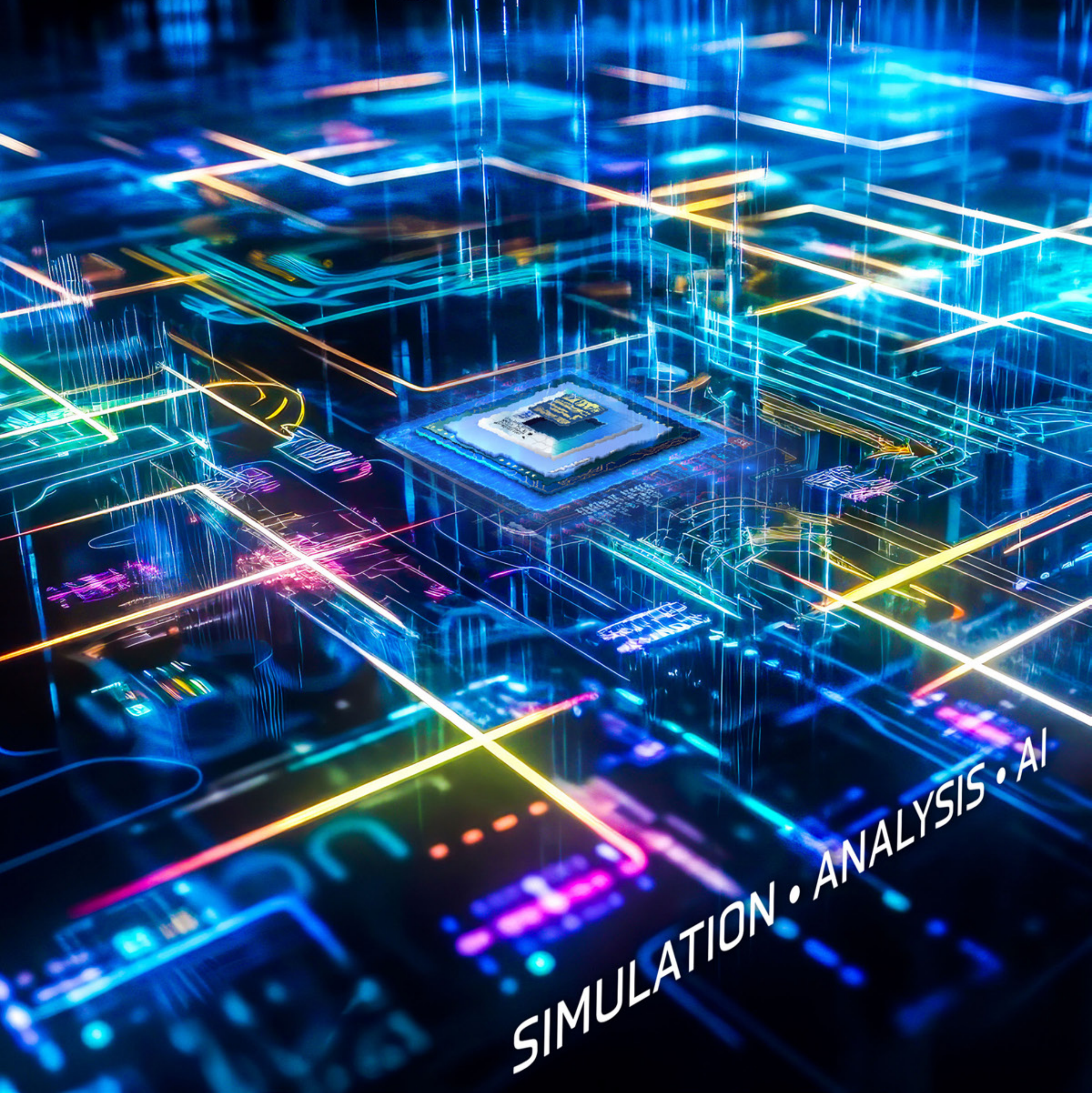
AUGUST 2023

I-Connect007

DESIGN007 MAGAZINE



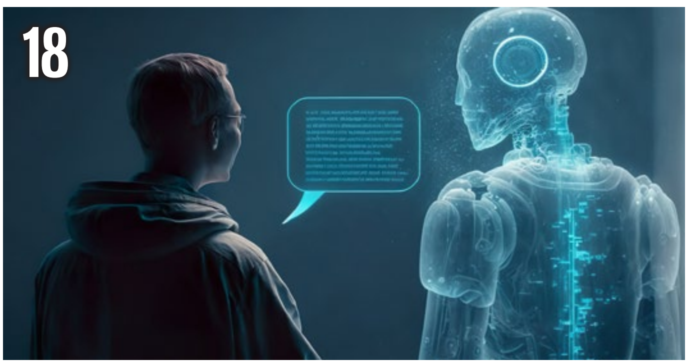
SIMULATION • ANALYSIS • AI

DESIGN007 MAGAZINE

Simulation, Analysis, and AI

As we learn in this month's issue, AI is already making inroads into the design cycle—and learning from its successes and missteps. Today's simulation tools are now benefiting from machine learning and artificial intelligence. We're still a long way from "push-button" operation, but as AI becomes more "intelligent" and learns from its design mistakes, there's almost no limit to what AI could potentially offer PCB designers and engineers in the not-too-distant future.

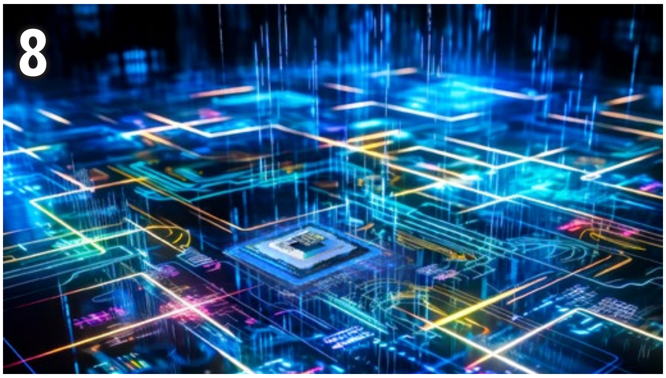
FEATURE ARTICLES



FEATURE INTERVIEWS

FEATURE COLUMNS

DESIGN007 MAGAZINE



COLUMNS

SHORTS

HIGHLIGHTS

DEPARTMENTS

Simulation, Analysis, and AI

The Shaughnessy Report

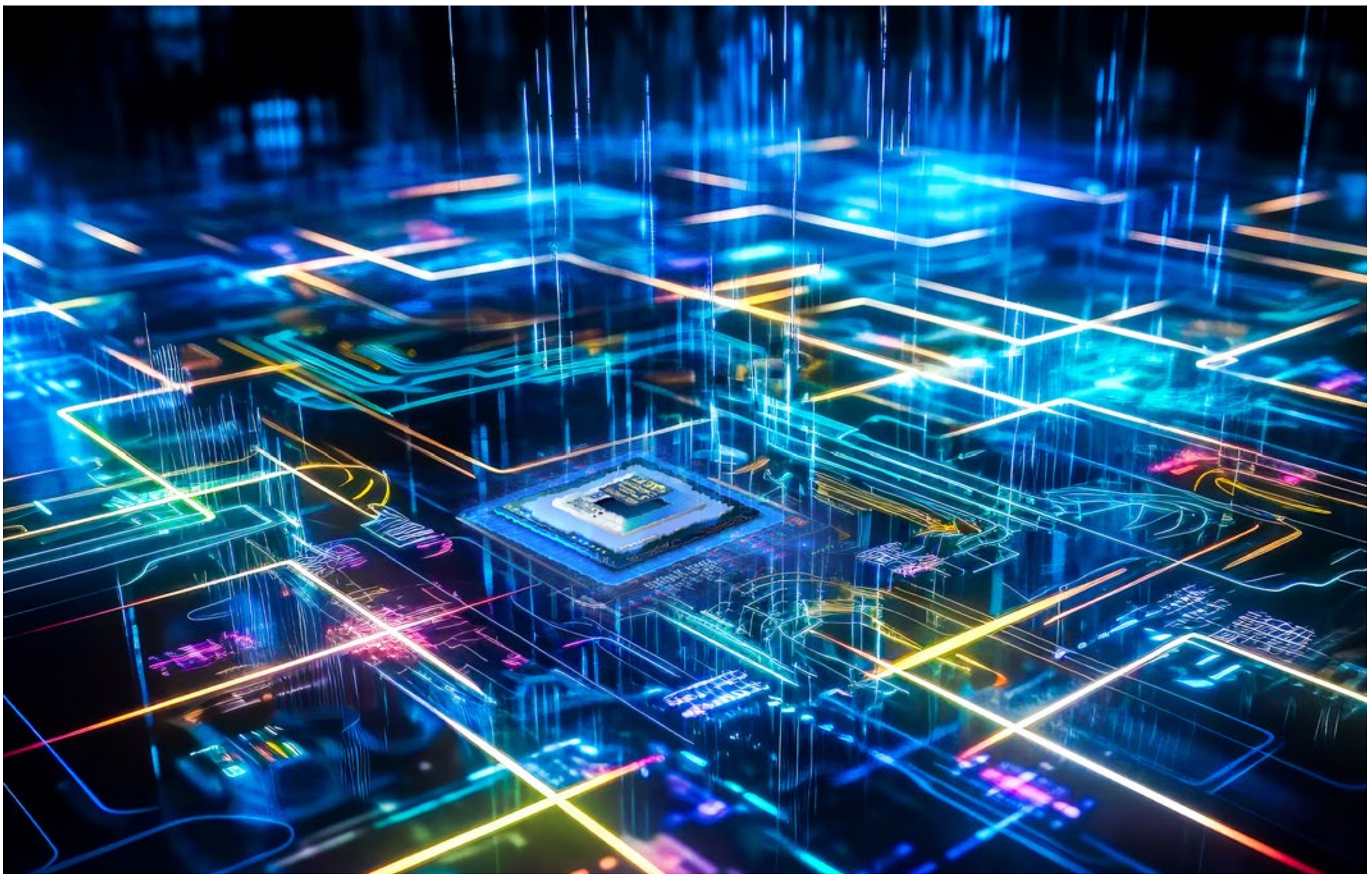
by Andy Shaughnessy, I-CONNECT007

Getting today's designs "right the first time" is critical, especially with costly advanced PCBs. Companies are slowly realizing that building two, three, or seven respins into the process—and budget—is like flushing cash down the toilet.

I can hear some of you thinking, "But that's how we've always done it. Why don't you knock it off with those negative waves?"

But even one respin is a waste of money, not to mention time and manpower. Multiple respins can be expensive, especially with complex high-speed designs. And respins are entirely preventable with a little extra effort in the design cycle.

Simulation and analysis software tools can help you in the fight to eliminate respins. They're not magical, but they can predict the future of your design. If you're designing high-performance, high-speed boards without using these tools, I predict that you're on your way to respins.



As we learn in this month's issue, AI is already making inroads into the design cycle—and learning from its successes and missteps. Today's simulation tools are now benefiting from machine learning and artificial intelligence. We're still a long way from "push-button" operation, but as AI becomes more "intelligent" and learns from its design mistakes, there's almost no limit to what AI could potentially offer PCB designers and engineers in the not-too-distant future.

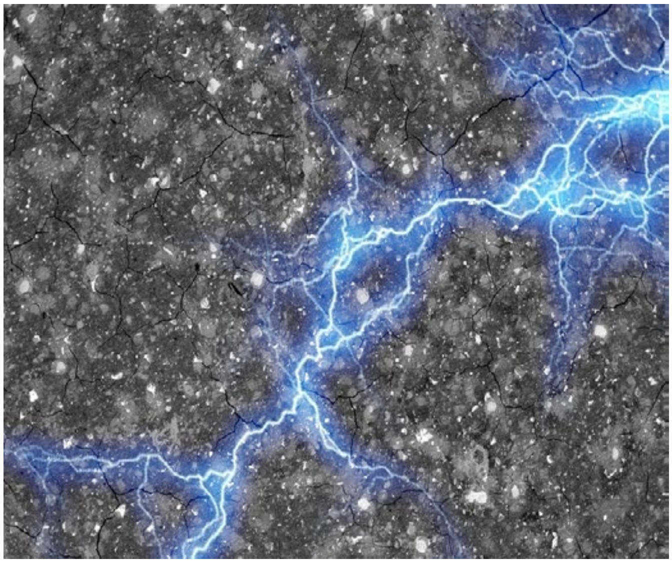
In his feature column, Barry Olney and guest Charles Pfeil discuss the integration of AI into the PCB design flow. David Wiens explains the relationship between PCB design and the digital twin, and why designers have been using digital twin for decades—whether they realize it or not. Sheldon Fernandez lays out the potential that AI holds for PCB design tools, and the difference between AI and machine learning. Brad Griffin discusses several changes that he would like to see in the PCB design flow, especially in signal and power integrity tools.

Yuriy Shlepnev walks us through some of the latest advances in 3D electromagnetic analysis tools and the challenges that they address. John Watson discusses the future of AI for circuit simulation tools, and he provides details of Altium's SPICE tool, which the company launched several years ago. Steve Watt focuses on the simulation and analysis challenges of today. Finally, Chris DeMartino explains the business "model" for Modelithics, which provides proven simulation models and services.

Who knows? Maybe AI in simulation tools will bring us closer to zero respins. **DESIGN007**

Andy Shaughnessy is managing editor of *Design007 Magazine.* He has been covering PCB design for 23 years. To read past columns, click here.

MIT Engineers Create An Energy-storing Supercapacitor from Ancient Materials



Two of humanity's most ubiquitous historical materials, cement and carbon black (which resembles very fine charcoal), may form the basis for a novel, low-cost energy storage system, according to a new study. The technology could facilitate the use of renewable energy sources such as solar, wind, and tidal power by allowing energy networks to remain stable despite fluctuations in renewable energy supply.

The two materials, the researchers found, can be combined with water to make a supercapacitor—an alternative to batteries—that could provide storage of electrical energy. As an example, the MIT researchers who developed the system say that their supercapacitor could eventually be incorporated into the concrete foundation of a house, where it could store a full day's worth of energy while adding little (or nothing) to the cost of the foundation and still provide the needed structural strength. The researchers also envision a concrete roadway that could provide contactless recharging for electric cars as they travel over that road.

Initial uses of the technology might be for isolated homes or buildings or shelters far from grid power, which could be powered by solar panels attached to the cement supercapacitors, the researchers say.

(Source: MIT News)

Integrating AI Into PCB Design Flow

Beyond Design

Feature Column by Barry Olney, IN-CIRCUIT DESIGN PTY LTD / AUSTRALIA
with Special Advisor Charles Pfeil

Today's PCB design tools are a far cry from the rudimentary tools we used 30 years ago, but even though the algorithms are now quite clever, they are still fairly basic as far as intelligence goes. Comprehensive design rules can be established to accommodate myriad constraints to limit placement and routing which advise us when we have overstepped the boundaries. But what if artificial intelligence (AI) was introduced into the mix? How could it enhance the PCB design environment? What would happen if we applied AI/machine learning techniques as part of the design flow?

PCB designers are used to complex designs and much of the design process is a fairly straightforward engineering challenge. However, the number of options and potential interactions has soared. There are many ways to achieve the same goal, but some ways are better than others in the context of system integrity or a specific application. Still, keeping track of all the possible tradeoffs and options in a complex design is approaching technological singularity—the hypothetical point when computer capacity rivals that of a human brain.

Google, Xilinx, Synopsys, Intel, Cadence, and Siemens are all working on AI applications for silicon design. Recently, machine learning (ML)-based techniques have been efficiently utilized in several applications,

where enhanced learning capability makes them unique to solve any complex/nonlinear problem. IC design has also benefited from ML techniques at different design levels, from device modeling to test of manufactured chips. IC layout is a very labor-intensive task that typically requires iteration. The performance of an IC depends on where things sit relative to each other. The distances between objects, the wire length, capacitance, and the inductance of the interconnects are also important. After the initial layout, the simulated values of variables are back-annotated to the design. This first layout may not be perfect, so further iterations are required. Reducing the number of iterations and, hence, the design cycle time by using AI, can be extremely cost-effective.

Much of the same methodology happens with PCB layout: We do a pre-layout simulation to determine the constraints, a post-layout simulation to verify the layout, and then the results are back-annotated. This process can also take many iterations.

Earlier this year, Cadence announced the release of the Allegro X AI cloud technology. Cadence states that it dramatically reduces design turnaround time (10X) by automating placement, power plane generation, and critical net routing. Cadence has been developing place and route (P&R) tools for IC synthesis for decades and has now adapted the technology for PCB P&R. Shorter interconnects and reduced crossovers are essential for both chip and PCB layout but critical routing incorporating signal integrity and flight time requirements is of greater importance for the PCB.

Unfortunately, most EDA vendors do not have the same resources to pour into R&D as the Big Tech companies. Such financial backing, coupled with creative engineering, reduces time-to-market. So, EDA tools will inevitably take much longer to develop. Then, most EDA companies are primarily interested in doing R&D that increases sales in the next cycle, not long term.

EDA tools will inevitably take much longer to develop.

Currently, EDA tools use algorithms to control auto-placement and routing. This is a set of instructions that a computer program follows to accomplish a task. PCB routers have gone through many different stages of development over the years, from third-party applications that were difficult to learn, use, and interface, to a cohesive, cloud-based, layout/router environment. IC and PCB routing applications have used many of the same algorithms over time, including:

- Lee's Algorithm (maze routing)
- Dijkstra's Algorithm (shortest path)
- Grid-based routing
- Rip-up and reroute
- Shape-based, push-and-shove
- Steiner Tree (rectilinear routing)
- Heuristic (topology) routing

It is not so much the algorithms that differ, but rather the environment that dictates which algorithm is most effective.

With the advent of AI, machine learning applies artificial intelligence that provides systems with the ability to automatically learn and improve from experience without being explicitly programmed. This is sort of like how proficient PCB designers know how to best tackle a complex task: Experience is the best teacher of all. These algorithms are procedures that are implemented in code and run-on data. Machine learning models are output by algorithms and are comprised of model data and a prediction algorithm. Machine learning algorithms provide a type of automatic programming where machine learning models repre-

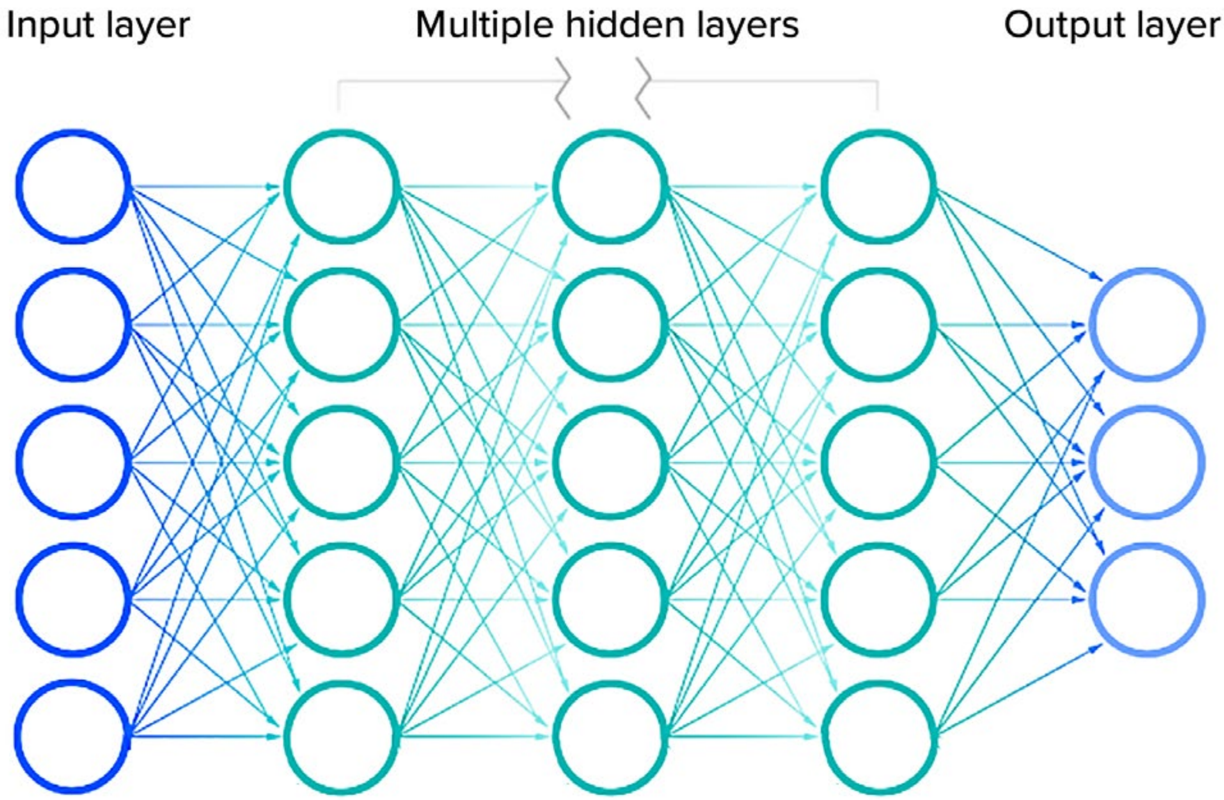


Figure 1: Machine learning model. (Source: IBM)

sent the program. In other words, algorithms are the building blocks that make up machine learning and artificial intelligence.

A machine learning model (Figure 1) is a model that has been trained to recognize certain types of patterns. A model is trained over a set of data, providing it with an algorithm that it can use to reason over and learn from those data. Specifically, machine learning models are computer programs that are used to recognize patterns in data or to make predictions. Machine learning models are created from machine learning algorithms. These algorithms adapt, evolve, and improve themselves based on the data they process. The models can be used to make predictions, categorize information, or discover patterns.

How Could AI Improve Placement and Routing of a PCB?

Skilled PCB designers have many years of experience laying out complex designs such as high-speed FPGA and memory circuits, so AI needs to absorb this same information in order to achieve the same (or better) results but in a much shorter timeframe. Data in the form of design rules and images of preferred routing strategies can be fed into the input layer of the machine learning model. Algorithms in the hidden layers of the model can then process this data, and recognize, predict, and create the resultant layout (Figure 2).

Sample images of steadfast routing patterns plus the netlist, appropriate rules for electrical and mechanical constraints, frequently used stackup configurations, system timing, and memory timing requirements (plus sample eye diagrams of working designs) are fed into the model where the learning algorithms recognize and process the images and data to predict an optimal solution.

Machine learning uses two types of techniques: supervised learning, which trains a model on known input and output data so that

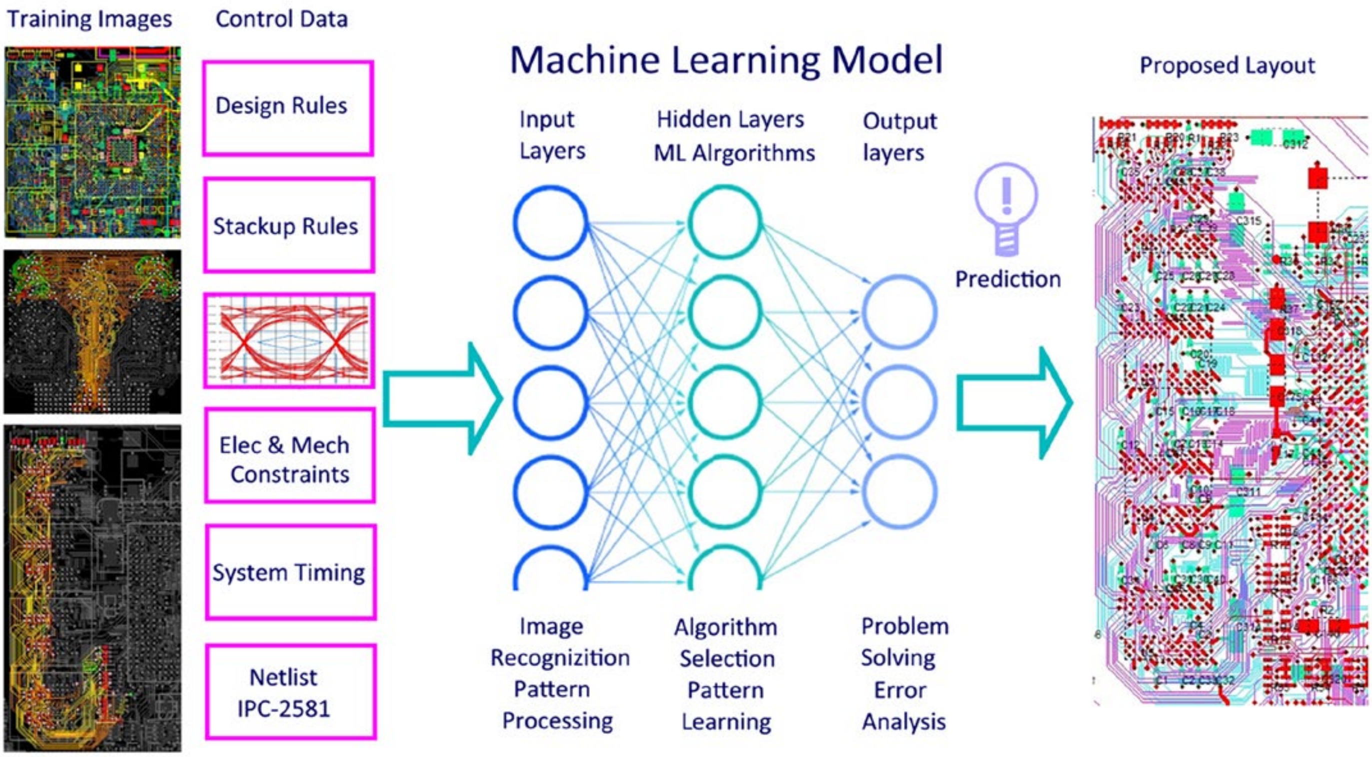


Figure 2: Machine learning model for PCB layout.

it can predict future outputs, and unsupervised learning, which finds hidden patterns or intrinsic structures in input data.

Choosing the right algorithm can seem overwhelming; there are dozens of supervised and unsupervised machine learning algorithms, and each takes a different approach to learning. There is no best method or one size fits all. Finding the right algorithm is partly just trial and error but is also based on the recognition of patterns learned, and experience with similar designs. The algorithm selection also depends on the technology used (e.g., DDR4), frequency, and rise time of the signal.

The biggest issue in training AI with PCB images is the availability of data. ChatGPT had access to the unlimited wealth of information available on the internet, however, PCB models only have access to the images and data fed into the machine learning model which, by comparison, is extremely limited. One way around this would be to create an open-source global repository for PCB layout images and databases that have endorsed functionality. However, convincing customers to share their designs may be difficult as most do not wish to publicly expose their intellectual property.

Key Points

- There are many ways to achieve the same goal, but some ways are better than others in the context of system integrity or a specific application.
- Reducing the number of iterations and, hence, the design cycle time by using AI can be extremely cost-effective.
- Shorter interconnects and reduced crossovers are essential for both chip and PCB layout but critical routing incorporating signal integrity and flight time requirements is of greater importance for the PCB.
- Currently, EDA tools use algorithms to control auto-placement and routing.
- With the advent of AI, machine learning applies artificial intelligence that provides systems the ability to automatically learn and improve from experience without being explicitly programmed.

- Algorithms are the building blocks that make up machine learning and artificial intelligence.
- A machine learning model is a model that has been trained to recognize certain types of patterns.
- AI needs to absorb and learn from information in order to achieve the same (or better) results.
- Finding the right algorithm is partly just trial and error but is also based on the recognition of patterns learned and experience with similar designs. DESIGN007

Resources

- 'Beyond Design: Artificial Intelligence in EDA Tools' by Barry Olney, May 2016, *The PCB Design Magazine*
- "What is artificial intelligence," ibm.com
- "What is Artificial Intelligence," by Alyssa Schroer, May 19, 2023, builtin.com
- "EDA Vendors Widen Use of AI," by Ed Sperling, Sept. 29, 2021, semiengineering.com

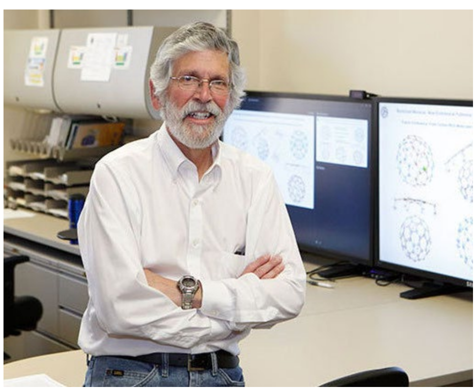
Charles Pfeil worked in the PCB industry for more than 50 years as a designer and owner of a service bureau. Now retired, he is the original product architect for Expedition PCB and an inventor of XtremePCB and more recently enhanced the Altium routing technology

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.

Researchers Achieve Historic Milestone in Energy Capacity of Supercapacitors

In a new landmark chemistry study, researchers describe how they have achieved the highest level of energy storage in a supercapacitor ever recorded.

Supercapacitors store electrical energy between two metal plates that are close together but separated by a surface that cannot conduct electricity. Supercapacitors are similar to batteries, except that batteries store and retrieve energy using chemical transformations, while capacitors store energy by using oppositely charged surfaces.

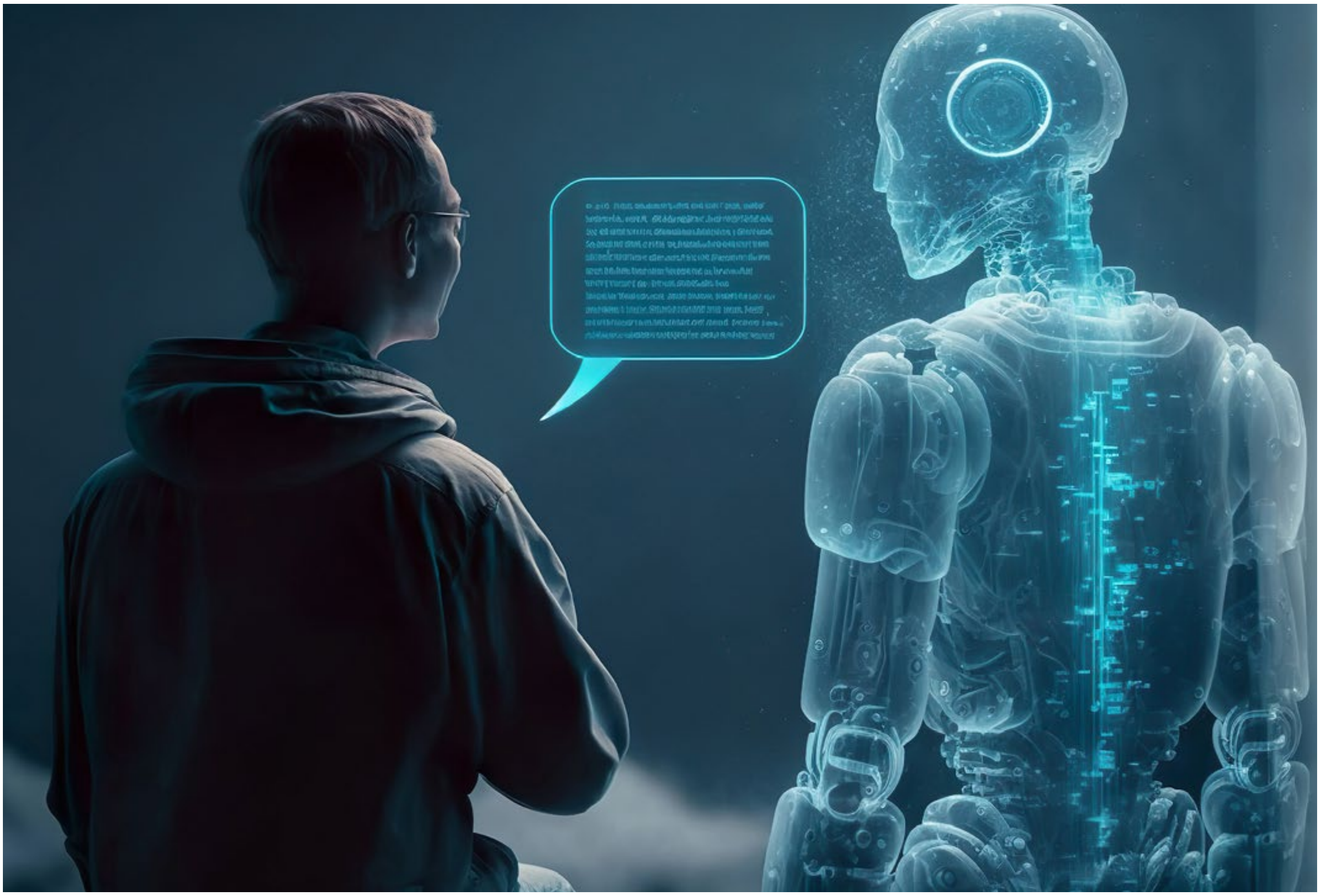


"This is a big step forward and gets us closer to achieving supercapacitors with high energy density, which would radically change how we store and manage energy," said Dr. Luis Echegoyen, a longtime faculty member within the University of Texas at El Paso's Department of Chemistry and Biochemistry.

Supercapacitors have high potential because they can charge much faster than batteries—within seconds to fractions of a second, according to Echegoyen. However, current supercapacitors can only store a low amount of energy, which limits their range of potential applications. If supercapacitors could be designed to store more energy, they would be physically lighter and charge much faster than batteries.

The new supercapacitor designed by Echegoyen and Dr. Marta Plonska-Brzezinska of the Medical University of Bialystok achieved a record level of storage, or capacitance, using a material with a carbon "nano-onion" core structure, which creates multiple pores that allow storage of a greater volume of energy.

(Source: University of Texas at El Paso)



How Far Can We Trust AI?

Feature Interview by the I-Connect007 Editorial Team

Since the advent of computers, engineers have been trying to create systems that "think" for themselves—making that leap from repeating an algorithm to actually inventing one itself. Sheldon Fernandez, CEO of DarwinAI, discusses the difference between true artificial intelligence and machine learning (ML), and whether we can trust what AI gives us. Is AI only as good as the training it's given by a human?

Andy Shaughnessy: Sheldon, what's the cutoff between AI and ML? How do we know?

In the 1950s, the original definition of AI was in asking how to get an artificial entity—something that's not human—to exhibit behavior that we would classify as intelligent. Could it play chess or solve a mathematical equation on its own? It was a general term which the original AI practitioners had. As the AI got more sophisticated, people said those type of questions about chess or mathematics weren't AI because the program wasn't really "thinking."

For example, when IBM's Deep Blue beat Garry Kasparov in chess in 1997, people said, "Well, it may be better than the best human, but it doesn't know that it's playing chess. It doesn't even know what chess is. It's just realizing an algorithm." Now, things are more sophisticated, and it depends on who you're talking to. People in my field wouldn't consider pattern matching or expert systems to be the arti-

Sheldon Fernandez

ficial intelligence we are typically talking about with second- or third-wave machine learning, deep learning, and such. What's the cutoff between these techniques and true artificial intelligence? There's something called artificial general intelligence (AGI)—AI doesn't have to be trained or taught about a task but can solve it anyway.

With ChatGPT, for example, in these very large language models, it's seeing scenarios that it's never seen before and can problem solve against them. A recent research paper from Microsoft makes the argument that you're seeing the glimmers of AGI in the system. Some would say that's the cutoff point, but it's a nebulous term right now.

Nolan Johnson: In an interview with Brad Griffin at Cadence Design Systems, he said they're looking at creating simulation parameters for optimization. Data shows that while a human might put together five or six different scenarios to simulate for fault and signal integrity, the AI generates a couple of dozen additional parameters. At that point, it looks more like basic problem solving. How do we get to a solution?

Think about imagination or creativity; AI thus far can be creative in a derivative sense. You provide it with a theme or boundaries, and it will give you a creative artifact in that genre or within those boundaries. The question that philosophers and practitioners will ask is something called meta creativity: Can AI make an intuitive leap and come up with something new altogether? Einstein came up with the general theory of relativity; Picasso created a form of art called Cubism. Will AI ever do that? Can it look at what you're doing, tell you that your design isn't efficient enough, and then come up with something completely new, a design that hasn't been thought of yet?

Shaughnessy: Some companies are using AI and they say it's definitely a step above machine learning because they've used it to create some products. But why is it so hard to get AI into production? Is it due to a generation gap?

It might be about hiring younger people. They can't imagine a world without AI; it's all they know. But it's more about understanding output. I ask ChatGPT something, but how do I trust that what it's giving me is accurate? There's no way to do that right now.

It's one thing to ask ChatGPT to write you an essay. But when you're asking it to design with certain parameters which will go into a critical system, how do you trust its output? That's a big gap with generative AI right now: There's no way to validate its output other than the human being that has to make sense of what it has generated. That's why I can see there being some skepticism to using this tool in a completely free manner. There has to be certain checks and boundaries around it.

Shaughnessy: There's much that remains to be seen. Sheldon, thank you for your time.

Thank you both. DESIGN007

PCB Design, Digital Twin, and Digital Transformation

Feature Interview by the I-Connect007 Editorial Team

There's been a lot of talk lately about digital twin and its use in PCB fabrication and assembly. Manufacturers have been quick to get onboard the digital twin train.

But what about PCB designers and design engineers? Can the front-end folks benefit from digital twin? We asked David Wiens, Xpedition product manager for Siemens Digital Industries Software, to weigh in on this topic. He's been involved with digital twin for years, and he explained what digital twin can potentially offer to PCB design, and why he believes designers have been using digital twin for decades, whether they realize it or not.

Andy Shaughnessy: David, let's start by sharing your thoughts on PCB designers and digital twin. What exactly is a digital twin?

Designers have always been working with twins of what is manufactured. You know, even designers working with mylar tape had a twin; it was just a physical twin. Since then, we've moved to digital twins and steadily increased the fidelity of that twin. Originally, it was just digitizing, and there was no intelligence. I came all the way from Intergraph, and we started with a mechanical design system. We were digitizing artwork in a mechanical environment, and it didn't have much electrical intelligence. Now, there is a lot of intelligence that comes along with the digital twin, and that enables the engineering team to make much smarter decisions. We wrote a series of columns on digital transformation for *Design007 Magazine* that ended December of last year[1].


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

If you think of a digital twin at the highest level, it's the entire system. A digital twin of an airplane represents the entire airplane, but it also represents all the bits inside, including the electronics, the cabling, and everything else that connects it. A digital twin of a system represents a hierarchy of smaller digital twins. If you want to build a real digital twin, you have to build a digital twin of the environment for that aircraft as well.


David Wiens

If you're at a design shop laying out a PCB and you may now know what it plugs into—okay, fine. That's your end-product; that's your system. But if you ultimately are working at, say, a large mil/aero company, then you're designing everything from electronics all the way up to the full body, working with lots of partners who are sharing their various derivatives of the digital twin with you to help make smarter decisions.

So, you can model digital twins at different levels of abstraction. Now, you would probably not want to do a full thermal model of an entire F-35, all the way down to the individual interconnect on those PCBs. At that scale, it's just not possible to mesh that whole thing out, so they would have to do different levels of abstraction, or create sub models. Even in a PCB, if you wanted to simulate every single interconnect on a board with a few thousand parts and maybe 10,000 connections, that still would take a really long time to simulate all that just for signal integrity, let alone the impact of signal integrity on power integrity, or power integrity on thermal. You have to start thinking about the different disciplines and how they interact together to produce a real digital twin of the design.

Dan Feinberg: I can see how digital twin could be a powerful tool for designers, but I imagine it could be used well beyond the design phase, for example, on the final product.

Absolutely, and the International Space Station is the perfect example. NASA said, "We need to maintain this thing, but we can't bring the space station down every time we want to look at it." What they would really like is a digital twin, so they can at least do root cause analysis to understand some of the problems remotely. If you create a high-fidelity digital twin of the product, then you've got something that you can query when it's actually out in the market. If a product is out there and it's having yield failures, why is that? Well, let's look at the data on what's happening in the real world as far as percentage failures, then look at what's failing, use that data and apply it to the digital twin, and figure out what's going on.

You can also create a digital twin of a production line, for instance, and you can send feedback from the production line back to the product engineering team. If you combine the production line model with the product model to be built, you can use that digital twin to train folks on the manufacturing line. You can model what a human does on the manufacturing line using the product that they're going to build. You know [what I mean]: I take it from this cart, and I put it in this hardware box, and I make sure that my elbows don't hit anything when I move around.

Now, I say this as if we're speaking of a seamless digital thread of technology, but in reality, we have broken digital threads all over the place. The idea is flawed that there's a singular digital twin model that can universally model everything. You have different models for different things: IBIS models for signal integrity, BCI ROM models for thermal, VHDL AMS or SPICE behavioral models, CAD models for footprints and layouts, ODB++ for manufacturing, and many others. Today, the engineer has to take all that disaggregated data and turn it into valuable information that's called the digital twin.

Shaughnessy: Do designers see or call what they are doing digital twin? What's the goal?

It doesn't really matter what they call it. But I bet they are doing some level of physical prototyping. The goal of the digital twin process is zero spin. Now, you could say one prototype spin is really, really good. Most teams say three to four spins is normal. But zero spin is the goal. You design it, you digitally verify it for all kinds of conditions like electrical, thermal, vibration, stress, manufacturability, and then boom, you're off to manufacturing to volume.

Design teams live in a virtual, digital world, so it doesn't bother me that a PCB designer doesn't say, "Hey, I'm doing digital twin." Do I need to convince them that they're doing digital twin to sell my stuff? No. But I would like them to leverage the digital twin they're working with as efficiently as possible. That means verifying the digital twin during the design process, rather than via multiple physical prototype spins. Engineering teams often "bake in failure" by planning for three to four spins. Now, that means you were doing pretty good design work. But three is the average, and a lot of designs go through six or more spins. But the Holy Grail is having teams not only use verification tools during the design process, but also leverage generative technologies and AI to enable a system that can make those decisions for them. Of course, we're years away from that.

Most teams say three to four spins is normal. But zero spin is the goal.

Shaughnessy: Most designers are still using Gerber, and very slowly switching over to other digital data formats like ODB++ and IPC-2581.

Gerber is a digital twin model, it's just a less complete one. ODB++ is an intelligent product model, and IPC-2581 is a fairly complete model as well. That means they take less work on the shop floor to recreate intelligence for manufacturing. Any time you have to re-create something, it's indicative of a broken digital thread, and that's an opportunity to introduce errors and waste time. But those formats still can't model everything in the manufacturing process. They don't have, for instance, the enclosure; it's just the board. It's just one board, so if you have a multi-board system, those are coming across as separate files. A manufacturing engineer often doesn't see the complete digital twin of the system—they have to rely on different models coming across at different times. So intelligent decisions about the complete system are limited.

Shaughnessy: Is there anything that designers can or should do differently regarding digital twin?

Mainly, I would want designers to be aware of the other disciplines that are hanging out around them. And not just the easy one, manufacturing, but also mechanical, cabling, thermal, signal integrity, all of these different things that have an impact on a design, and a digital twin can be built up to verify that. So, it's all about learning to leverage these different models in a way that that can produce a better result in the end. A lot of companies are going through different digital transformations. It's not just in engineering; it may be in their purchasing or manufacturing. But the more seamlessly they do these digital transformations, leveraging these digital twins, the greater the output for the PCB designer.

Barry Matties: I'm curious: Is the decision about digital twin a business office decision? How do they sell it?
I would say that engineers are already using digital twins (the only decision is how efficiently they're going to use them). But yes, management must be involved in the larger digital transformation. The value proposition of digital transformation is usually around quality, time, cost, and different variants of that. In terms of time and cost of the design process, if you adopt a cleaner process, you don't have to do those respins, and that saves you money on prototyping, and it saves you money on the engineering time spent going back and doing those cycles.

That should trigger the competitiveness in any management team, right? You have to start from those high-level benefits and work your way in to sell them on it. Depending on the size of the company, they may laugh at you and say, "Buzz off with your fancy words." But you'll find that really large companies have a digital transformation officer, just like you have a CTO, who is empowered to optimize the processes for the company's benefit.

The digital transformation is continuing, and it's a great time to get out of the box.

Matties: But for companies that aren't that large, is part of the challenge, aside from the business model and skill set model, that management may consider implementation too high of a hurdle?
Sure, and that's where lots of people make good money consulting on best practices. That's why we all engage with analysts and do surveys to quantify the benefits of digital transformation. But it often takes pressure of some form, such as a company facing abject failure, to consider doing something dramatically different.

Matties: Yes, if there's somebody out there to help mitigate the risk to a tolerable level, we'll see quicker transformation.
We may see the Holy Grail that I mentioned. It's a holistic digital twin that would allow you to look at the surface of a jet or car, and just zoom in and look under the covers. It would let me zoom in some more so I can probe the entire model, top to bottom, all the way down to the smallest via. I can't say that we won't see it in our lifetime. We've had some huge innovations in this space.

Shaughnessy: What advice would you give those who are interested in the digital twin?
Gosh, look outside of your box. Have the curiosity to see and evaluate other processes for what they're worth, and what they could bring back to you and your team. Doing the same thing day in, day out is boring. So, look around and see what other teams are doing. The digital transformation is continuing, and it's a great time to get out of the box.

Shaughnessy: Thanks for sharing all of this. I know you've been working on digital twin for quite a while. This was very enlightening.
It's my pleasure, Andy. Thank you. DESIGN007

References

1. "Digital Transformation: Leveraging Digital Twin to Optimize Electronic Systems," by David Wiens, December 2021, *Design007 Magazine* (part of a larger series on digital transformation).

Three Ways to Improve High-speed PCB Signoff, Part 1

Feature Article by Brad Griffin
CADENCE DESIGN SYSTEMS

Signal integrity (SI) and power integrity (PI) are top priorities for engineers designing today's high-speed, high-density circuit boards, and faster signoff of designs can be achieved by uncovering SI/PI issues early in the design process, before costly respins are required. Three key issues engineers need to overcome to sign off on high-speed PCB designs include power analysis, serializer/deserializer (SerDes) link compliance, and double data rate (DDR) memory interface compliance.

The power delivery network (PDN) must be sufficient, efficient, and stable, and the signal quality must meet memory interface and serial link compliance specifications. This article highlights a PCB design methodology that empowers PCB design teams to create successful products on time and on budget without waiting for SI and PI specialists who may not be readily available.

Design Analysis Frameworks

There are several important frameworks to consider when designing PCBs (Figure 1).

The design begins with the schematics, followed by the layout, and then, late in the layout phase, a detailed analysis is done to ensure the layout functions properly. During the design

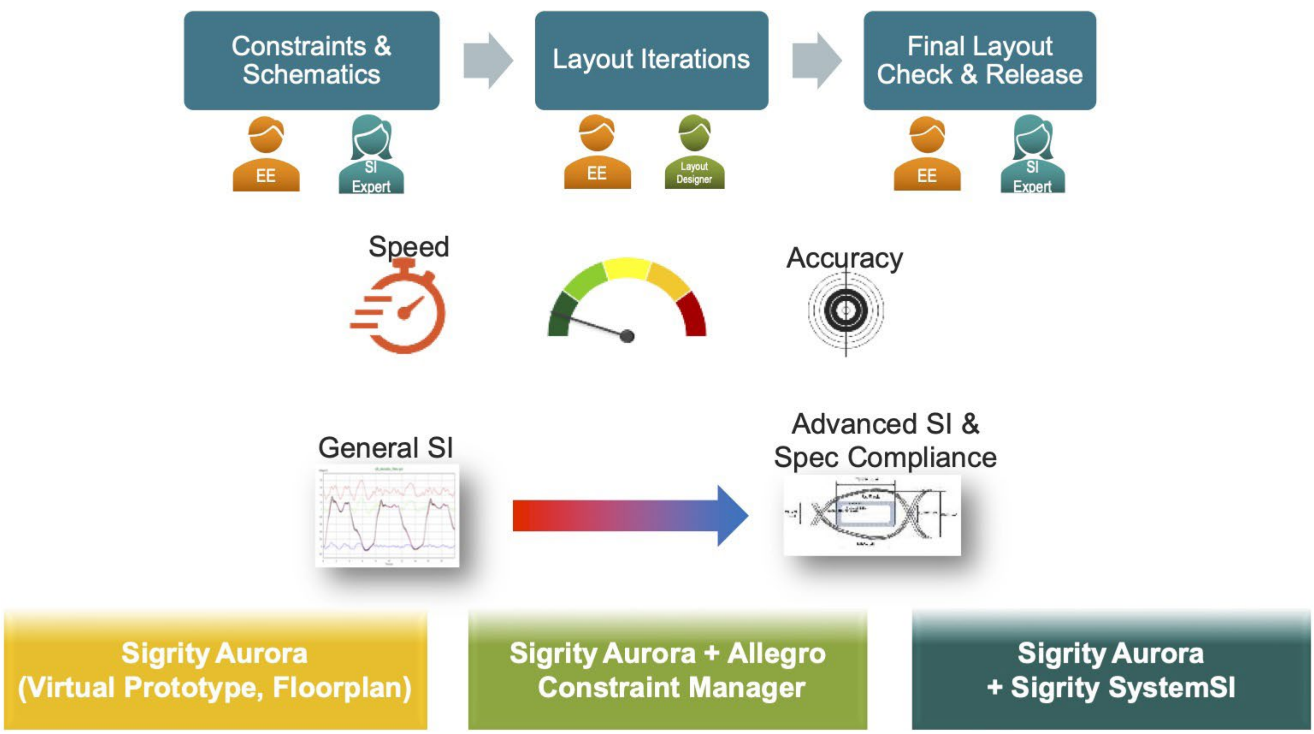


Figure 1: Design analysis frameworks.

cycle tasks, priorities and focus change, but one critical rule of thumb is that the earlier problems are uncovered and corrected, the better.

Electronic product development has traditionally embraced a workflow in which the detailed simulation, analysis, and optimization takes place at a very late stage in the design process, often as the final step of verification and signoff. However, this delay inevitably leads to costly issues that derail budgets and delay time to market. Defects in requirements and performance are uncovered that require additional cycles to address these issues, which could have been discovered and mitigated earlier in the design phase.

To succeed in today's highly competitive electronics markets, simulation and analysis are now being integrated from the earliest stages of the design process using an in-design analysis (IDA) methodology, where analysis and verification are moved from an afterthought in the workflow to an integral part of each phase of the design process at the chip, package, board, and complete system level. Figure 2 provides an example of IDA in the Allegro PCB Designer layout environment integrated with the Sigrity Aurora PCB analysis software.

The design team is an important factor in the process. Early in the design, SI engineers are often brought in to help with understanding design constraints and other issues related to SI. During the layout phase, a layout design expert will provide input. During the final layout verification phase, the SI engineer will be involved. Depending on the organization and time schedules, the SI expert may not be readily available, but the PCB designer is still responsible for delivering the design on schedule and working properly. Therefore, it is desirable for the PCB designer to be able to perform general SI/PI simulation independently of the SI/PI simulation experts. Embedding simulation workflows directly in the PCB design environment empowers the designer and helps ensure the design is correct and will be delivered on time.

Quick answers to general SI/PI issues such as crosstalk and impedance matching are often needed early in the design. Later, the detailed accuracy of the simulation is important, but in the beginning, designers usually just need to make sure they are on the right track. Simulation accuracy can often be at odds with simulation speed, requiring the engineer to make tradeoffs. If detailed accuracy is important at the start, the designer could reduce the scope of the work and consider just one piece of the design. In the event that the entire design must have an accurate simulation early on, analysis workflows can utilize sophisticated distributed computing that enables designers to achieve both speed and accuracy.

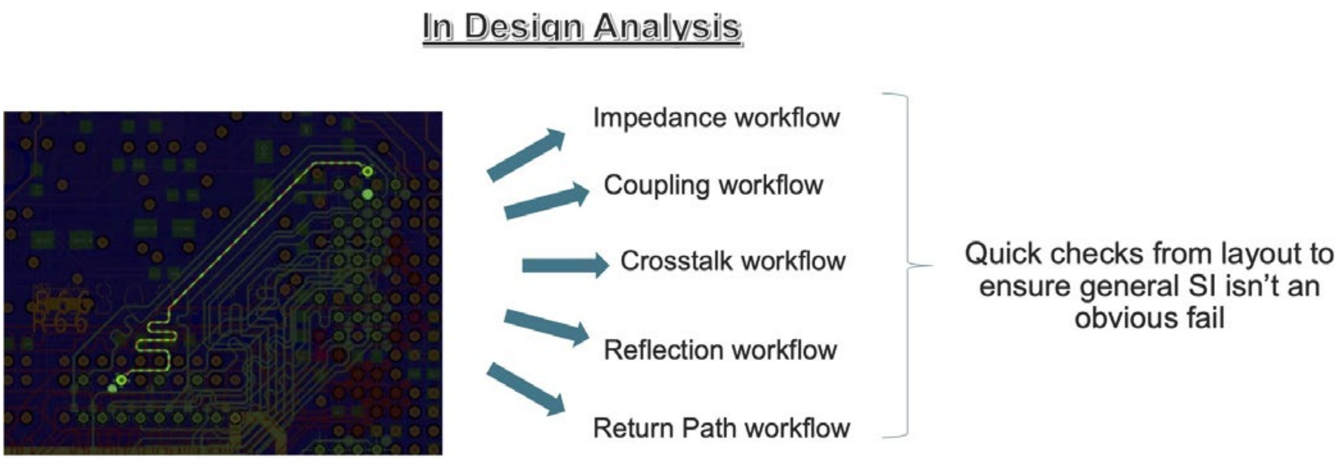


Figure 2: Cadence in-design analysis workflows.

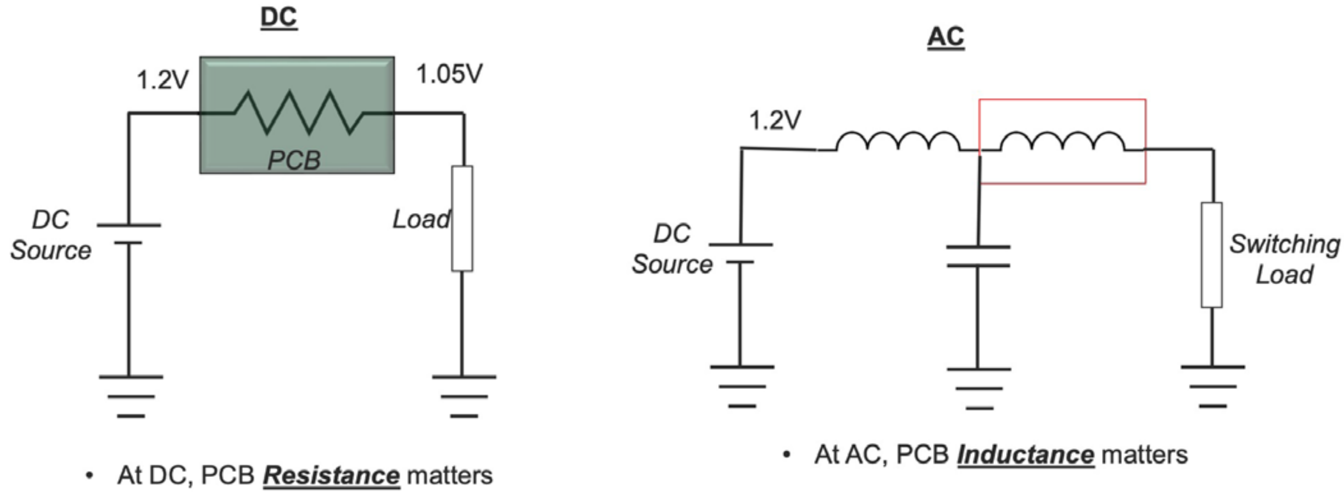


Figure 3: Sufficient and stable power for both DC and AC is required for components transmitting and receiving signals.

Power Design Analysis Workflow

A key challenge for PCB designers is delivering power to the load. The correct power for both DC (magnitude) and AC (ripple) are crucial for transmit (TX) and receive (RX) signals (Figure 3).

At DC, resistance matters, and at AC, inductance matters. Looking at DC, there is the DC source, the PCB, and the IC load where the current needs to go. The DC source will go through the copper on the PCB, which has a finite resistance and will therefore have an IR drop across it. The voltage coming at the load side will likely lose some amplitude from the original source. The question that needs to be analyzed is whether the voltage at the load is sufficient. Every IC has a requirement for the amount of voltage needed to power it, and the designer must make sure the IC is getting the needed voltage.

What is important for the DC analysis is the resistance of the PCB. For the AC analysis, the logic and input/output (I/O) circuits are toggling millions of transistors per second, and every time a transistor toggles, it requires immediate current, which is unlikely to come from the DC source. This is because the PCB is acting more like an inductor. At the higher frequencies at which the switching is taking place, the inducted impedance far overshadows the resistance impedance. Consequently, the inductance of the PCB begins to matter a great deal. Because the inductance is largely a function of the geometry of the PCB, the PCB layout becomes crucial.

The inductance from the voltage regulator module (VRM) will almost certainly be too high, so localized decoupling caps are used to provide the instantaneous current needed by the switching load. What becomes important is how much inductance there is between the capacitor and its load. The higher the inductance, the lower the effectiveness of the capacitor. This is an issue that must be considered by making sure to identify the capacitors with high-inductance connections to the load and reduce that inductance.

Power Workflow: Pre-Layout

At the beginning of the design flow, the designer has dozens to hundreds of schematic pages, as well as dozens of voltage rails, and the power connectivity needs to be set up correctly. With a large, complex power structure, the ability to visualize the location of the sources, sinks, passive drops, etc., is immensely useful in making sure there are no problems such as short circuits or unintended destinations for the power.

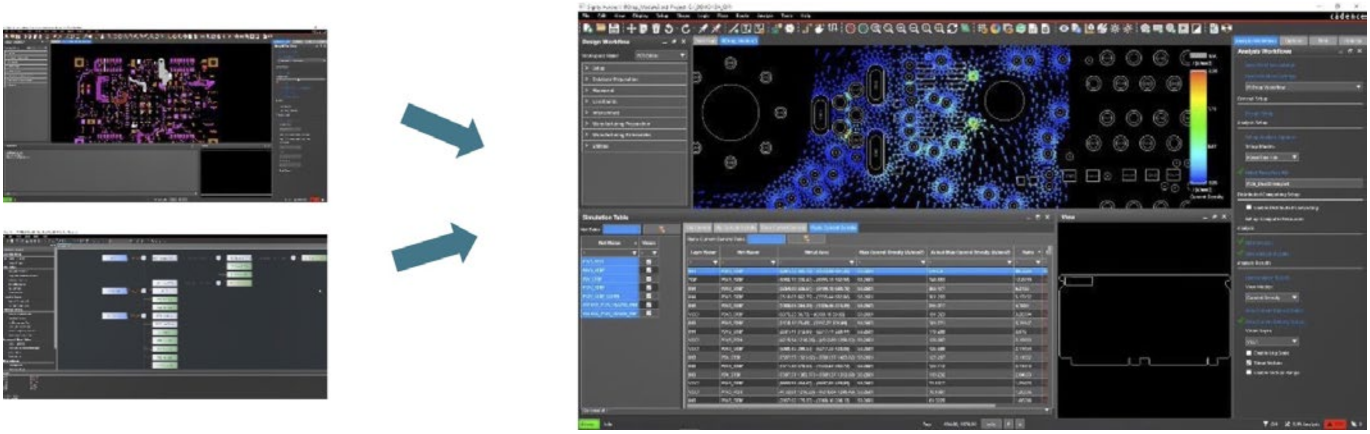


Figure 4: PCB layout and PowerTree combine to enable voltage drop analysis.

Our Sigrity PowerTree utility helps designers quickly and easily visualize their power connectivity from the schematic. It also allows them to run a quick simulation that will uncover any unnatural resistance drops or bad connections early in the design so corrections can be made in the schematic before they become a major problem later in the layout.

Power Workflow: Layout Analysis

In layout analysis workflows, engineers can collaborate with the layout designer on the board file using the previously created PowerTree file to analyze the DC and discover any current bottlenecks. When analyzing DC, it is important to understand the amount of VRM source current going to the IC destination, how much current the IC is drawing, and which VRMs are connected to which ICs. Because this information has already been captured in the PowerTree file, everything needed to run DC drop analysis is available in the layout environment, enabling IR drop, current, current density, via currents, etc., to be visualized (Figure 4).

A thermal impact can also be performed with the electrothermal co-simulation ability in the Celsius Thermal Solver to understand how high the current density is, how much heat is being generated, and if it is being dissipated appropriately.

The AC analysis is performed in a similar manner. The designer has the board and PowerTree files and can quickly run an AC analysis to see if the hundreds/thousands of decoupling caps are placed effectively and if the via breakouts from the capacitors are acceptable (Figure 5). The inductance for every capacitor on the selected via is shown, and the designer can quickly see any outliers in loop inductance

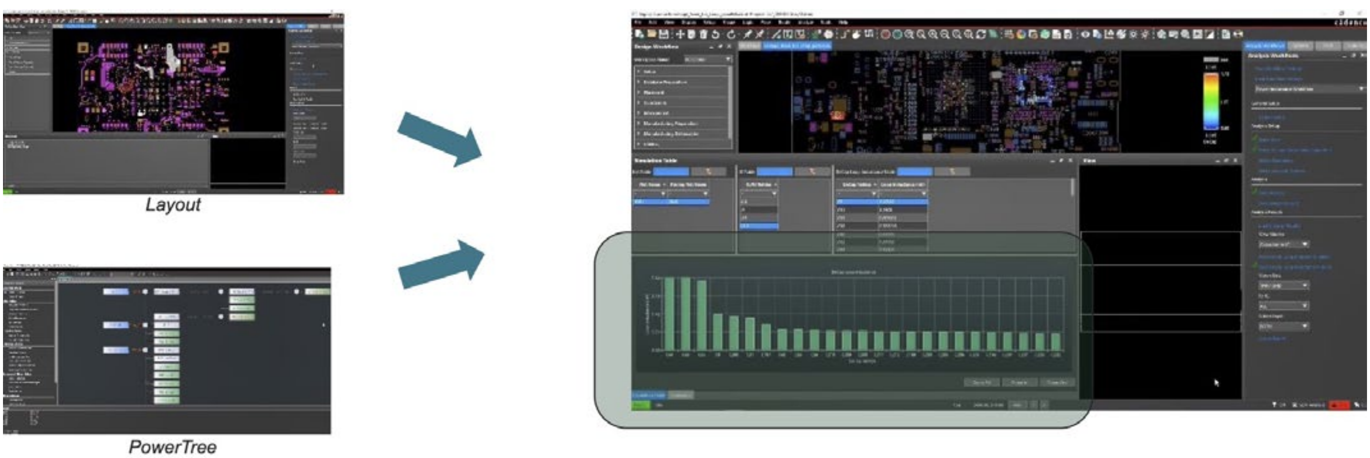


Figure 5: PCB layout and PowerTree combine to enable analysis of decoupling cap placement.


We are dedicated
to excellence through
innovation,
technology
and most importantly,
service.
COMMERCIAL • MILITARY • MEDICAL • BAREBOARD
U.S.C.
U.S. CIRCUIT
www.uscircuit.com
ISO
9001 CERTIFIED
DEPARTMENT OF DEFENSE
UNITED STATES OF AMERICA
ITAR
Registered International Traffic in Arms Regulations

to find poor decoupling placement. If there is a small decoupling capacitor that has unnaturally high inductance, the implication is that something is wrong with the layout and the designer will need to change the layout before moving on.

The key takeaway is that IDA can be utilized with information already obtained from PowerTree in the pre-layout phase, thus enabling the designer to quickly get information about the quality of the layout and act on any problems early in the design process.

SerDes Compliance Design Flow

A SerDes transceiver operates at extremely high frequencies, which presents many issues and leaves very little room for design errors. A SerDes design, without equalization, may not have an open eye at the receiver, so equalization at the TX and RX is simulated with IBIS-AMI models to show the eye opening. At high speeds, layer transitions are sensitive, the correct dielectric material must be selected, and even via placements become important to minimize channel impact.

The PCB designer designing high-speed SerDes channels typically works with an SI expert in the early phase of the design on the via structure. An optimized via structure can make the difference between passing and failing a serial link compliance test.

SerDes Design Challenges/Solutions

The basic rule of electrical engineering is that the higher the speed, the more details must be considered. Smaller structures that could previously be ignored in lower-speed designs can have a catastrophic effect at high speeds if not designed properly. Vias can cause impedance discontinuities and negatively impact signal quality. Therefore, via structures need to be carefully designed so that their behavior is predictable. This is traditionally addressed during the schematic phase by pre-designing each via within the structure to work at the speed of the design. This time-intensive manual process can be made more efficient with technology embedded in the design tool to design, simulate, and optimize vias for high-speed signaling. Sigrity Aurora workflows include a via wizard, which quickly generates Allegro-based via structures. With this automated flow, engineers can create their own via structures in the very familiar Allegro environment and

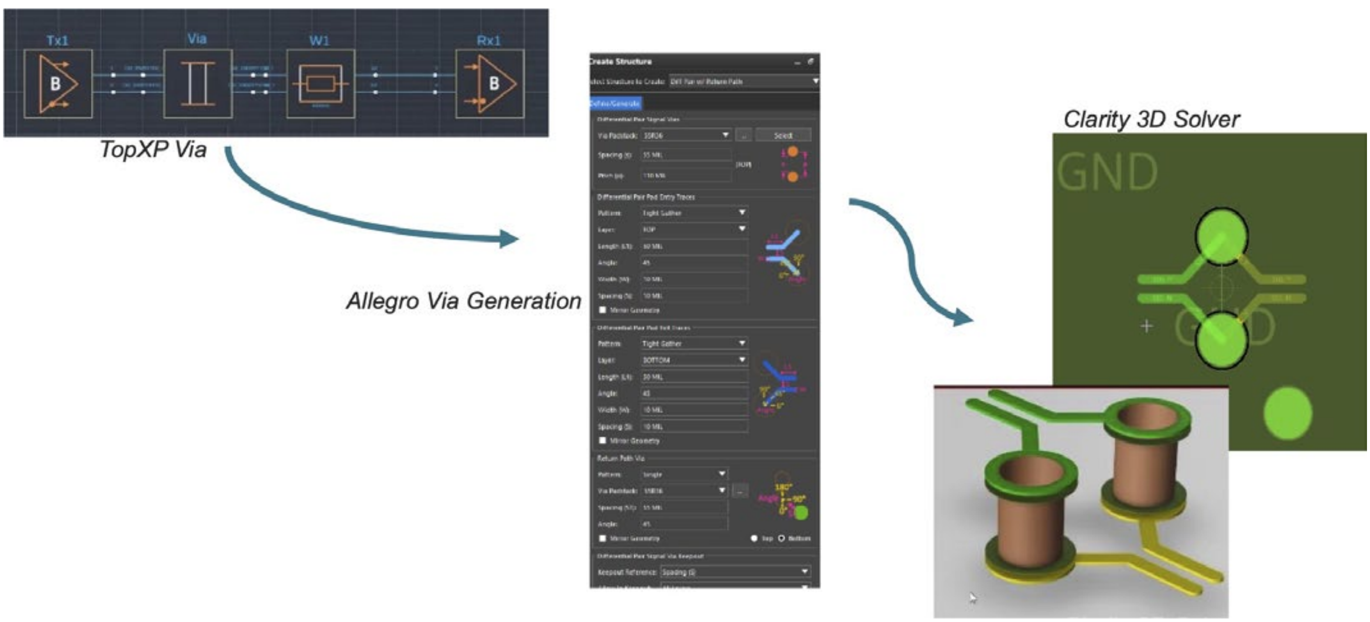


Figure 6: The Sigrity Aurora via wizard workflow efficiently generates Allegro-based via structures to be analyzed and optimized with Clarity 3D Solver.

then perform an analysis with the Clarity 3D Solver (Figure 6).

This is something the SI expert would traditionally do, but with IDA, PCB designers don't have to depend on the availability of the SI expert—they can do it themselves without being intimidated. The easy-to-use flow involves simply setting up the structure and opening Clarity 3D Solver from within the via wizard environment to run the simulation, assess the structure's validity, and make adjustments early in the process.

Part 2 of this article will appear in the September issue of *Design007 Magazine*. **DESIGN007**

Brad Griffin is product management group director at Cadence Design Systems and the author of *The System Designer's Guide to… System Analysis.*

IPC E-Textiles Committee A-Teams Are Shaping Some Much-needed Standards

By Chris Jorgensen
DIRECTOR, IPC TECHNOLOGY TRANSFER

The e-textiles industry recognizes IPC as a leader in standards development for the greater electronics industry, and in 2017 asked IPC for assistance to develop global standards for materials, design, and manufacture of e-textiles. Volunteers quickly adapted to IPC's processes by integrating themselves into the fabric (pun intended) of IPC's global standardization efforts.

There are now eight task groups under the IPC E-Textiles Committee that are developing standards and many new IPC Test Methods for conductive yarns; woven, knitted, braided, and embroidered e-textiles; printed electronics e-textiles; and even wearable e-textiles systems. The IPC E-Textiles Committee was among the first to embrace IPC's A-Teams approach, where dedicated groups of volunteers from an originating task group take on the lion's share of the work for their task group.

The first A-Team came together in 2018 and, from a list of topics identified by their subcommittee—IPC-8921, *Requirements for Woven and Knitted Electronic Textiles (E-Textiles) Integrated with Conductive Fibers, Conductive Yarns and/or Wire*—became IPC's first published e-textiles standard.

That A-Team's efforts established IPC as a true leader for standards development in the e-textiles industry. As the results of their work became more well known, it attracted more volunteers to form new E-Textiles Committee task groups. These volunteers found IPC to be a fast path: A proposed standard topic could lead to a task group approval, an A-Team formation, and then to starting their projects.

The IPC E-Textiles Committee now represents IPC's global footprint with eight A-Teams, all actively engaged in different standards projects. A-Team members hail from Belgium, Canada, France, Germany, Malaysia, Pakistan, Sri Lanka, Taiwan, the United Kingdom, and the United States. Each team has embraced IPC's challenge to come up with creative names for their teams. Let's meet the IPC E-Textiles Committee A-Teams and see what they're up to.

Click here to get details about each of the A-Teams, including their very clever names.

Best Drilling Practices for Better PCB Manufacturing

Connect the Dots

by Tim Totten, SUNSTONE CIRCUITS

Drilling is one of the most fundamental steps in the printed circuit board manufacturing process. Until the advent of the through-hole, PCBs were all single-sided with traces and components located on one side. With double-sided and multilayered boards so common now, a PCB without holes doesn't seem like a PCB at all. The drilling process creates the holes that connect the different layers of the PCB. Those holes allow for the connection of components. In fact, without holes, a double-sided PCB is just a coaster.

To maximize efficiency and reduce the error rate during the manufacturing process, PCB designers need to know several critical things about drilling. Some drilling information might vary from manufacturer to manufacturer, such as design limitations, tolerances, and optimal drill sizes. However, other factors can be controlled by the designer to maximize the quality of drilled holes across all manufacturers.

Since it is such a key step, understanding how drilling works during the manufacturing process can make your designs better and can help speed up production.

How Does Drilling Fit Into the Manufacturing Process?

Drilling happens early in the manufacturing process. For a multilayer PCB, drilling occurs directly after all the inner layers are combined and laminated together into a single manufacturing panel. Generally, panels are stacked together for the drill operation. Several physi-


PLM INTEGRATION
BOM OPTIMIZATION
SUPPLY CHAIN INTELLIGENCE
CHANGE MANAGEMENT

cal features help govern the size of the stack, panel thickness, smallest drill size, thickness of the copper, registration requirements, and even material type. The complete stack also includes a panel of entry material (such as aluminum) and a panel of backup material (phenolic material is a common backup), both of which help improve the drill quality and accuracy. Once all the boards are sandwiched between the entry and backup, the actual drilling can begin.

Precision drilling requires specialized machinery for accuracy and consistency. The drill machinery makes a huge difference in through-hole quality, and the best holes make for the best quality boards. Great drill machines are capable of high aspect ratio drilling, small via sizes, and tight positional accuracy.

Precision drilling requires specialized machinery for accuracy and consistency.

Once the drilling process is finished, individual boards can move on to the next stages of manufacturing. Electroless copper is added to the newly drilled through-holes, panels are imaged, copper electroplated, and then the boards move on to the etching process.

When Drilling Goes Wrong

When something goes wrong during the drilling process, it can result in one or more boards being damaged and discarded. The most common causes are material problems (image transfer from stacked pads/ground plane) and preparation errors which can lead to burrs and other hole quality issues.

Burring is a fairly common issue. If drilling happens too quickly, the stack isn't sandwiched correctly, or a drill bit gets too dull, burrs can form around through-holes. This can lead to short circuits, faulty connections, or uneven application of copper in through-holes.

The bottom board can receive the worst kind of burr: the "volcano effect." If you have ever drilled a hole too quickly through a piece of wood, you've seen the volcano—where the bit emerges on the other side, a large, conical burr can form.

While minor burrs can be fixed with a sander or by scrubbing, volcanoes are usually too severe, and will result in scrapped boards. To prevent burrs and volcanoes, sandwich the stack between entry and backup layers.

Designing for Efficient Drilling

Designers can take a number of steps to improve the efficiency of drilling and help cut down on errors. One of the most important improvements is to reduce the variety of through-hole sizes on the PCB design, which allows for fewer tool changes. This can be accomplished by paying close attention to the allowances for various through-hole requirements. If multiple allowance ranges overlap, select a through-hole size that fits in as many allowances as possible.

Keep in mind that tool changes take time. By cutting down on drill size requirements, the designer will not only greatly improve the speed of manufacture but reduce the number of extra steps needed to finish the manufacturing process.

Some design elements can increase the chances for burring. Among these are higher copper weights and anything that can keep layered boards from sitting flat against each other. Sometimes burring can be fixed with a hand sander, but that takes a lot of time and can slow down the manufacturing process.

By minimizing the number of hole sizes, a designer can reduce the amount of material

that needs to be drilled and removed from a board. In addition, a clever design can optimize spacing between the holes, which reduces the amount of motion a drill needs between drilling. Both design optimizations might seem minimal, but their savings can add up for large manufacturing runs.

Better Drilling for Better PCBs

Drilling plays an integral role in the PCB manufacturing process, and PCB designers need to understand how their designs affect drilling efficiency and accuracy. Optimized through-hole and pad sizes can facilitate the drilling process and ensure high-quality PCB manufacturing.

By working closely with the PCB manufacturer and following best practices, designers can achieve better efficiency and reduce board waste. Accurate drilling is key to a reliable and efficient board, and by focusing on these essential elements, designers can produce PCBs with fewer errors and failure rates that end up being cost-effective. DESIGN007

Tim Totten has been at Sunstone since 2004, where he has played an integral role in initiatives to enhance the drill department, including training his colleagues on new equipment, developing processes for new product lines, and learning how to write G-code.

Download *The Printed Circuit Designer's Guide to... Designing for Reality* by Matt Stevenson. You can view other titles in the I-007e library here.

BOOK EXCERPT

The Printed Circuit Designer's Guide to... Manufacturing Driven Design

Chapter 1: How We Got Where We Are

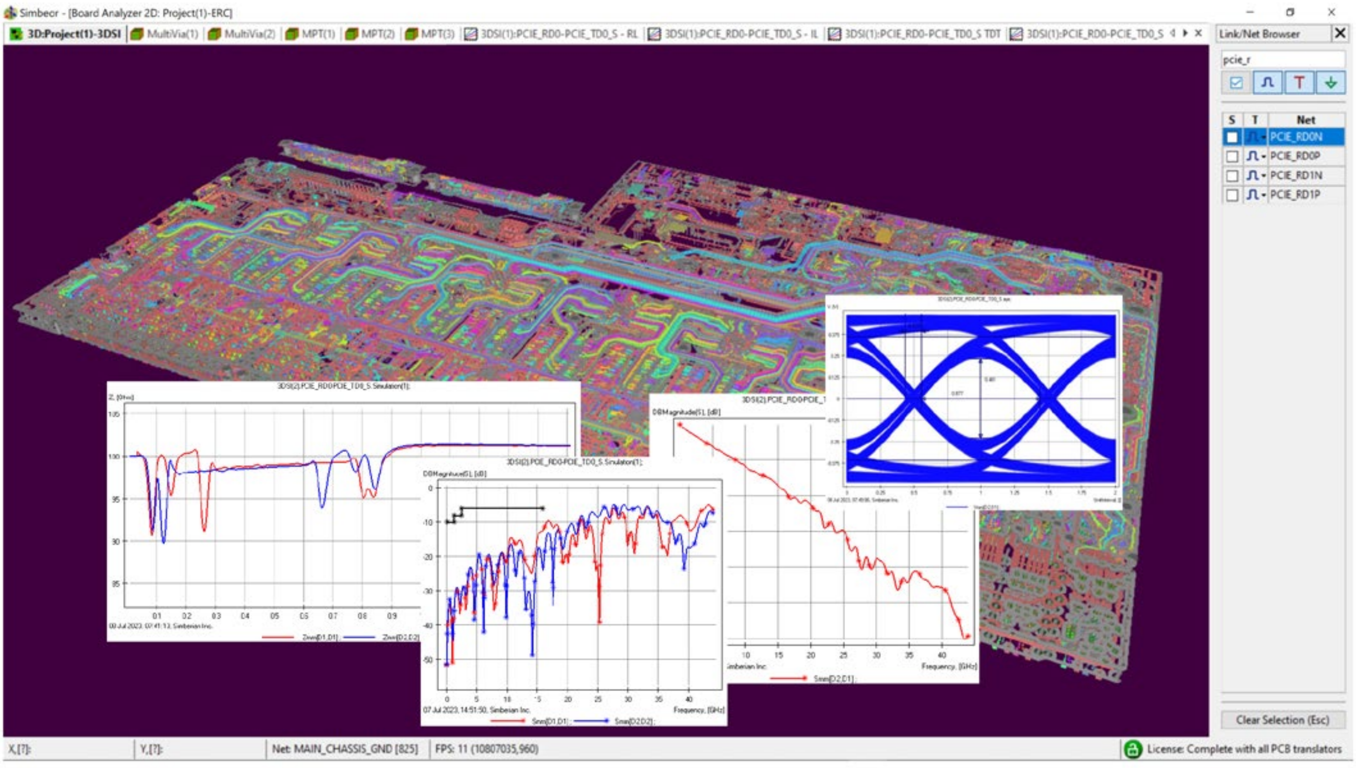
The Paradox of DFM

Before the development of PCB computer-aided manufacturing (CAM), the process of preparing a board for fabrication and assembly was as cumbersome as actually designing the board itself. Early PCB manufacturing involved several manual processes which remain largely intact today, mirrored by a modern digital equivalent. Manual design for manufacturing (DFM) began with PCB artwork that was produced by applying an opaque film on a clear piece of mylar. The examination and measurement process involved using an eye loupe, and any necessary edits were made by either carefully scratching off the tape with an X-ACTO® knife to provide additional clearance or filling in voids with a black marker.

Tasks such as drilling and component placement relied on custom-built mechanical programs. These were stand-alone solutions, created individually in a time-consuming and painstaking process involving specialized equipment such as drill bomb-sight machines. With such an intensive and manual process, there were many challenges to overcome to meet all the necessary operational complexities needed to produce a circuit board.

As the complexity of systems increased, these physical sheets were replaced by digital files, but the manual complexities persisted. During the 1980s, the design data transferred to manufacturing facilities was heavily split as a result of the myriad of solutions necessary to produce a PCB. Gerber data was used for artwork, Excellon drill format for mechanical drilling, typically a centroid file which assisted in component placement and, if you were very lucky, a netlist file in IPC-356-A format was included. This data came accompanied by a corresponding component bill of materials.

Continue reading.



3D Electromagnetic Analysis

Feature Article by Yuriy Shlepnev
SIMBERIAN

Data rates in PCB interconnects are increasing in all signaling protocols (PCIe, DDR, GDDR, Ethernet, USB, SAS, InfiniBand, CEI, OIF, 5G). Most of those high-speed signaling standards have one-lane data rates over 6 Gbps (GT/s) and some up to 112 Gbps with signal spectrum in microwave and even millimeter wave bandwidths[1]. Design of compliant interconnects at these data rates cannot simply rely on geometrical rules or rules of thumb. Signal distortion by reflections, dissipation and crosstalk can cause interconnect performance degradation or even failure. To avoid it, signal integrity compliance analysis and possible interconnect optimization is required.

Our software provides 3D electromagnetic analysis of PCB and packaging interconnects. It can be used for pre-layout design (stackup exploration, via hole design) and post-layout interconnect compliance analysis and optimization. Simbeor ensures the accuracy of the models by using advanced algorithms for 3D full wave analysis, benchmarking, and experimental validation. Simbeor and the "sink or swim" interconnect design process[2,7] remove all uncertainties and guarantee the first pass design success. Most importantly, it lets you solve electromagnetic signal integrity problems at a relatively low cost and with extreme ease. You don't need to be an expert in signal

integrity or have a PhD in electromagnetics to use it.

The first version was introduced in 2007. It was the first electromagnetic tool designed specifically for PCB designers and signal integrity engineers. Since then, its evolution is a chain of innovations in interconnect analysis and validation. In the past three years, with the major part of the work completed in "self-isolation," our team has elevated the tool to a new level.

Our software development kit was introduced for design exploration, machine learning, and for possible integration into other tools[3]. Post-layout geometry processing, visualization and model building were accelerated orders of magnitude. Electromagnetic analysis was also accelerated orders of magnitude with the domain decomposition technique. We had enough time—there were very few distractions—and sufficient expertise to re-think and re-design the post-layout process, making it suitable not only for SI engineers, but for any PCB designer.

The result is a tool called the SI Compliance Analyzer that can be used for fast, consistent post-layout signal integrity verification with simulation-based electrical rule checking (ERC), basic signal integrity analysis (Fast SI), and advanced 3D EM signal integrity analysis (3D SI). It is a solution for interconnect validation and compliance analysis tasks with one unified easy-to-use interface and the following operating modes:

- Electrical rule checking (ERC) uses the 2D quasi-static Simbeor field solver (SFS) for traces and component pads and fast EM models of via holes, to find reference integrity and localization violations, impedance continuity violations, and possible crosstalk noise. This mode can be used for interactive analysis of links in a fraction of a second or thousands of links with automation. It makes all geometry-based rule checkers obsolete and unnecessary.
- Fast SI uses SFS for traces and pads, fast EM models of via holes and precise decomposition for the basic signal integrity analysis of crosstalk noise, losses, delay, and skew for relatively slow signals (<10 Gpbs, >100 ps rise time), or preliminary analysis of high-speed links. It enables interactive analysis of links in seconds or hundreds of links with the automation in real time.
- 3D SI uses the 2D quasi-static field solver or a 3D EM solver for traces and 3D EM solver for via holes, component pads, and other discontinuities and precise decomposition for advanced signal integrity analysis of PCB/packaging interconnects (unlimited data rates, accuracy depends on geometry, materials, and link localization). It enables interactive analysis of links in minutes or hundreds of links with the automation in real time.

All those modes are designed to verify interconnect compliance with a particular signaling standard and quickly find the reason for failure if a compliance metric is violated. A few examples of interconnects analysis and optimization are provided to illustrate the process.

A perfect digital interconnect is a lossless transmission line with constant characteristic impedance and phase delay over the signal bandwidth and termination resistors matching the characteristic impedance. In such interconnect, bits sent by transmitter would flow smoothly into the receiver with no bit rate limits. Such an ideal transmission line is only imaginary and theoretical. The physics of our world does not allow that. To ensure that the digital signal is actually getting through, we have to build interconnect models that include all signal degradation factors important for a specific data rate. But, before building any model, the reference integrity and via localization must be verified and fixed if necessary.

Reference integrity analysis in ERC mode checks all reference conductors and stitching

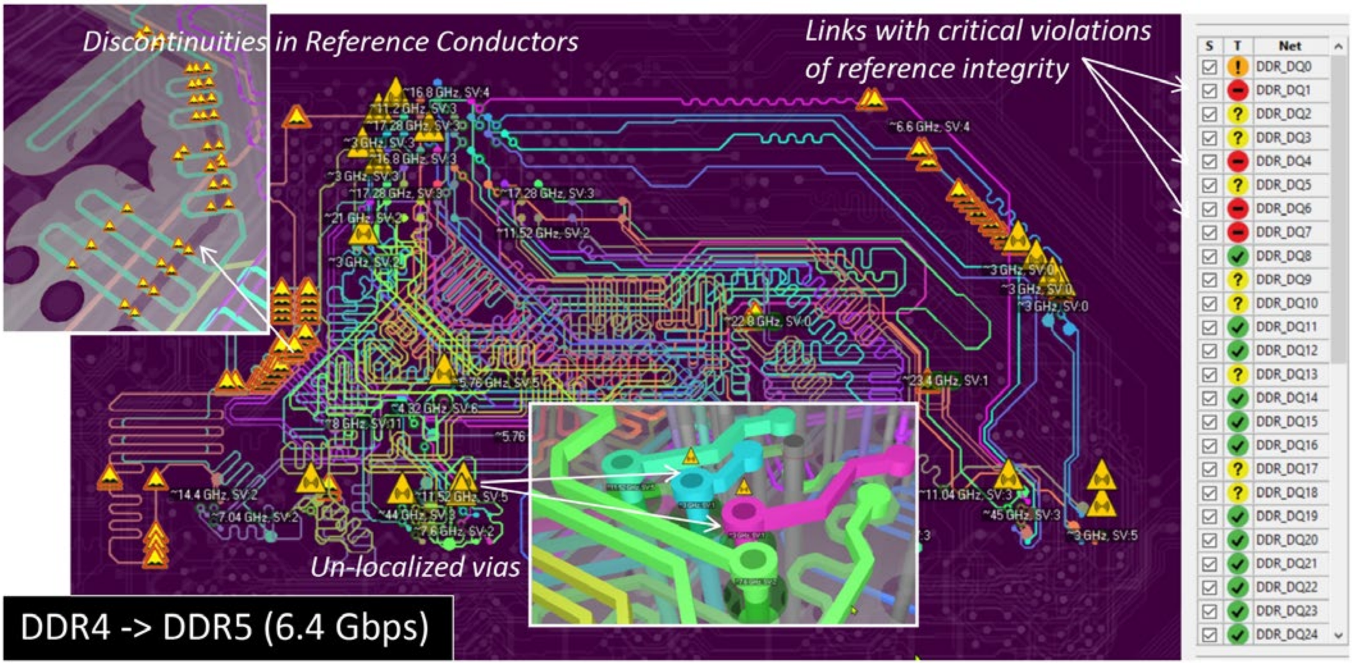


Figure 1: Reference integrity analysis for DDR data links in the OCP board.

vias, and via hole localization[4]. Basically, it is the analysis of the current return path. Figure 1 is an example of the reference integrity analysis for all DDR data links on the Open Computing Project (OCP) PCB.

The analysis is done for DDR5 data rate 6.4 GT/s and reveals some problems in the nets marked with the red stop signs on the right (severe reference integrity violations). Some traces go over the splits in the closest reference planes and some via holes are un-localized (begin leaking the energy at the Nyquist frequency). The severe reference integrity violations must be fixed in layout before proceeding with any other type of analyses. This is imperative. The PCB in this case was designed for DDR3 and as we can see cannot be used "as is" for DDR5.

When all reference integrity problems are corrected, we can proceed with the other types of compliance analysis. Impedance continuity analysis in ERC mode can be used to quickly check the impedance of interconnects, including vias and pads. (More on impedance and reflection.)[5,6] Figure 2 illustrates impedance analysis, showing how the reference conductors can change the impedance of traces on a design with traces going through BGA breakouts.

The software evaluated the effect of cut-outs and reference pads on the impedance. We can

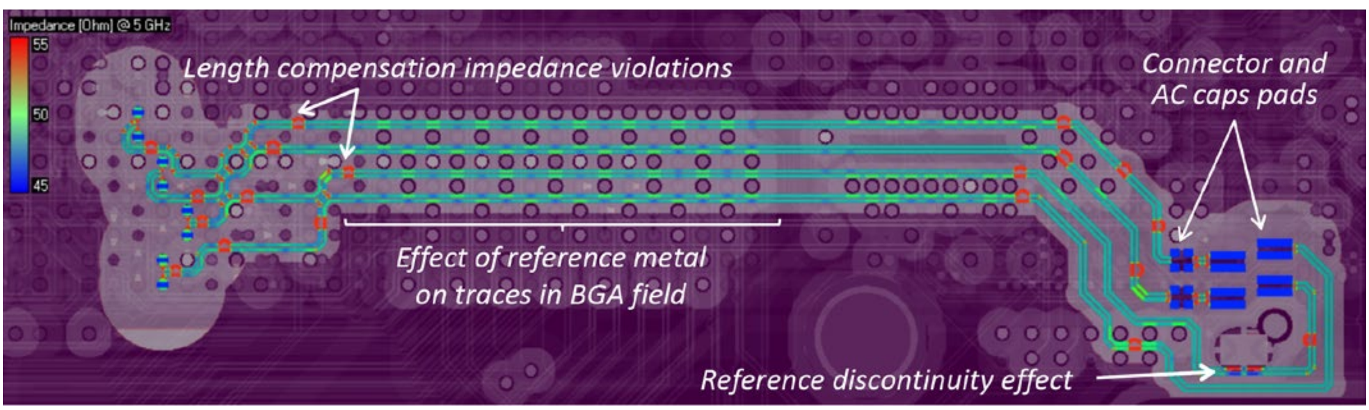


Figure 2: Impedance analysis that shows how BGA breakouts can alter impedance of traces.

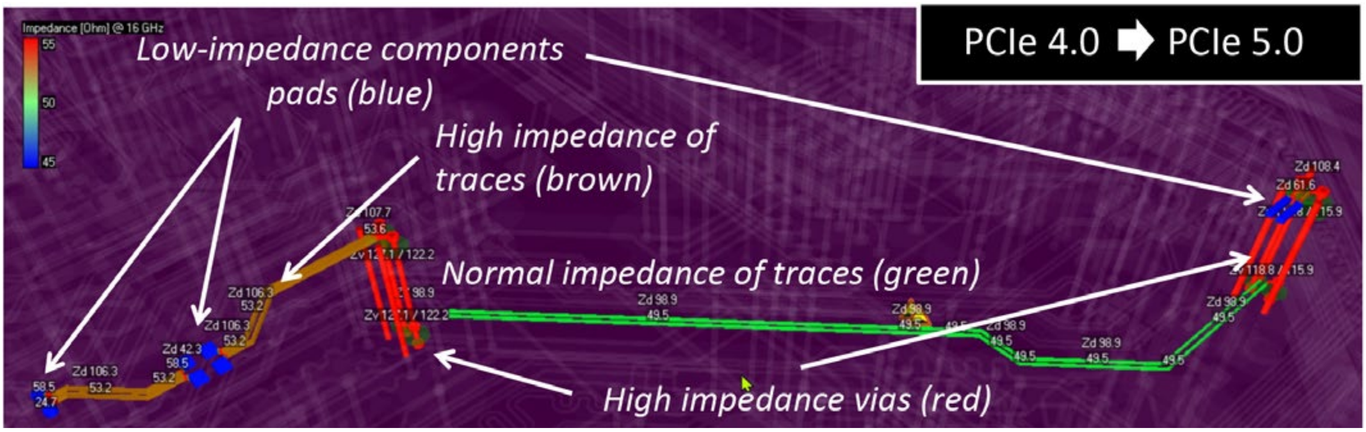


Figure 3: Impedance continuity analysis of the OCP board shows impedance violations.

see that the impedances of connector and AC coupling pads are below the target and the impedance of the length compensation sections is above the target—a layout mistake. The discontinuities in the reference conductors also create impedance violations—another layout mistake discovered by the tool.

Figure 3 is another example of the impedance continuity analysis for properly localized PCIe link on the OCP board.

The analysis is done at the Nyquist frequency of PCIe 5.0 signal running at 32 GT/s. The target differential impedance is 100 ohms, and we can see that the link may require some improvements. The question is how the impedance violations affect the signal transmission. Analysis of reflections in 3D SI analysis mode (or Fast SI for lower data rates) can answer that. Accurate 3D SI models are used here to compute return loss (RL) and do the TDR analysis as illustrated in Figure 4.

The return loss violates PCIe 5.0 standard mask (the black line on the bottom graphs) in this case; the main reasons for that are the

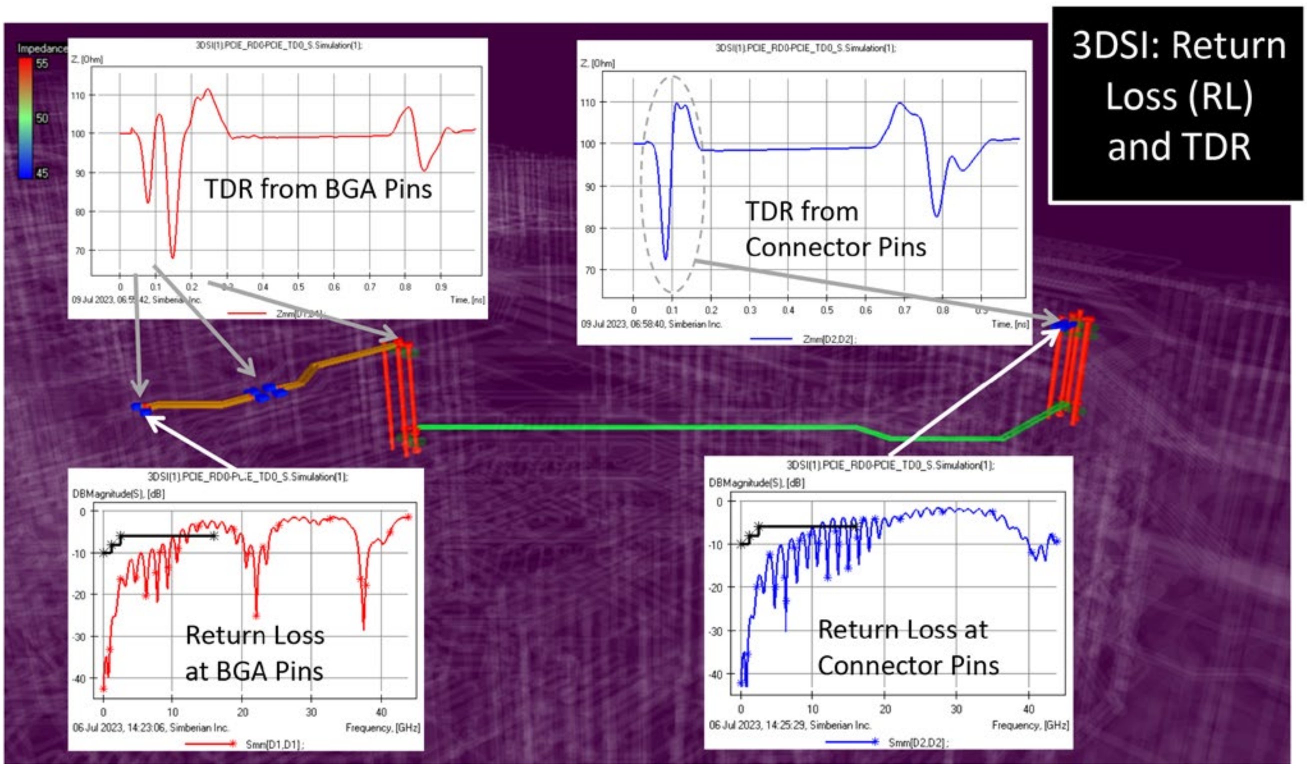


Figure 4: Examples of violations, primarily caused by reflections.

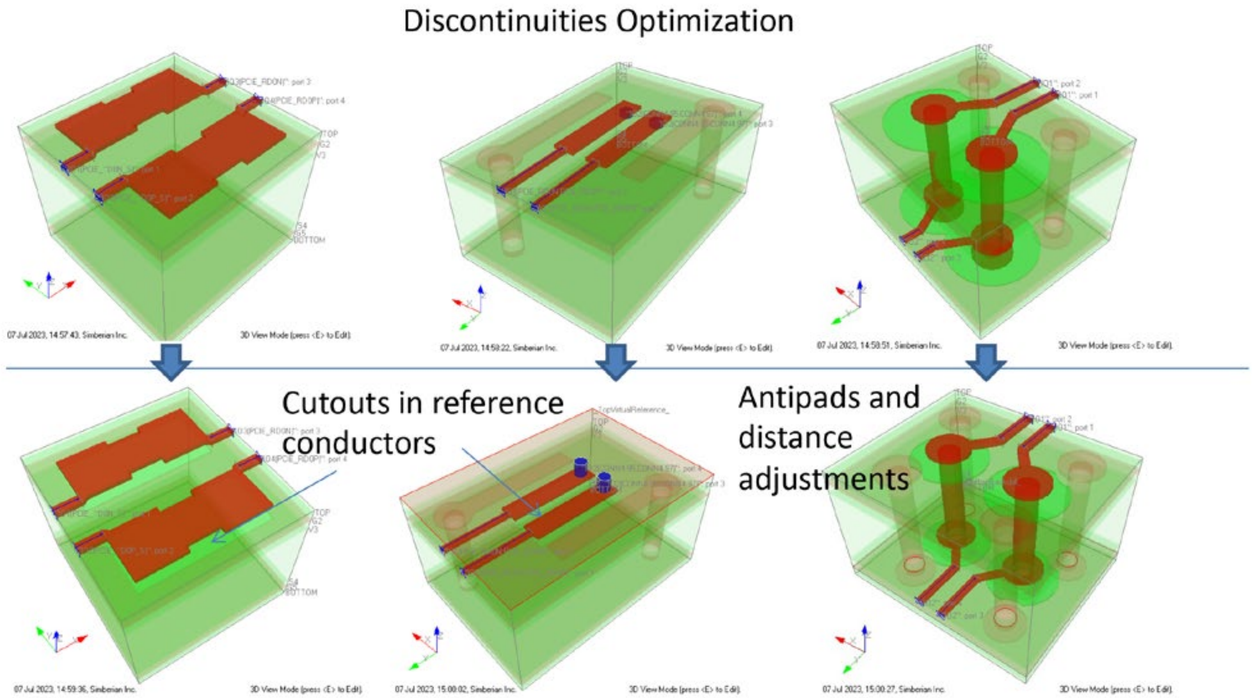


Figure 5: Before and after: The top row geometries are potentially problematic, while the bottom row images show these structures after optimization.

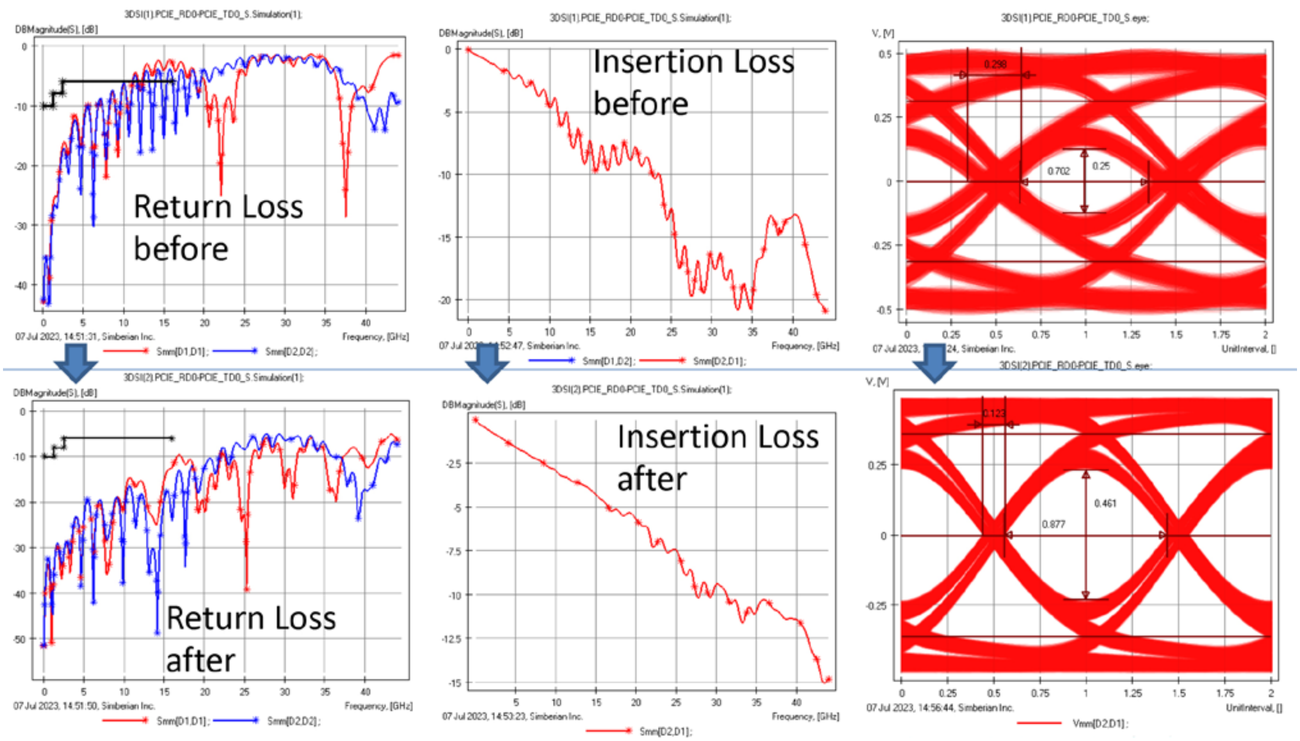


Figure 6: Results of the adjustments shown in Figure 5.

reflections from the component pads, via holes, and microstrip traces on the top side of the board (dips and spikes on TDR plots of the top graphs). That was also observed on the preliminary impedance continuity analysis in the ERC mode. The link requires optimization that can be done in this tool. Small cut-outs in the reference conductors below the signal pads

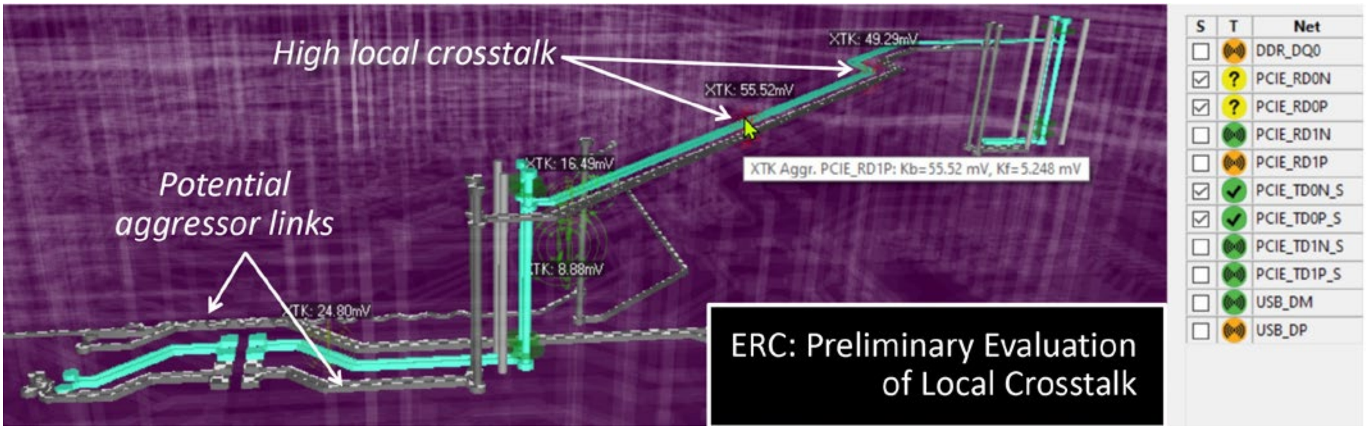


Figure 7: Crosstalk evaluation showing potential aggressor links.

can be used to reduce the capacitance of the component pads and adjustments of distance between the signal vias, and anti-pad size can be used to reduce the inductance of vias as shown in Figure 5.

The original geometries of the discontinuities are shown in the top row and the adjusted more optimal or less reflective discontinuities are shown in the bottom row. The results of such small adjustments are shown in Figure 6.

Return and insertion loss and eye diagram of the original link are shown in the top row and corresponding results for the adjusted link are shown in the bottom row. What a difference such small adjustments make. The link passes the return loss mask (black line) and the eye is larger, offering more margin for possible random unpredictable things. The analyzer can be used for control of all types of standard compliance metrics (RL, IL, Fitted IL, ILD, PSXT, MDXT, ICN, ICR). That includes crosstalk analysis. Local crosstalk evaluation can be done in the ERC mode as illustrated in Figure 7.

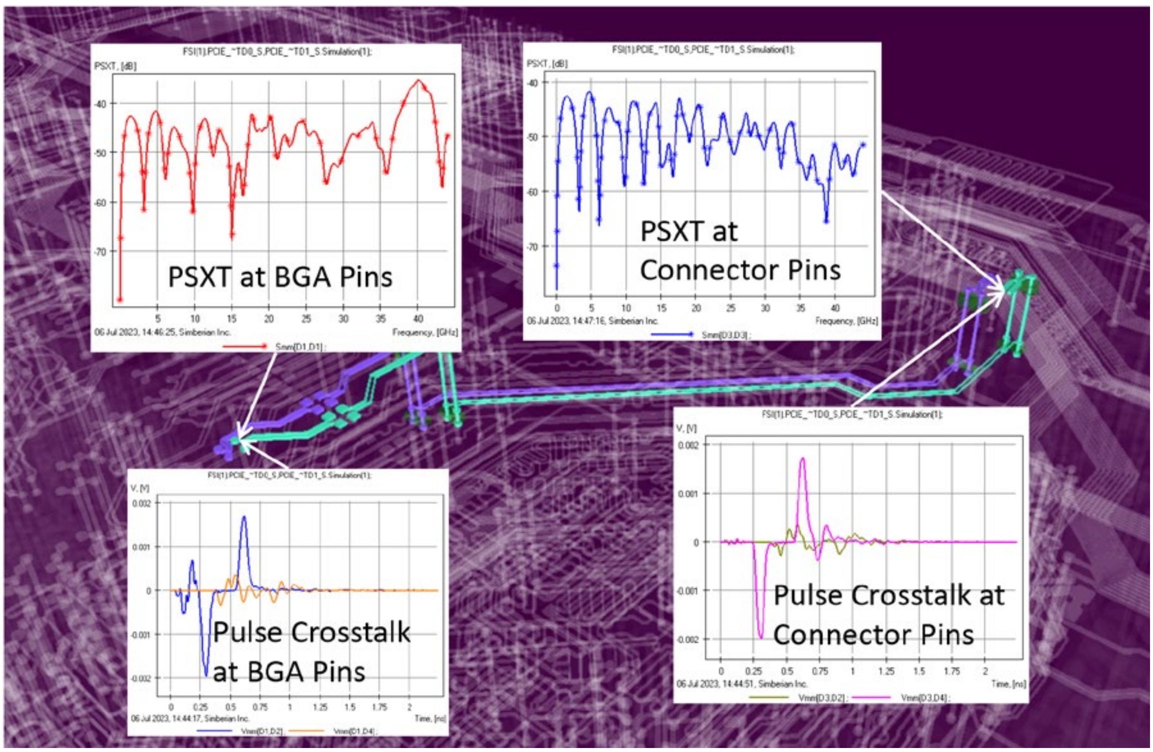


Figure 8: Frequency and time domain analysis shows a variety of metrics.

For a selected link, Simbeor finds all possible aggressor links and evaluates trace-to-trace and pad-to-pad coupling in mV (can be dB or %), assuming 1 V excitation with the rise time specified for the signal. It eliminates all types of geometrical proximity rules.

To evaluate the system-level impact of the crosstalk, Fast SI or 3D SI analysis modes can be used. Fast SI mode includes crosstalk between the traces and pads and 3D SI optionally adds crosstalk evaluation between the vias. The analysis can be done in frequency domain (PSXT, ICR, ICN, MDXT) as well as in time domain (step, pulse crosstalk or eye diagram with crosstalk) as illustrated in Figure 8.

The top graphs show power sum crosstalk (PSXT) at the BGA and connector pads and the bottom graphs show the pulse crosstalk for 32 GT/s signal with 25 ps rise time and 0.5 V excitation. 3D SI analysis is used for this example.

The upcoming release further pushes the boundary of possibilities with the extensions in EM solvers to build accurate models at frequencies over 50 GHz, and new capabilities can increase productivity of PCB designers. DESIGN007

References

1. "How Interconnects Work: Bandwidth for Modeling and Measurements," by Yuriy Shlepnev, Simberian App Note #2021_09, November 8, 2021

2. "Sink or swim at 28 Gbps," by Yuriy Shlepnev, *The PCB Design Magazine,* October 2014, p. 12-23.

3. "Machine Learning Applications for COM Based Simulation of 112Gb Systems," by A. Manukovsky, Y. Shlepnev, Z. Khasidashvili, E. Zalianski, DesignCon 2020.

4. "Life beyond 10 Gbps: Localize or Fail!" by Yuriy Shlepnev, April 13, 2018, Simberian App Note #2018_03.

5. "How Interconnects Work: Impedance and Reflections," by Yuriy Shlepnev, Simberian App Note #2021_11, December 22, 2021.

6. How Interconnects Work: Reflections from Discontinuities, by Yuriy Shlepnev, Simberian App Note #2022_01, January 10, 2022.

7. "40 GHz PCB Interconnect Validation: Expectation vs. Reality," by M. Marin and Y. Shlepnev, DesignCon2018. This was the award-winning paper from DesignCon 2018.

8. "Effect of PCB Fabrication Variations on Interconnect Loss, Delay, Impedance and Identified Material Models for 56-Gbps Interconnect Designs," by A. Manukovsky, Y. Shlepnev. Nominated for Best Paper Award at DesignCon 2019.

Yuriy Shlepnev is president of Simberian Inc.

Circuit Simulation, SPICE, and AI

Elementary, Mr. Watson

Feature Column by John Watson, CID, ALTIUM

According to the Federal Aviation Administration's flight time limitations and rest requirements, a commercial pilot is restricted to 36 flight hours in a week, 100 hours in 28 days, and 1,000 hours in any calendar year[1]. The cycle time depends heavily on the aircraft they fly and length of time in the air. It can often be long and grueling, so these limits are there to ensure the safety of the flight.

But before these pilots even start flying the friendly skies, they spend considerable time in a flight simulator. Airline pilots go through about a month of simulator training before they can fly on the line, and this comes after two to three weeks of systems and operations training.

Most airlines maintain a rigorous simulator training program of 18–24 simulated flights lasting four hours each. During these flights, they train on system reviews and procedures, including checklists and "flows"—special maneuvers, and real-world problem situations. All that can add up to 96 hours of simulator training before they walk aboard an actual plane. This level of training is done every six months to ensure that their regular and non-normal operation capabilities are up to standard.

Why? In the simulator, pilots have a controlled, safe environment to practice various flight maneuvers, emergency procedures, and critical decision-making without the risks

associated with actual flight. That allows pilots to gain experience in handling challenging situations and improves their ability to respond appropriately in real-world emergencies without crashing a plane if they make a mistake. Furthermore, conducting flight training in a simulator is more cost-effective than training in an actual aircraft. Simulators reduce fuel, maintenance, and operational costs, allowing pilots to gain valuable training hours without the high expenses of flying real planes.

Take Your Design for a Test Flight

In the same way, PCB design and circuit simulations provide the same advantages: a controlled, safe environment for testing a circuit without the considerable overhead of fabricating and assembling the PCB. That is all done through SPICE simulations (Simulation Program with Integrated Circuit Emphasis). SPICE is a computer simulation and modeling program used to predict the behavior of electronic circuits mathematically.

Circuit simulations have a rich history that has mirrored the ever-changing electronic innovations.

Many PCB designers feel that circuit simulation is something they will never need to learn, and it is best handled by that infamous "someone else." But in our present business environment, with the lack of PCB design and engineering resources, many designers are pulling multiple duties throughout the design process. Having individuals understand various disciplines, including PCB simulation, increases your overall value to any company. I always recommend that new PCB designers learn anything and everything they can that would be an asset to the company.

Circuit simulations have a rich history that has mirrored the ever-changing electronic innovations. Circuit simulation must keep up with ever-increasing technology and more complex circuits. It all began in the early 1970s at the University of California, Berkeley, with Dr. Laurence W. Nagel and his team. Initially designed for simulating integrated circuits (and part of Nagel's PhD dissertation), SPICE aimed to analyze and optimize the performance of electronic systems at the transistor level. It wasn't until the late 1990s to early 2000s that SPICE simulation became integral to comprehensive EDA software suites, integrated with schematic capture and PCB layout tools. This integration allowed designers to move between schematic design, layout, and simulation seamlessly.

The 2010s was the golden age of circuit simulations with significant advancements. With the advent of cloud computing, some EDA companies began offering cloud-based SPICE simulations, enabling scalable resources for more extensive simulations.

Furthermore, advancements in machine learning and AI techniques were integrated into SPICE simulators to optimize designs, perform parameter sweeps, and explore design spaces more efficiently, which we will discuss later in this column.

Altium's SPICE Environment

Altium has what can be described as an ongoing development of its SPICE tool. Although we originally released a circuit simulation tool in the early 2000s, the release of Altium Designer 22 in 2022 featured a fully revamped simulation tool, with a focus on several key points.

Circuit simulation was fully and seamlessly integrated into the Altium Designer PCB design environment. This integration allows design engineers to move seamlessly between schematic capture, PCB layout, and simulation, streamlining the design process.

A completely redesigned advanced simulation engine that can handle complex circuits and mixed-signal designs drives the platform. That facilitates the co-simulation of analog and digital circuitry. It can quickly analyze the interaction between the two domains, ensuring proper integration and functionality. It supports various analysis types, including transient, AC, DC, and noise. Also, parameter sweeping is a remarkable advanced simulation technique for sweeping different circuit variables, enabling them to analyze the circuit's response to varying component values. This feature helps understand circuit sensitivity and optimize design parameters.

As mentioned earlier, SPICE circuit simulations mirror the advancements in our industry, which means the challenges are constantly shifting and changing. The primary area is the simulation of high-speed designs. Access to frequency domain analysis to analyze circuit behavior in the frequency domain is essential for RF and communication systems. It helps ensure proper frequency response, gain, and filter characteristics. Directly tied to that is transient response analysis, which assesses circuits' transient behavior during startup or switching events. This analysis is critical for understanding circuit stability, settling times, and response to input changes.

Shrinking the Simulation Learning Curve

SPICE was once seen as a complex tool that would take years to master, but today's SPICE flavors, such as Altium SPICE, are a tad more user-friendly. Of course, based on what I've seen teaching this subject, the time necessary to learn it will vary from person to person. As a PCB design instructor, if I were to set up a curriculum for a circuit simulation course (hey, that's a great idea), I would put the following time estimates for each objective of the course:

- Basic simulation set-up (two to three weeks). Users start working with the circuit simulation tool, learning to add and configure component simulation models. To understand component behavior, fundamental simulation analyses like DC and AC analysis are explored. Users begin by running simulations on simple circuits and verifying the results.
- Advanced analysis (two to four weeks). Users progress to more complex analysis types, such as transient analysis, noise analysis, and parameter sweeping. Convergence issues and adjusting simulation settings will be covered. Signal integrity analysis for high-speed designs may also be introduced.
- Mixed-signal simulation (two to three weeks). Mixed-signal simulations, including simulating the interaction between analog and digital components, as well as co-simulation techniques and troubleshooting mixed-signal designs are addressed.
- Design optimization and rules (two to three weeks). Students would use simulation results to optimize PCB designs iteratively and make design improvements

- based on simulations. Curriculum would also focus on implementing design rule checks to ensure compliance with specific design constraints and industry standards.

To become proficient with the Altium SPICE tool, I estimate roughly eight to 13 weeks.

Circuit Simulation for Beginners

I would start with simple circuits and progress to more complex designs. Master the fundamentals of circuit simulation, such as DC and AC analysis, before moving on to advanced techniques.

- Familiarize yourself with different simulation models (e.g., ideal, behavioral, and transistor-level models) and their limitations. Understanding the accuracy and applicability of these models will help you interpret simulation results effectively.
- Take advantage of tutorials and online resources provided by the simulation tool's documentation and the broader community. Work through examples to grasp simulation techniques and gain practical experience.
- Learn to perform parameter sweeps to explore how varying component values impact circuit performance. Parameter sweeping helps you understand the sensitivity of your circuit to different parameters.
- Familiarize yourself with component datasheets and their corresponding SPICE models. Accurate component models are crucial for reliable simulations. And remember that circuit simulation is an iterative process. Don't be discouraged by initial results; use them to refine and improve your design iteratively.

Simulation, AI, and the Future

With AI's development, I believe our world will never be the same. I would put this on the level of the discovery of fire or the invention of the wheel. AI is not going away, so we should embrace it. I am sure it will change how circuit simulations are performed. Here are some ways I see AI impacting this whole area in the future.

Simulation

AI-based algorithms can enhance simulation efficiency and accuracy by leveraging computational models and approximations. This can significantly reduce simulation times for large and complex circuits while maintaining acceptable accuracy. Furthermore, those AI algorithms can automatically optimize circuit designs based on specified objectives and constraints. By leveraging machine learning techniques, AI can explore vast design spaces to find optimal solutions for various performance metrics, such as power consumption, signal integrity, and thermal efficiency.

Predictive Analysis

With the ever-growing and knowledgeable platform of AI, predictive analysis will describe the behavior of complex circuits by learning from past simulation results and identifying patterns in circuit performance. Take this to its logical conclusion. There will be an incredible improvement in circuit failure analysis; AI will be able to analyze your circuit and, from the simulation results, predict potential points of failure in the circuit design, improving circuit reliability.

AI is an intriguing area of research, and we are barely in the infancy of this area. Fasten your seatbelts, because we haven't seen anything yet. DESIGN007

References

1. "How many hours and cycles do pilots amass in 12 months?" by Jack Herstam, simpleflying.com, Dec. 29, 2022.

John Watson, CID, is a customer success manager at Altium. To read past columns, click here.

MilAero007 Highlights

Altium 365 GovCloud Offers Increased Security ▶

Altium recently launched Altium 365 GovCloud, a dedicated platform accessible only to—and managed solely by—U.S. persons. The company says that GovCloud can help customers to be in compliance with ITAR, EAR, and other requirements. Andy Shaughnessy spoke spoke with Bruno Blasigh, director of Cloud Security for Altium 365, about the new platform, how it functions, and how GovCloud can help to keep foreign entities from accessing your data.

Flexible Thinking: The Adjacent Possible ▶

In the inspirational and informative book titled *Where Good Ideas Come From*, author Steven Johnson uses the term "the adjacent possible." This term, which immediately captivated my mind, originated with a theoretical biologist named Stuart Kaufman, who used the term in his book, *Investigations*, to describe the circuitous path of biological evolution.

Filling Critical Traceability Gaps With AI ▶

Traceability means being able to track the origin of any given electrical component throughout the supply chain. For OEMs, this is no longer optional or "nice to have." Yet industrial traceability capacities are sorely lacking throughout industries. Today, the most widespread standard for traceability is "batch traceability," which aside from tracking the production lot, serial number, and exact board placement for components, fails to analyze the individual components themselves, thus jeopardizing the quality of the goods they compose.

Averatek Names New CEO ▶

Averatek is pleased to announce Scott Meikle, Ph.D. as CEO. Dr. Meikle has more than 30 years of experience in the semiconductor industry.

Designing Aerospace PCBs: A Galaxy of Challenges ▶

Jeffrey Boye designs aerospace PCBs at the Johns Hopkins University Applied Physics Laboratory. After a decade or so at the APL, some of his boards are currently floating in space. Jeffrey recently took a class with IPC instructor Kris Moyer titled "PCB Design for Military and Aerospace Applications."

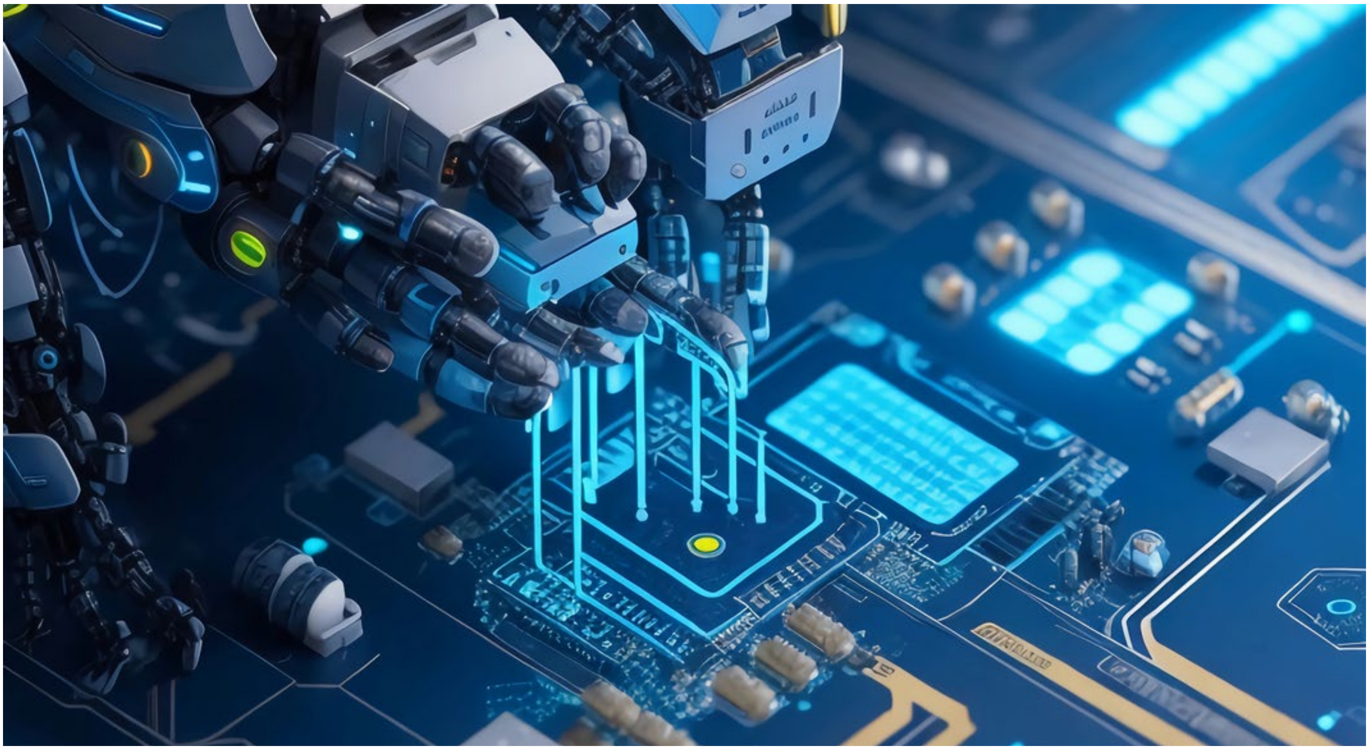
Summit Interconnect Welcomes Brian Kamradt as New Chief Financial Officer ▶

Summit Interconnect announced the appointment of Brian Kamradt as chief financial officer. Kamradt brings over 20 years of finance, accounting, merger and acquisition, and IT experience to Summit's leadership team. He will succeed Tom Caldwell, Summit's initial CFO, who will be retiring after leading the company's expansion for the past four years.

CERcuits Wins Second Place in US Army's xTechInternational Competition with 3D Ceramic Circuits Solution ▶

CERcuits BV, a Belgian technology company specializing in ceramic electronic circuits, proudly announces its achievement as the second-place winner in the prestigious xTechInternational competition organized by the U.S. Army.


In a hurry?
We're the fastest in the business!
• ISO 9001:2015 Certified
• AS 9100 Rev D Certified
• MIL-PRF-31032-C Qualified
• MIL-PRF-55110 Qualified
• ITAR Registered
U.S. AIR FORCE
Prototron Circuits
America's Board Source
Serving the industry for over 35 years
prototron.com
(520) 745-8515



Challenges in Modern PCB Design and Analysis

Feature Article by Steve Watt
ZUKEN

Navigating the complex landscape of modern PCB design presents numerous challenges for designers and the processes they use. Concerns about time to market, performance, system-level design, requirements, costs, and staffing are driving forces that shape the evolution of the modern PCB industry.

Zuken recognizes the need to address innovation in all areas of the design process while remaining responsive to customer-driven requirements. While we can make predictions about various challenges, concerns, and requirements, our customers play a vital role in shaping the company's development direction. They have the benefit of being "in the trenches," so to speak, on a daily basis, so they are the subject matter experts and can help identify gaps in our tools based on where their processes are heading.

System-level Analysis

The new era of electronic systems is characterized by increasing complexity. The demand for higher data rates, lower power consumption, and smaller form factors drive this surge in complexity. As a result, the challenges of managing signal integrity (SI), power integrity (PI), and electromagnetic interference (EMI) are also increasing.

To tackle these challenges, performing system-level SI, PI, and EMI analysis is imperative. This means the entire system must be considered, including different boards, cables, and other components. This comprehensive approach allows engineers to identify and address SI and PI problems before they lead to potential system failures. Engineers can proactively mitigate SI, PI, and EMI problems by delving into system-level analysis, which ensures robust system performance.

In addition to chips and boards, it is crucial for SI and PI analysis to extend to the package level, particularly when designs use innovative packaging technologies, such as chiplet and MEMS. The package itself can introduce unique SI and PI challenges, such as signal reflections and power noise, which necessitate detailed analysis and mitigation strategies. Understanding and addressing these package-level considerations is essential to ensure optimal SI and PI throughout the design.

With today's modern EDA tools, design engineers can leverage the power of system-level SI, PI, and EMI analysis to perform comprehensive evaluations using IBIS models. This advanced capability empowers design engineers to identify and mitigate potential problems at an early stage in the design process, which can help to reduce the time and cost of product development.

Requirements-aware Design and Optimization

The increasing complexity of electronic products is driving the adoption of system-level design and digital engineering methodologies. These new design paradigms take a holistic view of the product and then deconstruct it into subsystems. The architectural subsystems, which typically encompass elements such as PCBs, cables, enclosure data (mechanical), boot loader software, etc., necessitate collecting and documenting various requirements. When considering usage within a PCB data subsystem, the architectural requirements introduce unique specifications and functionalities for analysis, verification, and traceability in the electrical design process.

Ensuring optimized architecture prior to moving into the detailed design stage is more critical than ever. Entering the detailed design stage with a flawed architecture can lead to schedule and cost overruns, plus increase the risk of project cancellation. In cases where a requirement can't be met, such as cost restraints, it is necessary to revisit the architecture proposal and the specific requirement from a system-level perspective. Identifying unachievable design requirements during architecture verification can eliminate the need for costly design changes further downstream.

Artificial Intelligence in PCB Design: Enhancing the Design Process

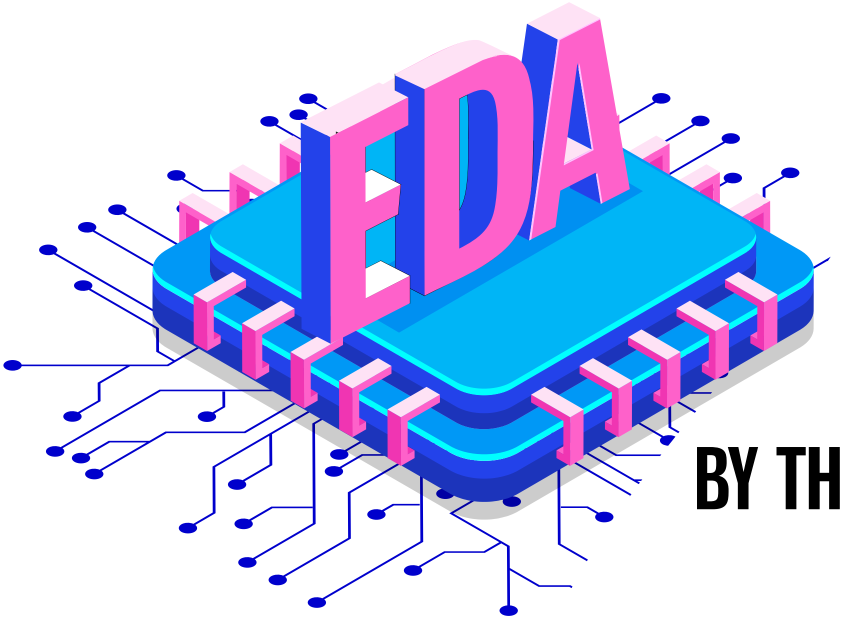
Artificial intelligence is a prominent topic across the technology world, and PCB design is no exception. We have dedicated substantial development efforts in this rapidly growing area of technology. The result is the release of various AI functionalities in our products with additional exciting features on the horizon.

Our primary objective is to ensure an accelerated design process that mimics human-like expertise. This does not imply that we see the future obsolescence of PCB designers—far from it. We see these innovations driving change in the design process. For instance, if AI handles the place and route stage of a design, it's not out of the question that a designer could spawn multiple sessions and assess the viability of each result.

Rather than extending maximum brute force effort on a particular design path and relying on trial and error, designers can quickly explore various layer stackups or line/vias combinations to determine the optimal path forward. This is a simple example of how AI technology could shift the focus of the designers' efforts, and we are confident that other possibilities will emerge as this groundbreaking technology continues to advance.

Zuken is leading the way in AI for PCB design, introducing various new functionalities to accelerate and enhance the design process. While human designers will always be crucial, AI presents new opportunities to reimagine how we approach design. DESIGN007

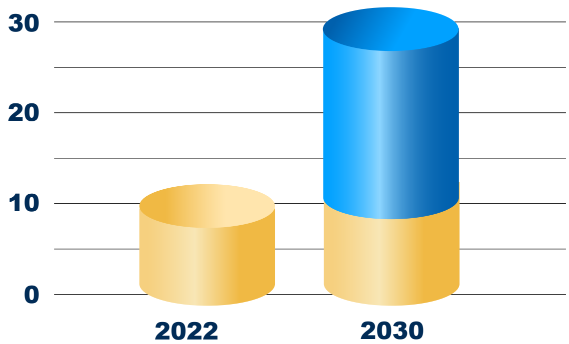
Steve Watt is PCB engineering manager with Zuken.



BY THE NUMBERS

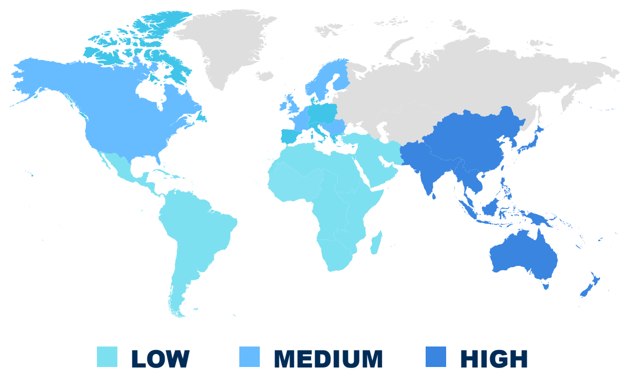
GLOBAL MARKET FOR EDA TOOLS

Market forecast to grow at CAGR of 11%



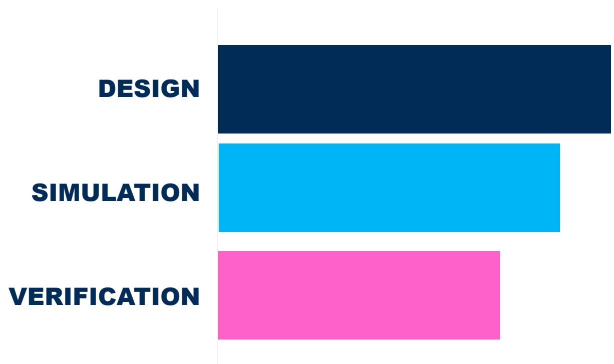
Source: Research and Markets

MARKET GROWTH BY REGION 2022-27



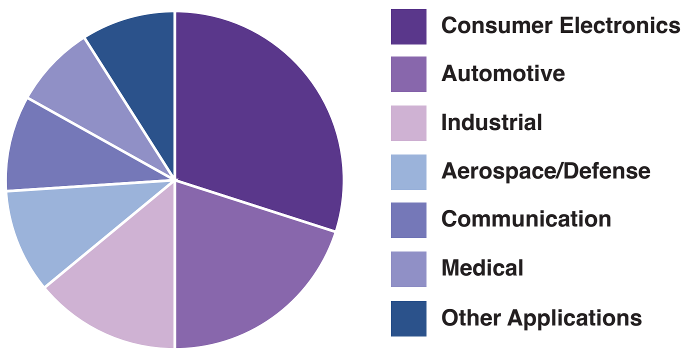
Source: Mordor Intelligence

EDA SOFTWARE MARKET SPLIT BY TYPE, 2023



Source: Fact.MR

EDA SOFTWARE MARKET SHARE BY APPLICATION



Source: Market.US

I-007e FAST FACTS

Model Citizens: Modelithics

A Q&A with Chris DeMartino
MODELITHICS

Simulation models are important weapons in the battle for signal integrity. Because semiconductor companies are often less than helpful in offering detailed models, engineers are left to spend time creating their own models or to acquire models from third parties.

We recently asked Chris DeMartino, an applications engineer at Modelithics, to discuss the company's focus on providing simulation models, primarily for the RF and microwave segments. In this interview, Chris explains the Modelithics business "model" and why the need for good models continues to grow at a rapid pace.

Chris, let's start with some background. Tell us about Modelithics.

Modelithics has specialized in RF, microwave, and millimeter-wave measurements, and measurement-based, as well as electromagnetic analysis-based modeling of RF and microwave components and semiconductor devices, since 2001. We offer an extensive library collection of models for both passive and active components that are compatible with multiple industry-standard EDA software tools. These libraries include scalable models (known as Modelithics Microwave Global Models) for RLC components, nonlinear models for diodes and transistors, and 3D models for full-wave electromagnetic (EM) simulations, as well as higher level system component models. In terms of measurement capabilities, we have a large repertoire of test-and-measurement equipment that make it possible for the company to measure S-parameters, IV, load pull, noise parameters, ESR, power, load-pull, and non-linear distortion characteristics.

Modelithics has partnered with a variety of EDA companies. How does this all work?

Modelithics has long-standing partnerships with many EDA software companies. Through these partnerships, we provide model "plug-n-plan" libraries for several different EDA software tools. Customers can obtain these libraries, which can be considered an add-on to the software tool itself. We work closely with the EDA companies to ensure the models stay compatible with the latest EDA software versions as well as new model features and capabilities. For example, new library versions are typically released immediately after a new EDA software version is released so that the EDA software companies and mutual customers can use the most recent EDA software versions with our libraries.

Chris, you say that you have the "world's best RF and microwave simulation models." What are some of the modeling challenges that you help your RF and microwave customers to address?

When Modelithics first started in 2001, the company's goal was to address a need in the industry for quality models for RF/microwave components held to a high accuracy standard. While some vendors have good models, we are unique in the industry for providing accurate models with standardized features for over 70 vendors. We pioneered the idea of a model information datasheet for microwave device models. Whereas all vendors have datasheets for their devices, we have extensive datasheets on the models we have developed for those devices, giving many details of the models' ranges of validity, scalability, and much more information benefitting designers. What we have accomplished is we have brought the type of first pass design success required for semiconductor-based ICs to the discrete PCB and thin-film board-based design space.

What was true when we started 22 years ago is true now. When simulations are performed with inadequate models, the simulated results will likely fall short of accurately predicting real-life performance. This discrepancy often results in multiple printed-circuit-board (PCB) iterations and/or a great deal of bench tuning, lengthened time to market, and considerable real and opportunity costs and inefficiencies.

Chris DeMartino

On the other hand, these models make it possible to accurately predict real-life performance on the first attempt. This first-pass design success not only lets companies reduce costs but it shortens schedules and improves overall productivity.

In addition to providing simulation models, you offer characterization and modeling services. Walk us through these services.

We have a large amount of test-and-measurement equipment at our disposal. For instance, in terms of S-parameter measurement capability, multiple vector-network-analyzer (VNA) test platforms cover frequencies ranging from 5 Hz to 170 GHz. Load-pull and noise-parameter measurements can be performed all the way to mmWave frequencies. We can also perform DC and pulsed IV measurements for diodes and transistors.

In terms of modeling services, some of these capabilities were mentioned earlier. For RLC components, we offer Microwave Global Models, which scale with respect to part values, substrates, and solder-pad dimensions. In terms of part-value scalability, a single Microwave Global Model can cover a full range of part values for a vendor part series, enabling these models to be ideal for tuning and optimization, while taking into account substrate-dependent parasitic behavior of the component.

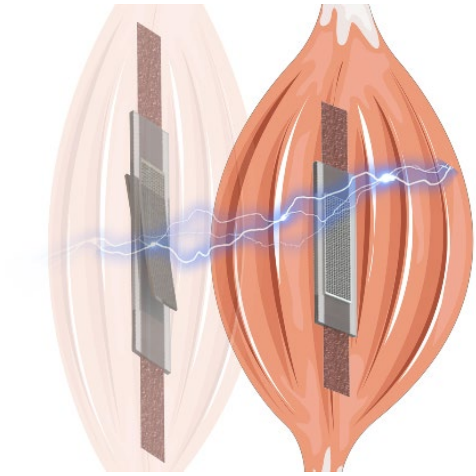
Gallium-nitride (GaN) nonlinear device modeling is another area that we are very focused on. These world-class GaN models are made possible by some of the measurement capabilities mentioned earlier. GaN models include advanced features like bias and temperature dependency and intrinsic IV access for waveform analysis.

Another area to mention is 3D modeling. We offer encrypted 3D models based on detailed physical dimensions and material properties. These models make it possible for designers to perform simulations that capture the coupling effects that can occur when components are located very close to one another.

I've always wondered why semiconductor companies are so reluctant to provide accurate models. What are your thoughts on this?

Developing accurate models can be a major challenge for companies. For one, since new models must constantly be developed, companies must allocate a great deal of resources just to modeling. A team of RF engineers focused on modeling would be the ideal scenario for a company that wants to develop enough models. However, companies must also decide how many RF engineers should be focused on modeling as opposed to product design. In certain cases, it may make more sense for a company to outsource model development so that it can focus more on design. This is where we can help. Instead of having to tie up resources to develop models, companies can turn to us. DESIGN007

Revolutionary Self-Sensing Electric Artificial Muscles



Taking inspiration from nature, a team of researchers at Queen Mary's School of Engineering and Materials Science has successfully created an artificial muscle that seamlessly transitions between soft and hard states while also possessing the remarkable ability to sense forces and deformations.

Dr. Ketao Zhang, a lecturer at Queen Mary and the lead researcher, explains the importance of variable stiffness technology in artificial muscle-like actuators. "Empowering robots, especially those made from flexible materials, with self-sensing capabilities is a pivotal step towards true bionic intelligence," says Dr. Zhang.

The cutting-edge artificial muscle developed by the researchers exhibits flexibility and stretchability similar to natural muscle, making it ideal for integration into intricate soft robotic systems and adapting to various geometric shapes.

The potential applications of this flexible variable stiffness technology are vast, ranging from soft robotics to medical applications. The seamless integration with the human body opens up possibilities for aiding individuals with disabilities or patients in performing essential daily tasks.

The groundbreaking study conducted by researchers at Queen Mary University of London marks a significant milestone in the field of bionics. With their development of self-sensing electric artificial muscles, they have paved the way for advancements in soft robotics and medical applications.

(Source: Queen Mary University of London)

Flex007 Highlights

Flexible Thinking: Stretching Conductors—and Design Possibilities ▶

Stretchable circuits, also referred to as elastic circuits or even "elastronics," are a subset of the venerable flexible circuits that have enabled countless numbers of today's electronics products, from toys to smartphones to the International Space Station. This most recent "member" of the interconnection family—stretchable circuits—has been designed and engineered to be resiliently bent, twisted, and/or stretched in support of the end product's need without negatively affecting its electrical function.

Designing and Manufacturing Wearable Biosensors ▶

To cost-effectively mass produce wearable biosensors, vertical integration of manufacturing and assembly operations is key. This involves printing conductive inks on flexible substrates and successfully performing converting operations (such as lamination of medical-grade hydrocolloids, adhesives, non-woven and foam layers, hydrogel dispensing and/or placement and final packaging) in surface-mount technology (SMT) components.

PCB Technologies Invests in Advanced Packaging ▶

Jeff De Serrano, president of PCB Technologies North America, gives an update on the company's recent move into advanced packaging—a move that stemmed from company leadership and vision for the future. He also shares his forecast on the PCB market, specifically around rigid-flex, along with some of the challenges the industry still faces.

American Standard Circuits Acquires Sunstone Circuits, Joins Forces to Enhance Customer Value in the PCB Industry ▶

American Standard Circuits (ASC) and Sunstone Circuits are excited to announce their merger, creating a dynamic partnership that will revolutionize the printed circuit board (PCB) manufacturing landscape. This strategic alliance brings numerous advantages to customers, establishing a new standard of excellence in PCB solutions.

Excello Circuits to Showcase Rigid-Flex/Flex & PCB Capabilities at AEMS 2023 ▶

Excello Circuits, a dedicated quick-turn rigid-flex, flex and PCB manufacturer, will be exhibiting at this year's Anaheim Electronics Manufacturing Show (AEMS), scheduled for Sept. 27-28, 2023. The annual gathering brings together some of the most innovative minds in the industry and we're excited to be a part of it.

IDTechEx: Printed Electronics Progresses Towards Greater Commercial Adoption ▶

How far is printed/flexible electronics along the road to widespread adoption? The emerging manufacturing methodology, which replaces etched copper laminate with printed conductive inks on flexible substrates, offers multiple benefits, including flexibility/stretchability, the potential for low-cost roll-to-roll production, digital manufacturing/rapid prototyping, and improved sustainability.

The Growth of Flex-hybrid Electronics in Mil-Aero Applications

Flexible Thinking

by Joe Fjelstad, VERDANT ELECTRONICS

Over the past several years, flexible electronics (FE) and flex-hybrid electronics (FHE) have enjoyed heightened attention in the electronics industry and have seen special interest and attention given by mil-aero companies. This is evidenced by June's NextFlex conference titled "Hybrid Electronics Commercialization Path for Aerospace Applications," an event at Boeing's Seattle facility. It was well attended by a diverse group of U.S. suppliers of materials, processes, equipment, and products for the aerospace industry as well as several participants from academia.

The reasons for interest are becoming ever clearer as the technology, like its traditional flexible circuit elder siblings, offers advantageous applications in the military and aerospace industries, where lightweight, conformable, and robust electronics are highly desirable.

It is thus worth considering some specific examples of FHE applications in the mil-aero environment.

Wearable Electronics

One application for flex-hybrid circuits is wearable electronics. While the first wearables in the most recent rise of interest in FHE have been for fashion application, such as LED-lighted dresses, the idea of wearable electronics for soldiers caught attention early on and the ground soldier of today is increasingly

dependent on electronics to carry out assigned missions.

This particular application is near and dear to me as I was a member of the Land Warrior Development Team at my former company, Pacific Consultants, where, in 2000-01, our small company of engineers and scientists demonstrated the concept of a local area network of soldiers wearing Smart uniforms, each with computers, radios, and heads-up displays to enhance situational awareness of team members. One significant benefit was preventing fratricide (which has unfortunately been the cause of far too many friendly fire casualties in former conflicts) by identifying friendlies on the display. Additional capabilities included gun sight cameras and night vision capabilities. The ultimate objective has always been to improve both the soldier's performance and safety on the battlefield. More advanced systems will include integrated sensors for monitoring vital signs, hydration levels, and environmental factors. Making the warrior safer and more effective is crucial.

Aboard the Aircraft

Another area of interest is flexible displays and control panels where FHE technologies can be used for aircraft and spacecraft. Advantageously, flexible displays can be integrated into not just flat, but also curved surfaces, providing critical information to pilots or crew members, while saving valuable cockpit space.

Antennas and Sensors

Likewise, conformal antennas and sensors are another area of interest, as FHE technologies facilitate the creation of conformal antennas and sensors that can be seamlessly integrated into the surface of military vehicles, aircraft, and naval vessels. Having conformal antennas can improve aerodynamics and reduce the overall weight of the vehicle while maintaining essential communication and sensing capabilities.

UAVs

Drones and unmanned aerial vehicles (UAVs), as well as unmanned autonomous ground vehicles, are beneficiaries of FHE. They make the UAVs lighter in weight, more efficient, and more mission capable. Flexible electronics enable unique design possibilities, leading to more advanced and more stealthy UAVs for reconnaissance, surveillance, and other military applications. The ongoing conflict in Ukraine has, for better or worse, become a proving ground of these increasingly important war-fighting technologies.

Flex-hybrid electronics, again like their traditional flexible circuit kin, are well suited to ruggedizing electronic systems in military and aerospace environments. It is well known that electronic components in mil-aero applications must withstand harsh conditions, including shock, vibration, and extreme temperatures. Flexibility is a key feature in mitigating such conditions.

Health Monitoring

Health monitoring is not just for the human soldier, it is also a concern for aircraft of every sort. Thus, interest in "smart skins" and aircraft structural health monitoring is being targeted for and enabled by FHE where flexible sensors both monitor and detect damage, fatigue, or stress in real-time, enhancing maintenance and safety. In recent years, prognostics has become an increased area of interest in both mil-aero and consumer environments; FHE tools will undoubtedly see increased use.

Electronic Warfare

With the ever-increasing reliance of electronics by militaries around the world, electronic warfare (EW) systems are, unsurprisingly, also of ever-increasing interest. Flex and flex-hybrid electronic technologies can foreseeably be used in EW systems for intelligence monitoring and gathering as well as for other electronic countermeasure applications.

Again, the conformal nature of the products enables their near seamless integration into various EW platforms.

Cybersecurity

Closely coupled with electronic warfare is, unsurprisingly, cybersecurity. Once again, flexible circuit technologies can offer enhanced security in military communications and data transmission. Sealed flexible circuit designs include built-in EMI and ESD shielding to make it difficult to physically tamper with the circuitry while at the same time providing an additional layer of protection against wireless electrical and electronic threats.

Satellites and Space

Last but certainly not least are satellite and space applications for flex and flex-hybrid electronics. The technologies have long been employed in such products to create flexible solar panels, lightweight antennas, and various electronic sensors of many varieties. Flex and FHE designs enable more efficient use of space while reducing weight, both long considered critical factors in the aerospace industry.

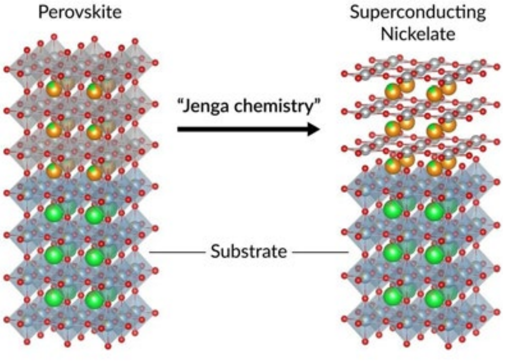
This is not an exhaustive recitation of where and how flex-hybrid electronics are helping to advance military and aerospace industries, but it is reasonably representative. As familiarity with flexible circuit and flex-hybrid electronics technologies continue to advance, there will unquestionably be more innovative applications coming into existence, providing ever greater efficiency, reliability, and versatility to the products. **DESIGN007**

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

A Foundation That Fits Just Right Gives Superconducting Nickelates a Boost

Researchers at the Department of Energy's SLAC National Accelerator Laboratory and Stanford University say they've found a way to make thin films of an exciting new nickel oxide superconductor that are free of extended defects.

Their first look at a superconducting nickel oxide, or nickelate, that does not have defects revealed that it is more like the cuprates, which hold the world's high-temperature record for unconventional superconductivity at normal pressures, than previously thought.



It's the latest step in a 35-year quest to develop superconductors that can operate at close to room temperature, which would revolutionize electronics, transportation, power transmission and other technologies by allowing them to operate without energy-wasting electrical resistance.

The research team, led by Harold Hwang, director of the Stanford Institute for Materials and Energy Sciences (SIMES) at SLAC, described their work in the journal Nature.

"Nickelate films are really unstable, and until now our efforts to stabilize them on top of other materials have produced defects that are like speed bumps for electrons," said Kyuho Lee, a SIMES postdoctoral researcher who contributed to the discovery of superconductivity in nickelates four years ago.

(Source: SLAC)

TOP TEN

EDITOR'S PICKS

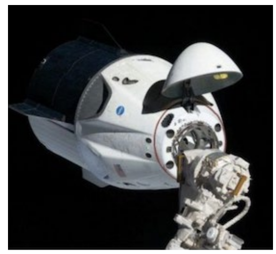
RF Antenna Design on the Bleeding Edge

At SMTA Atlanta Tech Expo and Forum, I met with PCB designer Albert Gaines, owner of HiGain Design Services. Albert has been working on some really interesting, fragmented aperture antenna designs, and some of this stuff is really pushing the limits. Albert and I discussed his work with RF, the differences between COTS and custom antennas, and his efforts to educate engineers about what they can and can't do.

New Book Strengthens Relationship Between Designer and Manufacturer

This book introduces a new process workflow for optimizing your design called *Manufacturing Driven Design (MDD)* and is a distinct evolution from DFM. When defining Manufacturing Driven Design, it is important to recognize that this is, foremost, an element of the design stage.

IPC to Hold PCB Design for Military & Aerospace Applications Training Course

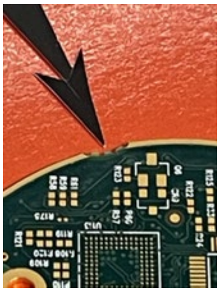


IPC announces its virtual class for PCB Design for Military & Aerospace Applications, Aug. 7–Sept. 13, 2023. This course addresses specific challenges encountered in military and aerospace applications, including the effects of vibration, shock, radiation, and altitude, extended operating temperature range, and other design considerations for high-reliability applications.

Mark Thompson's Biggest Problems With PCB Designs

What are the top problems I see with PCB design? From where I sit now on the assembly side, one of my biggest concerns related to PCB design is the lack of uniform part markings on the Gerber or ODB++ data, specifically the way customers reference diodes. We would prefer either an "A" depicting the anode side or a "C" or "K" for the cathode side.

What Are Hiring Managers Looking For?



Paul Farquhar recently took a few PCB design classes from John Watson, who doubles as a Palomar College professor when he's not working at Altium. I asked Paul to discuss what he learned in John's classes, as well as where he hopes to work afterward and how John and the college are working with industry to provide trained designers for the many open PCB designer positions.

Mil-Aero Design: Not Just Another High-Rel Board

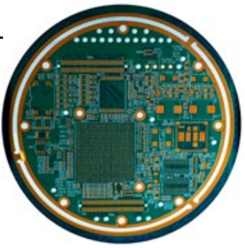
Meijing Liu, CID+, is a senior PCB designer for Microart Services, an EMS company in Markham, Ontario, Canada. She recently took a six-week mil-aero PCB design class from IPC's Kris Moyer, and she was surprised at how much content she was able to absorb in such a short time. I spoke with Meijing and we discussed some of her takeaways from the class.

Beyond Design: Standing Waves in Multilayer PCB Plane Cavities

Plane cavities in multilayer PCBs are essentially unterminated radial transmission lines. They form a transmission line that propagates electromagnetic (EM) energy within a plane cavity emanating from a feed point within the plane and outward in all directions. Like all transmission lines, it will reflect if not terminated. This creates standing waves—ringing.

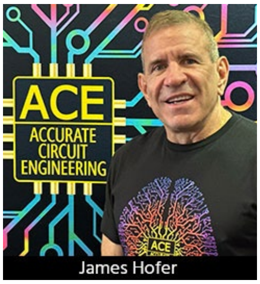
Elementary, Mr. Watson: Honey, I Shrunk the PCBs

As an industry, we live in our own version of "Honey, I Shrunk the Kids." PCB designs are shrinking smaller and smaller with each design spin. Our industry demands the latest and greatest, where innovations coming off the line must be smaller and sleeker and have all the latest new functions, which, as we know, determines the fit and form.



Are You Still Over-materializing?

During the recent IEEE IMS2023 Exhibition in San Diego, we caught up with James Hofer, general manager of Accurate Circuit Engineering (ACE). A few years ago, we talked to James about the over-materialization of boards. In this recent discussion, we wanted to know if anything has changed regarding materials and approach. James also shares his thoughts on the changes in the design community.



James Hofer

Introducing Pulsonix Version 12.5: Empowering PCB Designers with Enhanced Features and Efficiency

Pulsonix, the Electronic Design Automation (EDA) company delivering technology-leading PCB design solutions, announces the release of Pulsonix Version 12.5, a significant update that brings a host of new features and enhancements to the PCB design industry.

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

Career Opportunities

Technical Support Engineer

USA Region

ViTrox aims to be the world's most trusted technology company in providing innovative, advanced, and cost-effective automated Machine Vision Inspection Solutions for the semiconductor and electronics packaging industries. Located in Hayward, California, ViTrox Americas Inc. is actively looking for talent to join our expanding team.

Key Responsibilities:

- Delivering excellent and creative problem-solving skills for servicing, maintaining, machine buy-off, and troubleshooting advanced vision inspection machines at customer sites. Providing remote customer support to minimize machine downtime.
- Cultivating strong customer relationships and ensuring comprehensive customer service to drive repeat orders and support business development in machine evaluation.
- Proactively understanding customer needs and feedback to drive continuous improvement in existing technologies and new product development.

Qualifications & Requirements:

- A recognized diploma/advanced diploma/degree in Science and Engineering, preferably in Electrical & Electronics/Computer Science/Computer Studies or equivalent.
- 3+ years of relevant experience in servicing automated inspection equipment (SPI, AOI, and AXI).
- Strong communication and troubleshooting skills.
- Willingness to travel extensively across the USA.
- Positive attitude and flexibility to accommodate conference calls with headquarters.
- Applicants from the USA and Canada are welcome to apply.
- Training will be provided at our headquarters in Penang, Malaysia.

apply now

The Test Connection, Inc.

Rewarding Careers

Take advantage of the opportunities we are offering for careers with a growing test engineering firm. We currently have several openings at every stage of our operation.

The Test Connection, Inc. is a test engineering firm. We are family owned and operated with solid growth goals and strategies. We have an established workforce with seasoned professionals who are committed to meeting the demands of high-quality, low-cost and fast delivery.

TTCI is an Equal Opportunity Employer. We offer careers that include skills-based compensation. We are always looking for talented, experienced test engineers, test technicians, quote technicians, electronics interns, and front office staff to further our customer-oriented mission.

Associate Electronics Technician/ Engineer (ATE-MD)

TTCI is adding electronics technician/engineer to our team for production test support.

- Candidates would operate the test systems and inspect circuit card assemblies (CCA) and will work under the direction of engineering staff, following established procedures to accomplish assigned tasks.
- Test, troubleshoot, repair, and modify developmental and production electronics.
- Working knowledge of theories of electronics, electrical circuitry, engineering mathematics, electronic and electrical testing desired.
- Advancement opportunities available.
- Must be a US citizen or resident.

apply now

Career Opportunities

THERMAL SYSTEMS

Sales Engineer SMT North Mexico

Rehm Thermal Systems, a leading German manufacturer of reflow soldering systems with convection or condensation and drying and coating systems, has produced energy-efficient manufacturing equipment for the electronics and photovoltaics industry since 1990. We also offer tailor-made applications related to the soldering, coating and hardening of modules.

Responsibilities:

- This position is responsible for expanding our customer network and maintaining existing customer relationships in the Northeast Mexico region. The Sales Engineer would work closely with the German headquarters and the General Manager Rehm Mexico to implement the sales strategy.
- A candidate's proximity to Monterrey, Mexico, is a plus.

Qualifications:

- An engineering degree or comparable qualification with a strong technical background is required.
- Sales-oriented attitude, good communication skills and willingness to travel frequently within Mexico is essential.

We offer innovative products, a great dynamic work environment and exciting training opportunities in our German headquarters.

To learn more about Rehm Group, please visit our website at www.rehm-group.com.

Please send resumes to: Mr. Luis Garcia at luis.garcia@rehm-group.com.

apply now

TAIYO AMERICA

Europe Technical Sales Engineer

Taiyo is the world leader in solder mask products and inkjet technology, offering specialty dielectric inks and via filling inks for use with microvia and build-up technologies, as well as thermal-cure and UV-cure solder masks and inkjet and packaging inks.

PRIMARY FUNCTION:

1. To promote, demonstrate, sell, and service Taiyo's products
2. Assist colleagues with quotes for new customers from a technical perspective
3. Serve as primary technical point of contact to customers providing both pre- and post-sales advice
4. Interact regularly with other Taiyo team members, such as: Product design, development, production, purchasing, quality, and senior company managers from Taiyo's group of companies

ESSENTIAL DUTIES:

1. Maintain existing business and pursue new business to meet the sales goals
2. Build strong relationships with existing and new customers
3. Troubleshoot customer problems
4. Provide consultative sales solutions to customers technical issues
5. Write monthly reports
6. Conduct technical audits
7. Conduct product evaluations

QUALIFICATIONS / SKILLS:

1. College degree preferred, with solid knowledge of chemistry
2. Five years' technical sales experience, preferably in the PCB industry
3. Computer knowledge
4. Sales skills
5. Good interpersonal relationship skills
6. Bilingual (German/English) preferred

To apply, email: BobW@Taiyo-america.com with a subject line of "Application for Technical Sales Engineer."

apply now

Career Opportunities

BLACKFOX
Premier Training & Certification

IPC Instructor

Longmont, CO

This position is responsible for delivering effective electronics manufacturing training, including IPC certification, to adult 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 primarily conduct training at our public training center in Longmont, Colo., or will travel directly to the customer's facility. It is highly preferred that the candidate be willing to travel 25–50% of the time. Several IPC certification courses can be taught remotely and require no travel or in-person training.

Required: A minimum of 5 years' experience in electronics manufacturing and familiarity with IPC standards. Candidates with current IPC CIS or CIT Trainer Specialist certifications are highly preferred.

Salary: Starting at $30 per hour depending on experience

Benefits:
- 401k and 401k matching
- Dental and Vision Insurance
- Employee Assistance Program
- Flexible Spending Account
- Health Insurance
- Health Savings Account
- Life Insurance
- Paid Time Off

Schedule: Monday thru Friday, 8–5

Experience: Electronics Manufacturing: 5+ years (Required)

License/Certification: IPC Certification–Preferred, Not Required

Willingness to travel: 25% (Required)

apply now

Prototron Circuits

Sales Representatives

Prototron Circuits, a market-leading, quick-turn PCB manufacturer located in Tucson, AZ, is looking for sales representatives for the Southeastern U.S. territory. 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

Career Opportunities

KYZEN

Where Science and Care Converge.

Regional Manager

West Region — Two Positions

General Summary: Manages sales of the company's products and services, Electronics and Industrial, within the Pacific Northwest or Southwest Region. Reports directly to Americas Manager. Collaborates with the Americas Manager to ensure consistent, profitable growth in sales revenues through positive planning, deploy-ment 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

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

Career Opportunities

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

KOH YOUNG AMERICA

Technical Service & Applications Engineer

Full-Time — Flexible Location

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

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

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 to 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

Career Opportunities

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

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

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

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.

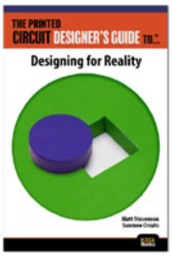
apply now

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

NEW! Manufacturing Driven Design

by Max Clark, Siemens

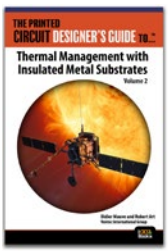
This book introduces a new process workflow for optimizing your design called Manufacturing Driven Design (MDD) and is a distinct evolution from DFM. When defining Manufacturing Driven Design, it is important to recognize that this is, foremost, an element of the design stage. Manufacturing certainly plays a critical role in this process change, and manufacturers do certainly benefit from the improved process, but it is design teams that ultimately own their overall product workflow; they are the ones who need to drive this shift. Design teams are already invested in the success of their product; they just need to be empowered to control all the factors that go into this success. Get empowered now!



Designing for Reality

by Matt Stevenson, Sunstone Circuits

Based on the wisdom of 50 years of PCB manufacturing at Sunstone Circuits, this book is a must-have reference for designers seeking to understand the PCB manufacturing process as it relates to their design. Designing for manufacturability requires understanding the production process fundamentals and factors within the process. Read it now!



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!

The Companion Guide to... Flex and Rigid-Flex Fundamentals

This compact guide, written as a companion to The Printed Circuit Designer's Guide to... Flex and Rigid-Flex Fundamentals by topic experts at American Standard Circuits, is designed to provide additional insights and best practices for those who design or utilize flexible and/or rigid-flex circuit boards. Topics covered include trace routing options, guidelines for process optimization, dynamic flexing applications, rigid-to-flex transition and more. Find out what it's all about.

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

BUSINESS DEVELOPMENT: **TAMARA SITES**
tamara@iconnect007.com

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

ART DIRECTOR/PRODUCTION MGR. **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 © КОНСТЯНТИН БАТЫЛЬЧУК

DESIGN007 MAGAZINE

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

August 2023, Volume 12, Number 8
DESIGN007 MAGAZINE is published monthly
by IPC Publishing Group, Inc., dba I-Connect007

ADVERTISER INDEX


Problems solved!
PCB007 MAGAZINE MAY 2023
PCB007 MAGAZINE JUNE 2023
Growing Your Workforce
DESIGN007 MAGAZINE JUNE 2023
DESIGN007 MAGAZINE JULY 2023
Advanced, Complex, and Emerging Design Strategies
DESIGN007 MAGAZINE
SMT007 MAGAZINE JULY 2023
SMT007 MAGAZINE AUGUST 2023
FLYING HIGH WITH DIGITAL TWIN
SUBSCRIBE NOW