, thereby allowing us to stop doing the dull and boring things and freeing us up to start doing only the cool and interesting things.

Remember that the first iPhone didn't appear until 2007 (15 years ago), the modern era of AI and ML only kicked off circa 2012 (10 years ago), and that consumer VR in the form of the Oculus Rift made its first appearance in 2016 (only six years ago). Who among our number would have predicted any of these applications and technologies 20 years ago? So, how accurate will any predictions we make be here? Well, let's take some guesses, and then in 2030 and 2040, we'll look back to see how well we did.

To read the entire column, click here.

The Rapidly Expanding Realm of Stretchable Circuits

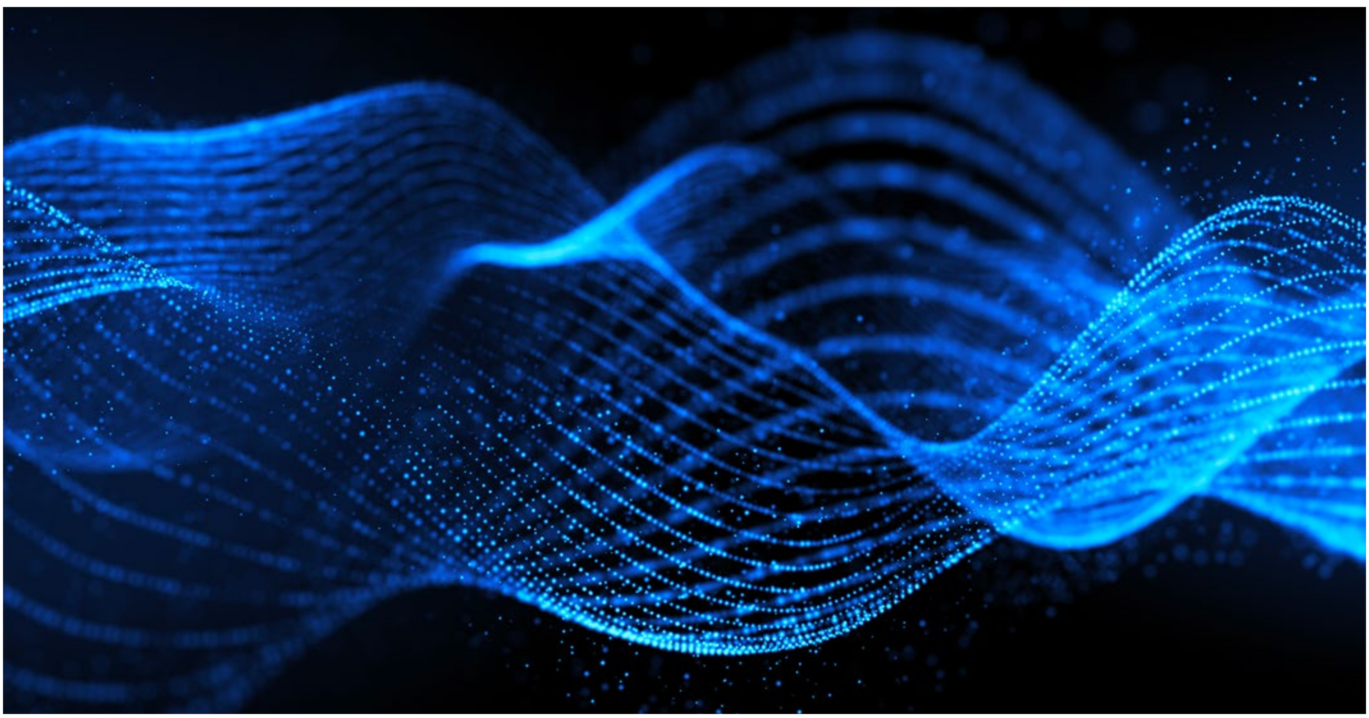
Flexible Thinking

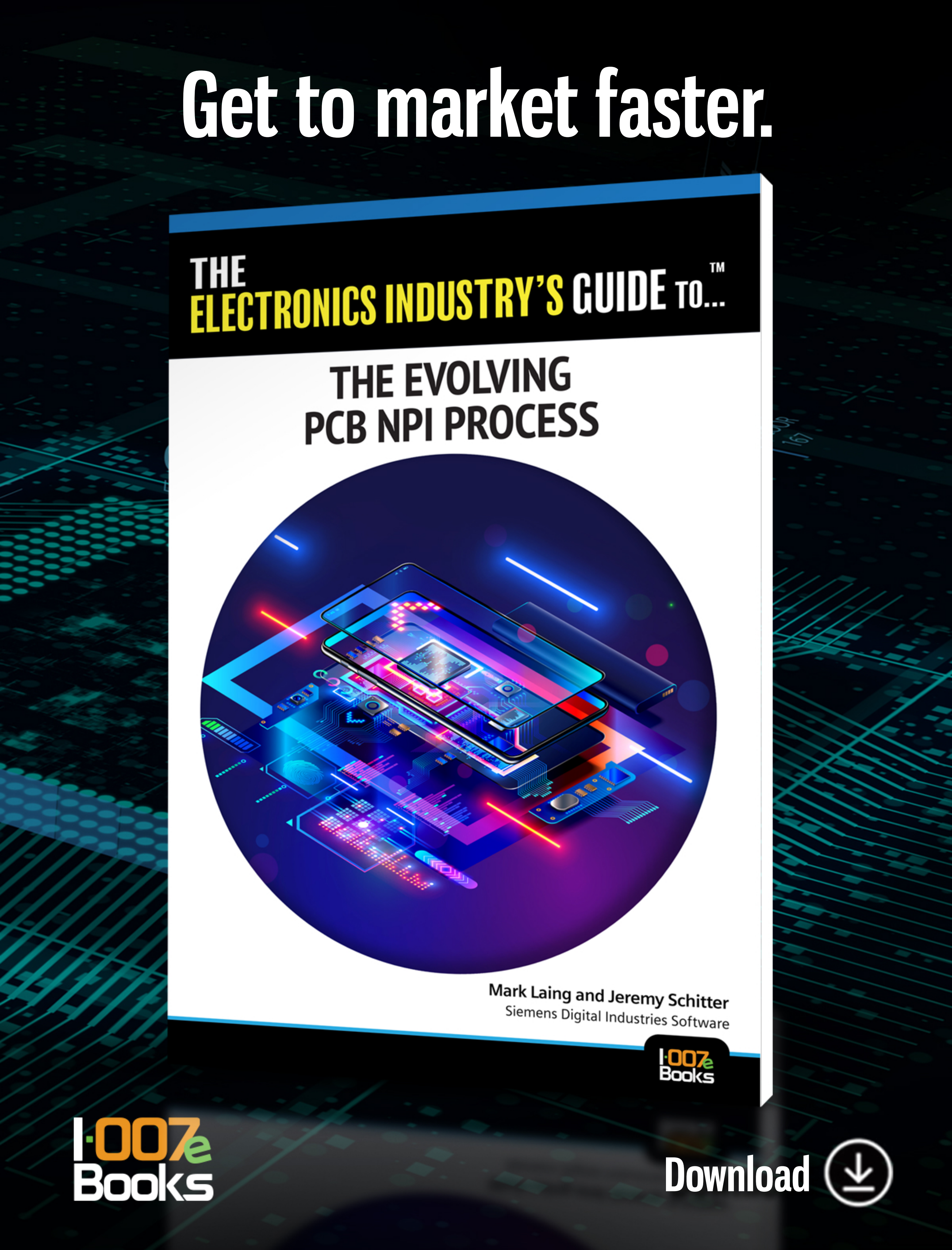
by **Joe Fjelstad**, VERDANT ELECTRONICS

Flexible circuit technology has been rising ever higher on the radar of those charged with designing next-generation electronic products for every imaginable application, from the mundane to the highly exotic. The technology is being embraced by a growing fan base as they become increasingly aware of flex circuit technology's numerous benefits. They are being driven to new heights by industry and government collaborations such as NextFlex and FlexTech, and their laser-like focus on what was formerly called printed polymer thick film circuits—now rebranded as flex hybrid electronics (FHE) and printed electronics.

More than a simple technology rebranding and marketing effort, NextFlex has created an impressive operation in San Jose, California, where material equipment and process solution providers can showcase their offerings to members and provide space where engineers and local students can familiarize themselves with them. Given the looming talent gap in the field of interconnection fabrication technologies within the United States, this development is most welcome.

The general focus on printing technologies for their flexible circuit products has boosted enthusiasm among designers looking for new ways to use the technology to realize their visions—from the practical to the fanciful. According to a recent study and report by market research group Lucintel, the stretch-





Get to market faster.
THE ELECTRONICS INDUSTRY'S GUIDE TO...™
THE EVOLVING PCB NPI PROCESS
Mark Laing and Jeremy Schitter
Siemens Digital Industries Software
I-007e Books
I-007e Books
Download

able and conformal electronics market is expected to reach approximately $479 million by 2025 through a compound annual growth rate (CAGR) of 89%[1]. If it holds true, that is an impressive number.

One area receiving ever greater attention in recent times has been wearable electronics, especially regarding physical wellness monitoring for sports and medicine. While not an imperative (wireless solutions are an option), a key feature increasingly sought in such products has been stretchability and conformability to the contours of the subject being monitored—the human body. We take for granted the various movements of our physical bodies, especially our appendages. However, we less often think about the rising and falling of our chests as we breathe, or movements in our throats as we swallow, but these motions should be accounted for if we need to reliably keep contact with the skin during use to monitor internal functions such as pulse and/or heart rhythm. A circuit that can stretch, and repeatedly and reliably recover during operation, can be a very important feature.

A circuit that can stretch, and repeatedly and reliably recover during operation, can be a very important feature.

In my book *Flexible Circuit Technology 4th Edition,* I co-authored a chapter on stretchable circuits with Professor Jan Vanfleteren. At the time, the topic was new, but nevertheless, there was a great deal of interest in stretchable circuit technology in the European Union, with research funded by the same. That interest has remained over the past decade and a range of new materials, both substrates and conductors, have been developed and entered the fray.

Developing stretchy materials has been less of a challenge than making printed conductors reliably adhere to them over repeated stretching events. The continuing effort to create better and more stretchable conductors has yielded some interesting results, including some liquid metal solutions.

The stretchable circuit branch of flexible circuits has resulted in the need to create dynamic testing tools to account for the different stresses that might occur with these newest members of the flexible circuit family. During my visit to FlexCon, co-located with SEMICON® West in San Francisco last month, I visited the Bayflex Solutions booth. The company is the exclusive distributor of mechanical endurance testing equipment for North America and Europe for Yuasa System Co., Ltd. Yuasa describes itself as a global leader in accurate, repeatable, and reliable mechanical test equipment capable of flexing, bending, folding, stretching, rolling, and twist testing flexible circuits, including data collection and analysis. In the real world, however, stretching can be more complex than simple linear stretching. The company displayed a device where a dome-shaped tool was pressed down into a stretch circuit and then lifted upward in a repeating motion. This machine was not found on the company website but appeared to be a new and legitimate type of test, capable of approximating the motion of a circuit placed, for example, over a knee or elbow during flexing.

Finally, they displayed a twisting test machine, which I found interesting because testing was not a normal part of the traditional mechanical testing regimen for flexible circuits, but makes sense when you consider the wider range of motions sought from flexible/stretchable circuits both for monitoring motion in humans and enabling motions in robots.

In summary, stretchable circuit technology is rapidly expanding its position and role in the realm of flexible circuitry, enabling a host of new applications, including consumer electronics, healthcare monitoring, artificial skins for robots, decorative textiles, automotive systems, and aerospace and defense. It is a welcome and enabling addition to the family of flexible circuit technologies. DESIGN007

References

1. "Stretchable and Conformal Electronics Market, Global Forecase to 2022, Technology & Telecommunications, Lucintel.

Joe Fjelstad is founder and CEO of Verdant Electronics and an international authority and innovator in the field of electronic interconnection and packaging technologies with more than 185 patents issued or pending.

To read past columns, click here. Download your copy of Fjelstad's book *Flexible Circuit Technology, 4th Edition*, and watch his in-depth workshop series.

Print, Recycle, Repeat: Scientists Demonstrate a Biodegradable Printed Circuit

A team of researchers from the Department of Energy's Lawrence Berkeley National Laboratory (Berkeley Lab) and UC Berkeley have developed a fully recyclable and biodegradable printed circuit. The researchers, who reported the new device in the journal Advanced Materials, say that the advance could divert wearable devices and other flexible electronics from landfill, and mitigate the health and environmental hazards posed by heavy metal waste.

"When it comes to plastic e-waste, it's easy to say it's impossible to solve and walk away," said senior author Ting Xu, a faculty senior scientist in Berkeley Lab's Materials Sciences Division, and professor of chemistry and materials science and engineering at UC Berkeley. "But scientists are finding more evidence of significant health and environmental concerns caused by e-waste leaching into the soil and groundwater."

In a previous study, Xu and her team demonstrated a biodegradable plastic material embedded with purified enzymes such as Burkholderia cepacian lipase (BC-lipase). For the current study, Xu and her team simplified the process even further. Instead of expensive purified enzymes, the biodegradable printed circuits rely on cheaper, shelf-ready BC lipase "cocktails."

To test its shelf life and durability, the researchers stored a printed circuit in a laboratory drawer without controlled humidity or temperature for seven months. After pulling the circuit from storage, the researchers applied continuous electrical voltage to the device for a month and found that the circuit conducted electricity just as well as it did before storage.

Next, the researchers put the device's recyclability to test by immersing it in warm water. Within 72 hours, the circuit materials degraded into its constituent parts. By the end of this experiment, they determined that approximately 94% of the silver particles can be recycled and reused with similar device performance.

Now that they've demonstrated a biodegradable and recyclable printed circuit, Xu wants to demonstrate a printable, recyclable, and biodegradable microchip.

(Source: Berkeley Lab)

The Printed Electronics Roundtable, Part 2

Interview by Andy Shaughnessy
I-CONNECT007

We recently held a roundtable with a team of printed electronic circuit experts from companies that run the gamut: John Lee and Kevin Miller of Insulectro, Mike Wagner of Butler Technologies, Tom Bianchi of Eastprint, and John Voultos of Sheldahl Flexible Technologies.

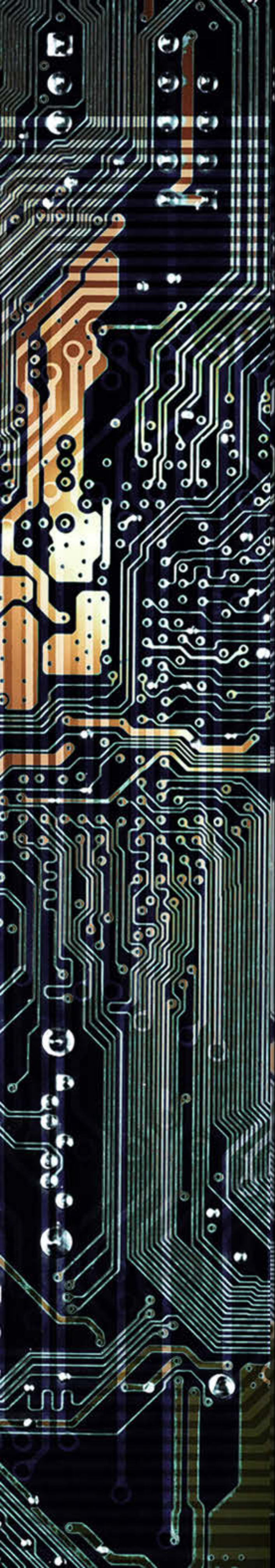
In the first part of this roundtable, the team dispelled a variety of myths surrounding PEC. In this second part of the roundtable, the participants discuss what designers and fabricators need to know to jump into printed electronics, and some of the drivers behind this growing technology.

Andy Shaughnessy: We've looked at some of the myths about PEC. Now, how would a fabricator or PCB designer get involved in doing printed electronics? What resources are available? What's the first step?

Tom Bianchi: Talk to Eastprint, Butler Technologies, or John at Sheldahl. That's the first step.

Mike Wagner: Prior to the pandemic, one easy way was going to specific shows that had printed electronics as their theme. Eastprint was there and you could find the material suppliers they use, or Butler Technologies. You can talk to everybody—the material suppliers and the integrators—and come out of there with a whole set of suppliers to produce a product for you. You still have to seek those out. Sometimes, the shows are virtual, and others are going back to in-person, but I would get involved with those groups, whether it's online forums or actual printed electronics shows, and you'll meet the right crowd. That's how I got involved—by going to these shows and networking at these events.

Kevin Miller: When I started in the printed electronics world, I'd spent 25 years in buil ding and supporting the printed circuit boards



Kevin Miller

industry. So, I did start bringing the technology to a few of the flex-rigid shops because there was some interest in it. We did some presentations showing the technology and manufacturing requirements.

A lot of the PCB manufacturers started out as screen printers as their process for inner layer and outer layer processing. They started with screens. Twenty to 30 years ago, you would go into a shop and there would be a screening room with 30 screeners in there because they didn't have the dry film technology or the other processes. So, there was interest. Not to say that anybody has gone that route. I think they just understand it. There is some crossover between the two types of companies—printed electronics and printed circuit boards. We are seeing more interest in some of the additive processes that we can provide because it takes out many of the difficult process steps in building a printed circuit board.

Shaughnessy: But for a traditional PCB shop, what is the cost and steps of entry into PEC? What does that look like?

Miller: From my standpoint, you would start by hiring an individual who has spent many years working in a printed electronics facility. As far as capital equipment, they already have a lot of the equipment inside their printed circuit board shops. They all make their screens and put legend on the boards, so hey have some of this equipment and understand it. It would be a more sophisticated screening process for the shop, and they would need to upgrade this process. Of course, there are needs like the big dryer. Also, the amount of dollars it would take to get into it would depend upon whether they wanted to build prototypes or volume. Of course, they would need to have the footprint available to expand.

Shaughnessy: Right. There are so many myths floating around about this, and we get asked, "How do I get involved with this?"

Miller: It's not an easy process. It's not like these guys just pick somebody off the street to run this place. There's a lot of technology that you must learn and there are investments. They're no different than the guys that have invested in printed circuit board shops. The biggest difference is that they're probably not dealing with the plating chemistry and waste treatment that these circuit board guys are doing, because it's all additive. I'm sure that there's some of that, but it's not like a PCB shop.

Shaughnessy: Are the standards keeping up with printed electronics?

Bianchi: Yes. There were a lot of ASTM standards for membrane switches that apply to the different use cases for printed electronics. Many are applicable from the actual curing of the ink and dielectric and adhesion properties to the base substrate. But from a finished product standpoint, our perspective is that the OEM is the one that validates the finished product and to the standards that they need.

We can provide the screen-printing standards. I don't know what your opinion is on that, Mike. I haven't been at any standards meetings in a long time.

Wagner: The last one was probably 2018 or 2019. I served on the committee, and then we just started transitioning into anything new in printed electronics. But it's so diverse compared to the general membrane switch standards. No standards really exist right now for wearables or stretchable type things. That's a totally different set of variables.

There are different folks in those industries working on those. IPC was involved, as well as some of the textile associations. Outside of printing, they're also looking for the same threads, conductive threads, and things that they might be knitting with. So, they need standards for the garment itself, whether it's knitted into the garment or applied through a TPU process. It's not where it needs to be, because there are so many unique applications. I don't know if they can harness all of that. They must take it chunk by chunk. As Tom said, we see more customers doing the end validation and testing, maybe asking, "Can it do this?" Then we take it from there.

Shaughnessy: When PEC first started to spread, the OEMs seemed to be the ones driving the development. When customers come to you, are they knowledgeable about PEC or do you have to educate them?

Bianchi: We see both cases. For some customers, we demonstrate the technology, and they find an application for it. There are other times they come up and say, "This is how we want to use your printed electronics technology." We help them design for manufacturability. Sometimes, they can come up with a concept, but they don't always engineer it to be cost-effective manufacturing.

Wagner: I concur with Tom. From the stretchable and wearable standpoint, some customers, say from a printed heater standpoint, don't know that the technology even exists—that I can print a heater that stretches. They're still hung up on different technologies to do that. So, educating them is important. Then they can approach you and say, "Hey, this is what we're trying to accomplish. Can your technology work in this application?" Then we start the dialogue.


John Voultos

John Voultos: It's still about education. The average electrical engineer graduates from college and they understand all about a printed circuit board. But flexible circuits, printed electronics—we must elevate our game for those who are graduating. As they're joining EMS companies, make them aware that this technology exists and why they should be considering and evaluating it. It should not be a solution, "Oh, by the way, we need to use printed electronics. Oh, by the way, we need to use a flexible circuit, etched copper." We should be educating the industry, the young. It's about form-fit-function.

Mike Wagner

Tom Bianchi

Shaughnessy: I've been trying to find a speaker with a lot of experience in PEC, a Rick Hartley or Lee Ritchey in this segment. There doesn't seem to be a lot of PEC instructors. Where would you tell somebody new to PEC to find resources: classes, books, websites, etc.?

Bianchi: Well, three of us at least are on the printed electronics committee for the PRINTING United Alliance, which is the printed electronics and industrial applications group. This is an association that can certainly direct OEMs and engineers to the right resources for our niche.

Wagner: There are a handful of colleges that have integrated printed electronics into their curriculum. Western Michigan has the Center for the Advancement of Printed Electronics. I've been there and seen their curriculum. They're teaching people how to make screens for printing conductive inks, studying how gravure works, and how flex works with conductive ink. They have the equipment. They're doing research projects for the government and corporations. There are some people in the industry that have graduated from that program who are working in printed electronics. There are others, like Cal Poly, Clemson, and maybe Georgia Tech.

Bianchi: And at UMass-Lowell there's the Printed Electronics Research Collaboration.

Miller: Not only do they have printed electronics, but they have a plastics degree as well. They put them together on different types of products. We've been in contact with them.

Bianchi: I think Kevin and Insulectro probably have done the best merge on those PCB manufacturing and printed electronics manufacturing. When Insulectro first started promoting, I said, "Who is Insulectro?" They were so heavily PCB-based but now, they're a staple in the printed electronics world. DESIGN007

Stay tuned for Part 3 in the October issue of *Design007 Magazine*.

TOP TEN
EDITOR'S PICKS

HyperLynx: There's an App for That

Todd Westerhoff, product marketing manager for signal integrity software, talks about a new capability called HyperLynx Apps that offers a new take on traditional signal and power integrity analysis, and how that fits in with the Siemens plan to put SI and PI tools into the hands of more designers early in the design cycle.

John Watson Wants You—to Sign Up for His PCB Design Class


John Watson

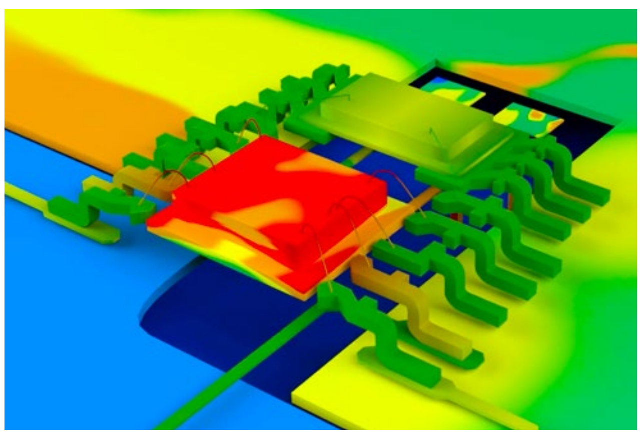
I-Connect007 columnist John Watson is teaching an introductory class on PCB design at Palomar College this fall. It's an online class, so you don't have to live in San Diego to attend. In this interview, John talks about the genesis for the class and its benefits.

Fein-Lines: Live in Person—PCEA/Orange County Meets Up Again

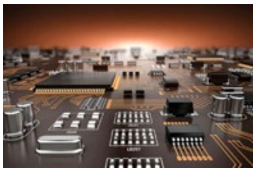
If the United States is ever going to regain some of its historically large PCB fab market share, design and fabrication know-how must play a significant role. "Fabricators are an important part of the design with manufacturing practice," says Scott McCurdy, president of the Orange County chapter of the Printed Circuit Engineering Association (PCEA), which recently had its first live event in more than two years.

Webinar Review: Thermal Integrity of High-Performance PCB Design

In a new CadenceTECHTALK webinar, "How Static and Dynamic IR Drop Analysis Can Help PCB Designs and Challenges," product manager Melika Roshandell and SerDes SI/PI engineer Karthik Mahesh Rao explain how the EE and ME can both use the Celsius Thermal Solver to achieve their disparate objectives.



DFM 101: Final Finishes—ENIG and ENIPIG



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

Dana on Data: Time for a Data Format Revolution

Starting in the 1950s, the Gerber data format, complemented with several paper and electronic files, was used to transfer the physical PCB data from designers to fabricators and assemblers. RS-274-D and RS-274X gave us incremental improvements to the Gerber format, but still required several additional files to transfer all the data. The 274X format is still the most predominant data transfer package in use today.

Connect the Dots: Controlled Impedance—The Devil is in the (Math) Details

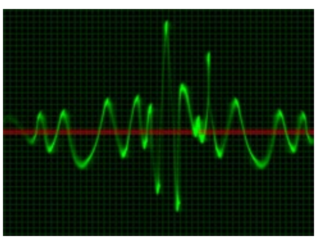
Controlling impedance is critical to signal integrity and board performance in devices powering everything from high-speed digital applications to telecom and RF communication. It is common practice for designers to include impedance-related notes with their PCB designs and rely on the manufacturer to determine the proper trace parameters.

Excerpt: *'The Printed Circuit Designer's Guide to… Thermal Management with Insulated Metal Substrates, Vol. 2'*

As the second in this two-part series, *The Printed Circuit Designer's Guide to… Thermal Management with Insulated Metal Substrates, Volume 2,* by Didier Mauve and Robert Art, builds on the material presented in the first book by describing up-to-the-minute products and design techniques for thermal management with IMS.

The Pulse: Field Solver Finesse for Modelling Transmission Lines

When I-Connect007 asked me to contribute for this issue on field solvers, I wondered what more could be added to this extensively discussed subject, but as a supplier and developer of field solvers, Polar still gets asked the same questions both by experienced customers who are perhaps exposed to a new scenario and, as is most welcome, by new entrants to the industry.



Altium, IPC Education Foundation, and Arduino Announce Student Electronics Design Competition

Altium LLC is partnering with the IPC Education Foundation (IPCEF) and Arduino to launch the second annual student electronics design challenge. This competition aims to engage, educate, and enhance PCB design capabilities while developing STEM solutions to environmental challenges.

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

Career Opportunities

EMA Design Automation®

Technical Marketing Engineer

EMA Design Automation, a leader in product development solutions, is in search of a detail-oriented individual who can apply their knowledge of electrical design and CAD software to assist marketing in the creation of videos, training materials, blog posts, and more. This Technical Marketing Engineer role is ideal for analytical problem-solvers who enjoy educating and teaching others.

Requirements:

- Bachelor's degree in electrical engineering or related field with a basic understanding of engineering theories and terminology required
- Basic knowledge of schematic design, PCB design, and simulation with experience in OrCAD or Allegro preferred
- Candidates must possess excellent writing skills with an understanding of sentence structure and grammar
- Basic knowledge of video editing and experience using Camtasia or Adobe Premiere Pro is preferred but not required
- Must be able to collaborate well with others and have excellent written and verbal communication skills for this remote position

EMA Design Automation is a small, family-owned company that fosters a flexible, collaborative environment and promotes professional growth.

Send Resumes to: resumes@ema-eda.com

apply now

TAIYO AMERICA

Field Service Technician

Taiyo Circuit Automation designs and manufactures the world's finest dual sided soldermask coating and vertical drying equipment. Since 1981, we have served the printed circuit board industry with highly reliable innovative machinery, engineered to exceed.

PRIMARY FUNCTION:

The Field Service Technician is responsible for troubleshooting and providing technical services on Taiyo Circuit Automation's mechanical and electro-mechanical products and systems.

ESSENTIAL DUTIES:

1. Identify mechanical issues and implement process control solutions for process improvement and new projects
2. Consult with maintenance, operations, engineering, and management concerning process control and instrumentation
3. Specify, install, configure, calibrate, and maintain instrumentation, control system and electrical protection equipment

QUALIFICATIONS/SKILLS:

1. 3 years of experience with equipment, preferably in PCB or related electronics industry
2. 3 years of experience in similar process industries with hands-on experience in operations, maintenance and project implementation—OMRON, Koyo, Allen Bradley experience preferred
3. Experience in installation and calibration of process control elements and electrical measurement devices
4. The ability to read and understand electrical, pneumatic diagrams and control systems

REQUIRED EDUCATION/EXPERIENCE:

1. High school graduate
2. Associate degree in Industrial Engineering Technology, Mechanical or Electrical Engineering, preferred.
3. PLC experience

Email: BobW@Taiyo-america.com (Subject: "Application for Field Service Technician for TCA")

apply now

Career Opportunities

Altium®

DevOps Engineer

Altium is a publicly traded global company responsible for the most widely used PCB design software in the industry. Altium 365® is our cloud-based design and collaboration platform; it gives more power to every contributor in the electronics product chain, from the PCB designers to manufacturing. Our R&D teams are the driving force behind Altium 365 and all our technological accomplishments.

- The primary role of the DevOps Engineer is to help continue our transition to a cloud-based SaaS model as part of the production engineering team
- The team's top priorities are product reliability, security, feature delivery, and automation
- DevOps is responsible for the CI/CD process, streamlining automation for provisioning and deployment, scalable infrastructure, uninterrupted service, other DevOps activities

Required Skills and Experience:

- Analysis, troubleshooting
- 4+ years' DevOps/SRE/ Linux/Windows
- AWS (EC2, RDS, S3, Storage, Route53, and network appliances
- Architecting and securing cloud networking

Altium offers a culture built and managed by engineers. We don't micromanage; we define the goals and give engineers the freedom and support to explore new ideas for delivering results. In doing so, we all have a hand in shaping the future of technology.

https://careers.altium.com/

apply now

INDIUM CORPORATION®

Supplier Quality Manager

Headquarters, New Hartford, NY

JOB SUMMARY:
The Supplier Quality Manager is responsible for maintaining and improving the quality of Indium Corporation's supplier base as well as compliance with identified quality standards and risk mitigation. This position will work cross-functionally with Supply Chain, Operations, and our suppliers. The role will ensure that the quality levels of all Indium Corporation suppliers and products meet customer requirements while supporting the company's growth, vision, and values.

REQUIREMENTS:

- Bachelor's degree in business, supply chain or a science-based discipline
- Minimum 3 years in a supply chain role supporting or leading supplier quality
- Obtain and/or maintain International Automotive Task Force (IATF) auditor certification within first 3 months of employment
- Able to work independently or lead a team, as needed, to meet goals
- Excellent oral and written communication skills
- Knowledge of quality standards
- Proficiency in MS Office

apply now

Career Opportunities

Koh Young America

Technical Service & Applications Engineer

Full-Time — Midwest (WI, IL, MI)

Koh Young Technology, founded in 2002 in Seoul, South Korea, is the world leader in 3D measurement-based inspection technology for electronics manufacturing. Located in Duluth, GA, Koh Young America has been serving its partners since 2010 and is expanding the team with an Applications Engineer to provide helpdesk support by delivering guidance on operation, maintenance, and programming remotely or on-site.

Responsibilities

- Provide support, preventive and corrective maintenance, process audits, and related services
- Train users on proper operation, maintenance, programming, and best practices
- Recommend and oversee operational, process, or other performance improvements
- Effectively troubleshoot and resolve machine, system, and process issues

Skills and Qualifications

- Bachelor's in a technical discipline, relevant Associate's, or equivalent vocational or military training
- Knowledge of electronics manufacturing, robotics, PCB assembly, and/or AI; 2-4 years of experience
- SPI/AOI programming, operation, and maintenance experience preferred
- 75% domestic and international travel (valid U.S. or Canadian passport, required)
- Able to work effectively and independently with minimal supervision
- Able to readily understand and interpret detailed documents, drawings, and specifications

Benefits

- Health/Dental/Vision/Life Insurance with no employee premium (including dependent coverage)
- 401K retirement plan
- Generous PTO and paid holidays

apply now

Flexible Circuit Technologies

Electrical Engineer/PCB/CAD Design, BOM/Component & Quality Support

Flexible Circuit Technologies (FCT) is a premier global provider of flex, rigid flex, flex heaters, EMS assembly and product box builds.

Responsibilities:

- Learn the properties, applications, advantages/disadvantages of flex circuits
- Learn the intricacies of flex circuit layout best practices
- Learn IPC guidelines: flex circuits/assemblies
- Create flexible printed circuit board designs/files to meet customer requirements
- Review customer prints and Gerber files to ensure they meet manufacturing and IPC requirements
- Review mechanical designs, circuit requirements, assembly requirements, BOM/component needs/ and help to identify alternates, if needed
- Prepare and document changes to customer prints/files.
- Work with application engineers, customers, and manufacturing engineers to finalize and optimize designs for manufacturing
- Work with quality manager to learn quality systems, requirements, and support manager with assistance

Qualifications:

- Electrical Engineering Degree with 2+ years of CAD/PCB design experience
- IPC CID or CID+ certification or desire to obtain
- Knowledge of flexible PCB materials, properties, or willingness to learn
- Experience with CAD software: Altium, or other
- Knowledge of IPC standards for PCB industry, or willingness to learn
- Microsoft Office products

FCT offers competitive salary, bonus program, benefits package, and an outstanding long-term opportunity. Location: Minneapolis, Minn., area.

apply now

Career Opportunities

KYZEN
Where Science and Care Converge.

Regional Manager

Midwest Region

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

DETAILS OF FUNCTION:

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

QUALIFICATIONS:

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

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

fernando_rueda@kyzen.com

apply now

pluritec®
MACHINES FOR PRINTED CIRCUIT BOARDS

Field Service Engineer

Location: West Coast, Midwest

Pluritec North America, ltd., an innovative leader in drilling, routing, and automated inspection in the printed circuit board industry, is seeking a full-time field service engineer.

This individual will support service for North America in printed circuit board drill/routing and X-ray inspection equipment.

Duties included: Installation, training, maintenance, and repair. Must be able to troubleshoot electrical and mechanical issues in the field as well as calibrate products, perform modifications and retrofits. Diagnose effectively with customer via telephone support. Assist in optimization of machine operations.

A technical degree is preferred, along with strong verbal and written communication skills. Read and interpret schematics, collect data, write technical reports.

Valid driver's license is required, as well as a passport, and major credit card for travel.

Must be able to travel extensively.

apply now

Career Opportunities

Manncorp™

SMT Field Technician
Hatboro, PA

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

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

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

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

apply now

ventec INTERNATIONAL GROUP 騰輝電子

European Product Manager
Taiyo Inks, Germany

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

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

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

What's on offer:
- Salary & sales commission--competitive and commensurate with experience
- Pension and health insurance following satisfactory probation
- Company car or car allowance

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

apply now

Career Opportunities

Prototron Circuits

Sales Representatives

Prototron Circuits, a market-leading, quick-turn PCB manufacturer located in Tucson, AZ, is looking for sales representatives for the New England and Northern California territories. With 35+ years of experience, our PCB manufacturing capabilities reach far beyond that of your typical fabricator.

Reasons you should work with Prototron:

- Solid reputation for on-time delivery (98+% on-time)
- Capacity for growth
- Excellent quality
- Production quality quick-turn services in as little as 24 hours
- 5-day standard lead time
- RF/microwave and special materials
- AS9100D
- MIL-PRF- 31032
- ITAR
- Global sourcing option (Taiwan)
- Engineering consultation, impedance modeling
- Completely customer focused team

Interested? Please contact Russ Adams at (206) 351-0281 or russa@prototron.com.

apply now

MivaTek Global

Field Service Technician

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

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

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

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

More About Us

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

apply now

Career Opportunities

INSULECTRO

Are You Our Next Superstar?!

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

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

apply now

eptac

TRAIN. WORK SMARTER. SUCCEED.

Become a Certified IPC Master Instructor

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

Qualifications and skills

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

Benefits

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

apply now

Career Opportunities

Career Opportunities

ARLON
ELECTRONIC MATERIALS

Arlon EMD, located in Rancho Cucamonga, California, is currently interviewing candidates for open positions in:

- **Engineering**
- **Quality**
- **Various Manufacturing**

All interested candidates should contact Arlon's HR department at 909-987-9533 or email resumes to careers.ranch@arlonemd.com.

Arlon is a major manufacturer of specialty high-performance laminate and prepreg materials for use in a wide variety of printed circuit board applications. Arlon specializes in thermoset resin technology, including polyimide, high Tg multifunctional epoxy, and low loss thermoset laminate and prepreg systems. These resin systems are available on a variety of substrates, including woven glass and non-woven aramid. Typical applications for these materials include advanced commercial and military electronics such as avionics, semiconductor testing, heat sink bonding, High Density Interconnect (HDI) and microvia PCBs (i.e. in mobile communication products).

Our facility employs state of the art production equipment engineered to provide cost-effective and flexible manufacturing capacity allowing us to respond quickly to customer requirements while meeting the most stringent quality and tolerance demands. Our manufacturing site is ISO 9001: 2015 registered, and through rigorous quality control practices and commitment to continual improvement, we are dedicated to meeting and exceeding our customers' requirements.

For additional information please visit our website at www.arlonemd.com

apply now

BLACKFOX®
Premier Training & Certification

IPC Instructor

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

Independent contractor, possible full-time employment

Job Description

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

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

Qualifications

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

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

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

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

apply now

Career Opportunities

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

CAD/CAM Engineer

Summary of Functions

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

Essential Duties and Responsibilities

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

Organizational Relationship

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

Qualifications

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

Physical Demands

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

apply now

U.S. CIRCUIT

Plating Supervisor

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

Competitive benefits package.
Pay will be commensurate
with experience.

Mail to:
mfariba@uscircuit.com

apply now

Career Opportunities

APCT
Passion | Commitment | Trust

APCT, Printed Circuit Board Solutions: Opportunities Await

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

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

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

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

apply now

YOUR JOB AD HERE

For information, please contact:
BARB HOCKADAY
barb@iconnect007.com
+1 916.365.1727 (PACIFIC)

I-Connect007
GOOD FOR THE INDUSTRY

Educational Resources

The Electronics Industry's Guide to... The Evolving PCB NPI Process

by Mark Laing and Jeremy Schitter, Siemens Digital Industries Software

The authors of this book take a look at how market changes in the past 15 years, coupled with the current slowdown of production and delivery of materials and components, has affected the process for new product introduction (NPI) in the global marketplace. As a result, companies may need to adapt and take a new direction to navigate and thrive in an uncertain and rapidly evolving future. Learn how to streamline the NPI process and better manage the supply chain. ***Get it Now!***

The Printed Circuit Designer's Guide to...

Thermal Management with Insulated Metal Substrates, Vol. 2

by Didier Mauve and Robert Art, Ventec International Group

This book covers the latest developments in the field of thermal management, particularly in insulated metal substrates, using state-of-the-art products as examples and focusing on specific solutions and enhanced properties of IMS. **Add this essential book to your library.**

High Performance Materials

by Michael Gay, Isola

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

Stackups: The Design within the Design

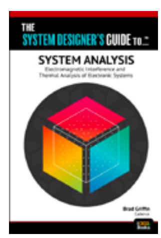
by Bill Hargin, Z-zero

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

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

by Brad Griffin, Cadence

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



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

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

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

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

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

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

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

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

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

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

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

TECHNOLOGY SPECIALIST: **BRYSON MATTIES**

COVER DESIGN: **SHELLY STEIN**

COVER IMAGE: **ADOBE STOCK © KNUT**

DESIGN007 MAGAZINE

DESIGN007 MAGAZINE®
is published by IPC Publishing Group,
3000 Lakeside Dr., Suite 105N, Bannockburn, IL 60015

September 2022, Volume 11, Number 9
DESIGN007 MAGAZINE is published monthly,
by IPC Publishing Group

ADVERTISER INDEX


Problems solved!
START PLANNING NOW WITH OUR
IPC APEX EXPO PRESHOW COVERAGE
Cybersecurity & Counterfeit
PCB007 MAGAZINE
JUNE 2022
PLATING
PCB007 MAGAZINE
JULY 2022
PCB007 MAGAZINE
AUGUST 2022
DESIGN007 MAGAZINE
JULY 2022
HURTS
DESIGN007 MAGAZINE
AUGUST 2022
LESS IS MORE
DESIGN007 MAGAZINE
SEPTEMBER 2022
THE SUPPLY PAIN SCALE
JULY 2022
SMT007 MAGAZINE
AUGUST 2022
MANAGING THE SUPPLY CHAIN PAIN
SMT007 MAGAZINE
SEPTEMBER 2022
SEEING THE LIGHT
SUBSCRIBE NOW