SEPTEMBER 2021

I-Connect007

DESIGN007 MAGAZINE

Collaboration by Design

DESIGN007 MAGAZINE

Collaboration by Design

In recent surveys, designers pointed to a lack of collaboration with fabricators as among their biggest ongoing challenges. And some respondents noted that when they did work with their fabricators, it was late in the design cycle, usually fixing simple design errors that could have been avoided by communicating with the CAM engineer earlier in the process. Does that sound like your design team's relationship with your fabricator?

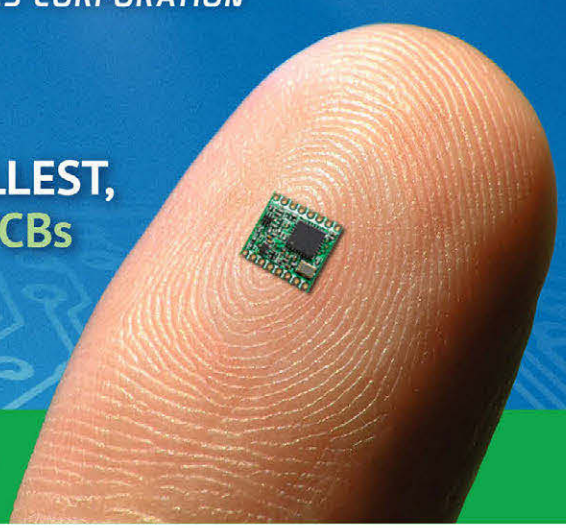
ENABLING NEXT-GEN SYSTEMS

DOMESTIC MANUFACTURING READY

Advanced additive manufacturing down to 15 µm features.
IC substrates and highly integrated HDI PCB solutions.
Phased array beamforming technologies.
100% American owned. 100% American made.
America's only HUBZone PCB manufacturer.

Calumet
ELECTRONICS CORPORATION

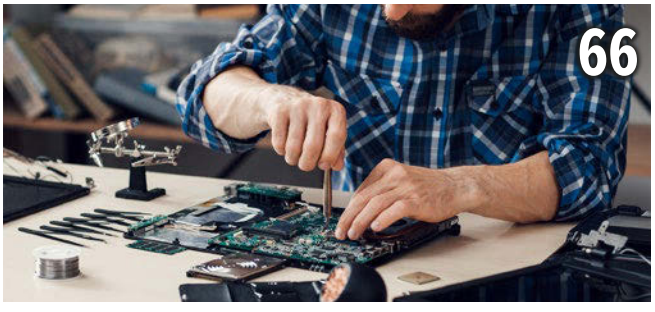
AMERICA'S SMALLEST,
LIGHTEST, FASTEST PCBs



Leaders in Averatek A-SAP™

NADCAP | HUBZONE | AS9100D | IPC-1791 | TRUST ITAR
ISO9001 | MIL-PRF-31032 | MIL-PRF-55110 | CAGE CODE | UL

DESIGN007 MAGAZINE



isola
THE BASE FOR INNOVATION

THE NEW TERRAGREEN® 400G

Extremely Low Loss Laminate and Prepreg

UNLEASH THE POWER WITHIN

Our most advanced ultra high speed, material solution for next generation 5G infrastructure and computing applications.

FEATURES

- HIGH SPEED DIGITAL
- 5G APPLICATION
- HALOGEN FREE

MARKETS

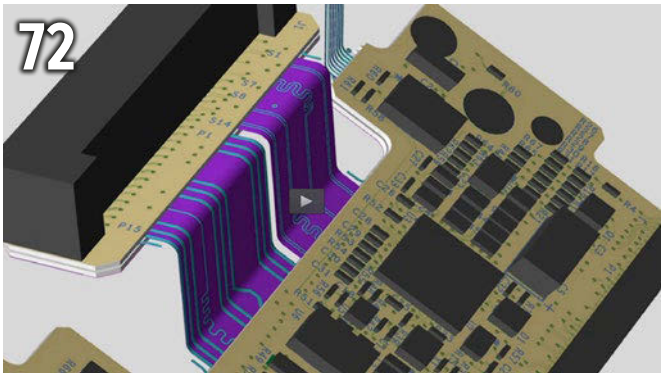
- COMPUTING, STORAGE & PERIPHERALS
- AEROSPACE & DEFENSE
- NETWORKING & COMMUNICATIONS

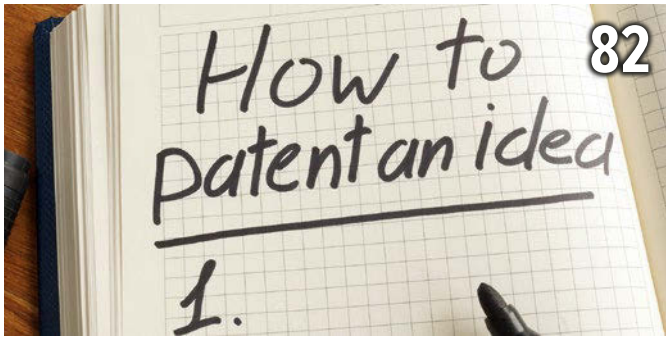
info@isola-group.com
800-537-7656
isola-group.com

FLEX007

This Month in Flex

By their very nature, flexible and rigid-flex circuits can benefit greatly from the "left-shift" of manufacturing-aware capabilities earlier in the design process. This month, Patrick McGoff of Siemens EDA outlines how DFM analysis tools can shrink the distance between flex designers and manufacturers—and the design cycle.





SUMMIT ACQUIRES EAGLE ELECTRONICS

READ THE PRESS RELEASE HERE

SUMMIT INTERCONNECT

EAGLE electronics Inc.

summit-pcb.com

eagle-elec.com

Collaboration by Design

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

For the past year, we've been discussing $X = X_c - 1$, the idea that big gains in efficiency can be made just by making one small change to a process. We've spoken with technologists across the PCB design, fabrication, and assembly segments, and they've shared stories about how one small tweak to a process often leads to savings of hundreds of man-hours and thousands—or millions—of dollars.

There seems to be a general realization that the most effective way to make process improvements is not through a revolutionary, disruptive change, but in small increments that can be made almost immediately, without making a department learn an entirely new process. And let's face it: Disruptive changes can turn managers and employees into very unhappy campers. Thus, it was with an eye on $X = X_c - 1$ that we began planning this issue of *Design007 Magazine*.

We began, as we often do, by looking at our survey data. In recent surveys, designers pointed to a lack of collaboration with fabricators as among their biggest ongoing challenges. And some respondents noted that when they did work with their fabricators, it was late in the design cycle, usually fixing simple design errors that could have been avoided by communicating with the CAM engineer earlier in the process. Does that sound like your design team's relationship with your fabricator?

After poring over the survey results, we realized that many of our readers are in rocky relationships—if you can call them relationships—with their chosen fabricators. There's not only a lack of collaboration, but sometimes no communication until that fateful 5:30 p.m. Friday phone call from the board shop putting a job on hold. A successful design almost feels like an accident at that point.

Fortunately, some design teams work with their manufacturers as true partners. We discovered some designers and fabricators who have their process locked in, akin to the gears in a fine Swiss watch. So, for this issue, we asked a variety of contributors to discuss their collaborative relationships, as well as the advantages, incremental and otherwise, that accrue from such collaboration.

We kick off with an interview with Patrick Davis of Cadence Design Systems. Patrick draws on his experience on the manufacturing floor to address key areas where designers and fabricators can work together to streamline the design process. Altium's John Watson has a feature column about the need to view PCB design as a "team sport." Calumet's Ian Huibregtse explains why a good board shop will welcome your questions and act as a "sounding board" for PCB designers. Rich Tighe discusses the role of EDA tools in helping bridge the designer-fabricator gap. Columnist Tim Haag explains why designers must build solid relationships with every fabricator they use, and the benefits of today's EDA tools and intelligent data transfer formats. And Patrick McGoff of Siemens EDA outlines how DFM analysis tools can shrink the distance between flex designers and manufacturers—and the design cycle.

We have a collection of columns from our regular contributors Barry Olney, Phil Kinner, Kelly Dack, Matt Stevenson, Tara Dunn, and Joe Fjelstad. We also have another article by American Standard Circuits' Anaya Vardya that continues his "DFM 101" series. This month, Anaya focuses on layer stackup.

So, check out this month's features; you might learn exactly what you need to know to establish a tight relationship with your fabricator. And once you're in a true collaborative relationship with your fabricator, you start to see the benefits of the incremental changes described by $X = X_c - 1$.

It's almost the end of summer, and it's going to be a busy fall. We hope to see you at PCB West, SMTA International, PCB Carolina, and IPC APEX EXPO. Let's hold a good thought that these shows are permitted to take place as live, in-person events. See you next month! **DESIGN007**

Andy Shaughnessy is managing editor of *Design007 Magazine.* He has been covering PCB design for 20 years. He can be reached by clicking here.

Collaboration: The Key to Streamlining Your Design Process

Feature Interview by the I-Connect007 Editorial Team

We recently spoke with Patrick Davis, product management director of Cadence Design Systems, about the benefits of collaboration between PCB designers and fabricators—and the drawbacks to throwing designs "over the wall." Patrick explains why designers and fabricators often seem to be talking past each other, the role that EDA software companies can play in facilitating collaboration, and why IPC-2581 may be the right means for connecting all of design's stakeholders.

Andy Shaughnessy: Patrick, please share your thoughts on collaboration between designers and fabricators, and where it all begins.

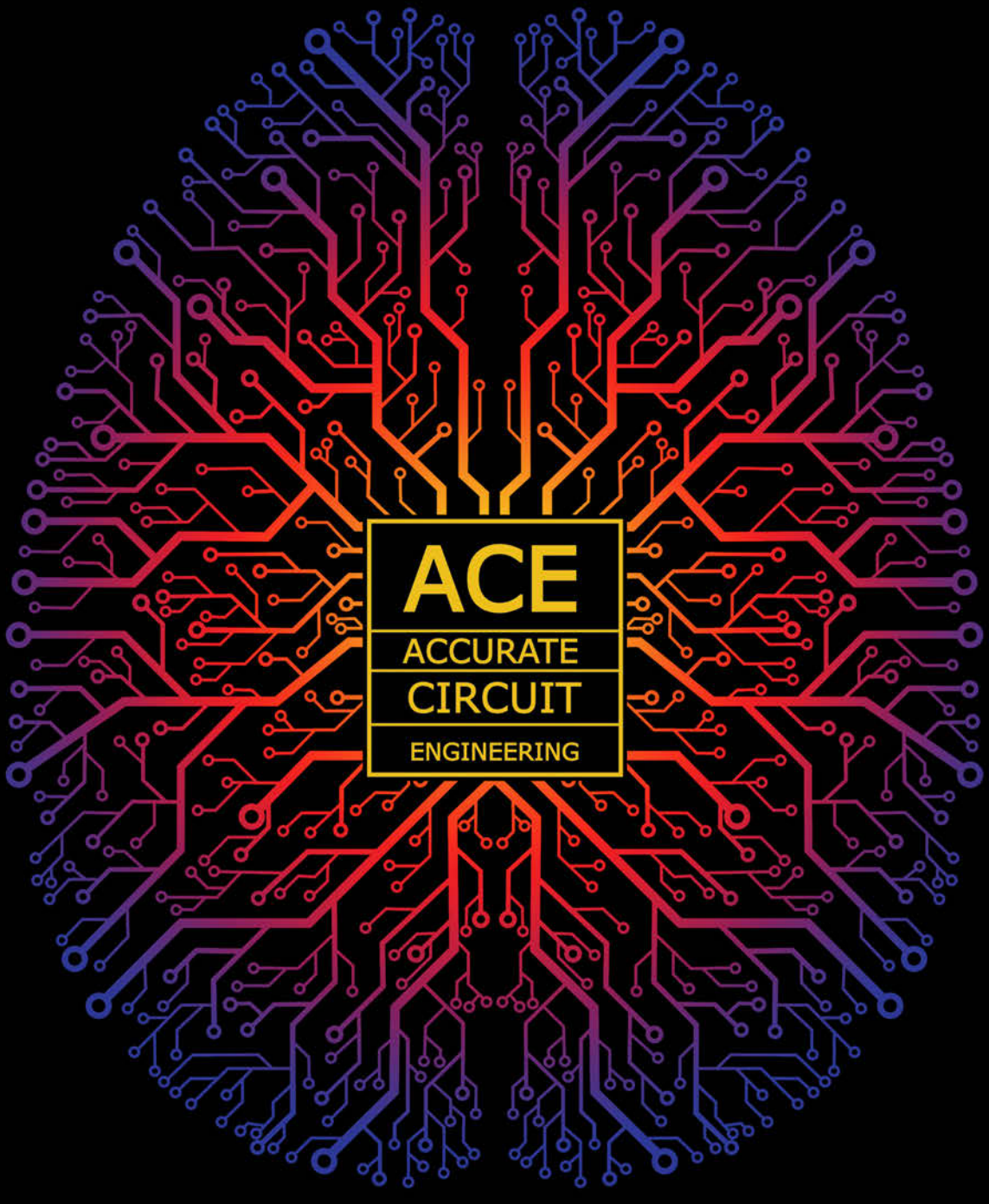
Patrick Davis: I think the collaboration is extremely important. It's that last mile, as they call it, when you come out of the design process and you're trying to get it to the fab shop, the assembly shop, in an efficient manner. Right now, the communications are through email, PowerPoint, Excel spreadsheet, carrier pigeon, and a few other ways to get your data back and forth.

We want to streamline that and make a digital thread to pull everything together. I like the idea of digital rope because the ends spread out, you have a lot of places in the very beginning and a lot of places at the very end, but it's all one common rope going back and forth.

Cadence believes that this is pretty important, why we are so heavily invested in the IPC-2581 standard, and why we're trying to push that forward. We think it's the right tool to be able to help everybody communicate up and down the line, through the entire design process.

Shaughnessy: Designers say there needs to be more collaboration with fabricators, but a lot of times it just doesn't happen. There's often very little communication with the fab.

Don't Overthink Your Prototype Builds



Go with the team that can make your ideas a reality...

RF • MICROWAVE • HIGH-SPEED • QUICK TURN EXPERTS

ACE
ACCURATE
CIRCUIT
ENGINEERING

Accurate Circuit Engineering
3019 Kilson Dr
Santa Ana, CA 92707

(714) 546-2162 • sales@ace-pcb.com

For technical assistance, call:
James Hofer
(714) 227-9540

www.ace-pcb.com

Patrick Davis

Davis: That's true. We're making it convenient to communicate within the tools themselves. Here's a little of my background: I ran design teams for years at a contract manufacturer, and I did fab and assembly. My door opened onto the shop floor, and when the "Blue Smurfs" came into my office, it meant that my team screwed up—so I'm extremely familiar with that world.

Now, Cadence has a set of tools called DesignTrue and the DesignTrue DFM Portal. We ask fabricators to put in their design rules and say, "This is the rule set that we can build to." Then a designer says, "I have a design with an aspect ratio of 15:1, and I have 3.5-mil space and trace. Who can build this?" The designer can simply look up who builds this type of board in the portal and has the option to contact them to build the design.

Barry Matties: What prevents it from being thrown over the wall, so to speak?

Davis: You actually can throw it over the wall, as long as you set your rules up and you know which wall you're throwing it over. You can't just throw it over the wall blindly unless you're designing a simple power supply board that has no technology like impedance control, and you do not care too much about the design. For example, a two-layer design that has 20-mil space and trace with vias the size of Texas.

But as soon as you start to put technology in the design, like controlled impedance, microvias, blind and buried vias, smaller trace and space, you need to know where the board is going and talk to them. Otherwise, you will have some big surprises when you fire up the design.

Matties: Who typically decides on the fabricator in the process? Is it the designer or is it some purchasing agents somewhere?

Davis: In my experience, it's 50-50. Often, when you have smaller manufacturers and smaller companies, designers will come in and say, "If you do not have a fab shop picked out, I know a few that I like to work with." When you get to the really big players, they usually give you a list of two or three companies that they work with. The big companies will usually have factory reps who can make sure you are designing the board within manufacturing tolerances.

Matties: So, collaboration, obviously, is most important when you're going into a new fabrication facility.

Davis: Yes, especially when it's a new facility that you're not familiar with, or if are you pushing the edge of technology.

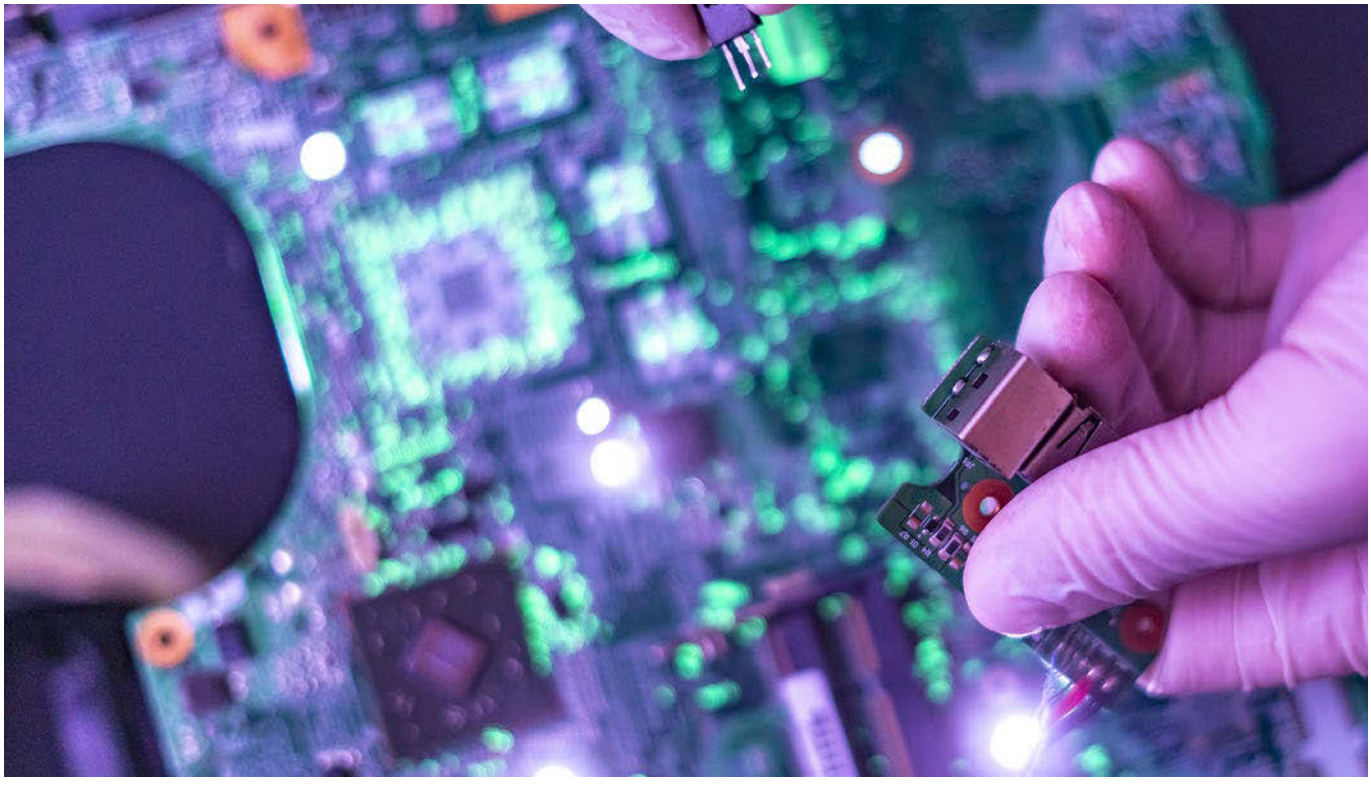
Matties: What are some other ways that designers can streamline their processes?

Davis: Designers can streamline their process by understanding how the board is going to be manufactured and understanding the end use

of the product. For example, if you are doing a demo board with a new chip on it and we're only making five of them, I am going to treat it a lot differently than if I'm making 400,000 of these boards for a new product launch. There are two different aspects of the manufacturing side you're looking at. Who else is involved with it? What are the mechanical, the electrical, and the physical imitations that are being put into it? Those are the big aspects that really control how you're going to flow with this.

With fabrication, there's an interesting game to play: Many people send off the preliminary data to fab for an initial DFM. I don't do it very often. The reason is because the fab shops are very low-margin businesses, and they are moving very, very quickly. When you send preliminary data to the fab shop and ask for a DFM, they're going to say, "Sure," and you get what comes out of their CAM tool and nothing more. Then when they start to tool it up and the CAM engineers really look at the data and read the notes, they often come back with, "We can't quite do this," "This doesn't quite work," or "Can you change this?" This is because, while space and trace issues are easy to find and fix, there are a lot of other issues that take a deep dive into the data to figure out. Please keep in mind, this is not all shops; there are a few out there that do it right, but not many. So, how do you shorten that cycle? You have all the DFM rules in the CAD tools so there is no question whether the board can be built. That is what Cadence has done with DesignTrue.

Matties: I hear a lot of people say that if more design engineers used simulation, it would streamline that process and maybe eliminate



a spin. But what I'm hearing you say is, even if you do simulation, as soon as I come down to the manufacturer and their execution of the project, it depends on their ability.

Davis: You are correct, and there's another thing that kicks me in the butt every now and then. Fab shops change. One person leaves a fab shop, goes someplace else, suddenly the quality suffers, and you're thinking, "Who left? Where'd he go?" When that happens we try to figure it out, then we may start working with that shop.

> Fab shops change. One person leaves a fab shop, goes someplace else, suddenly the quality suffers...

Matties: Right. Once they figure out a solution, if they don't necessarily document it, it stays in the mind of whoever solved the problem. It's tribal knowledge. If you lose that person, you lose customers.

Davis: And that is a real problem at all levels. The only time that you get away from that is when you go to crazy high-end manufacturers where their process is stringent, especially if they're doing DoD, military, or aviation where it is dictated, "This the way it's going to be." The DFM process on a board like that takes weeks sometimes, but it will be done right.

Another problem that fabricators have is saying yes to everything, and they shouldn't always say yes; they should push back more, especially when you're pushing the edge. There are a couple of companies in Japan that make phenomenal boards. But they're also really good at saying, "No, you're 0.001 millimeters out of spec on this. No thank you and go away." We're talking about a half-million-dollar order. "Sorry, we can't build this; we can't guarantee it." But they're dead on when they say they can build it. They will also say it will take three weeks, and they don't let anybody influence their processes. Can they expedite it? Sure, but it's still three weeks.

Shaughnessy: How much do regional differences figure into the whole collaboration formula? Asia and North America have completely different work cultures and management styles, not to mention language.

Davis: That's true. The international flair and the regionalization make it difficult, especially when you have time zones and a lack of English communication. English is the "default standard," but when you're designing and building in China, it makes it a little bit easier to pick up the phone and call them. Often, when you get something coming back from companies in some regions, their English is challenging—I think that's the polite way of putting it. You get these spreadsheets with translated English. It doesn't exactly work great for Microsoft spreadsheets.

Matties: How important are the tools today to help streamline the process? Is AI becoming a factor in design?

Davis: I think it is going to become a factor. It's just a matter of time. I think what will end up happening is people are going to start to figure out how to do predictive analysis.

Matties: The market opportunity, I would think, is huge for a tool like this. Why wouldn't the tool companies jump on it?

Davis: They should.

Happy Holden: I'm glad to see you're invested in IPC-2581.

Davis: We're 2581-centric because this is the

way it should be. I don't need dumb data. I need something that allows us to actually communicate back and forth efficiently. Plus, we still don't really know how to communicate. It's like two different languages, and I'm not talking about English and Chinese.

> I don't need dumb data. I need something that allows us to actually communicate back and forth efficiently.

You have the CAM operator sitting at the fab, and then you've got the DFM guy and the front-end engineer from an assembly shop, the designer and the engineer who designed the board, plus the purchasing person who's working on the BOM. You have a chasm, because you've got these silos around everybody. How do you collapse those silos down to just one way of communicating? And I'm going to bring this back around to the IPC-2581 standard. Inside of the 2581 standard is a DFM piece, which allows for this kind of communication to go back and forth. This is something that we are very actively trying to develop and have industry support for; this would solve the problem. The fact is, this is how you should communicate issues within a design. Let it be the BOM, the notes, the silk screen color, anything that is now done via email or a spreadsheet, would be done inside the 2581 format. That way you have full traceability of all changes and a single common means of looking at the issues.

Shaughnessy: In a recent interview, a CAM expert said he'd be happy getting fewer documents. He said they don't need drawings of everything, especially if it's not a critical part. What do you think?

Davis: I've heard this from manufacturing people for decades, right up until they needed more data. So we decided that we're going to give you everything in a format where you don't have to go look for it. Here's what you want, and here's everything else. Here's all your fab data, and we're also giving you the assembly data.

Matties: So, you think the way to solve the communication is through the IPC-2581?

Davis: I do. It's an open standard that everybody can use because it's not going to be limited to just Cadence's tools. It has to be for everybody because we don't have the tools that run the pick-and-place machines. We don't have the tools that run the fab machines. That's not our world. Our world is getting it right to that door.

Matties: Is there anything that we haven't covered today, Patrick, that you feel we need to include in this conversation?

Davis: The one thing I'd like to touch on is the checklist. I like checklists. I think designers and fabricators should use checklists. I understand that there are huge 400-page checklists, and guess what? If they get that big, they're not real. I'm a pilot, so, I live by my checklist. And one of the things that I do is make my custom checklist from the stuff that I always screw up.

When I was at my previous company, we had checklists that were ever-living. "We screwed that one up, and everybody missed it, so add that to the checklist. Make sure that it's not somebody else's name." And it's the customization of the checklist that helps; you're not looking for everything, but you're looking for the things that you miss. It's constantly evolving, and you have that feedback. That's something that will help a lot of people.

Matties: This has been great, Patrick. Thank you.

Davis: Thank you all for your time. DESIGN007

How to Increase the Reliability of Conformal Coatings

Sensible Design

by Phil Kinner, ELECTROLUBE

For this month's column, I'm going to examine the main methods for achieving the best outcome for your conformal coating process, ultimately increasing the protection afforded by your coating and lifetime of your product.

Let's start at the very beginning, at the design stage, factoring in careful consideration for coatings and highlighting the potential for problems during production. This will have a positive impact in the long term and avoid excessive time and expense, resolving problems at a later stage. Another important factor for successful coating is selecting the right coating for the job. This could be an acrylic, a polyurethane, a silicone, a UV cure, a two-part (2K) system, or even a water-based coating. Whilst the design of the board doesn't directly affect the choice of coating material, it will determine which methods can be used to apply the coating. Certain materials, such as moisture curing, 2K or UV curable materials, are difficult to use in a dip process due to the material curing prematurely, so it could be said that the choice of coating is indirectly related to the board design. Similarly, board design will affect the production yield. Consider maximising the distance between components that must be coated and those that must not. The greater the distance between them, the more the production staff will thank you. As a rule, 2 mm is the absolute minimum clearance between coat/no-coat areas, but 5 mm will make an enormous difference in the coating cycle time and success.

Factors such as operating temperature range, temperature excursions (such as thermal shock and thermal cycling), corrosion, condensation, and resistance to chemicals, solvents, and water are all key in determining the successful outcome for a coating.

How Important is Coating Application?

Do not underestimate this element. Probably the single biggest influence on coating success is as a direct result of how well the coating is applied. The application method of

PegaClad Series

for Sub-6 GHz Antenna (Dk 3.38, 3.45, 3.65)

PegaClad 1&2

for mmWave

Developed for Sub-6 GHz antenna applications, TUC's PegaClad 338, PegaClad 345, and PegaClad 365 - with respective Dks of 3.38, 3.45, and 3.65 - are advanced materials designed to minimize passive intermodulation and meet designers' needs. PegaClad 338, PegaClad 345, and PegaClad 365 can be used in multilayer and double-sided circuit board designs requiring excellent thermal reliability.

TUC's PegaClad 1 and 2 are designed for high frequency, very / super low loss applications, including mmWave printed circuit board designs. Their electrical properties are equivalent to PTFE laminates, yet they perform in multilayer circuit boards requiring excellent thermal reliability in harsh environments.

TUC's PegaClad Series and PegaClad 1 and 2 laminates also exhibit excellent moisture resistance, improved CTE, superior chemical resistance, and thermal stability, and are compatible with modified FR-4 processing.

tuc
Taiwan Union Technology Corporation
www.tuc.com.tw

TUC Asia
Raymond Foo
TEL : +886 3 5551103 x 129
E-mail :
raymond.foo@tuc.com.tw

TUC USA
Dana Korf
TEL : +1 408 643 1144
E-mail :
dana.korf@tuc.com.tw

TUC Europe
Marko Holappa
TEL : +358 40 770 9618
E-mail :
marko.holappa@tuc.com.tw

a conformal coating is as important, if not more important, than the selection of the right material. I can't stress this enough, because even the best coating, when applied poorly, may not provide the level of protection required or that it is capable of. Significant elements to consider include the thickness the coating is applied, the level of coverage achieved and how well the coating adheres to the board and its components. It is imperative to choose the best method for applying the coating.

There are numerous methods to apply coatings, which include hand applications such as brush, aerosol and atomised spray, and machine applications such as dip and selective coating. When applied correctly they form a thin film that conforms to the contours of the PCB and its components, as well as the exposed metal of component leads and solder joints. Typically, conformal coatings are applied in the 25–250-micron range.

The application method chosen for a particular assembly will depend upon which existing equipment is available to the manufacturer, the coating processes in use, the takt time (the average time interval between the start of production of one unit and the start of the next), and the design of the assembly. This includes those areas of the circuit, which *must* be coated and those which *must not,* connectors, switches and so forth. In addition, it is incredibly helpful to define don't care areas, which may or may not be coated. The best application method will ensure that each board coated receives good coverage on all required metal surfaces at a sufficient thickness to afford protection against the environment. These requirements will change from board design to board design, and environment to environment, and invariably they need to be tested and verified ahead of the production run.

Coating Thickness

Achieving the correct coating thickness is essential; bear in mind that if the coating is too thick it can lead to entrapment of solvents which can result in blow-out bubbles, delamination, or lack of adhesion. Similarly, it can cause the coating to crack as it cures or as the result of changes in temperature, or due to mechanical shock and vibration. Conformal coatings should not be applied at thicknesses greater than necessary and should never exceed the maximum thickness they were designed to be applied at.

Is it Necessary to Clean Prior to Coating?

The potential presence of residues on the substrate can have a critical impact on coating performance. If the substrate is not clean, the residues present may interfere with the curing mechanism, lead to poor adhesion of the coating to the substrate, and/or trap conductive/ionic materials under the coating. Without meticulous attention to preparation or pre-coat cleaning regimes, corrosive residues bridging the PCB's conducting tracks can cause failures over time. It is important to test the reliability of no-clean flux residues from your assembly process with your coating process, to ensure compatibility and adequate performance in your design.

What Will the PCB Board Layout Be?

The design of the board should include consideration of the placement of components that should not be coated. Selective spray equipment or the application of a peelable coating mask can be used to help avoid such areas. Alternatively, gel materials can be used to form a "dam" to contain the coating and avoid capillary effects transferring material to unwanted areas, such as connectors. Another aspect to consider is how close together the components are. If there is not a sufficient gap between the components, it can lead to the coating bridging the gap rather than correctly conforming to the board. This can have several effects including the coating building up to become too thick and crack, or it can leave pockets under the coating where solvent can gather and cause corrosion.

By placing connectors and components that must not be coated along one edge of the assembly, the conformal coating application process will be simplified. It is also advisable to avoid large arrays of discrete components, which can pose a huge coating challenge due to the high levels of capillary forces present. Tall components can also present challenges of their own by the creation of shadowed or hard to reach areas. Splashing is another associated problem. The trick is to avoid placing tall components next to "must-coat" components to avoid this eventuality.

Know Your Component Zones

Lastly, it's worth noting that being aware of what kinds of components should be coated or should not be coated will provide more flexibility to the coater in assembly operations. When a component will not be coated, it becomes a "keep out" zone. The engineering drawing should not only identify components but identify the tolerance on that keep out zone. It is important to be very specific on the tolerance. The manufacturer needs to know exactly where you do and don't want coatings. It is best practice to specify the areas that need to be coated and the areas that don't, as well as the "don't care" areas, to help the coating process run as smoothly as possible. Avoid vague statements at all costs in an engineering drawing. This is especially true when specifying coating around connectors.

Over the following months I hope to provide more useful tips and design advice that will help make life a bit easier for those who are responsible for making the decisions on protecting components and circuitry. I hope the foregoing will help you with your coating process. Look out for my next column where I will be exploring more coating tips and techniques. **DESIGN007**

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

PCB Design—It's a Team Sport

Elementary, Mr. Watson

Feature Column by John Watson, ALTIUM

During the struggle with COVID, the situation was driven by the fear of not knowing how long things would be affected, how bad things would get, or even what the new normal would look like after it was over.

As a result, we re-invented the way that we worked. Kitchen tables became our offices. Our entire industry had to re-invent the way we did things. It's during the difficult times that the most valuable lessons are learned. Now that things are beginning to open up and move forward, we see what can only be described as a Brave New World unfolding. We don't know what the new normal will ultimately be, post-COVID, but welcome to it.

One of those hard lessons was about the value of the team and collaboration. I have repeatedly heard how many of us have a newfound respect and appreciation for the teams we work with inside our companies. Out of necessity, we had to find new ways to collaborate.

"No Man is an Island," a poem by John Donne, could not be truer today. This principle is especially true when it comes to the PCB design process. Since no single person is involved in PCB design, I would contend that to succeed, PCB design must be a team sport.

I can hear a few designers saying, "My team is me, myself, and I." I fully understand your situation. For you, it's not an individual doing specific tasks but rather a single person doing all the different roles to finish the PCB design. Sometimes you put on the component librarian hat, other times the electronics engineer hat, and other times you may be the purchaser or the project manager. Although it is just you

Candor
Circuit Boards

Innovative solutions for complex circuit board manufacturing

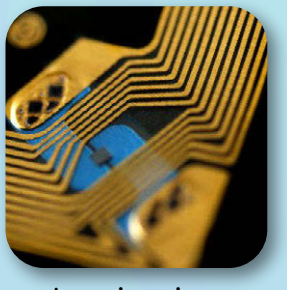
Click to see what we bring to each sector of the industry:

RF/Microwave PCBs

Power Electronics

High-Speed Data

Automotive Electronics



Lamination

Startups

Our technology produces what our customers dream!

www.candorind.com | (416) 736-6306

doing everything, there are still underlying principles of collaboration needed to finish the project successfully.

Importance of Collaboration

Collaboration refers to specific individuals working together as a team in an intellectual endeavor or directly working together to perform a particular task.

During a typical PCB design process, we have multiple people working on a single design, sometimes simultaneously. It becomes rather challenging to mesh the PCB data with a specific role. If not handled correctly, data management could turn into a real problem. That is all done while you manage what each person is doing while keeping anything from falling through the cracks.

There are endless advantages to developing collaboration within your team. But collaboration needs development; it is not something that just happens. Some designers tend to be islands who like to handle things on their own. Nevertheless, the growth of your team and the collaboration between team members is one of the critical ingredients of a successful company.

Leveraging Your Talent

Everyone has strengths and weaknesses, and joining forces helps develop the overall team. As the saying goes, "Iron sharpens iron, so a man sharpeneth the countenance of his friend."[1] During this process, the hope is that we can learn from each other. However, what are we talking about sharpening? Abrasion is a process of wearing down through friction. When people with various personalities work together and do things differently, there is going to be friction. That tension can make us better and provide a chance to grow and learn, or it can destroy the team entirely.

Developing Better Communication

The key to good collaboration is communicating effectively. Individuals on your team need to connect to other individuals in order to get across a certain point. Working together requires us to share ideas, which allows each of the individuals to learn new things and grow. Often preceding a team's breakdown is the collapse of communication. There are various reasons why communication breaks down in a team; it is essential to identify those causes and remove them as quickly as you can.

Expediting Projects

Time is usually a prominent driver in any PCB project. The big question for managers and sales teams is "When?" Fortunately, the addition of even one designer to the team multiplies the work effort and, fortunately, the results.

The real key to successful collaboration is to assure that one designer's work does not overwrite the other's efforts. That will involve using advanced tools and techniques. A good strategy is to arrange for each designer to work within certain zones or areas. For example, you can have specific people routing a section of the design and then consolidate those changes into a master design.

A primary tool that I would consider a requirement on any PCB design team is the version control system (VCS) design repository such as an SVN or GIT system. When multiple people work on a single design, after the first person commits the design back to the VCS, the others must conduct a compare and merge of the changes. But the VCS will manage the workload and document the progress by saving a version of the PCB project at a specific moment in time.

Roles and PCB Data

There is a sort of symbiotic relationship between the PCB data and the roles involved. Understand these roles; it is not mainly about individuals or positions but the tasks and stages in the process.

As we now all know, PCB design begins with the electronic component. Within that com-

Component Part	Electronic Engineer	PCB Librarian	PCB Designer	Mech Engineer	Purchasing	Quality
Information						
Parameters	X	X	X			
Sourcing			X		X	
Supporting Doc	X	X	X	X		X
Models						
Schematic Symbol	X	X	X			
Footprint (Decal)		X	X			
3D Model		X	X	X		
Circuit Sims	X					

Table 1.

ponent is the unified data model, which holds all the data necessary to create a PCB. The first role involved in the PCB design process is the librarian. I have always believed that the librarian holds one of the most critical positions and roles. The first rule is to be best friends with your component librarian.

A little review is in order when we break down that component. It has two major categories: the information and the models. First, there is the general component information of name, description, the specific component parameters, and sourcing information. On the other side are the component models, consisting of schematic symbols, footprint, 3D models, and sometimes (if you can get them) simulations or IBIS models. Each of these items in the unified data model plays its part at a specific design process point. Depending on what role you have, it is the particular information of importance to you.

For example, the mechanical engineer focuses on the 3D model because he is concerned with how the PCB will go together mechanically. When you take a deep dive, break up the component, and tie it to specific roles, you begin to see the importance of providing all the information inside the component. Furthermore, the stronger the provided information, the better support it gives to a specific role. Therefore, it is better to give too much information than not enough.

In Table 1, we see the breakdown of the component and how it provides the required information for a specific role.

Conclusion

Think of the saying, "Many hands make light work." An enormous task is made easier with many working together. In the face of a daunting task, we come together as a team and solve the problem.

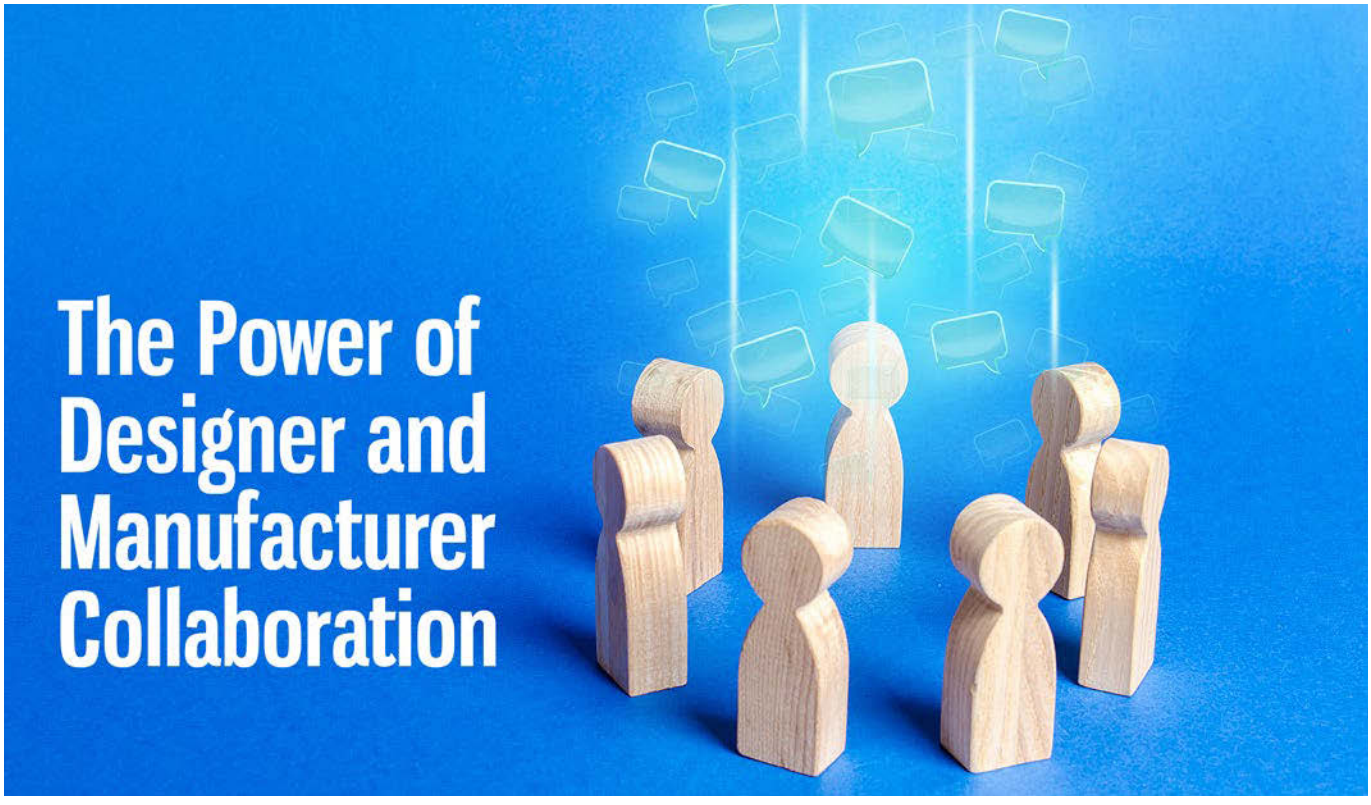
COVID turned our industry upside-down. But, let's not forget the vital lessons we learned that we are a part of a team. It's not about the individual parts but rather the strength we exude when we join together—with our strengths and our weakness. There was a motto that we heard repeatedly throughout the pandemic: "We are all in this together." Let's hope that never changes. **DESIGN007**

References

1. Proverbs 27:17

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

The Power of Designer and Manufacturer Collaboration



Feature Article by Ian Huibregtse, CID
CALUMET ELECTRONICS

Do you have a preferred PCB manufacturer? Do you have a contact in their engineering group? Do you have a contact in their quality department? What technologies could your PCB manufacturer introduce that would improve your products the most?

Whether you are beginning board layout, building the first run of a prototype, or you're making a change to a mature product several years into its life span, there is no incorrect time to consult your PCB manufacturer. Circuit boards today are dense with features that push the limits of manufacturing technology, and as designers employ these technological advances it's easy to venture off the well-documented path of standard manufacturing.

A PCB manufacturer can offer answers, advice, and a partnership to ensure your next generation electronic system is successful. We have integrated PCB designers and process engineers into our front-end engineering team and established a dedicated product realization team to ensure that complex problems are approached head-on and in direct communication with designers and supply chain managers.

A robust PCB manufacturer can serve as a consultant. If you are looking to try something new, push the limits of feature density, incorporate embedded passive components, stick to a tight schedule, or reduce materials or manufacturing costs, it is in their best interest to deliver you the best product they can. This is especially true as you grow your arsenal of technologies. While it could be your first foray into sequential lamination, your manufacturer has most likely been building a variety of sequential lamination boards for some time. Communicating directly with your manufacturer early in the process allows them to help you identify design pitfalls and manufacturing difficulties before they cause delays and before a new design is released.

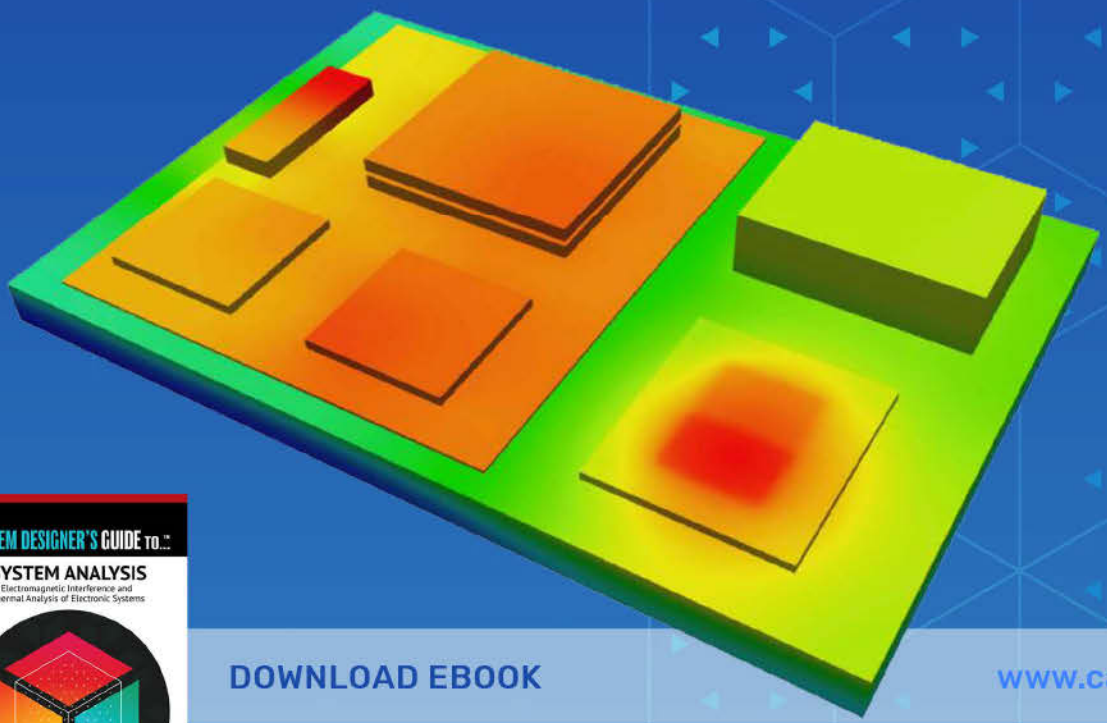
PCB front-end engineering talent is in short supply on a national scale and all new board designs pass through an extensive DFM process, therefore it is important to find a PCB manufacturer with an advanced front-end

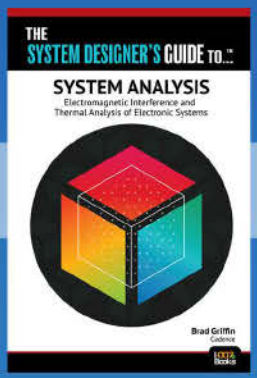
cadence®

Thermal and Stress Analysis of 3D-IC Systems

The explosion of data, both in volume and speed has increased the need to detect and mitigate thermal issues early in the design process to reduce the number of design iterations. This requires complete thermal analysis and signoff from chips to enclosures in a system. On a 3D-IC system, it is even more compelling. The massively parallelized Celsius™ Thermal Solver with its FEM and CFD engines and transient and steady-state capabilities offers the most complete thermal analysis, up to 10X faster and with virtually unlimited scalability.

WATCH NOW





DOWNLOAD EBOOK

www.cadence.com

engineering team. One should avoid shops that offshore their front-end engineering to other countries in favor of shops that have in-house expertise.

By maintaining a partnership with your PCB manufacturer, the handoff will be as seamless as it would be to a member of your own team. Just like the recipe for a good team, a partnership with a PCB manufacturer requires both parties to have technical knowledge, open communication, and an understanding of your design goals. It also requires mutual trust. Design advice from a qualified manufacturer should drive the design to higher reliability and lower cost.

This partnership relies on you letting your manufacturer know where you are and where you plan to go in the future. Helping your manufacturer to grow with you and build the technologies you want to implement creates a robust supply chain. Even if you are a smaller designer and you aren't anticipating being influential in the future of your manufacturer, communicating your goals will result in higher quality products. These partnerships aren't built overnight, but they have the potential to drive technology and innovation, while reducing cost, reducing waste, and building more reliable products in the future.

We hear from designers all the time who aren't sure if we would even want to answer their questions. As a board designer and DFM engineer, I can tell you that we are ready to answer your questions at any stage in your design process. We have acted as a sounding board for new ideas, introducing new milling and plating processes to reduce downstream manufacturing costs and increase part reliability. We review stackups for material availability, dielectric performance, ease of acquisition, impedance requirements, lamination characteristics, and cost.

We are driving the technologies that bring 25-micron lines and spaces to U.S. domestic manufacturing, largely driven by discussions with designers who push the envelope of technology. A true collaboration is the solution to bringing back domestic capability and capacity. Our largest customers know that demanding higher yields won't get them better prices. They know that developing partnerships through open communication can lead to better designs and manufacturing improvements.

A great example of partnership is one that we recently developed with a customer designing a new phased-array radar board. The customer could not procure boards (with advanced technologies beyond standard capabilities) at required volumes in less than a 26-week lead time. Instead of moving on to the next supplier, the customer agreed to work with us to develop the specific capabilities and we dedicated a team of engineers to complete the qualifications and prototyping in a nine-month time frame, three months sooner than anticipated. At the end of the prototyping phase, we had reduced the manufacturing lead time to five weeks, simultaneously reducing costs through supply-chain collaboration including a switch to a more manufacturable and domestically sourced material supplier, who offered superior lead times and technical support. By investing time and energy into this collaboration, this customer now has a technologically capable PCB manufacturer that they can work with on current and future designs, and we have new capabilities to offer our entire customer base.

Whether you are incorporating a new technology for the first time or just want a second look at your impedance stackup, it is better to discuss designs in advance than have your manufacturer put your order on hold. Reach out, make connections, and build a partnership. Your PCB manufacturer wants to help you. **DESIGN007**

Ian Huibregtse, CID, is a senior CAM engineer/developer at Calumet Electronics.

AUCTION

33 Melford Dr, Scarborough, ON M1B 2G6, Canada

PCB Auction of LazerTech - High Quality PCB Manufacturer

Premier Multilayer & Prototype Facility - *Equipment As Late As 2019*

- Orbotech LP:9008 IHS Laser Drum Photo Plotter
- Western Magnum XRL 240 Dry Film Laminator
- Dynachem Model 300 Dry Film Laminator with Dust Removal System
- Print Process Expo-Aligner Fully Automatic Artwork Alignment System (2010)
- Excellon Century 2001 - 5 Spindle Controlled Depth Drill
- Excellon Mark VI-C - 5 Spindle Drill, 110k RPM
- Lauffer - 6 Daylight Vacuum Multi-Layer Press - 26" x 30", w/6 Daylight Cold Press
- T.M.P. - 3 Daylight Vacuum Multi-Layer Press - 26" x 30" w/6 Daylight Cold Press
- Eidschun/SIA Phoenix Cobra Bond Automatic Line

Entertaining Pre-Auction Offers on Major Machines. For All Questions About Equipment and Bidding
Please contact Tyler Gardner: +1 203-500-2529 • tgardner@thebranfordgroup.com

- IPS Semi Automatic Electroless Line (2010)
- Numerous 500/650/1000 AMP Rectifier's
- ENIG - Electroless Nickel Immersion Gold Line (29 cells)
- Donggual 4500 KC Flat Semi-Auto Screen Printer (2019)
- Circuit Automation DP-1500 2X Auto LPI Solder Mask Machine
- Circuit Automation DP-10 Automatic LPI Solder Mask Machine
- ATG A-5 Flying Probe Tester (2015)
- Polar Impedance Tester CITs 880s Impedance Tester (2018)
- Camtek Orion 404 AOI System w/806 Elite Eye-Q add on (2015)
- Numerous Power Sheers, Compressors, Vaccuum Systems & More

MONDAY
SEPT 13
8:00 AM (EDT)

The Branford GROUP
Equipment Auctions Worldwide

203.488.7020
TheBranfordGroup.com
service@thebranfordgroup.com

WWW.THEBRANFORDGROUP.COM

PDN Trends and Challenges

Beyond Design

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

Power distribution network (PDN) planning is a relatively new technology that has become an essential, interrelated component of signal integrity analysis. However, mainstream PCB developers have yet to adopt PDN analysis as a common design process. But, now that the technology is proven and the uptake costs have decreased dramatically, there is no reason why all designers should not take advantage of the technology to improve the reliability and performance of their products. In this month's column, I will delve into the latest PDN trends and challenges.

Today's high-performance processors employ low DC voltages with high transient currents and high clock frequencies to minimize the power consumption and hence the amount of heat dissipated. Unfortunately, the lower core voltages, higher currents and faster edge rates all impact the PDN design as well as signal integrity. The goal of robust PDN planning is to design a stable power source, tak-

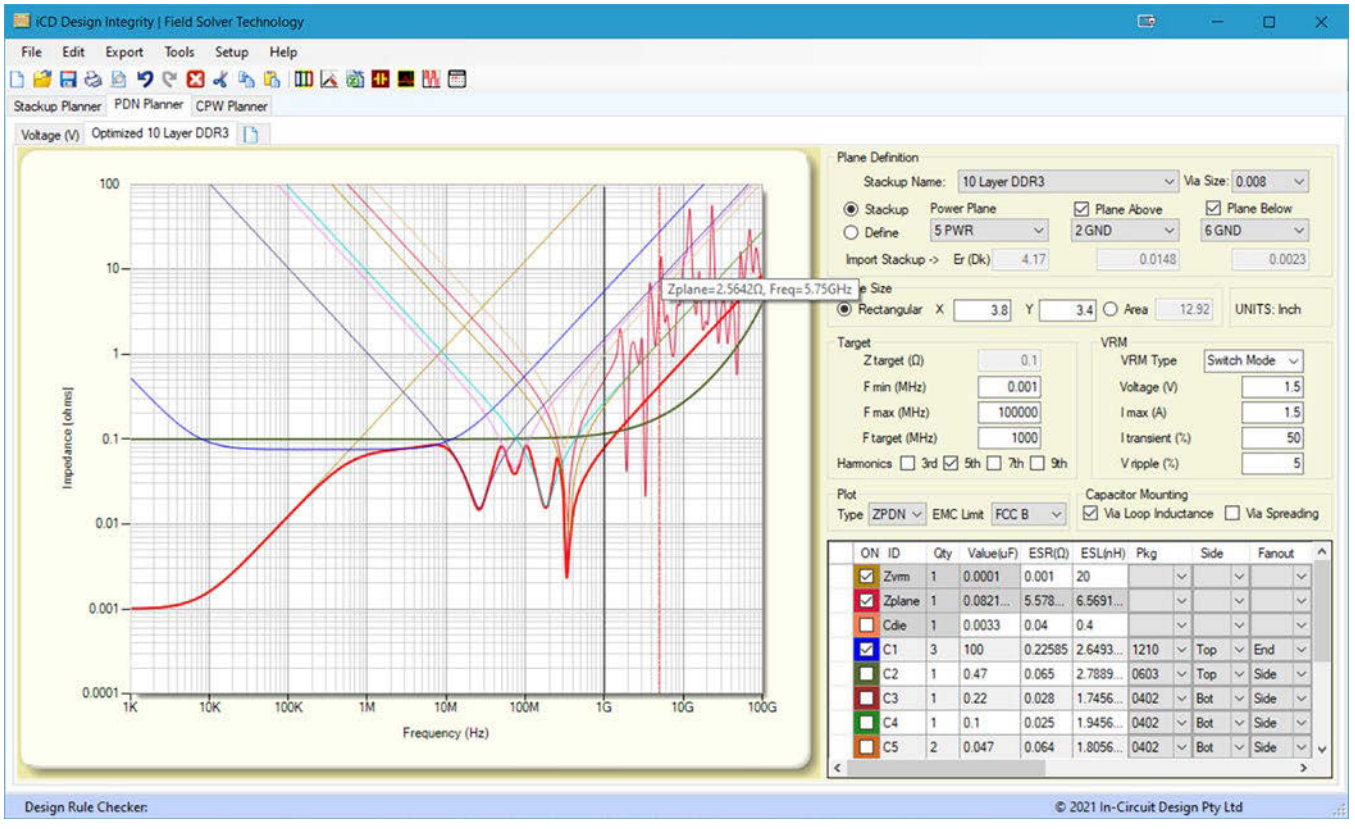


Figure 1: AC impedance profile of a DDR3 supply. (Source: iCD PDN Planner)

Our capabilities are simply out of this world.

We manufacture printed circuit boards for a wide range of advanced applications using just about any type of substrate currently available.

EXPLORE OUR CAPABILITIES

American Standard Circuits

Creative Innovations In Flex, Digital & Microwave Circuits

www.asc-i.com | (630) 639-5444

ing the above into account, for all the required onboard power supplies. And this all takes up valuable real estate—in some cases, up to 30% of the board surface area. One solution is to combine up to eight DC-to-DC converters in the same IC package. The resultant IC has one power input and outputs eight power rails. This approach reduces the overall PDN size by allowing the use of small 0201 capacitors combined with an integrated multi DC-to-DC converter.

Taking this strategy further creates a new challenge. As we move toward complete systems in a package (SiP) components with more functionality, even more power inputs are required. The SiP package itself may require up to 12 individual sources to power the increased functionality. Then, each supply requires discrete filtering components.

The trend in lower DC voltages also requires tighter voltage noise tolerances and higher currents. Market demands are forcing product designers to create PDNs with greater density, higher power efficiencies, and lower costs, making the process even more challenging.

The target impedance approach to analyzing the PDN is the combination of the worst-case transient current and the voltage noise specification, which act together to set the maximum allowable AC impedance with assured performance. But, as current demands increase and voltage noise fluctuation tolerances reduce, we must lower the AC impedance even further with higher density capacitance. This shift requires the use of more expensive, tighter-tolerance parts, such as capacitors capable of surviving higher currents—creating more heat in a reduced space.

In practice, accurately calculating the transient currents and the precise requirements for the target impedance can be difficult. Since we typically do not know the transient noise current excitation very accurately, it is customary instead to design the PDN to meet the required AC impedance profile. Also, it seems that the current portion of the target impedance equation varies from point-to-point, on the board, depending on a host of intricate relationships. One must always apply engineering judgment in translating the information available into the requirements for a cost-effective PDN design.

With the continuous trend to smaller feature sizes and faster signal rise times, planar capacitor laminate or embedded capacitor materials (ECMs) are becoming a cost-effective solution to further improve power integrity. This technology provides an effective approach for decoupling high-performance ICs whilst also reducing electromagnetic interference.

Plane pair cavity resonances contribute to emissions. Smaller plane separation implies less area of equivalent magnetic current at the plane pair edge, or equivalently less local fringing field volume, and therefore lower emissions for a given field strength.

Embedded capacitance technology comprises a very thin dielectric layer (0.24 – 2.0 mil) that provides distributive decoupling capacitance and takes the place of conventional discrete decoupling capacitors over 1 GHz. These ultra-thin laminates replace the conventional

Manufacturer	Material	Description	Thickness (mil)
3M	ECM	Embedded Capacitance Material (ECM)	0.24, 0.47, 0.55
DuPont	Interra HK04	Ultra-thin laminate	0.5, 1.0
Integral Technology	Zeta Bond	High Tg Epoxy Based adhesive film	1.0, 1.5, 2.0
Integral Technology	Zeta Lam SE	Low CTE C-stage dielectric with a Hi Tg	1.0
Integral Technology	Zeta Cap	Hi performance polymer coated copper	1.0
Oak-Matsui Technology	FaradFlex	Planar capacitor	0.31,0.47,0.63,0.94
Samina	ZBC1000	Buried Cap, hi-performance decoupling	1.0
Samina	ZBC2000	Buried Cap, hi-performance decoupling	2.0

Table 1: Embedded capacitor materials available in the ICD Dielectric Materials Library.

power and ground planes and have excellent stability of dielectric constant and dielectric loss up to 15 GHz. The thinner layers of ECM also significantly reduce the capacitor mounting inductance.

These ultra-thin laminates allow a significant layer count reduction in PCBs with better signal performance. Having a low dielectric constant, combined with very high withstanding voltage, these glass-free films change the design rules for a given via diameter and trace width, while still conforming to the manufacturing needs of the PCB fabricator.

Several technology trends are also enabling denser PDN design. For instance, increasing the converter switching frequency reduces the size of the PDN but is less efficient—producing more heat. Also, decoupling capacitors are tending to be smaller so they can be placed in closer proximity to the load and minimize the parasitic loop inductance. But, this means they need to be of higher quality to withstand the heat. These capacitors have low equivalent series resistance and inductance and take less mounting space which results in lower overall loop inductance.

When you cannot shrink the traditional PDN any further, it may be time to take a new approach involving switched tank converters. Traditionally, we use switched capacitor converters to step down the source voltage, however these require large banks of capacitors. Switched tank converters use resonant tanks which require much less space, so it is possible to put the power delivery devices much closer to the processor core. This enables faster power switching to accommodate changes in consumption of the core.

One of the best ways to lower design costs is to minimize respins of the board. Simulation is the key to resolving trade-offs. And, simulation allows the designer to perform what-if analysis of the PDN before the board assembly is produced. The AC impedance curve is a summation of all of the effects of the power source that you choose: bulk bypass capacitors, high-frequency decoupling capacitors, mounting inductance, the PCB substrate stackup, and the IC package. So it is imperative to be able to extract the plane data from the stackup and import the information into the PDN profile (Figure 2). This allows one

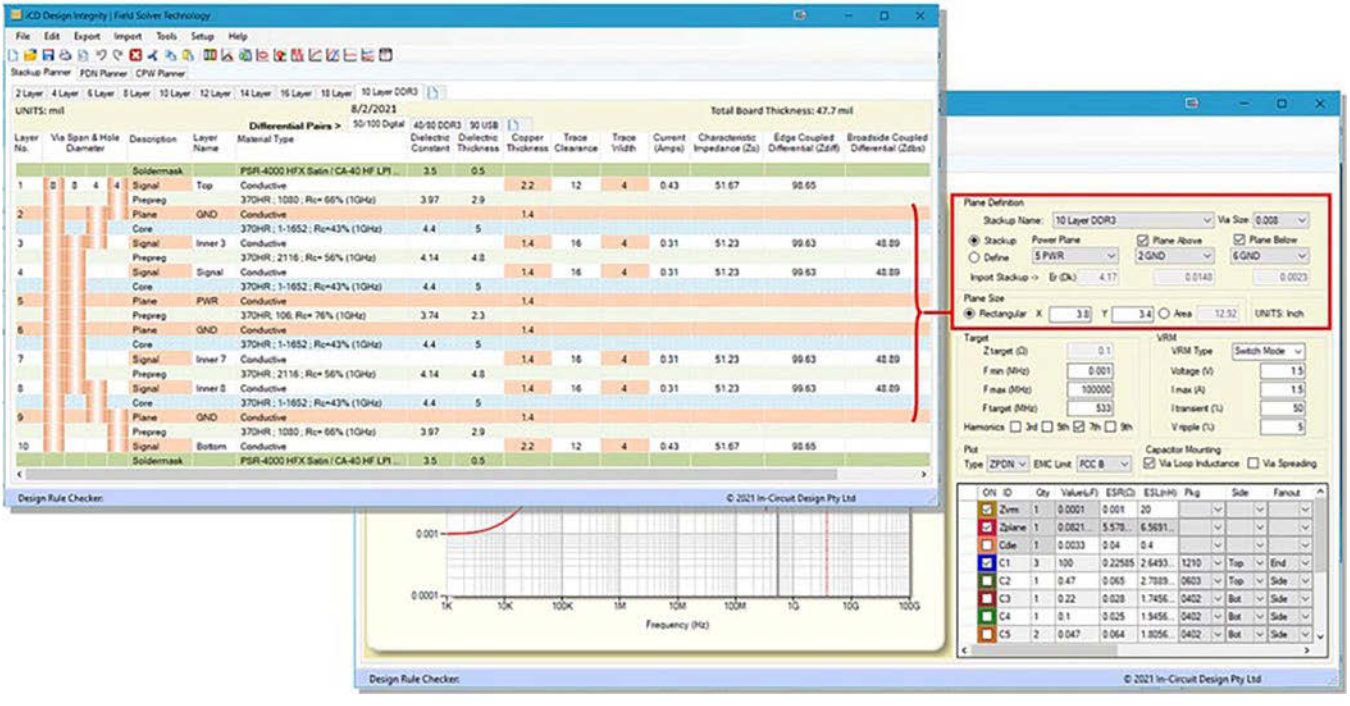


Figure 2: Plane data is extracted from the stackup to build the plane definition of the PDN. (Source: iCD Design Integrity)

to use planar capacitance during the analysis. Planar capacitance is important as it dramatically reduces inductance at frequencies where decaps are out-of-range.

Capacitor manufacturers are now mainly providing capacitor profiles in the form of S-parameter (S_{11}) data which makes analysis more difficult. Gone are the days when you could look at the V-curves from the capacitor ESL, ESR, and value. S-parameter data can be imported into the iCD PDN Planner to model the combination of decaps, planar capacitance, and mounting inductance.

PDN issues impact the project schedule. By incorporating PDN simulation into your design process you can avoid the likelihood of a costly respin and a delay in the schedule. PDN Planners are now very affordable and so there is no reason why every product designer should not have access to the simulation tool.

Key Points

- PDN analysis is now a proven technology that should be adopted by mainstream developers.
- The goal of robust PDN planning is to design a stable power source for all the required onboard power supplies.
- One solution to reduce the PDN size is to combine up to eight DC-to-DC converters in the same IC package.
- The trend in lower DC voltages also requires tighter voltage noise tolerances and higher currents.
- The target impedance approach to analyzing the PDN is the combination of the worst-case transient current and the voltage noise specification, which act together to set the maximum allowable AC impedance with assured performance.
- As current demands increase and voltage noise fluctuation tolerances reduce, we must lower the AC impedance even further with higher density capacitance.
- Planar capacitor laminate or embedded capacitor materials are becoming a cost-effective solution to further improved power integrity.
- Embedded capacitance technology provides distributive decoupling capacitance and takes the place of conventional discrete decoupling capacitors over 1 GHz.
- Ultra-thin laminates allow a significant layer count reduction in PCBs with better signal performance.
- Increasing the converter switching frequency reduces the size of the PDN but is less efficient—producing more heat.
- Switched tank converters create resonant tanks which require much less space than traditional switched capacitor converters.
- Simulation is the key to resolving trade-offs. And, simulation allows the designer to perform what-if analysis of the PDN before the board assembly is produced.
- It is imperative to be able to extract the plane data from the stackup and import the information into the PDN profile. This allows one to use planar capacitance during the analysis.
- Capacitor manufacturers are now mainly providing capacitor profiles in the form of S-parameter (S_{11}) data. **DESIGN007**

Resources

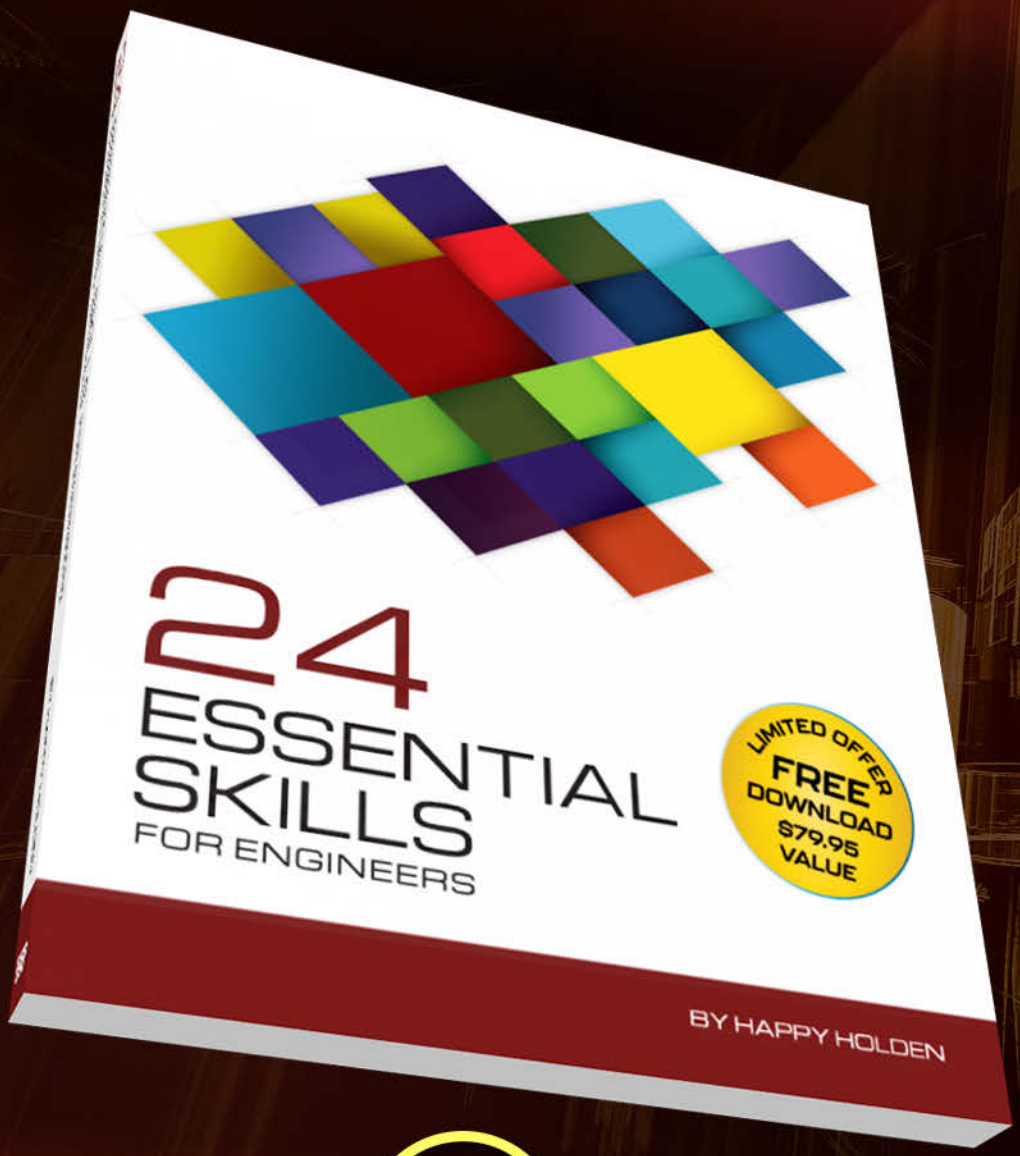
1. Beyond Design: Learning the Curve, Plane Crazy Part 1 and Part 2, The Target Impedance Approach to PDN Design, Impact of PDN Impedance on EMI, by Barry Olney.

2. KEMET and Mouser Electronics: 7 Experts on New Approaches for Power Distribution Network Design - Mighty Guides

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

Engineers: Here are 24 Real-world Skills You Didn't Learn in School

Industry veteran Happy Holden shares his strategies for overcoming engineering challenges.



1-800eBooks

SPONSORED BY

Calumet
ELECTRONICS CORPORATION

DFM 101: Layer Stackup

Article by Anaya Vardya
AMERICAN STANDARD CIRCUITS

Introduction

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

DFM

Design for manufacturing (DFM) is defined as the practice of designing printed circuit boards that meet not only the capabilities of the customer's assembly manufacturing process, but also the capabilities of the board fabrication process, at the lowest possible cost. While not a substitute to early design engagement with the PCB fabricator, these articles will provide guidelines that will help to "design for success."

Multilayer Construction Guidelines

A printed circuit board stackup refers to the material construction of the PCB, including core dielectrics, prepreg dielectrics and copper weights. Many factors need to be considered when developing a stackup, including

Figure 1: Multilayer PCBs are similar in basic construction to typical multistory buildings.

SIEMENS

Learn More

FROM HOME GARAGE TO ENTERPRISE

Making DFM Accessible For Any Business Size

Siemens solution enable companies to optimize the entire Electronics Manufacturing process in a smart and efficient way. Our lean manufacturing experts will walk you through the design-to-manufacturing flow and show you how to leverage the digital twin to improve your processes.

siemens.com

overall thickness restrictions, circuit width and spacing and impedance values required.

Here are some guidelines that should be followed to assure design success:

- Design multilayer boards with an even number of layers
- Whenever possible, only one core thickness should be used
- If specifying the dielectric thickness when, for example, it may be required for impedance reasons, the dimensions should be selected from available core or pre-preg thicknesses
  - Dielectric thicknesses made up of prepreg depend on the type or the combination of prepreg that is suitable and of achievable dimensions and tolerances
  - It is beneficial to discuss special dielectric requirements during the design stage with your PCB vendor if possible
- Maintaining a balanced layup in relation to the Z-axis median of the board will assure minimum bow and twist. This balance includes the following: dielectric thickness of layer, copper thickness of layers and its distribution and location of circuit and plane layers. A higher number of layers normally will mean an increased number of plane layers. Planes should be balanced around the Z-axis median line of the layup, and ideally located internal to the board. If accepted multilayer design rules are adhered to, boards will meet a maximum allowable bow and twist specification of 0.010 inch per inch (1%) or better.
- If the prepreg opening is greater than 0.016" use filler cores
- Outer layer circuitry: Circuit area and distribution between the front and back of the board should be balanced as closely as possible. The addition of plating thieving of low pattern density of external plane area should be considered. Each outer layer is typically a signal layer built on half-ounce copper
- Thickness tolerance: As the overall thickness of a multilayer board increases, the thickness tolerance should also increase. A good rule is to specify a tolerance of +/– 10% of the overall thickness.

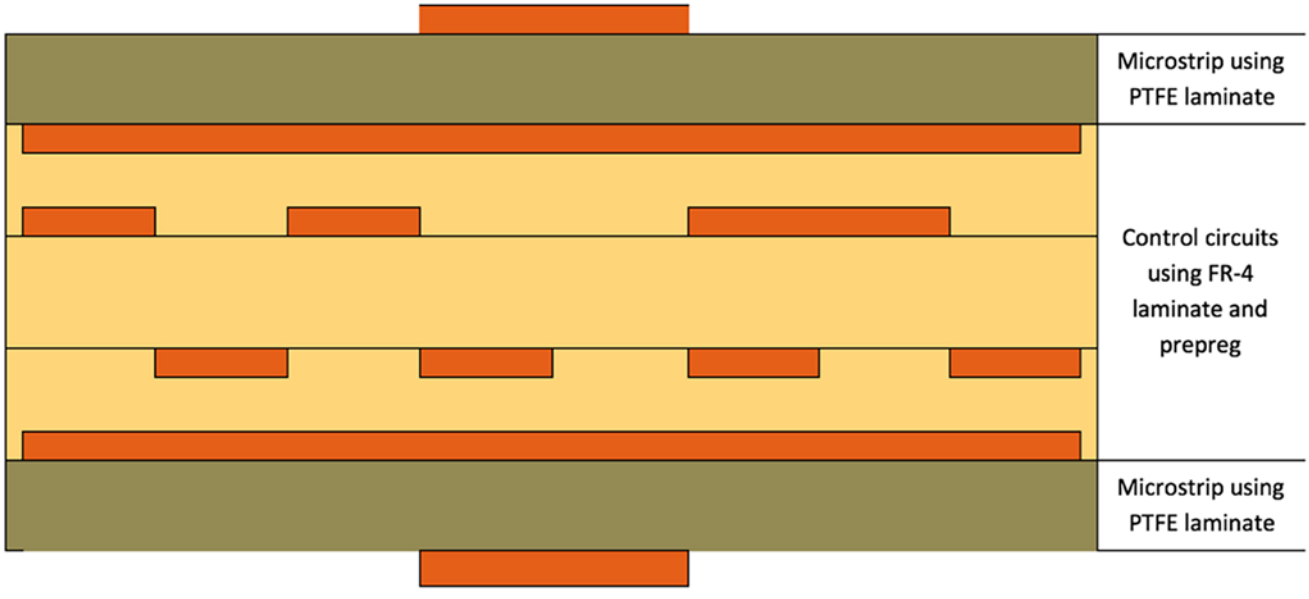


Figure 2: An example of a FR-4/PTFE hybrid material set.

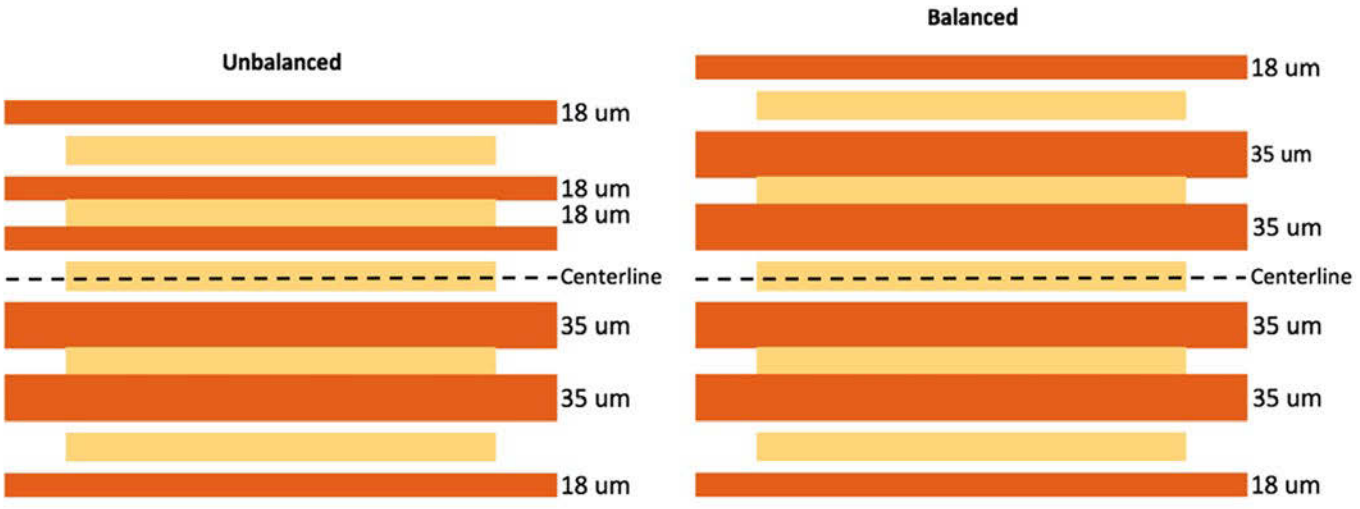


Figure 3: A balanced construction is important with any PCB design, but is even more critical with a hybrid material set using mixed dielectrics.

Always indicate where the thickness measurement is to be taken, for example:

– Glass to glass at rail guides

– Over gold contacts

– Over solder mask

Pure Build vs. Hybrid Build

The term "pure build" refers to a multilayer PCB material construction that is composed of the same type of material throughout the stackup, such as a construction entirely of FR-4, or one entirely of PTFE or other high-frequency material. A hybrid multilayer PCB uses materials with significantly different critical properties than those associated with a traditional "pure" multilayer PCB. The reasons for using a hybrid construction in RF multilayer PCB designs are typically driven by cost, reliability, and/or electrical performance.

Balanced Stackup Structure

Regardless of the material used, maintaining a balanced construction (lay-up, stackup, etc.) in relation to the Z-axis median of the board will assure minimum bow and twist. This balance includes the following: dielectric thickness of layer, copper thickness of layers and its distribution and location of circuit, and plane layers. Whenever possible, planes should be balanced around the Z-axis median line of the lay-up, and ideally located internal to the board.

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

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

PCB007 Highlights

Standard of Excellence: The Beauty of a Partnership ▸

Partnerships are more important than ever. Not only with your vendors, but with your competitors as well. I believe that if we can all start working together in this new post-pandemic world, life will be much better for all of us individually but also better for our industry as a whole.

EMC and Technica Introduced New Extreme Low Loss High Speed Materials at DesignCon 2021 ▸

EMC and Technica welcomed OEMs, ODMs, and PCB fabricators to their booth at DesignCon 2021 to learn more about these and other high-speed products.

The Right Approach: The Laws of Respect, Intuition, and Magnetism ▸

Good leadership always makes a difference; unfortunately, so does bad leadership. This leadership truth continues as we will be talking about the seventh, eighth, and ninth of the 21 Irrefutable Laws of Leadership.

The Demand for Copper ▸

Nolan Johnson spoke with Michael Coll and Chris Stevens of Nippon Denkai, home of the last-standing ED foil manufacturer in North America, about the demands and projections they're currently seeing in the copper market.

Punching Out! 2021 Mid-Year PCB/EMS M&A Update ▸

M&A activity is booming in the U.S. as the nation emerges from the COVID pandemic. Thanks to video conferencing and adjustments in procedures, M&A was also pretty busy in 2020. Now that the economy has opened and everyone is traveling, more deals are being discussed.

Ventec, Taiyo America Sign Exclusive Distribution Agreement for Mainland Europe, UK ▸

Ventec International Group Co., Ltd., is pleased to announce it will be taking over the exclusive distribution of Taiyo products in mainland Europe, the UK and Ireland.

Dan's Biz Bookshelf: Damn Good Advice (for People with Talent) ▸

If you want a book that not only shows how to be creative and spark your own creativity but provides real-life examples of what true originality looks like, this is the book for you. It is a small book filled with 119 ways to spark your own creativity. I love this book. I always keep it on my desk so that when I get stuck for an idea to write about, I look through it and I always find something that will get my engine running again.

Riding the Wave of Copper Inflation Pricing ▸

In this interview, Ventec's Mark Goodwin discusses the rising inflation hitting the electronics industry at the same moment as shortages of copper and other raw materials. He believes that, while the PCB industry has endured cycles like this before, this one feels different.


We are dedicated
to excellence through
innovation,
technology
and most importantly,
service.
COMMERCIAL • MILITARY • MEDICAL • BAREBOARD
U.S.C.
U.S. CIRCUIT
www.uscircuit.com
ISO
9001 CERTIFIED
ITAR

Collaboration by Design: The Role of EDA Tools

Feature Article by Rich Tighe
NOAH MEDICAL

Rapidly designing and building successful electronic circuit boards is always helped by good communication and collaboration among engineering teams (internal and/or external), purchasing departments, and EMS providers.

Collaborating effectively and efficiently with colleagues throughout the design cycle leads to better results. Parts selection and availability have always been important, but they have become a primary concern during the global pandemic-induced shortages. By using circumspect design techniques and a design tool that facilitates collaboration and provides clear insight into the supply chain, PCB designers can stay on top of parts issues and produce successful products.

Sharing Access

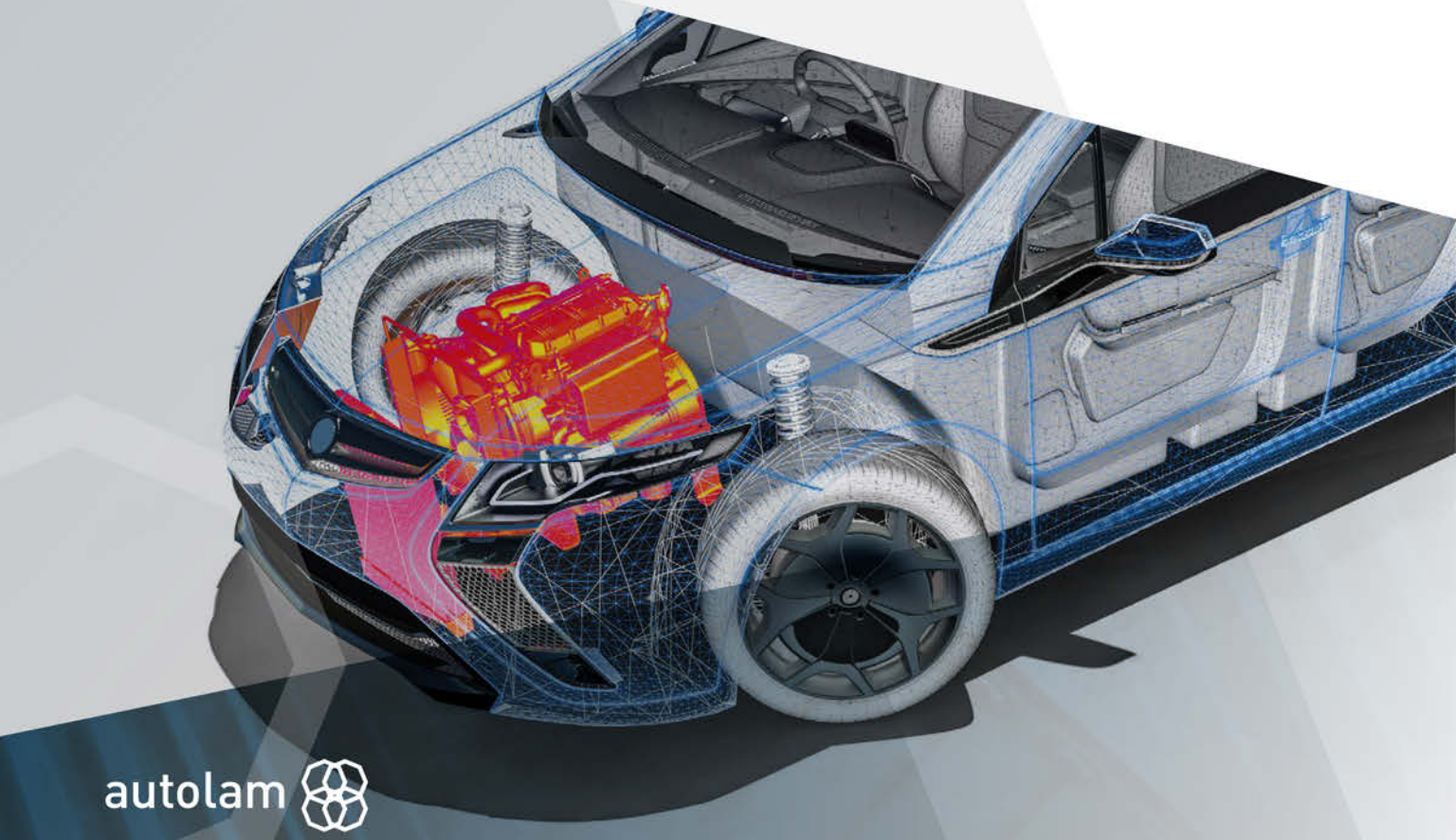
From the beginning of a new design through its completion, many colleagues can contribute to its success if provided access to the design. This may include mechanical, system, manufacturing, and service engineers. Sharing access to the design in a cloud-based depository, such as Altium 365, lets each team member review the design according to their perspective and needs. They can add comments or questions in the schematic or PCB views which become visible (and resolvable) by the designer in the CAD software. This makes reviewing the design and adding changes or answering concerns a breeze.

I have long enjoyed turning the mechanical engineer's MCAD PCB STEP model into the board's shape (mounting holes and all) with a button push. I'm no longer nervous waiting for the board to arrive or surprised when it looks and fits exactly as intended. Being able to do this while iterating between the ECAD and MCAD tools has sped several recent designs. No more emailing STEP files back and forth or sifting through shared drive storage for the Tuesday 3 p.m. file.

One design had particularly tight and variable height constraints and needed all the space it could get. When the design was ready to send to manufacture, both the EE (me) and ME were confident it was going to be right the

ventec
INTERNATIONAL GROUP
騰輝電子

autolam: Base-Material Solutions for Automotive Electronics



autolam

Automotive electronics technologies are evolving at an increasing rate. Paying attention to the properties of materials at the substrate level is the first step towards achieving the most stringent performance targets of today's automotive manufacturers. autolam offers the solutions demanded by the diverse and unique requirements of automotive applications today and in the future.

IATF 16949:2016 CERTIFICATION

venteclaminates.com

first time. The mechanical engineer can validate the 3D PCB when the design is completed with all the parts (including 3D bodies on all the components) to have the final, accurate model to include in their assembly.

Outside Contributors/Consultants

I recently had the opportunity to coordinate the design work of several consultants and bring their finished designs into my company's repository. They were able to do their work in our version-controlled cloud repository, which gave us all the benefits mentioned early (visibility, MCAD validation, and parts availability monitoring). The version control is implemented so well that it is almost invisible. We added the company templates and part numbers so that when the design was done, there was no final cleanup or documentation work needed.

This version-controlled ongoing access made it easy for the consultants to do their work and the inside team got visibility into the emerging design—to get started on specifying cables, checking the mechanical fit, and managing the components (more on that later). When complete, the design was already housed in our organization's home environment, ready for any future updates either by the original consultant or the internal team.

Supply Chain

Engineers often check component inventory at one or more distributors. Some have been known to base their design on the stock of a single distributor. I know I certainly used to do so. While this was always a little risky, pandemic-induced supply chain shortages have made it crucial to "see" the larger picture of parts availability. Altium's Octopart has offered this capability for a while.

In the past, I made it a standard practice to screen all BOMs for component availability, as well as NRND/obsolescence status or weird cost increases that can hint at high demand or EOL (end-of-life) issues. All PCB designers should make a habit of doing this for every design, regardless of their EDA tools.

Rich Tighe

BOM tools such as LiveBOM have proven especially important during this unprecedented time, giving us quick and regular stock status at a large number of distributors in one comprehensive view from within the design. All the red, green, and blue LEDs in one design went out of stock at all the top distributors overnight. But I knew about this in time; I specified substitutes before the design left for manufacture, and I avoided that dreaded phone call.

LEDs are a trivial example, easily substituted or respecified, but in the case of key ICs (especially microcontrollers) it has become more essential than ever to see the availability and possibly pre-buy BOM parts early in the design phase to avoid costly redesign or delay. The EMS provider I work with has agreed to pre-kit parts to assure stock when the design is ready for manufacture. Searching for parts by spec and availability directly in the ECAD tool really speeds parts selection and library

While others struggle just to keep pace, we stay ahead of the curve by constantly investing in the equipment, people and resources needed to be the best total PCB solutions provider in the industry.

VIEW OUR CAPABILITIES

EAGLE electronics Inc.

www.eagle-elec.com

creation. Many manufacturers provide easily acquired symbols and footprints which are a few clicks away from landing in the schematic. When manufacturers provide prebuilt symbols and footprints, I am more inclined to use their parts and it really speeds the design process.

Packages and Manufacturing

Sharing data packages directly with your manufacturer allows high confidence that the right version of the package is the one getting built, traceable by its unique release identifier and not "the files we sent Tuesday afternoon." Designers can make minor changes in response to CM questions or concerns and that keeps the documentation completely in sync. Keeping the schematic and BOM matched is good practice. With high-quality, consistent board packages, manufacturers can trust that the product they build is what is intended.

As electronics designers we often know the right thing to do. Having tools that make it easy for designers to do the right thing—such as collaboration, parts screening, and creating consistent, validated build packages—enables designers to quickly create successful, cost-aware designs. **DESIGN007**

Rich Tighe is an electrical engineering manager for Noah Medical.

Review: Siemens PCB Stackup Planning Webinar

by Cherie Litson, CID+
LITSON1 CONSULTING

I love getting to know new tools in the industry, especially when they address issues that haven't been included into the layout software yet. This Siemens webinar on stackup design, presented by Z-zero founder and HyperLynx alum Bill Hargin, offers a variety of tips for designers and design engineers. Bill walks you through the Z-Planner Enterprise tool and shows some cool features that can help anyone with planning their layer stack-ups. While some of what he's included can be found in other tools, there are a couple of things I found that were unique.

First, I have to say that I totally agree with Bill's statement "The PCB stackup is the central nervous system of the design." Also, I have experienced the issue of ordering boards from different fabricators and had different results in noise, costs, and other electrical performance.

His argument on manually adding stackup information is dependent upon which tool(s) you're currently using, but Z-Planner also interfaces with other PCB design software and signal integrity software. The work that has gone into developing this tool is evident and much appreciated. The tool has 150 different materials based on Dk and Df. This is to support selection of the material so that you don't over or under design it and possibly help to cut costs.

The "Library" contains the list of materials. This is editable and configurable depending on your needs. Then the "Material Mapper" allows you to select the acceptable loss factor and gives you a material selection. Some will be expensive, and some won't. The user still needs to communicate with the fabricator via a spreadsheet and email. I didn't see a way to add relative costs to the materials list.

The tool supports all the process stakeholders—from the OEM team to different fabricators. Using customer specs and running SI/PI analysis, comparisons can be made from the fabricators stackups. Communication with the fabricator via a spreadsheet and email is still necessary to make this work but does give us a tool that has more options to be able to come closer to consistency.

This webinar isn't for beginners. The viewer should already know how to set up a layer stackup for their design; this webinar doesn't guide you in this way. You have to learn that bit of information on your own from other sources. To watch this webinar, click here.

To read this entire review, click here.

Cherie Litson, CID+, MIT, is the owner of Litson1 Consulting and an instructor at EPTAC and Everett Community College.

Integrated Tools to Process PCB Designs into Physical PCBs

Visualize

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

Verify

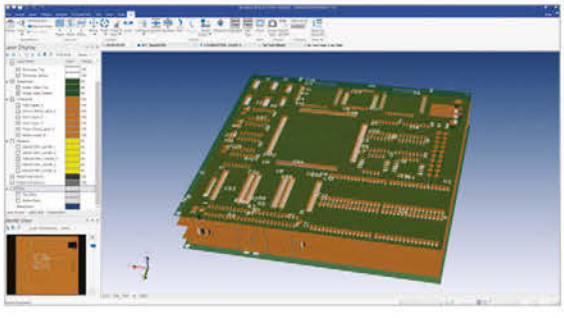
Ensure that manufacturing data is accurate for PCB construciton.

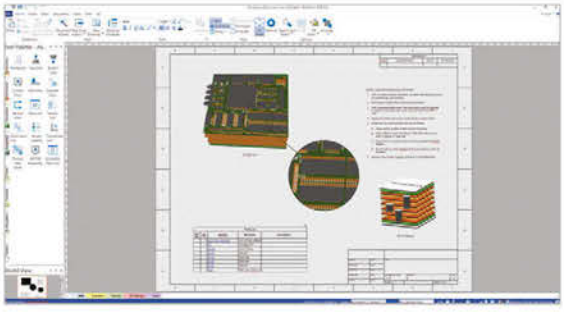
Panelize

Minimize costs in both PCB fabrication and assembly through maxiumum panel usage.

Document

Quickly and easily specify the PCBs form, fit and function.





CAM350®

Verify and Optimize PCB Designs for Successful Manufacturing

BluePrint-PCB®

Create Comprehensive Documentation to Drive Fabrication, Assembly & Inspection

DownStream Technologies, offers an integrated manufacturing data preparation solution for PCB post processing including manufacturing data visualization, stack-up definition, PCB panel visualization and support for 3D PCB documentation. This environment allows PCB designers and engineers to visualize, verify and document a printed circuit board before manufacturing.

DownStream Technologies

800-535-3226 | downstreamtech.com

NOW WITH 3D

The Collaborative PCB Design Process—

A Necessity for Efficient Manufacturing

Tim's Takeaways

Feature Column by Tim Haag, FIRST PAGE SAGE

It's been a few months since the June edition of *Design007 Magazine* was published, but since I missed the Star Trek theme of that issue, I hope you won't mind if I start with a related anecdote. *Star Trek* presented many fun gadgets and gizmos to its audience that have influenced ingenuity and invention for over five decades now. Some of these influences are pretty obvious, such as flip cellphones, which enjoyed a lot of free marketing due to *Star Trek's* communicator. Be honest, back when you had one of those phones, how many of you were tempted to say "Kirk to Enterprise" when you flipped it open? Other innovations, like phasers and photon torpedoes, thankfully remain solely in the realm of science fiction (at least for now).

Other than the starship itself, though, I think that most people would agree that the most interesting gadget on *Star Trek* was the transporter system. I am sure that the ability to beam out of danger, or a potentially embarrassing moment in life, has tilted many imaginations other than just mine. But did you know that unlike phasers and warp speed, which are very standard accoutrements for a science fiction show, that the transporter was invented solely for budgetary reasons?

When Gene Roddenberry was developing *Star Trek,* he realized that he couldn't afford to land a large spaceship multiple times in every episode. Even with the special effects of the time, it would have been too expensive to make it look even halfway realistic, yet he

APCT

Passion I Commitment I Trust

APCT'S LATEST EQUIPMENT INVESTMENTS MEAN

Further Advanced Technology Capability, Faster Lead Time, More Capacity

- New Electroless Lines
- New Lamination Presses
- New Plasma Systems
- New Direct Optical Registration Systems

Advanced Technology | Cycle Time Speed | Knowledgeable People

512.548.7435 | APCT.com

Printed Circuit Board Solutions

APCT.com

APCT Santa Clara HQ	APCT Anaheim	APCT Orange County	APCT Wallingford	APCT Global
408.727.6442	714.921.0860	714.993.0270	203.269.3311	203.284.1215

still needed some method or device to get his characters in and out of the action quickly. To resolve this problem, the transporter beam was conceived and became a basic fixture of the show. The actor would step into the transporter chamber on the sound stage, and then step back out with the camera still running. During post-processing the special effects department would cut and splice that footage together to make it seem like the actor had disappeared. To complement this vanishing act, footage of falling sand filmed under a high-intensity light would be optically merged with the disappearing actor to complete the illusion of futuristic transportation.

I probably should have preceded that revelation with "Spoiler Alert" before writing it, and hopefully I haven't ruined the show for you. My point, however, was to describe how financial necessity is often the mother of invention, and not just in 55-year-old science fiction television programing either. We can see how efforts to cut expenses and become more efficient can help create a better product in our own industry as well.

We can see how efforts to cut expenses and become more efficient can help create a better product in our own industry as well.

Circuit board design used to be a more complicated and lengthy process than it is now with the need to build scores of test circuits, develop multiple prototypes, and toiling with manual design operations. I'm not sure how many designers now have ever had to work with X-Acto knives, Rubylith, and opaque tape on a light table like I used to long ago, but if you did you will know what I'm talking about. Even after CAD systems were introduced into the design cycle, it could still be a lengthy process. The one good thing about all the time it took was that it gave ample opportunity for everyone to be involved. However, with so much automation in use, and with the ever-increasing need for faster time to market, many designs today seem to fly through the design cycle at warp speed. The great thing about speed, of course, is that when everything is going well, it is a glorious achievement. But if something goes wrong, that fast pace can produce a wreck on a wreck.

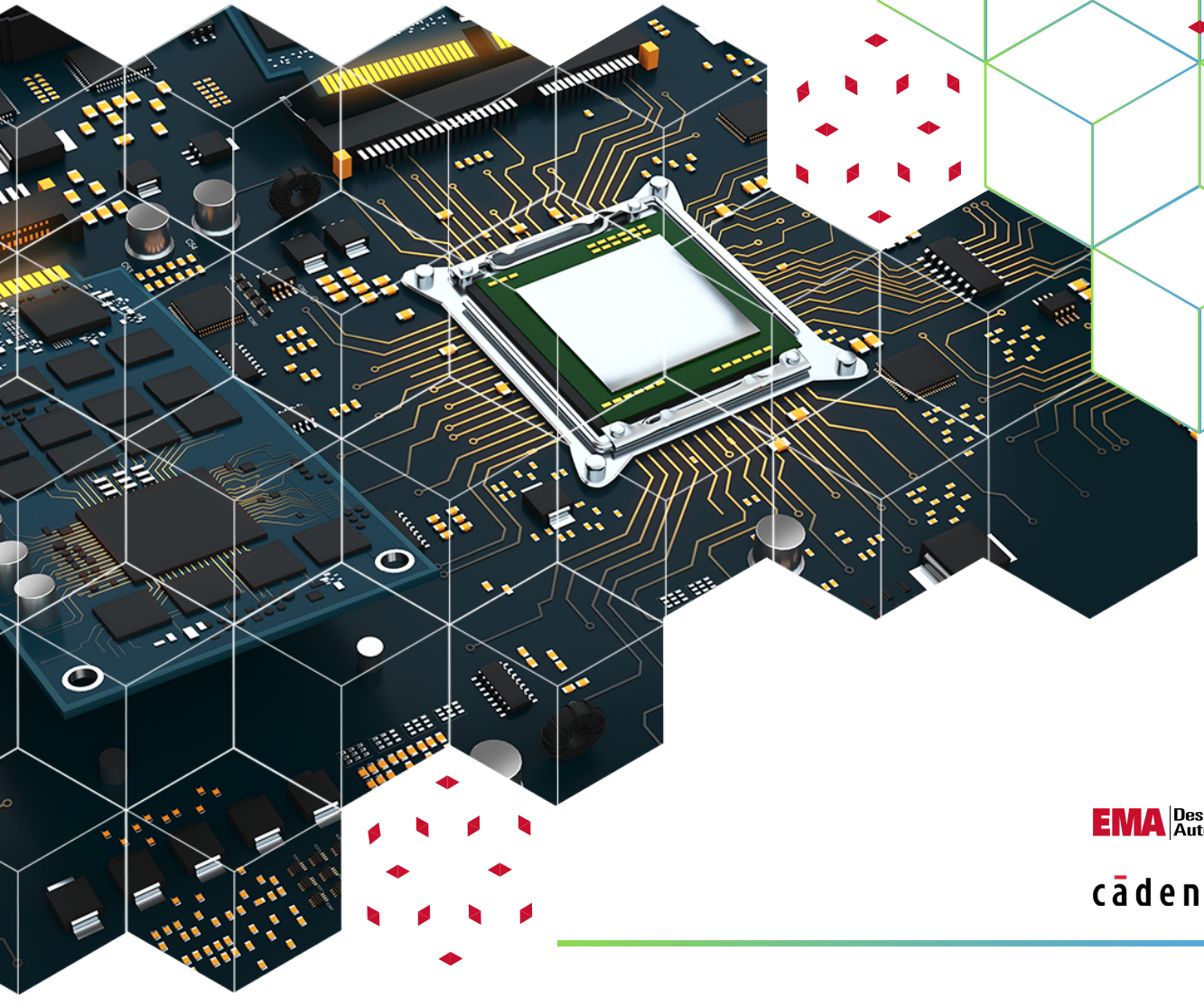
It has been reported to me that some designers have said that they don't even know which fabricators will be building their designs until the job is already out the door. On the flip side, I have talked with PCB contract manufacturers who want more interaction with PCB designers but aren't getting it. They have told me that they prefer to work together with designers prior to PCB layout to review the design and offer their input. Instead, the first they often know of a design is when they receive the completed manufacturing files. This forces them to find ways to make what they already have work, instead of helping the designers optimize their layout for better production yields and circuit board performance.

In our rush to get the product "out the door," are we sacrificing quality for market share? Are we spending more effort in trying to push a poor design through production rather than taking the time to improve it for error-free manufacturing, higher yields, and less expense? Are we finding out that here, too, financial necessity is going to be the mother of invention; in this case, the invention will be a more collaborative relationship between circuit board design and manufacturing? The plain and simple truth is that the need for better collaboration should be at the heart of all PCB process improvement. Here are some ideas on how we can go about doing it:

More Innovation,
Less Frustration.

When designing state-of-the-art technology, you need software that can keep up. The design features in Allegro provide real-time, actionable insights and guidance, allowing designers to consistently push the limits of technology. Turn your ideas into reality with the powerful features integrated directly within **Allegro® PCB Designer**.

Learn More



EMA Design Automation

cadence

1. Build a business relationship with your manufacturer prior to designing the board.

The business of PCB manufacturers is to build circuit boards, and they have a lot of experience building a great many types for different applications. With their experience they can add a lot of value to the project while it is still in the design phase. For instance, a good PCB manufacturer will review the board's bill of materials to ensure that the parts specified are available and at a good price. If there are problems procuring any of the listed components, the manufacturer will often be able to make recommendations on component or circuitry changes to help. By engaging with the manufacturer early on, you can leverage their knowledge and experience to help design a better board.

2. Involve all members of the design team when working with the manufacturer.

Many business relationships rely on a single point of contact, but that practice can hamper the productivity that a fully collaborative partnership can make possible. It is important that all stakeholders in the design have accessibility to their manufacturing partner in one form or another to speedily resolve issues and get questions answered. As an example, a part change recommendation from the manufacturer should be passed through the design team's CAD librarians as soon as possible to get new schematic symbols and PCB footprints ready for the design.

3. Use the engineering and other professional resources available at the manufacturer.

PCB manufacturers often have engineering teams that can help with schematic development or changes, design reviews, layout alterations or enhancements, and test development. In many instances, design modifications can be handled by the manufacturer, who will follow up with a completely new set of design documentation of their work. While this may not fit every design team's workflow, it is a benefit that should not be ignored to help with unscheduled changes or overflow work.

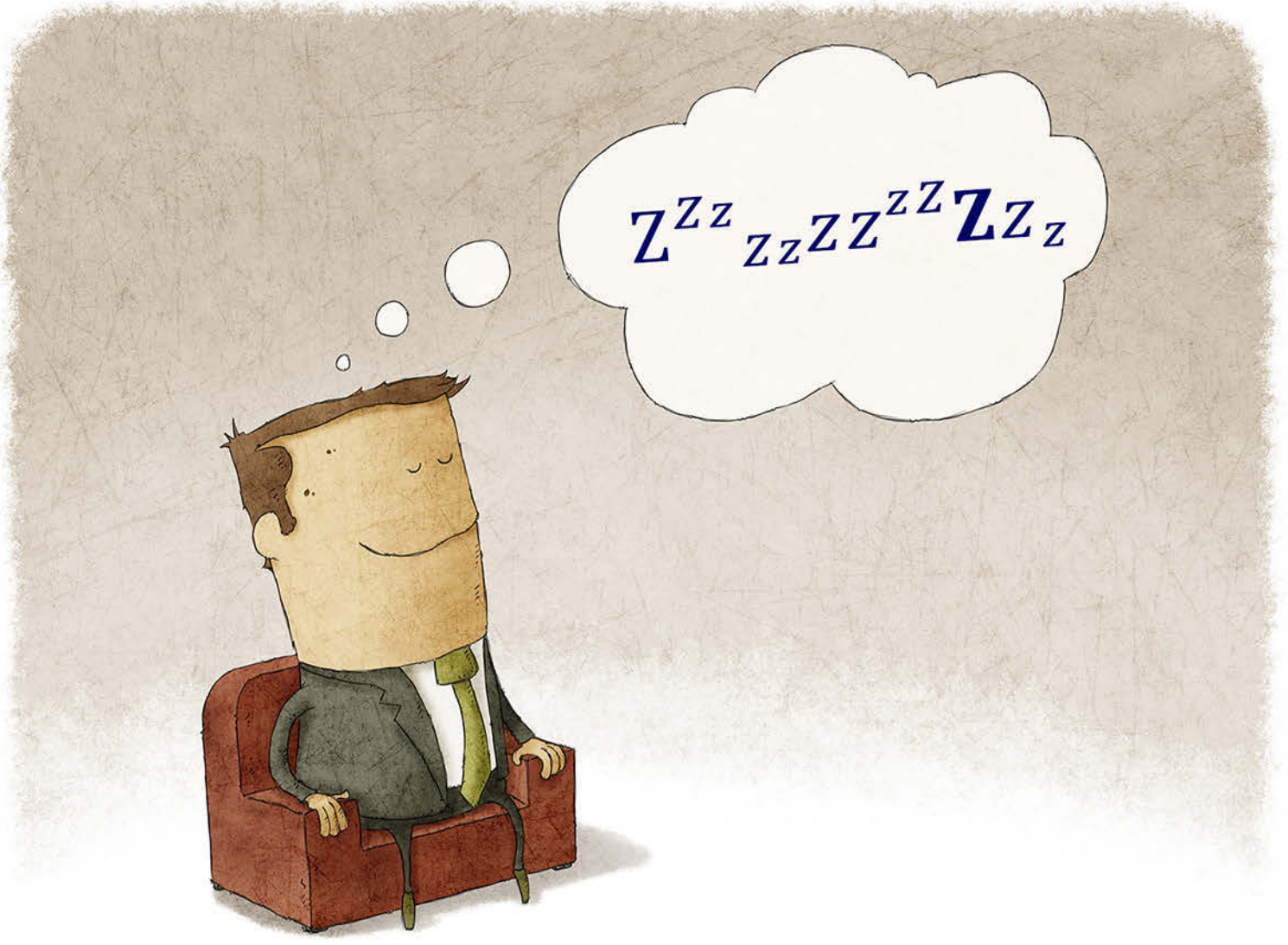
4. Explore the use of more collaborative tool functionality with your manufacturer.

Many design tools now feature utilities that help enhance the communication between circuit board design teams and PCB manufacturers. Using data transfer standards, such as IPC-2581, these design utilities allow design teams and manufacturers to efficiently exchange circuit board layer stackup information and fab and assembly files. Systems like these help the collaborative PCB design process by making it easier for designers and their manufacturing partners to communicate with each other as they work together to build a better product.

More and more designers and manufacturers are facing the financial necessity of finding new ways to improve the efficiency of the business partnership. Fortunately, by continually building on the collaborative relationships they already have, they are already on their way to further streamlining their design processes. As with all aspects of our industry, it will be rewarding to see how these collaborative relationships grow and what new benefits come from them. Until next time everyone, keep on designing. **DESIGN007**

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

We DREAM Impedance!



Did you know that two seemingly unrelated concepts are the foundation of a product's performance and reliability?

- Transmission line impedance and
- Power Distribution Network impedance

DISCOVER MORE

iCD software quickly and accurately analyzes impedance so you can sleep at night.

iCD Design Integrity: Intuitive software for high-speed PCB design.

"iCD Design Integrity software features a myriad of functionality specifically developed for PCB designers."
– Barry Olney

iCD
In-Circuit Design Pty Ltd

www.icd.com.au

Interaction—The Cure for Industry Contraction

The Digital Layout

by Kelly Dack, CIT, CID+, PCEA

Introduction

In this month's column, I share reports from our PCEA leadership on a flurry of activity over the past month. Next, I highlight some inspiring words that Chairman Steph Chavez recently delivered to our executive staff. Then, I wrap it up with a list of events for 2021 and 2022.

PCEA Updates

It seemed that the electronics trade show industry had been shrinking over the past year only to swell with a sudden, extreme realization that some venues were opening and the dormant plans for last year are now in the latent stage of metamorphosis and coming back to life. August provided a positive swell of relief in the form of a significant trade show called DesignCon, August 16-18, at the San Jose McEnery Convention Center. PCEA was happy to participate by setting up our first trade show booth ever. PCEA executive staff gives a special nod to Eriko Yamato, our PCEA events coordinator, on design, coordination and delivery of our booth, as well as some very special T-shirts to hand out during the show. Michael Creeden, PCEA vice chair and treasurer/coordinator of our PCEA sponsors, manned the booth throughout the show with some help from PCEA media coordinator Tara Dunn. Mike reports that while the show numbers were slightly down, he enjoyed the spirit of a "family reunion" and quite a few were interested in hearing about the value of becoming associated with the PCEA.

Figure 1: Michael Creeden shows off the new PCEA trade show booth.

PCBs are complex products which demand a significant amount of time, knowledge and effort to become reliable. As it should be, because they are used in products that we all rely on in our daily life. And we expect them to work. But how do they become reliable? And what determines reliability? Is it the copper thickness, or the IPC Class that decides?

Every day we get questions like those. And we love it. We have more than 490 PCB experts on 3 continents speaking 19 languages at your service. **Regardless where you are or whenever you have a question**, contact us!

What's your PCB question?
www.ncabgroup.com/PCBquestion

Reliable answers. Reliable PCBs.

NCAB GROUP

Figure 2: Michael Creeden and Tara Dunn looking forward to greeting show attendees at the PCEA booth.

Message from the Chairman

As I was closing out my weekly "post-a-notes" of action items, I noticed that many of them are regarding PCEA. One common theme that puts a huge smile on my face is the fact that PCEA is truly active and alive within the industry. Here are just a few highlights:

1. PCEA had its first major physical presence (booth) at an industry conference at DesignCon 2021.
2. PCEA Ohio had its initial kickoff meeting to jump start this new chapter.
3. An industry veteran has contacted me acknowledging his willingness to initiate a new PCEA chapter in Texas.
4. Our education committee is set to publicly release an official process for publishing industry technical content on our website.

And now, I am looking forward to our next major event. My focus is a PCEA booth and presence at PCB West 2021. It will be the first face-to-face event I will have attended since this pandemic hit the public. I can't wait for that integration with friends and fellow industry colleagues at this event.

Stephen Chavez

As I reviewed the list of PCB West 2021 presenters, I noticed several of them are PCEA members. I love this fact. Let us look forward to and continue to show a strong support for our fellow PCEA members, as well as our support the PCEA presence within the industry and at our next major event.

Best Regards,

Stephen V. Chavez MIT, CPCD, CID+, Chairman

Next Month

As Steph mentions, our PCEA educational committee has been hard at work crafting policies for publishing educational materials on the PCEA website and making them available to our members. Tomas Chester, our new head of the education committee, tells me that the committee will continue to review educational article submissions and even web links to outside existing resources. He hopes they will serve as an educational resource to the PCB engineering community. With some of the best minds in PCB engineering education on our PCEA educational committee, I look forward to hearing more from Tomas and reporting on what they come up with.

Upcoming Events

Here is our list of upcoming events. Hope to see you at any or all of these.

PCB West 2021
October 5–8, 2021
Santa Clara, California

SMTA International 2021
November 1–4, 2021
Minneapolis, Minnesota

productronica
November 16–19, 2021
Munich, Germany

PCB Carolina
November 10, 2021
Raleigh, North Carolina

AltiumLive 2022
January 26–28, 2022
San Diego, California

IPC APEX EXPO 2022
January 22–27, 2022
San Diego, California

PCB East 2022
April 11–13, 2022
Marlborough, Massachusetts

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

Refer to our column and pce-a.org for upcoming industry events. If you have not yet joined the PCEA collective, please visit our website and find out how to become a PCEA member.

Conclusion

We've all been waiting for our industry to cease contraction and expand back to a form so robust that it will enable the flow of information and ideas again. How great it was to experience this through our PCEA representation at DesignCon last month. Now is the time to tap in to the PCEA for what is coming your way in the future. As Steph indicates, this organization is committed to participation in the trade show industry and to reach out with educational support to its membership. Who knows what great educational, presentational, and organizational endeavors will grow from the PCEA by placing a simple blue trade-show booth on a show floor? I am excited to hear about any potentially historic and monumental leadership opportunities which may come about in the trade show industry as the PCEA sets up its next event right around the corner. Could we be looking forward to any big announcements? Yes, please.

See you next month or sooner. DESIGN007

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

Robots Who Goof: Can We Trust Them Again?

When robots make mistakes, reestablishing trust with human co-workers depends on how the machines own up to the errors and how humanlike they appear, according to University of Michigan research.

"Robots are definitely a technology but their interactions with humans are social and we must account for these social interactions if we hope to have humans comfortably trust and rely on their robot coworkers," said Lionel Robert, associate professor at the U-M School of Information.

Robert and doctoral student Connor Esterwood recruited 164 participants to work with a robot in a virtual environment, loading boxes onto a conveyor belt. The human was the quality assurance person, working alongside a robot tasked with reading serial numbers and loading 10 specific boxes. One robot was anthropomorphic or more humanlike, the other more mechanical in appearance.

The robots were programed to intentionally pick up a few wrong boxes and to make one of the following trust repair statements: "I'm sorry I got the wrong box" (apology), "I picked the correct box so something else must have gone wrong" (denial), "I see that was the wrong serial number" (explanation), or "I'll do better next time and get the right box" (promise).

When the robot was more humanlike, trust was even easier to restore for integrity when explanations were given and for benevolence when apologies, denials and explanations were offered.

Esterwood said this study is ongoing with more research ahead involving other combinations of trust repairs in different contexts, with other violations.

(Source: University of Michigan)

MilAero007 Highlights

XRF Analysis Adds to Spirit's Test Services ▸

Spirit's new Hitachi EA6000VX bench analyzer can inspect the material composition of your product. XRF inspection is essential in military and aerospace supply chains to verify that leads and finishes contain the correct ratio of lead to prevent whiskering.

Electronics Industry Summit With U.S. DoD Spawns More Dialogue, Collaboration ▸

The U.S. government, the electronics industry, and academia must continue to step up their joint efforts to address risks and gaps in the defense electronics supply chain.

Mars Rover Mission Only Just Beginning One Year After its Launch ▸

Of all the exciting aspects a space mission entails for a planetary scientist who makes a living studying the outer limits, it's still the launch from ground zero that gets Briony Horgan's heart-pounding the most.

Catching Up With AMI's Jim Barry ▸

Columnist Dan Beaulieu met Jim Barry 25 years ago while working on a project at Eltek Ltd. Jim discusses his new company's plans, including its move deeper into the mil/aero market.

Defense Speak Interpreted: Decoding the Military's COCOM ▸

Have you ever followed Defense activities around the world and been confused by terms like CENTCOM or SOUTHCOM? Who's in charge of worldwide Defense activities—just "a big guy at the top" or regional commanders? How do Army, Navy, and Air Force stay coordinated around the world in various geographies?

New Managing Director for NCAB Group Denmark ▸

After three successful years as managing director for NCAB Denmark and 30 years in the PCB industry, Jan Kronblom Thomsen will leave NCAB Group to pursue other interests. Peter Jensen will take the position as the new managing director, starting his position September 1.

TT Electronics Appoints Emma Darke as Group Sustainability Director ▸

TT Electronics, a global provider of engineered electronics for performance-critical applications, announced the appointment of its first-ever group sustainability director.

Summit Interconnect Announces PCBAA Membership ▸

Summit Interconnect is proud to be a founding member of the new Printed Circuit Board Association of America. The PCBAA was established in 2021 in response to increasing geopolitical challenges facing U.S. microelectronics manufacturers.

Raytheon: A Radar for Any Mission ▸

Raytheon Technologies is redefining the radar with what's known as the "software-defined aperture," making single arrays far more capable and flexible through secure software upgrades. The digital transformation in radar development offers benefits in every domain—land, sea, air, space, and cyberspace.

www.prototron.com

Serious About Speed

For over 30 years Prototron Circuits has led the pack when it comes to providing quality circuit boards FAST.

Be it Class 3, AS9100, ITAR or MIL-PRF-31032, Prototron has the speed you need.

Prototron Circuits

America's **Board Source**

Collaboration to Shorten the Learning Curve

PCB Talk

Feature Column by Tara Dunn, AVERATEK

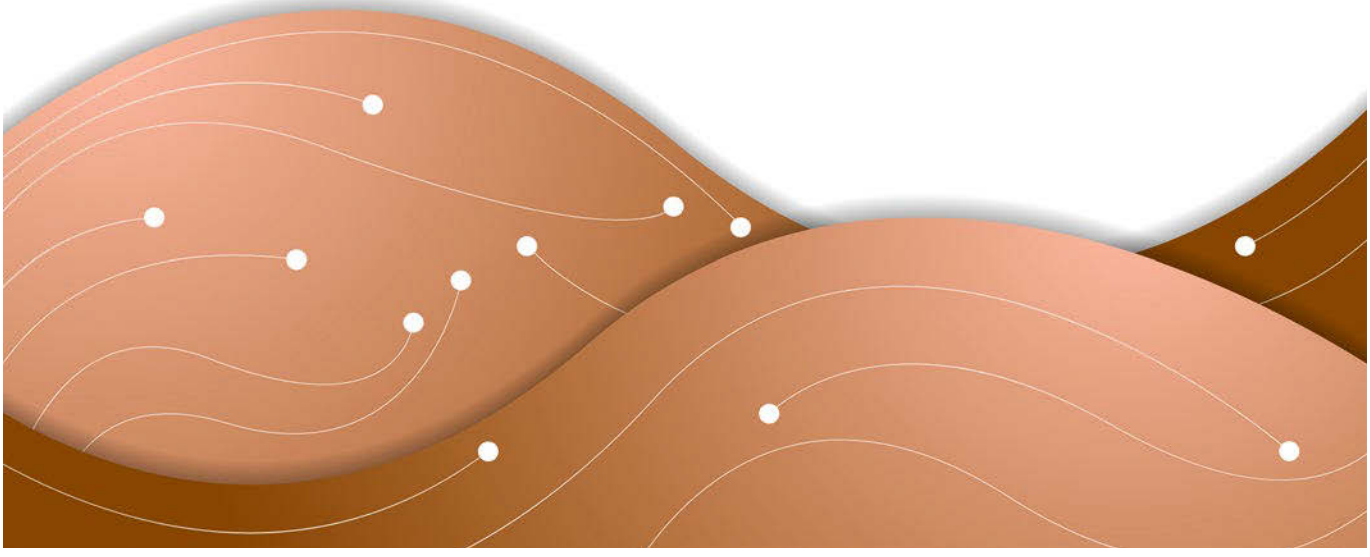
For those of you following my column or who have attended one of my sessions on flex and rigid-flex, you know that the number one message I try to drive home is to work with your fabricator early in the design process, especially when you are new to designing with flexible materials. I can tell countless stories about flexible circuit missteps:

- "The flex that didn't flex"
- "The flex conductors that cracked when flexed"
- "The flex that took eight weeks longer and cost an additional $2,000 than necessary because uncommon materials were specified"

The list could go on and on.

I can also tell countless stories about designers working with fabricators early in the process resulting in the most robust circuit, successfully dynamically flexing, on time and within budget. The key to success is working in partnership with your fabricator. Did you notice that I reiterated that key point twice, just in my opening statements? That is how strongly I feel about that message.

As I have shifted my focus to additive electronics, specifically circuitry that spans the gap between subtractive etch and IC scale, this message becomes even more pertinent. While there are certain high-profile, high-volume designs that have been fabricated with semi-additive PCB processes for several years, the technology is now also serving the low-volume, high-mix market domestically and offshore. As with any shift in technology in the PCB industry, this comes with a learning curve for both the design community and the fabri-

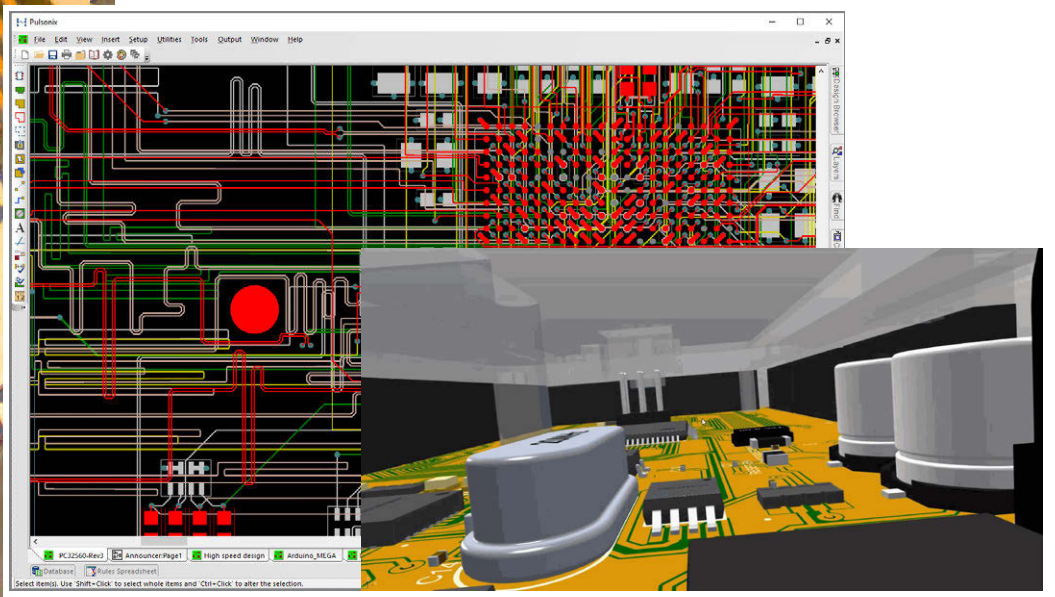


Pulsonix PCB Design Software

PCB design done right

New Version 11.0 Increases Speed up to 80%

Multi-threading and GPU processing in this new release create a gigantic leap in speed, dramatically reducing the time it takes to perform many PCB design operations, such as Design Rule Checking (DRC), Copper Pouring, and Gerber generation. That's PCB done right!



Advanced PCB Design Features

- Intuitive placement and routing
- Dynamic copper pour
- Full rules engine for automatic DRC
- High-speed design rules
- Heads-up display length matching
- Rigid-Flex, embedded components, and chip-on-board
- Revision control and PLM integration

Try Pulsonix yourself at pulsonix.com/trial

PULSONIX

cators, as we all learn how to best apply and take advantage of these new capabilities.

As I diligently work to disseminate information and to help shorten this learning curve for both designers and fabricators, I am routinely asked for "design rules" for SAP and mSAP. That is a fair request; after all it is what we have been requesting from fabricators for decades. Only this time, it is a little trickier. There is technology capability involved with creating a PCB trace and space with semi-additive or modified semi-additive processes that is dependent on each fabricator's equipment. These processes change the manufacturing constraint for trace formation from the etching process to the photolithography process; feature size capabilities will differ depending on equipment capabilities and photoresist. In fact, SAP technology can go far below the 25-micron trace and space that we see in marketing materials and if your fabricator has photolithography equipment that goes to 5 microns, they can produce at that feature size.

As I diligently work to disseminate information and to help shorten this learning curve for both designers and fabricators, I am routinely asked for "design rules" for SAP and mSAP.

Semi-additive and modified semi-additive processes also have very little direct impact on other PCB fabrication processing steps that come before or after the trace and space formation processes. Drilling is a good example. If your fabricator can only offer a 6-mil mechanical drill with subtractive etch processing, they can still only offer that feature size after installing a semi-additive process, unless they also make a capital investment in drill equipment at the same time. Materials is another good example. If your fabricator is currently running flexible materials, rigid-flex, and exotic materials, these are all compatible with semi-additive processes, but as we have all experienced, not every fabricator specializes in these materials. That is independent of the circuit formation, semi-additive processes.

I think that is fairly intuitive. Here is where things get a little trickier and we start entering the territory of moving through a learning curve. Semi-additive PCB fabrication techniques don't require every layer to be done with semi-additive technology; in fact, it is common to mix layers of subtractive etch with semi-additive layers. One thing to consider is what we are terming "layer pairs." As an example, if layer two is using semi-additive processes, the related layer on that core should also use semi-additive processes. It is not strictly required, but the fabrication process will be much simpler, resulting in improved yield and reduced cost. This is also related to fabricator capabilities and preferences—reinforcing the need for designers and fabricators to work collaboratively with this new technology.

Let's look at another example. The starting point is an eight-layer design with five signal layers. As a starting point, we took the manufacturing capability to 35 microns, a very comfortable feature size for semi-additive processing. This resulted in a reduction to a four-layer design with two signal layers and two power ground layers. Sounds impressive, doesn't it? It is and would certainly be a win for design simplification and manufacturability. But, working collaboratively, we realized that the impedance control in that particular stackup would be impacted with traces with that tight pitch. After further review, it was decided to use 50-micron traces and 22.5 micron spacing to meet the impedance requirements. The

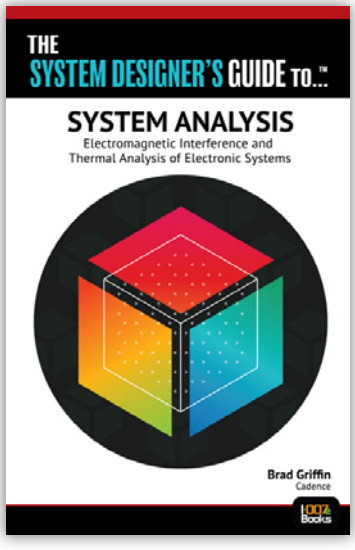
routing was still accomplished in those four layers. Because we were having a detailed dialogue through this process, it was also suggested that reducing this design from eight layers to six layers, adding in additional power and ground layers, would still result in a simplified construction while at the same time potentially yielding power benefits that had previously not been available. Interesting to ponder, isn't it?

One certainty I can bring to this discussion is that semi-additive processes can significantly simplify complex designs. Some are working to simplify existing designs; others can create a new design utilizing these new manufacturing capabilities. Designers have a learning curve to navigate. Fabricators have a learning curve to navigate. Collaborating with and supporting both as the industry adopts these new technologies is sure to shorten the learning curve for all. **DESIGN007**

Tara Dunn is the vice president of marketing and business development for Averatek. To read past columns or contact Dunn, click here.

Excerpt: *The System Designer's Guide to... System Analysis*

Chapter 2: Challenges in Design and Development of Electronic Systems



The exponential growth in data center infrastructure for IT networking introduced numerous challenges, from limited ecosystems to high-performance computing issues. There are many constraints on building data centers and updating the equipment in them. Planning is critical in managing increased capacity in the existing data center space. Increased rack density disturbs the prevailing power distribution infrastructure. When more devices are added to the existing space, temperature increases, and the need for containment solutions and precision cooling arises. Also, the components must be able to handle higher temperatures. Managing load capacity/phase power and weight of the equipment is another challenge. In addition, racks also have energy efficiency issues, and rack depth can cause incompatibility with newer designs. While other challenges in the context of hardware and software exist—like advanced node implementation at 7nm and verification of complex domain-specific architectures—this section focuses mainly on system analysis aspects. Data centers require high-computing devices in small footprints. With a decrease in transistor size, an advanced node is created. Small form factor brings several gains like higher density and faster switching. However, at the same time, advanced nodes take the design and integration complexity to a new level.

Decreasing metal pitch leads to coupling effects and signal integrity issues. Increasing wire and via resistance requires more advanced and variable wire sizing and tapering techniques to compensate. Server signals, chip complexity and cost, power management and electromigration, achieving performance goals, lithography limitations, process complexity and variability in extraction, timing, signal integrity analysis, and modeling, package complexity, shorter time-to-market, and project management (engineers/project cost) are some of the critical challenges in advanced node chip design.

To download this free eBook, published by I-Connect007, click here.

To view the entire I-Connect007 eBook library, click here.

Designing PCBs for Electronic Hardware Products

Connect the Dots

by **Matt Stevenson,** SUNSTONE CIRCUITS

We asked an expert what factors designers should consider as they lay out their boards

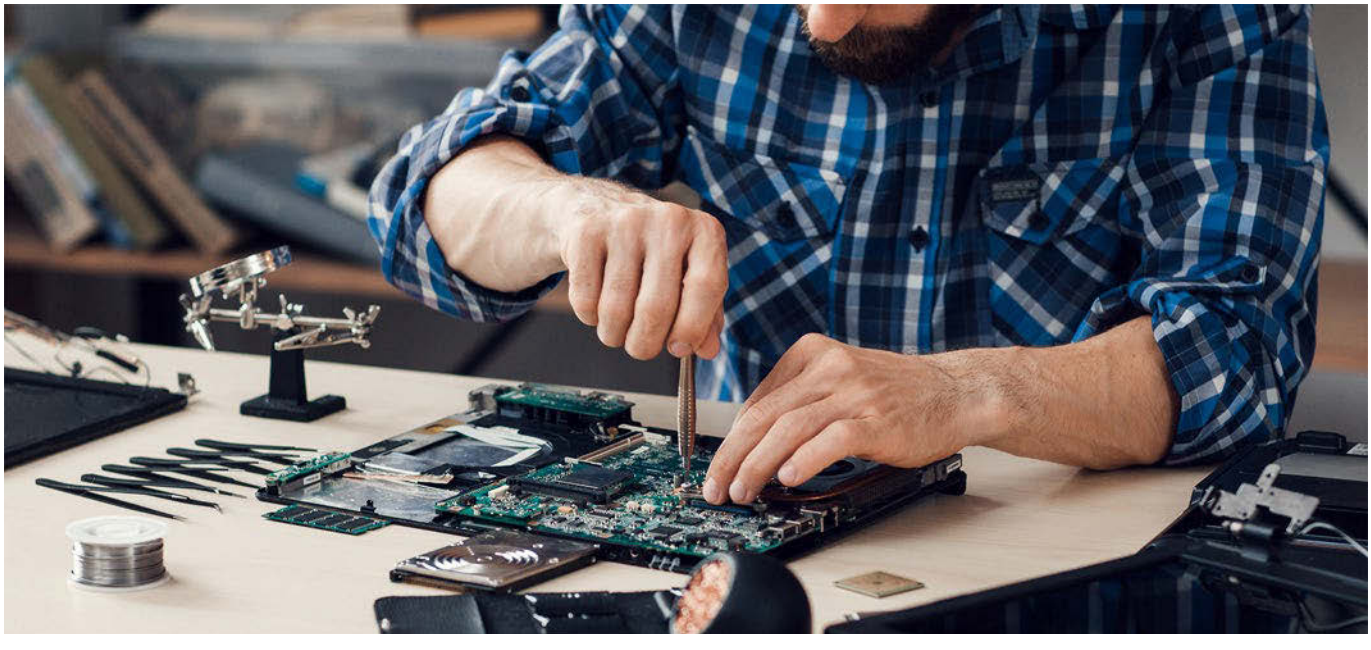
Every day we field questions from PCB designers about how to improve functionality of their boards or make them more manufacturable. Hardware entrepreneurs and designers for organizations developing new electronic products are an important subset of our customer base. Uncertainty surrounding the cost of product development is a big challenge as they compete with large corporations on the innovation front.

To help them make design choices that won't cause cost overruns down the line, we often turn to our trusted partner John Teel of Predictable Designs for counsel. Teel has dedicated himself to helping entrepreneurs create new electronic products, guiding them through the electronic product engineering process from initial cost estimates to schematic design and on through programming and testing.

We sat down with Teel recently to discuss what PCB designers need to be thinking about in terms of the bigger product development picture. This article surfaces several important takeaways from that discussion.

1. Simplify your product, but don't compromise on key features.

The process of product development is complex and difficult enough even with a relatively simple product. If you are building something more complex, it will take longer and cost more to develop and manufacture. By wager-



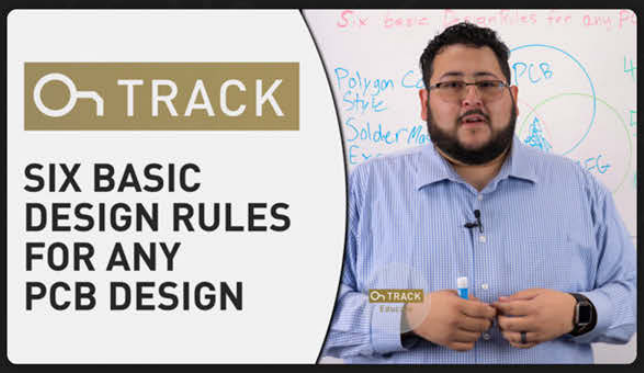
ALTIUM ACADEMY

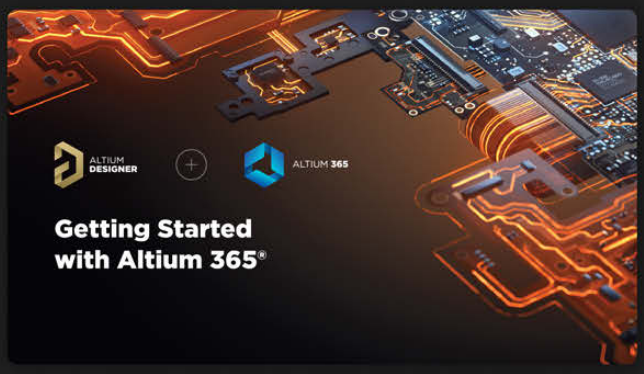
Online Education for PCB Designers and Electrical Engineers

Access over 350 tutorial videos

- On-demand Webinars
- What's New with Altium Designer
- Getting Started with Altium 365
- Whiteboard Series

Watch Now





Altium | YouTube

ing more in terms of time and money, product developers magnify their risk.

Before attempting to develop a product that is perhaps needlessly complex, look at your product from the customer perspective. Determine what product features are crucial to delivering value and then develop the absolute simplest product possible that meets the need.

"Non-fixed cost is a real risk for entrepreneurs, because projects often go over budget in the design phase," Teel said.

If an electronic device is less complex, the board designs probably will be too—making this early phase a critical one for staying on budget. Less onerous PCB designs take less time to complete, require less collaboration with manufacturers on the front end, and are less costly during PCB prototyping.

They also help get products to market faster and with fewer unpleasant surprises.

2. Embrace iterative product prototyping.

As with PCB prototyping, electronic product prototyping is all about learning. There are multiple phases of prototyping during product development, beginning with a proof-of-concept (POC) prototype. This early-stage version rarely functions or looks like the final product, because it has only one goal: prove the fundamental concept of the product at the lowest possible cost.

The next phase of prototyping should separate appearance from functionality. The "looks-like" prototype, usually created using a 3D printer, exists to demonstrate the look, feel, form, and aesthetics of the new product. Following the looks-like, the "works-like" prototype will have functional internal electronics and require development of a custom PCB to house and connect the product's discrete electronic components.

PCB design and prototyping often represent the most expensive phase of product development. It is critical for the hardware entrepreneur to foster relationships with reliable partners for PCB manufacturing and assembly. Without them, successfully building both engineering (works-like, looks-like) and preproduction prototypes can become a slow and costly process.

John Teel

"I don't ever have to tell them what to do," said Teel. "I send the design files, and they make it happen."

3. Seek expert guidance throughout the development process.

To have a better chance of bringing new product ideas to life, the development process should be as predictable as possible. Seek guidance from experts who have learned from experience what works and what does not.

Even experienced product developers can benefit from a fresh set of eyes when facing obstacles. On a recent collaboration between Sunstone and Teel's Predictable Designs, we encountered a surprisingly complex challenge related to vias: tunnels that connect different layers of a PCB. The PCB design for a high-powered, pocket-sized personal device used blind and buried vias—connections not visible

IPC BUILD ELECTRONICS BETTER®

PCB DESIGN COURSES

DRIVE QUALITY AND INNOVATION

Introduction to PCB Design II	Military & Aerospace Applications	Rigid-Flex Boards
Upcoming class:	Upcoming class:	Upcoming class:
8/31–10/21	8/30–10/6	8/30–10/6

Look for these additional courses!

- PCB Design for Manufacturability
- PCB Design Embedded Components
- Introduction to PCB Design I
- PCB Design for Advanced Packaging
- PCB Design for Extreme Environments
- PCB Design for Printed & Wearable Devices

Learn more about our PCB Design Courses.

through one or either of the PCB's outer layers.

"Sunstone helped me understand that what I had come up with just wouldn't function as needed and had the potential to increase the cost of the project two- or three-fold," said Teel.

Sunstone and Predictable Designs engineered an alternative, ensuring the product retained its small size and power capability. Design input like this from your PCB manufacturer can prevent production cost overruns and costly rework of board designs.

Collaboration is vital to success for entrepreneurs and smaller organizations competing with bigger companies in the product innovation space. The big companies can throw more money and resources at a product idea, so it behooves the independent product developer to seek guidance from trusted experts beginning with concept creation all the way through product launch.

We believe this offers innovators the best method to reduce overall cost of development, speed the product to market, and realize their full potential. **DESIGN007**

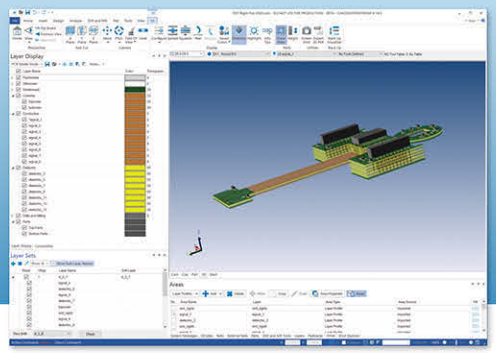
Matt Stevenson is the VP of sales and marketing at Sunstone Circuits. To read past columns or contact Stevenson, click here.

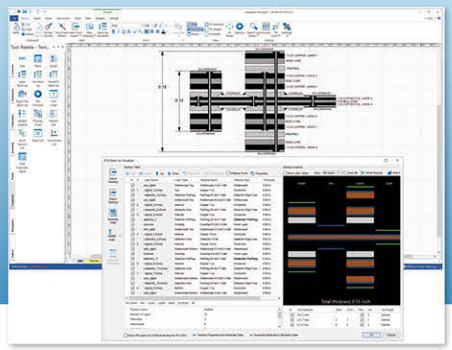
Support For Flex, Rigid Flex and Embedded Component Designs Now Available.

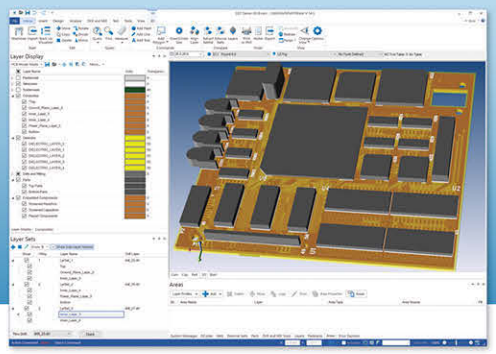
BluePrint-PCB®

CAM350®

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







DownStream Technologies

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

FLEX007
A SPECIAL DESIGN007 MAGAZINE SECTION

Applying DFM Analysis to Flexible PCBs

Flex007 Article by Patrick McGoff
SIEMENS DIGITAL INDUSTRIES SOFTWARE

The use of flexible printed circuit boards (PCBs), whether stand-alone or to connect rigid boards, has rapidly grown over the past decade. This growth has been driven by the proliferation of smaller form factors of products, such as wearable electronic devices. Because of their unique properties, flex circuits require special manufacturing methods as compared to rigid PCBs and can be less tolerant of issues that negatively affect manufacturability.

This article describes design-for-manufacturing (DFM) methods for flex circuits to help identify and correct problems during design. Applying these DFM techniques can save costs and ensure the project stays on schedule.

Flex circuit designs have unique properties and materials that separate them from rigid printed circuit boards (Figure 1). The flex circuits' thinness makes them more delicate, and

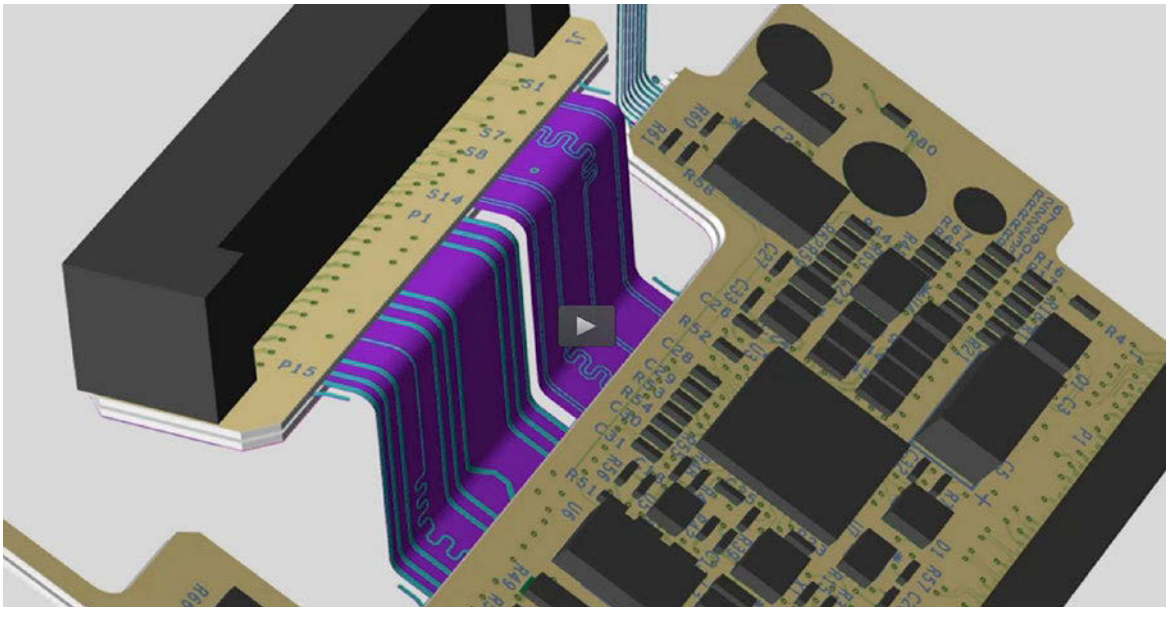
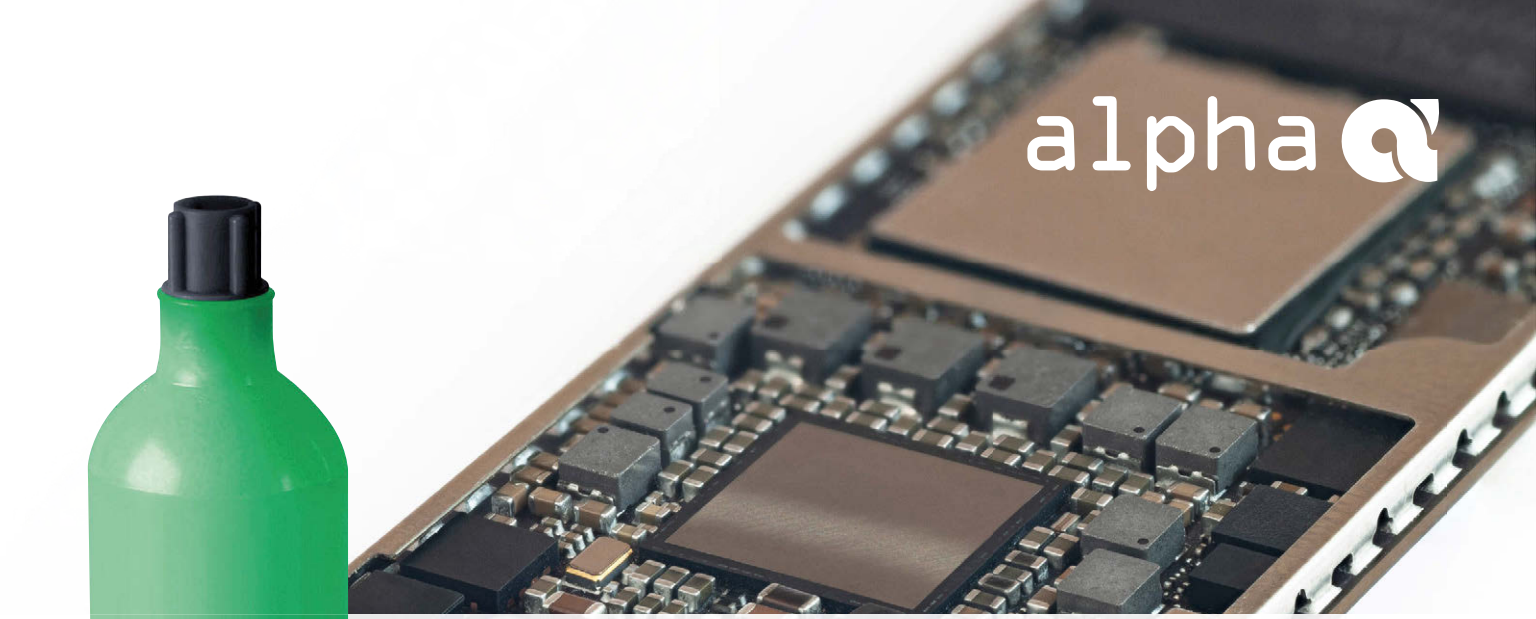


Figure 1: Flexible circuits introduce new challenges compared to rigid PCBs such as bending and flexing as well as different adhesion characteristics of copper traces.



ALPHA® OM-372

High Electrochemical Reliability
Ultra-Fine Feature No-Clean Solder Paste

ALPHA OM-372 is designed for superior performance on assemblies with ultra-fine pitch components requiring excellent stencil transfer efficiency and high electrical reliability, such as those found in mobile and wearables, computing, and medical assemblies.

ALPHA OM-372 Key Features:

- Best-in-class electrochemical reliability on fine pitched low standoff packages.
- Ultra-fine feature printing and reflow capability down to 008004 components.
- Minimum post reflow residue provides high reliability performance for fine pitch, high density designs when flux is entrapped under devices
- Excellent HiP/NWO Performance on high I/O count packages.
- No-Clean, Zero-Halogen

MacDermid Alpha ELECTRONICS SOLUTIONS

www.macdermidalpha.com

Conductors	Cores/prepregs/other materials	Adhesives and other materials
Rolled annealed (RA) copper	Polyimide	Pyralux® FR flex-core copper-clad laminate
Electro-deposited (ED) copper	Teflon®	FR406 No-Flo® and FR406 Lo-Flo® resins
Aluminum	Pyralux® FR flex core	DuPont CB028 silver conductor resin/ epoxy
Carbon	Mylar®	Silver mask
Silver ink	Kapton® polyimide	Stiffeners
Inconel®	Polyimide	None
Constantan	Teflon®	None

Table 1: Common construction methods for flexible circuits.

they are more susceptible to accidental damage during manufacturing. This negatively affects the yield, increasing the per-item cost. Let us explore why running concurrent DFM is even more important with flexible circuitry than for rigid PCBs.

Flexible circuits are typically built using a combination of material types (Table 1) and methods (Figure 2). These various materials present a challenge for manufacturing. When combined with typical FR-4 PCB materials, the complexity increases. The greatest challenge is posed by differences in the thermal expansion characteristics of the various materials. These characteristics need to be addressed in DFM reviews.

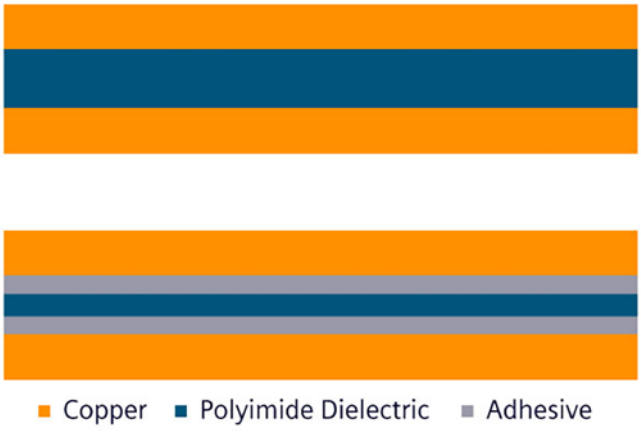
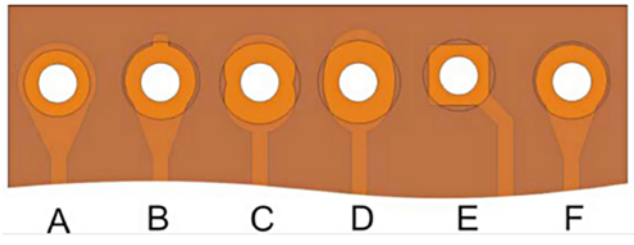


Figure 2: Flex circuitry can be constructed with or without adhesives. Adhesive-less circuits are used in applications needing higher performance, while those using adhesives are often used for low-layer count applications.

Designs that have both rigid and flexible circuitry contain all circuit construction elements; separate rigid circuits are connected by flex circuits to form a single assembly, eliminating the possibility of mating and signal routing errors. While rigid-flex can be more costly, in actuality, they may be more cost-effective and increase yield with a reduction of touch steps during assembly.

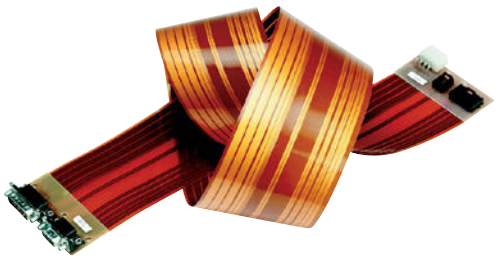
10 DFM Challenges for Flex and Rigid-Flex

Flex circuits and the interface between rigid and flex segments can have problems that do not exist with standard rigid PCBs. The following are 10 potential problems and how to solve them proactively.



Various pad designs to help facilitate their capture by the cover layer: (A) Standard filleted pad with full pad capture, (B) Standard filleted pad with hold down tab, (C) Overlapping pad design, (D) Oval pad design, (E) Corner entry to square pad, (F) Plated through holes normally require only filleting.

Figure 3: Techniques to provide support for surface mounting of components on the flex circuit.



all flex

Flexible Circuits • Heaters • Assemblies

- Design application assistance
- Component assembly
- Fast response
- Quick delivery

All Flex Flexible Circuits & Heaters
1705 Cannon Lane
Northfield, MN 55057
(800) 959-0865
(507) 663-7162

allflexinc.com

1. SMT components on the flex material

Placing surface-mount components on the flex circuit is possible by using selectively bonded stiffeners where required; several are shown in Figure 3. Copper on a flexible substrate is more susceptible to delamination than on a typical rigid board; consequently, surface mount pads require support, typically using a coverlay overlap onto pads and vias. Enlarged pads can be used instead of a coverlay, which provides greater area on the pad to adhere to the laminate. Using both support methods produces a pad with excellent adhesion.

2. Make pads larger than holes in coverlay

Making pads larger than the access holes in the coverlay (Figure 4) can significantly increase bond strength while decreasing delamination possibility. This technique is similar to solder-mask-defined pads for rigid PCBs. With flex designs, the coverlay adds additional anchoring to the copper features.

3. Bend area conductor spacing

Bend areas are another area that require careful design. The rule of thumb for conductor spacing is 5 mm between the rigid board and the bend area of the flex circuit (Figure 5). Bends closer than this are much more susceptible to breakage. Also, without proper spacing, the coverlay under the rigid area can peel and expose copper.

4. Sharp trace angles at bend points

There is a second issue with bend areas relating to trace direction. Flex circuit reliability and quality can be improved by ensuring that traces flow into the flexible circuit bend with little or no change in trace direction, also seen in Figure 5. If a change must be made in the direction of the traces in the bend, use curves instead of angled paths to reduce the chance of cracking the traces in the bend.

5. Conductor width

Conductor width is also especially important. A safe rule-of-thumb for conductor widths is to design them at least five times as wide as they are thick. Our recommended practice is using the thinnest copper trace that will safely carry the intended signal and then adjust trace width accordingly.

6. Plane hatching

Stress at the bend points can be relieved using a technique called plane hatching. While a honeycomb shape (hexagon) will alleviate the most angular stresses, any hatching will have a strain reduction when compared with that of a solid shape.

7. Pad fillets improve strength

Pad fillets (example F in Figure 3) improve the yield and strength of flexible PCBs. They should be employed in instances where the diameter of the pad exceeds the connecting trace width. You should also avoid acute angles at the interface between traces and pads with fillets to minimize the stress at the junction.

8. Interface region

If a coverlay does not extend over the entire rigid area, the area is called an interface region as shown in Figure 6. This is where the flex and rigid components meet. Typically, the area extends about 1.2 mm from the edge of the rigid board and into the flex area for a combined 2.5-mm region. The interface region can be more fragile than the rest of the design, requiring special design considerations. These include avoiding vias, mounting holes, slots, plated through-holes close to where flex and rigid join and designing the angle where traces enter rigid sections of the circuit to be anything other than 90 degrees.

9. Relaxed tolerance

Tolerances for flexible circuits must be more relaxed than for rigid PCBs. The materials and

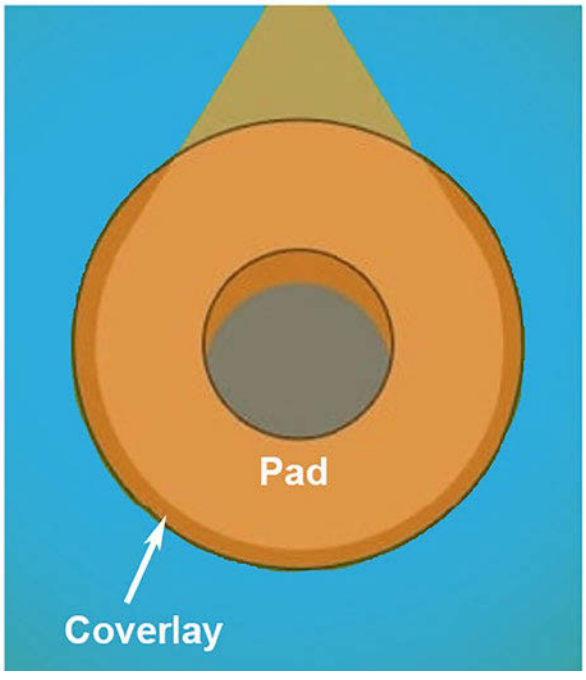


Figure 4: Pads that are larger than coverlay openings add strength.

construction techniques used in flex circuits can shrink, expand, stretch, and compress in different ways than rigid PCB counterparts. Relaxing tolerances allows for these events.

10. Thermal expansion

Most adhesive-based flex base materials are unfilled, which may result in unchecked thermal Z-axis expansion. This makes vias and other features prone to delamination and causes potential intermittent responses after components have been soldered to the surface of the flex material. To combat this, include a more robust pad stack, as compared to the rigid areas, for components soldered on the flex material.

Releasing to Manufacturing

Once designs are completed and validated, the product model data must be sent to the fabricator and assembler with specific manufacturing instructions. These must be included at a minimum to receive a correct board:

- Class of product
- Materials to be used for construction
- Base material and coverlayers
- Metal foil type and thickness
- Definition of holes, hole position and hole size
- Stackup showing number of layers and cross-sectional view
- Coverlayer or covercoat opening locations
- Board outline with datum and dimensions
- Marking requirements, materials and locations
- Bend and flex locations and direction of bend
- Stiffener locations and bonding requirements
- Special process and/or finish requirements
- Test point locations
- Special testing requirements

Figure 5: Conductor spacing should be 5 mm between the rigid PCB and bend. Also, traces should not change direction within the bend area.

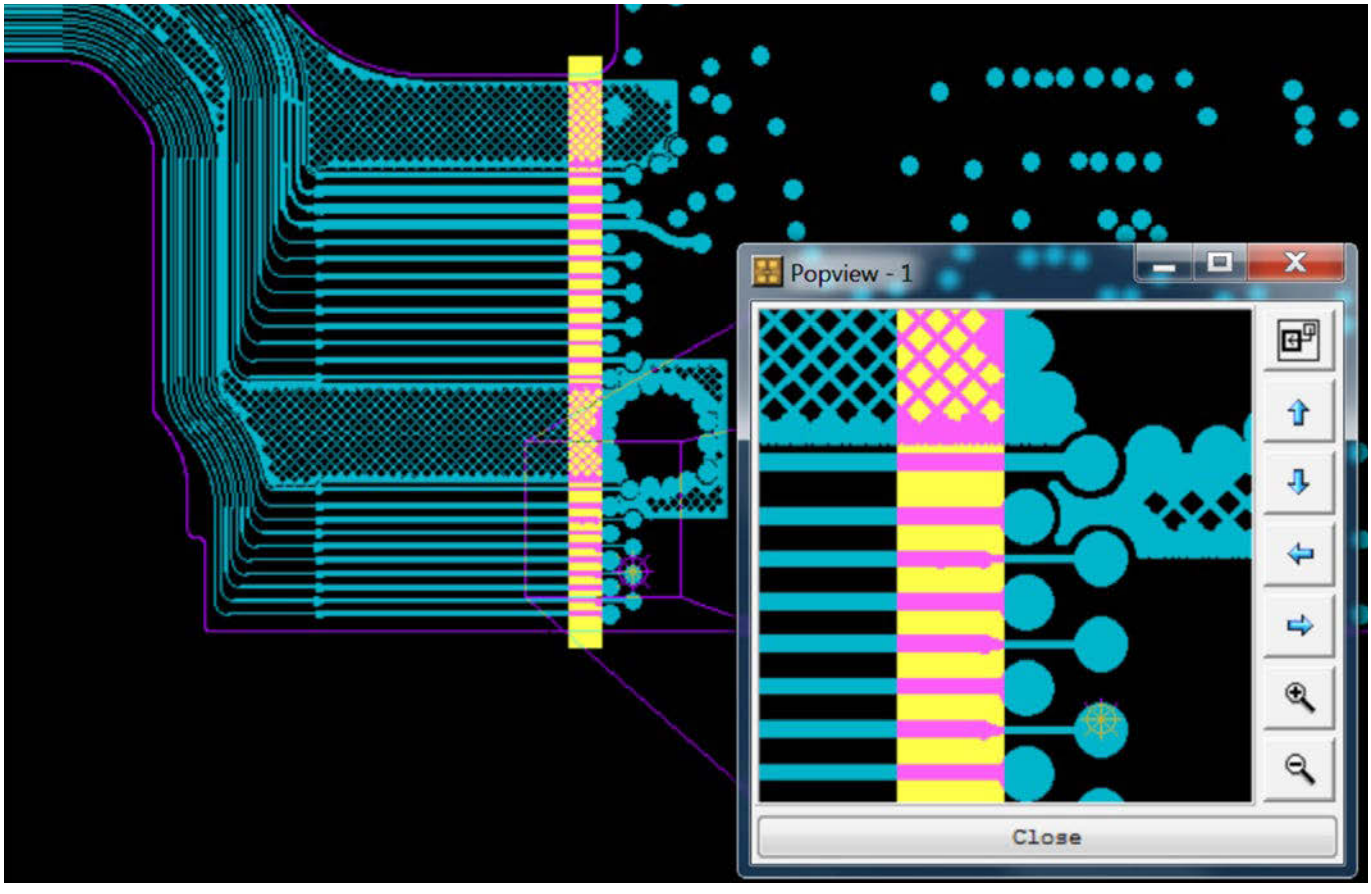


Figure 6: Interface region issues include trace width changes, via proximity and trace angle changes.

Output to the Manufacturer

Still in use by many companies today for submitting designs to the manufacturers are Gerber files, or "Gerbers." The Gerber file only describes part of the process. It must be augmented with a combination of data files and drawings with notes. Often, these files and drawings are generated with different software applications and frequently contain conflicting information. Transmitting clear and understandable product models to the manufacturer enhances the likelihood that the product will be produced correctly on the first pass. In reality, about 80% of the flex documentation packages received by manufacturers require some sort of clarification—more wasted time that could have been avoided.

Rather than providing manufacturers with multiple files of different formats along with their ancillary files, all the information manufacturing needs can be communicated with a single data format. Not only does that significantly increase efficiency, it also reduces the potential for errors, misunderstandings, and callbacks. The most widely used intelligent and comprehensive data format is ODB++. With ODB++, you can compile a manufacturing product model that mitigates the need for drawings, documentation, and other files necessary when using Gerbers, as illustrated in Figure 7. All the relevant information to manufacture your board is contained in data fields to be used by the fabricator and assembler.

Flex and rigid-flex circuit designs are increasingly popular with the miniaturization of technology and the explosion of wearable devices. PCBs involving flexible circuitry require unique manufacturing processes. To ensure and optimize flex and rigid-flex circuits for manufacturing, a dedicated DFM solution combined with an intelligent, comprehensive manufacturing product model are the best

Product Summary section -> Attribute	Value	Units
◢ Board Requirements		
Board Thickness	0.089200	Inch
Additional Requirements		
Board Outline Tolerance Plus	5.000000	Mil
Board Outline Tolerance Minus	5.000000	Mil
Board Thickness Tol Plus	3.000000	Mil
Board Thickness Tol Minus	3.000000	Mil
Board Thickness Type	over mask on plated copper	
Bottom Legend Color	white	
Bottom Soldermask Color	yellow	
Flammability Rating Standard	UL94V-0	
General PCB Standard	IPC 6012A	
Glass Transition Temperature (Tg)	110.000000	
Legend Sides	Both	
PCB Acceptability Standard	IPC 6012A	
Peelable Mask Side	none	
Plated Edge	Yes	
Plated Slots	No	
Qualification and Performance Standard		
Soldermask Sides	Both	
Top Legend Color	white	
Top Soldermask Color	green	

Figure 7: With the ODB++ Manufacturing Product Model, all neccessary instructions and data models are sent in one intelligent package.

combination of tools to improve the yield, cost, and reliability for flexible circuit designs.

Valor NPI is such a solution that brings the manufacturing knowledge into the application to provide the expertise to run DFM on PCB designs. Flex circuitry is more difficult to manufacture than rigid PCBs, and the best way to ensure that designs are manufacturable is to run DFM checks throughout the layout process as well as running DFM on the manufacturing release process.

Applying DFM to flex and rigid-flex designs results in more time spent designing and less time fixing. Identifying and rectifying issues early on can save time, money and, best of all, redesigns because of manufacturing issues. **FLEX007**

Patrick McGoff is market development manager for the Valor Division of Siemens Digital Industries Software.

Flex007 Highlights

Printed Circuits Completes Qualification of Notion Direct Soldermask, Nomenclature Printer ▸

Rigid flex circuit board manufacturer Printed Circuits has completed the installation of the process line and qualification of their new Notion direct soldermask and nomenclature printer.

Flexible Circuits Uses Essemtec FOX2 for High-Rel Flex-Centric Electronic Interconnect Assemblies ▸

Flexible Circuits, Inc. (FCI), a leading supplier of the highest reliability and complex flex-centric electronic interconnect assemblies to the U.S. defense and aerospace markets, purchased a FOX2 multi-function combined pick-and-place and dispensing system.

Altix Receives Repeat Order for Multiple AcuReel Platinium Contact Printers ▸

Altix is delighted to have taken their partnership with Ätztechnik Herz to the next level. Ätztechnik Herz pushes the boundary of etching technology to offer cost-efficient solutions at blazing speeds thanks to its Speedline service.

South Coast Circuits Hires Amanda Burgesser as VP of Business Development ▸

South Coast Circuits, a leading provider of PCB manufacturing and ATE services, announced Amanda Burgesser has joined the company as vice president of business development. In this position, she will spearhead sales and interactions with customers on all levels to bring in new business and expand on existing relationships.

Flexible Thinking: The Importance of Asking 'Why Not?' When Inventing ▸

With such an impressive list of benefits, it might seem as though flexible circuit technology has reached its improvement limits. However, the principle of continuous improvement does not rest, and it demands that we persist in our efforts do and make things better over time.

AT&S' Growth Continues Unabated in Q1 21/22 ▸

AT&S recorded a very positive revenue development despite unfavorable negative currency effects in the first quarter of 2021/22.

Eltek Reports 2021 Second Quarter Financial Results ▸

Eltek Ltd., a global manufacturer and supplier of technologically advanced printed circuit board solutions, announced its financial results for the quarter ended June 30, 2021.

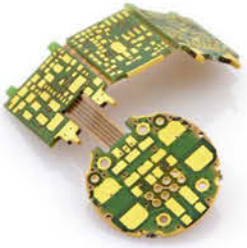
Submerged Sensors to Control Wearable Electronics ▸

Flexible and waterproof sensors that could unlock new applications for wearable electronics have been developed by scientists in Korea. Published in the journal Science and Technology of Advanced Materials, the study shows how the pressure sensor can control a phone, to take photos and play music, even when the sensor is fully immersed in water.

DuPont Reports Q2 2021 Results ▸

Net sales totaled $4.1 billion, up 26 percent versus the year-ago period as reported and up 23 percent on an organic basis.

Focused on Flex



schmoll optronics

LaserμFlex series

The versatility and precision of laser processing makes it the ideal tool for machining a wide range of materials.

Typical applications:

- Polymer cover foil
- Covered polymer laminates
- Thin, rigid-flex materials
- Inner layers and prepregs
- Separation or decap from tracks
- Structuring cavities
- Micro-via and through-hole drilling

1991
2021

BÜRKLE
NORTH AMERICA, INC.

For more information call us: +1 (336) 660-2701

burkleamerica.com

Intellectual Property—How It Works for the Benefit of All

Flexible Thinking

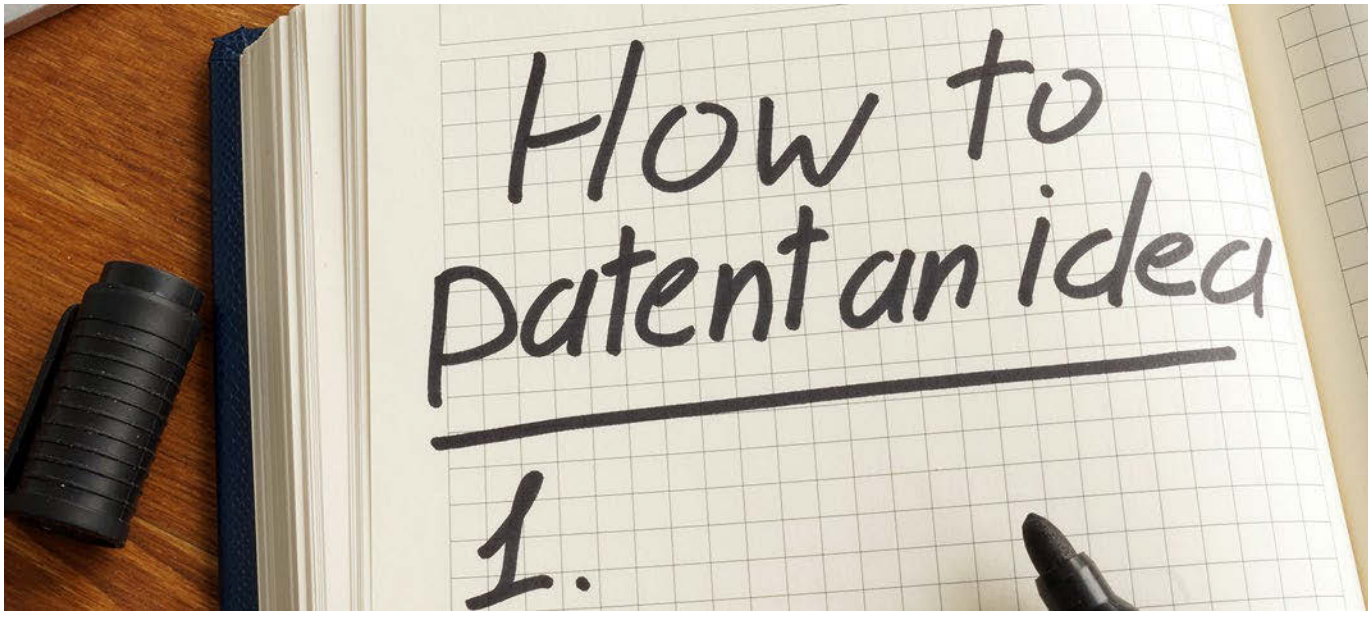
by Joe Fjelstad, VERDANT ELECTRONICS

Innovation is the lifeblood of technological progress. It has been the driving force in electronics for over a century. In general, intellectual property (IP) refers to innovations, those creations of the human mind. This includes everything from physical inventions to computer programs to trademarks and, as well, to trade secrets. But it also includes works of art and music which are commonly protected by copyrights.

If one has high interest in physical innovations related to electronics, one of the best places to explore and discover it is at the U.S. Patent and Trademark Office (USPTO) website. Using the Boolean search tools the patent office provides there, it is possible to find anything they have in their huge database of millions of patents. Because patents tend to precede actual products, the USPTO's website is a great place to get a glimpse of the wonderful things that might be lying in the road ahead. They have information not just on issued patents but also many patents which are pending in the published applications section. Following are some thoughts and observations on the important matter of intellectual property and patents.

Why Patents?

The patent system is arguably among the most important gifts that the Founding Fathers of the United States left us with. Its foundation



PREMIER GLOBAL SUPPLIER to **AUTOMOTIVE**

FLEXIBLE CIRCUITS | EMS/ASSEMBLY | CONTRACT MANUFACTURING

Industry leading Flex Design Support

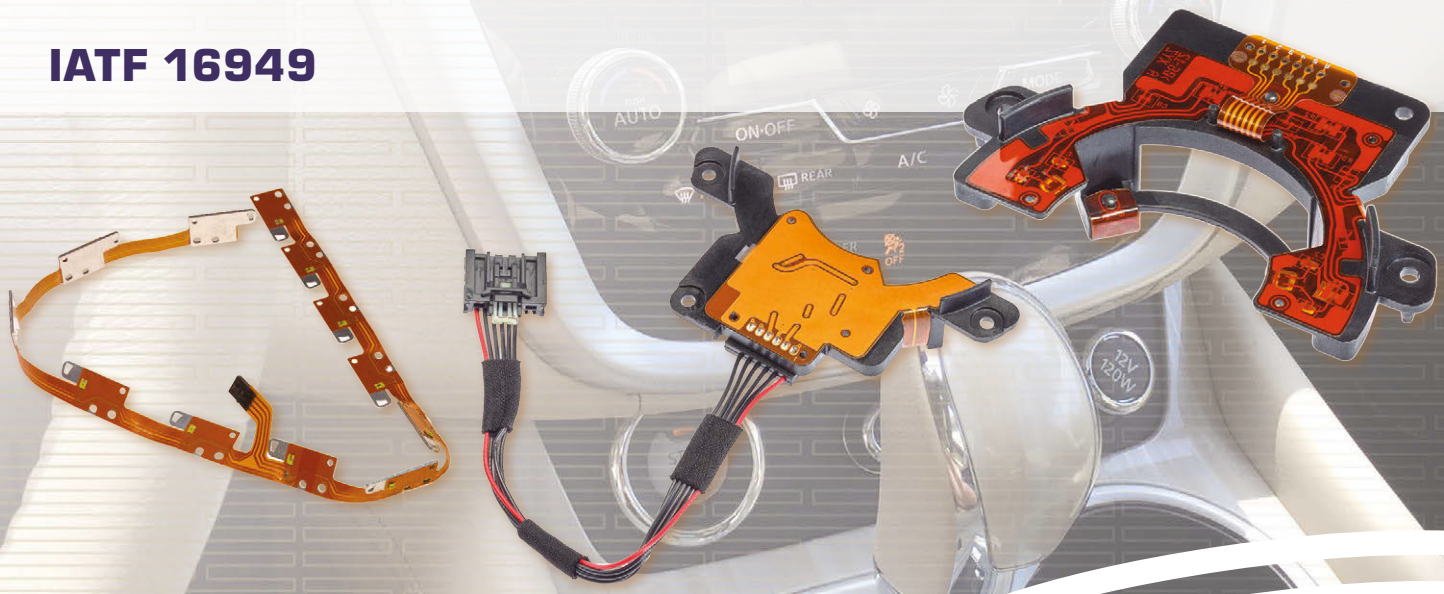
Flexible Circuits, Rigid Flex, Membrane Switches, Plastic Moldings

Specialized EMS/Assembly Services

Assemblies and Product Module Builds for Automotive

IATF 16949

EXPLORE the POSSIBILITIES!



FLEXIBLE CIRCUIT® Technologies

Flexible Circuit Technologies
9850 51st Ave. N. | Plymouth, MN 55442
www.flexiblecircuit.com | +1-763-545-3333

can be found in the U.S. Constitution, Article 1, Section 8, Clause 8, which reads in its entirety:

"The congress shall have power... To promote the progress of science and useful arts, by securing for limited times to authors and inventors the exclusive right to their respective writings and discoveries."

The idea for providing patents is one that the founders of this nation borrowed from the old world in England in the Statute of Monopolies of 1624, wherein the English Parliament endowed inventors with the sole right to their inventions for a period of 14 years. It also seems clear from the words that there was an understanding and likely even a prescient knowledge that the protection of the rights of inventors, by granting them limited monopoly over their ideas for a period of time, was key to getting them to share ideas and discoveries that would be important to the building of a new nation. There was intent to offer protection only to the writings and discoveries of authors and inventors and then only to the end of promoting science and the useful arts which would benefit the nation.

How It Benefits

This protection of intellectual property and right to practice safely one's ideas has proven to be a magnet for many who came to this country to seek out their fortunes in this new land of opportunity with government guarantees and, in the process, to make life better not only for themselves but for everyone else in society. It continues to this day.

The concept of the patent is now universally embraced at high levels on an intellectual basis, if not in actual practice, everywhere down to the level of business. Today, while a great many countries have their own patent systems, since 1970, the Patent Cooperation Treaty (PCT), an international patent law treaty to provide a unified procedure for filing patent applications to protect inventions from member countries, has been in place. A patent application filed under the PCT is referred to as international application (also known as a PCT application), however, the PCT does not lead to the granting of an international patent, as one must still file separately for a patent in each nation where protection is sought.

This is an ongoing effort to try to find cooperation among the many different countries of the world to make something more useful and enforceable.

This is an ongoing effort to try to find cooperation among the many different countries of the world to make something more useful and enforceable. However, concerns over commerce and competitiveness have caused many business leaders to turn a blind eye to patents, especially in some of the newer members of the international business community where the idea of intellectual property is not given much weight. In fact, the more cynical (practical?) folks in most industries commonly assume that a patent is of no value until it is tested in court and backed by the weight and force of governments which can restrict trade. This widely-held belief is something that must delight litigators around the globe, for they make their greatest personal gains from conflict. They are always assured a win, no matter what the outcome is for their client. As many haggard veteran inventors have observed: "If you think getting a patent is expensive, try defending one."

There is another other form of intellectual property such as trademarks and copyrights which are likely familiar to most readers. Copyright law can be traced to the English Statute of 1710, which secured to authors of books the

BENDING THE POSSIBILITIES

FLEXFINER

BY TAIYO

AVAILABE IN TWO COLORS

AMBER

BLACK

- Excellent bendability
- High Resolution by photolithography
- High Reliability
- Simplified process
- UL94 VTM-0 certified
- Low spring back force
- Resolution Line/Space=50/50μm
- Solder mask defined opening (30μm diameter)

OUR BIGGEST FLEX YET!

Contact your local sales representative today!

Visit www.taiyo-america.com for more information.

TAIYO AMERICA

UNMATCHED INNOVATION

Follow us on social media

sole right of publishing them for designated periods.

Perhaps the most common, however, is the trade secret, special knowhow, or a formula that cannot be easily gleaned. The most famous trade secrets are often associated with foods—recipes and flavored drink mixes such as Coca-Cola. The trade secret protects the individual's or business's intellectual property by not disclosing the details of the product as would be required by a patent.

In closing, there is an amusing anecdote of historical note worth sharing. Charles H. Duell was the U.S. Commissioner of Patents in the late 19th century. Mr. Duell incredulously suggested in 1899 that it might be a good idea to shut down the patent office because "everything that can be invented has been invented." The number of U.S. issued patents granted as of December 1899 was 640,000. The number of patents issued at the time of writing is 11,089,721.

Clearly, Mr. Duell was just a wee bit off. **FLEX007**

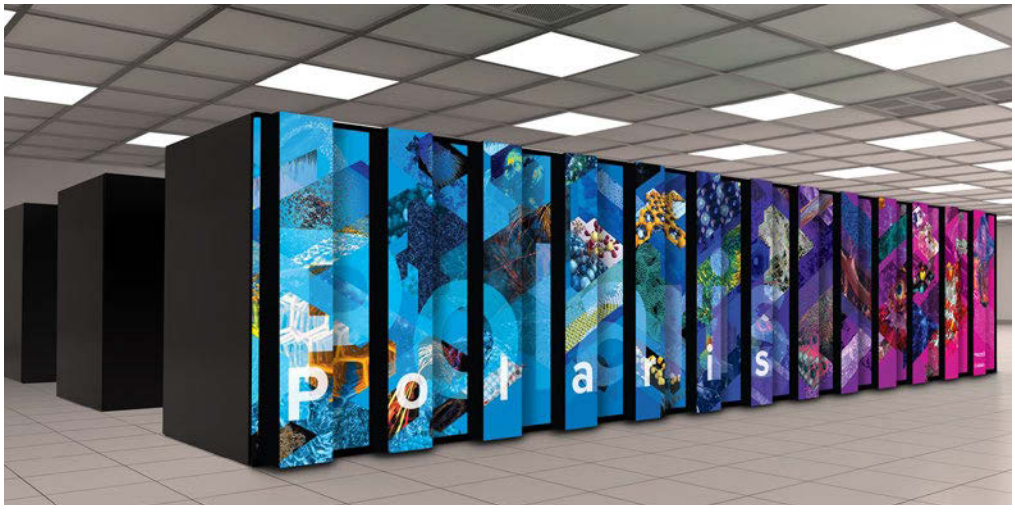
Joe Fjelstad is founder and CEO of Verdant Electronics and an international authority and innovator in the field of electronic interconnection and packaging technologies with more than 185 patents issued or pending. To read past columns or contact Fjelstad, click here. Download your free copy of Fjelstad's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

Argonne National Lab and Hewlett Packard Enterprise Prepare for Exascale Era

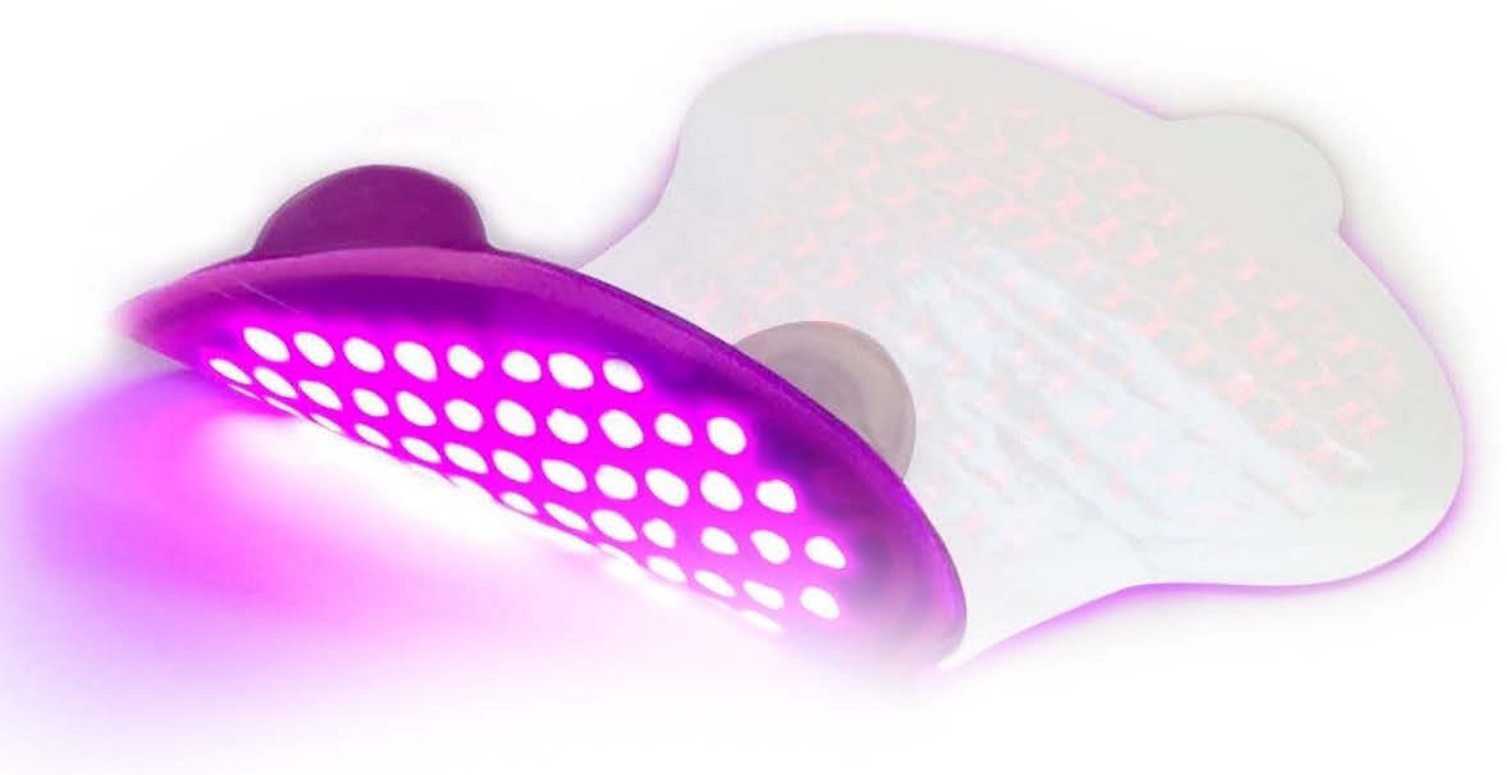
The U.S. Department of Energy's Argonne National Laboratory and Hewlett Packard Enterprise unveiled a new testbed supercomputer to prepare critical workloads for future exascale systems that will deliver up to four times faster performance than Argonne's current supercomputers.

The new system, which Argonne has named Polaris, will be built by HPE, and hosted and managed by the Argonne Leadership Computing Facility (ALCF), a U.S. DOE Office of Science User Facility. Polaris is designed with industry-leading high-performance computing (HPC) and AI solutions to advance investigations into society's most complex and pressing issues, from understanding the biology of viruses to revealing the secrets of the universe. It will also augment Argonne's ongoing efforts and achievements in areas such as clean energy, climate resilience and manufacturing.

"Polaris is well equipped to help move the ALCF into the exascale era of computational science by accelerating the application of AI capabilities to the growing data and simulation demands of our users," said Michael E. Papka, director at the ALCF. "Beyond getting us ready for Aurora, Polaris will further provide a platform to experiment with the integration of supercomputers and large-scale experiment facilities, like the Advanced Photon Source, making HPC available to more scientific communities." (Source: Argonne National Laboratory)



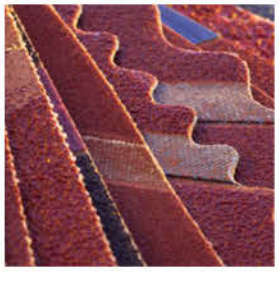
WE WERE PRINTING ELECTRONICS LONG BEFORE 'PRINTED ELECTRONICS' WAS A THING.



From simple switches to complex, functional, multi-layer prints with printed components, we have the capability to meet your needs and can enable your ideas moving into the future. Request a consult today!

Book a consult






Sheldahl®

A Flex Company

www.sheldahl.com

TOP TEN

EDITOR'S PICKS

David Duross

Fresh PCB Concepts: Does the Assembly Process Damage a PCB? (Part 2—Time to Delamination)

In Part 1 of this series, published back in April, I commented upon the effect the assembly process has upon a printed circuit board. There is another gauge that can be used to help a designer or contract manufacturer understand this point and that is the time to delamination test. These are referred to as either the T260 or T288 tests.

DFM 101: PCB Controlled Impedance

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

Dana on Data: The Critical Importance of the Fab Product Engineer

Dana Korf

Billions of dollars are spent yearly on CAD and CAM software to produce complex PCB designs and fabricate PCBs. The final technical manufacturing decisions generally are made by one person for each design. This is the PCB fabricator product engineer. But I don't think most design, procurement, or NPI teams understand how critical this person is to the data transfer success and liability protection.

The Digital Layout: Say What You Do—Do What You Say

In this month's column, Kelly Dack provides answers to some common questions about joining an electronics industry organization. PCEA Chairman Steph Chavez conveys his excitement and anticipation of events to come and how the PCEA will be involved.

Connect the Dots: The Split Planes Challenge

Losing track of voltage in your PCB design can lead to explosive problems. Your CAM tool will not manage split planes for you.

Altium, IPC to Co-Locate AltiumLive and IPC APEX EXPO

AltiumLive, Altium's annual printed circuit board (PCB) conference for the electronics design community, is co-locating with the IPC APEX EXPO, the premier event for the electronics manufacturing industry being held at the San Diego Convention Center January 22-27, 2022.

Lightning Speed Laminates: The Benefits of Understanding Engineering Disciplines for PCB Design

When a PCB designer understands the different aspects of PCB technology, they will certainly be able to deliver circuit designs that will be more robust and have better manufacturing yields. The high-volume manufacturing yields of a PCB are directly related to profit, and that is the connection between good circuit design and a successful circuit-product.

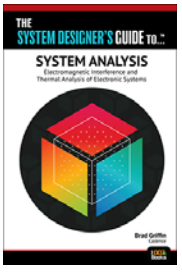
Elementary, Mr. Watson: Epic Fails with Design Rules

Various sciences, including physics, mathematics, and chemistry, are significantly involved throughout the PCB design process, rules that can sometimes be bent but not broken. However, the rules that designers break and ignore altogether and very often are the design rules.

Whizz Systems: The Silicon Valley CM

The I-Connect007 editorial team spoke with Whizz Systems in the months prior to COVID. As we come out of lockdown, we decided to check in with them again. In this part of the interview, Muhammad Irfan and Dan Williams discuss Whizz Systems' unique methods for navigating the many handoffs involved in the design and manufacturing process. They also share their thoughts on the trends they've seen as a CM working out of Silicon Valley.

Excerpt: *The System Designer's Guide to... System Analysis,* Chapter 1



System designers for 5G, automotive, high-performance computing (HPC), IoT, and other advanced applications have been facing growing challenges in EM interference and thermal issues. These are prevalent in all electronic devices. Data centers play a key role in this high-performance computing (HPC) era, and EMI/thermal issues have a huge impact on the performance of data centers. This book explains scenarios and issues based on the context of data center electronic systems.

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

Career Opportunities

Is your team growing?

Find industry-experienced candidates at I-Connect007.

For just $750, your 200-word, full-column ad will appear in the Career Opportunities section of all three of our monthly magazines, reaching circuit board designers, fabricators, assemblers, OEMs, suppliers and the academic community.

In addition, your ad will:

- be featured in at least one of our newsletters
- appear on our jobConnect007.com board, which is promoted in every newsletter
- appear in our monthly Careers Guide, emailed to 26,000 potential candidates

Potential candidates can click on your ad and submit a resume directly to the email address you provide, or be directed to the URL of your choice.

No contract required. Just send over your copy and company logo and we'll do the rest!

Contact barb@iconnect007.com to get your ad posted today!

+1 916.365.1727

I-Connect007
GOOD FOR THE INDUSTRY

smt007 pcb007 pcb design007

Career Opportunities

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

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

Laminator Technician

Nature of Duties/Responsibilities

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

Education/Experience

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

Wet Process/Plating Technician

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

Purpose

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

Nature of Duties/Responsibilities

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

Education and Experience

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

Production Scheduler

Main Responsibilities

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

Education and Experience

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

apply now

Career Opportunities

Fuji America Corporation is a rapidly growing electronics assembly equipment distributor. We support the factories of the future and smart factories globally. We offer an exciting and challenging career for a software support engineer and an applications engineer who want to join our growing company.

Software Support Engineer

As a software support engineer for Fuji America Corporation, you will be a customer-facing technical advisor with the opportunity to solve technically complex problems for our proprietary software. As a trusted advisor to our customers, you will have influence over a broad range of solutions that create business value. As a valued member on our team, the software support engineer will use advanced troubleshooting methods and tools to solve technically complex problems. These highly complex, escalated problems require broad and in-depth product knowledge, as well as exceptional troubleshooting skills.

- Field installation of proprietary software/automation equipment throughout North America
- Field troubleshoot, repair, training, and process support of proprietary software
- Provide remote and on-site technical support
- Troubleshoot Windows 10/Windows server installing, configuration, and support
- Networking experience—setting up and supporting networks.
- Exposure and/or experience with Oracle or Microsoft SQL server databases
- Strong verbal communication skills with both customer and other technical depts.
- Flexibility to travel and perform job assignments on short notice
- Strong aptitude with current computing applications and networking processes

Experience

- Bachelor of Science in computer science or related field preferred

Applications Engineer

As an applications engineer, you will be responsible for doing cycle time and studies in preparation to make recommendations of Fuji products for customers' applications. Support implementation of activities within the technical center such as customer visits, demonstrations, evaluations, testing, inspection of Fuji products, including peripheral equipment from other vendors.

- Assist sales representatives in technical aspects relating to machine and software functions and utilization.
- Assist sales representatives and customers with providing CTA (Cycle Time Analysis) to them for recommending Fuji products to customers' specific applications. This includes the sFAB machine as well as all other SMT machines.
- Schedule and perform product demonstrations on all available types of equipment and software to potential and existing customers.
- Test and evaluate existing as well as new technologies on equipment and software performance and reliability.
- Assist in the coordination of any new FAC projects by utilizing your full potential.
- Responsible for the setup of the equipment and its demonstration for various trade shows.
- Assist FAC staff in any technical issues which may require attention.
- Assist in the coordination of design and manufacture of customs tooling for placement equipment.
- Perform inventory checks every six months according to the schedule and manner regulated by the company, if applicable.

Experience

- Minimum five years programming/computer experience
- Bachelor's degree preferred

apply now

Career Opportunities

Application Engineer (m/f/d)—Fulltime, Germany

Our company is expanding its product portfolio into custom made products. This creates the need for an Application Engineer to provide technical support to our existing sales team and customer base.

Responsibilities:

- Analysis of incoming technical data and handling of engineering questions
- Technical consultation of customers (incl. new customer specifications and discuss with relevant technical and quality teams worldwide)
- Support and consultation for new projects
- Lead and/or participate in local, cross-location/global cross-department projects of various scale
- Develop and provide function-related trainings to existing and new staff in order to transfer and optimize know-how
- Provide technical solutions

Skills:

- Technical expertise in battery power solutions and technologies for Rechargeable and Primary cells and Battery Packs
- Mechanical background or knowledge to be able to discuss and manage other products, like custom made connectors, cable assemblies and keypad touch panels.
- Written and spoken English and German, any other European language a plus.
- Highly technical with a commercial flare.
- Self-motivated, ambitious, and eager to grow in a dynamic organization.

Interested? We are looking forward to your application!

Please send your application to **hr@cmit.support**. For any inquiries, please contact Mrs. Amélie Filler. For more information visit www.cml-globalsolutions.com

apply now

Sales Manager (m/f/d)—Worldwide Locations

CML Group is a leading provider of Printed Circuit Boards. We develop tailor-made sourcing concepts for our customers worldwide creating strong partnerships and reliable connections.

For the expansion of our target markets, we need you to generate new business, drive new projects from RFQ stage and manage the customer relationship.

Your Profile:

- Profound sales and technical expertise in printed circuit board industry
- Local market knowledge and ideally a customer base of contacts in one or more of the listed countries
- Have successful track records in developing new business opportunities
- Excellent command in spoken and written English and one additional local language
- Highly self-motivated, ambitious, eager to grow in a dynamic organization
- Able to work independently and have good communication skills and leadership skills
- Self-employed/contractor/commission-based agent also welcome

Your Target Markets:

- Europe: Spain, France, Germany, Netherlands, UK, Denmark, Sweden, Norway
- USA: New Jersey, Florida, Georgia, Michigan, San Jose, Bay area, Pacific Northwest and Canada
- Others: Singapore, Thailand, Malaysia, Australia, Brazil, Turkey, Russia, and South Africa

Interested? We are looking forward to your application!

Please send your application to **hr-china@cml-eurasia.hk**. For any inquiries, please contact Ms. Grace Feng. For more information visit www.cml-globalsolutions.com

apply now

Career Opportunities

PCB Field Engineer– North America Operations

ICAPE Group is a European leader for printed circuits boards and custom-made electro-mechanical parts. Headquartered in Paris, France, we have over 500 employees located in more than 70 countries serving our +2500 customers.

To support our growth in the American market, we are looking for a PCB Field Engineer.

You will work in our North America technical center, including our U.S. technical laboratory, and will be responsible for providing technical and quality support to our American sales team.

You will have direct customer contact during all phases of the sales process and provide follow-on support as required.

RESPONSIBILITIES INCLUDE

- Feasibility recommendations
- Fabricator questions and liaison
- Quality resolutions
- Technical explanation (for the customer) of proposals, laboratory analysis or technology challenges

REQUIREMENTS

- Engineering degree or equivalent industry experience
- 5 years' experience with PCB manufacturing (including CAM)
- Excellent technical understanding of PCBs
- Experience with quality tools (FAI, PPAP and 8-D)
- Good communication skills (written and oral)

Communication skills are essential to assist the customer with navigation of the complex process of matching the PCB to the application.

SALARY

Competitive, based on profile and experience. Position is full time in Indianapolis, Ind.

apply now

Prototron Circuits

Sales Representatives

Prototron Circuits, a market-leading, quick-turn PCB shop, is looking for sales representatives for all territories.

Reasons you should work with Prototron:

- Serving the PCB industry for over 30 years
- Solid reputation for on-time delivery (99% on-time)
- Excellent quality
- Production quality quick-turn services in as little as 24 hours
- AS9100
- MIL-PRF- 31032
- ITAR
- Global sourcing
- Engineering consultation
- Completely customer focused team

Interested? Let's have a talk.
Call Dan Beaulieu at
207-649-0879
or email to
danbbeaulieu@aol.com

apply now

Career Opportunities

The Test Connection, Inc.

Rewarding Careers

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

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

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

Associate Electronics Technician/ Engineer (ATE-MD)

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

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

apply now

Test Engineer (TE-MD)

In this role, you will specialize in the development of in-circuit test (ICT) sets for Keysight 3070 (formerly HP) and/or Teradyne (formerly GenRad) TestStation/228X test systems.

- Candidates must have at least three years of experience with in-circuit test equipment. A candidate would develop and debug our test systems and install in-circuit test sets remotely online or at customer's manufacturing locations nationwide.
- Candidates would also help support production testing and implement Engineering Change Orders and program enhancements, library model generation, perform testing and failure analysis of assembled boards, and other related tasks.
- Some travel required and these positions are available in the Hunt Valley, Md., office.

apply now

Sr. Test Engineer (STE-MD)

- Candidate would specialize in the development of in-circuit test (ICT) sets for Keysight 3070 (formerly Agilent & HP), Teradyne/ GenRad, and Flying Probe test systems.
- Strong candidates will have more than five years of experience with in-circuit test equipment. Some experience with flying probe test equipment is preferred. A candidate would develop, and debug on our test systems and install in-circuit test sets remotely online or at customer's manufacturing locations nationwide.
- Proficient working knowledge of Flash/ISP programming, MAC Address and Boundary Scan required. The candidate would also help support production testing implementing Engineering Change Orders and program enhancements, library model generation, perform testing and failure analysis of assembled boards, and other related tasks. An understanding of stand-alone boundary scan and flying probe desired.
- Some travel required. Positions are available in the Hunt Valley, Md., office.

Contact us today to learn about the rewarding careers we are offering. Please email resumes with a short message describing your relevant experience and any questions to careers@ttci.com. Please, no phone calls.

We proudly serve customers nationwide and around the world.

TTCI is an ITAR registered and JCP DD2345 certified company that is NIST 800-171 compliant.

apply now

Career Opportunities

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

KOH YOUNG TECHNOLOGY

Logistics Assistant

Koh Young America is looking for a Logistics Assistant to assist and oversee our supply chain operations. Working alongside a Logistics Specialist, you will coordinate processes to ensure smooth operations using a variety of channels to maximize efficiency. You must be an excellent communicator and negotiator well-versed in supply chain management principles and practices. Also, you should be meticulous with a focus on customer satisfaction. These attributes are ideally complemented by a Bachelor's in Supply Chain Management or equivalent professional experience in the manufacturing industry.

This position is in our Duluth, Georgia, headquarters, where we serve our customers within North and South America. We offer health, dental, vision, and life Insurance with no employee premiums, including dependent coverage. Additionally, we provide a 401K retirement plan with company matching, plus a generous PTO policy with paid holidays.

Koh Young Technology, founded in 2002 in Seoul, South Korea, is the world leader in 3D measurement and inspection technology used in the production of microelectronics assemblies. Using patented 3D technology, Koh Young provides best-in-class products in Solder Paste Inspection (SPI) and Automated Optical Inspection (AOI) for electronics manufacturers worldwide.

apply now

Career Opportunities

EAGLE electronics Inc.
MANUFACTURERS OF QUALITY PRINTED CIRCUIT BOARDS

Maintenance Technician

Inspects work-related conditions to determine compliance with prescribed operating and safety standards. Operates power-driven machinery and uses equipment and tools commonly used to maintain facilities and equipment. Replace filters, belts, and additional parts for repairs and preventive maintenance. Moves objects weighing up to 150 lbs. using a hand truck or pulley. Cleans work area and equipment. Works with cleaning fluids, agents, chemicals, and paints using protective gear. Works at elevations greater than ten feet, climbing ladders, while repairing or maintaining building structures and equipment. Assists skilled maintenance technicians/workers in more complex tasks and possible after-hours emergency repairs. Must meet scheduling and attendance requirements.

apply now

Plating Operator

Plating operator for printed circuit boards. No experience necessary, will train. Must be able to work with chemicals, lift up to 50 pounds, and have good math skills. Minimum high school/GED or equivalent. All shifts (1st, 2nd, 3rd), 8 hours per day minimum, Monday thru Friday. Saturday and Sunday work is common allowing for steady overtime pay.

apply now

EAGLE electronics Inc.
MANUFACTURERS OF QUALITY PRINTED CIRCUIT BOARDS

Water Treatment Operator

Responsible for operating waste treatment plant, our operation that converts wastewater in drains and sewers into a form that's metal free to release into the environment.

Control equipment and monitor processes that remove metals from wastewater. Run tests to make sure that the processes are working correctly. Keep records of water quality and pH. Operate and maintain the pumps and motors that move water and wastewater through filtration systems. Read meters and gauges to make sure plant equipment is working properly. Take samples and run tests to determine the quality of the water being produced. Adjust the amount of chemicals being added to the water and keep records that document compliance.

apply now

Drilling Operator

Drilling operator for printed circuit boards. Minimum 2 years of experience. Minimum high school/GED or equivalent.

All Shifts (1st, 2nd, 3rd), 8 hours per day minimum, Monday thru Friday. Saturday and Sunday work is common allowing for overtime pay.

apply now

Career Opportunities

ventec
INTERNATIONAL GROUP
騰輝電子

Technical Support/ Sales Engineer, UK

We are looking to expand our UK technical & sales support team. As a technical support/sales engineer (home office/Leamington Spa) you will assist potential and current customers in appreciating the benefits of using--and optimizing the use of--Ventec materials in their printed circuit board manufacturing processes, and so enhance customer loyalty and satisfaction, spread the use of Ventec materials, and grow sales. You will provide a two-way channel of technical communication between Ventec's production facilities and UK/European customers.

Skills and abilities required for the role

- HNC, HND, degree or equivalent in a technical/scientific discipline
- Sales experience/negotiating skills
- Printed circuit board industry experience an advantage
- Good written & verbal communications skills
- Ability to work in an organized, proactive and enthusiastic way
- Ability to work well both in a team and independently
- Good user knowledge of common Microsoft Office programs
- Full driving license essential

What's on Offer

- Excellent salary and benefits commensurate with experience

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

Please forward your resume to
anthony.jackson@ventec-europe.com

apply now

SIEMENS

Marketing Coordinator/Writing Strategist: Embedded Software

Location: Portland, Oregon or USA (remote)
Job Number: 242982

Seeking a technology communications change maker! Siemens Digital Industries Software is looking for a content creator for its embedded software group. The ideal candidate for the Brand Marketing coordinator/writing strategist position will work closely with engineers and managers to write, edit and produce compelling technology marketing content (magazine articles, blogs, technology papers, multi-media, customer success stories and promotional materials). Do you possess creative energy and enjoy storytelling with an energetic team?

Requirements:

- Strong writing and editing skills
- Education and/or experience in technology, science, journalism and/or English
- A technical background or experience (such as a BS or an associate's degree in engineering or computer science) is preferred
- 1-3 years of experience in writing about technology solutions
- Basic knowledge of online publications, digital platforms and social media is useful to meet project specifications in a fast-paced environment
- Ability to research and collect data, repurpose existing materials, collaborate with subject matter experts, and translate technical information into compelling marketing communications content that engage audiences

Creative materials will be used globally, in a high-energy environment, supporting the world's leading industrial software company.

apply now

Career Opportunities

MivaTek Global

Product Manager

MivaTek Global is preparing for a major market and product offering expansion. Miva's new NG3 and DART technologies have been released to expand the capabilities of Miva's industry-leading LED DMD direct write systems in PCB and Microelectronics. MivaTek Global is looking for a technology leader that can be involved guiding this major development.

The product manager role will serve as liaison between the external market and the internal design team. Leadership level involvement in the direction of new and existing products will require a diverse skill set. Key role functions include:

- **Sales Support:** Recommend customer solutions through adaptions to Miva products
- **Design:** Be the voice of the customer for new product development
- **Quality:** Verify and standardize product performance testing and implementation
- **Training:** Conduct virtual and on-site training
- **Travel:** Product testing at customer and factory locations

Use your 8 plus years of experience in either the PCB or Microelectronic industry to make a difference with the leader in LED DMD direct imaging technology. Direct imaging, CAM, AOI, or drilling experience is a plus but not required.

For consideration, send your resume to N.Hogan@MivaTek.Global. For more information on the company see www.MivaTek.Global or www.Mivatec.com.

apply now

MivaTek Global

Field Service Technician

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

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

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

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

More About Us

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

apply now

Career Opportunities

Sheldahl®

A Flex Company

Sheldahl, a leading provider of flexible interconnect products and electronic materials, is seeking candidates to join their diverse and skilled team.

We are looking for people who demonstrate:

- Intense collaboration
- Passionate customer focus
- Thoughtful, fast, disciplined execution
- Tenacious commitment to continuous improvement
- Relentless drive to win

Positions in America include:

Project Manager – Northfield, MN

Candidate will provide timely cost estimation and project budget definition, be responsible for maintaining customer relations, participate in meetings, etc.

apply now

Program Manager – Specialty Films

Candidate will work with our Specialty Films in the Aerospace, Medical, and Commercial Aviation markets providing timely cost estimation and project budget definition, maintaining customer relations, participate in meetings, etc.

apply now

Program Manager

We are looking for a candidate with a passion for customer service and a commitment to continuous improvement.

Responsibilities:

- Provide timely cost estimation and project budget definition; recommend pricing and estimate lead time.
- Maintain excellent relations with both new and existing customers.
- Review new applications and provide technical support.
- Keep apprised of relevant applications, quality and regulatory standards.
- Participate in contract review and price negotiations.
- Ongoing margin analysis; identify potential necessary price adjustment opportunities and cost reduction projects.
- Participate in the creation and maintenance of technical documentation.
- Manage the coordination of product life cycle activities with team including account management, customer service, purchasing, operations and quality on customer matters.

Requirements:

- Effective technical communicator
- Four-year Engineering degree or equivalent work experience
- PMP preferred
- 7-10 years Product Engineering or Product Management experience
- Well versed in Advanced Technical Materials (Aerospace and Defense preferred)
- Self-starter with trouble shooting/problem solving skills
- Computer savvy, quick learner
- Open to travel

Preferred Experience:

- Project management and planning, ERP systems, CRM Software, spreadsheets
- Experience with cost and project modeling

Benefits:

- Full range of medical benefits
- Life Insurance
- Matching 401K
- PTO
- Tuition reimbursement
- Employee discounts at local retailers

apply now

Career Opportunities

SIEMENS

Siemens EDA
Sr. Applications Engineer

Support consultative sales efforts at world's leading semiconductor and electronic equipment manufacturers. You will be responsible for securing EM Analysis & Simulation technical wins with the industry-leading HyperLynx Analysis product family as part of the Xpedition Enterprise design flow.

Will deliver technical presentations, conduct product demonstrations and benchmarks, and participate in the development of account sales strategies leading to market share gains.

- PCB design competency required
- BEE, MSEE preferred
- Prior experience with Signal Integrity, Power Integrity, EM & SPICE circuit analysis tools
- Experience with HyperLynx, Ansys, Keysight and/or Sigrity
- A minimum of 5 years' hands-on experience with EM Analysis & Simulation, printed circuit board design, engineering technology or similar field
- Moderate domestic travel required
- Possess passion to learn and perform at the cutting edge of technology
- Desire to broaden exposure to the business aspects of the technical design world
- Possess a demonstrated ability to build strong rapport and credibility with customer organizations while maintaining an internal network of contacts
- Enjoy contributing to the success of a phenomenal team

***Qualified applicants will not require employer-sponsored work authorization now or in the future for employment in the United States. Qualified Applicants must be legally authorized for employment in the United States.*

apply now

U.S. CIRCUIT

Plating Supervisor

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

Competitive benefits package. Pay will be commensurate with experience.

Mail to:
mfariba@uscircuit.com

apply now

Career Opportunities

BLACKFOX
Premier Training & Certification

IPC Instructor

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

Independent contractor, possible full-time employment

Job Description

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

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

Qualifications

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

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

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

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

apply now

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, penalization 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

Career Opportunities

Now Hiring

Director of Process Engineering

A successful and growing printed circuit board manufacturer in Orange County, CA, has an opening for a director of process engineering.

Job Summary:

The director of process engineering leads all engineering activities to produce quality products and meet cost objectives. Responsible for the overall management, direction, and coordination of the engineering processes within the plant.

Duties and Responsibilities:

- Ensures that process engineering meets the business needs of the company as they relate to capabilities, processes, technologies, and capacity.
- Stays current with related manufacturing trends. Develops and enforces a culture of strong engineering discipline, including robust process definition, testing prior to production implementation, change management processes, clear manufacturing instructions, statistical process monitoring and control, proactive error proofing, etc.
- Provides guidance to process engineers in the development of process control plans and the application of advanced quality tools.
- Ensures metrics are in place to monitor performance against the goals and takes appropriate corrective actions as required. Ensures that structured problem-solving techniques are used and that adequate validation is performed for any issues being address or changes being made. Develops and validates new processes prior to incorporating them into the manufacturing operations.
- Strong communication skills to establish priorities, work schedules, allocate resources, complete required information to customers, support quality system, enforce company policies and procedures, and utilize resources to provide the greatest efficiency to meet production objectives.

Education and Experience:

- Master's degree in chemical engineering or engineering is preferred.
- 10+ years process engineering experience in an electronics manufacturing environment, including 5 years in the PCB or similar manufacturing environment.
- 7+ years of process engineering management experience, including 5 years of experience with direct responsibility for meeting production throughput and quality goals.

apply now

Now Hiring

Process Engineering Manager

A successful and growing printed circuit board manufacturer in Orange County, CA, has an opening for a process engineering manager.

Job Summary:

The process engineering manager coordinates all engineering activities to produce quality products and meet cost objectives. Responsible for the overall management, direction, and coordination of the engineering team and leading this team to meet product requirements in support of the production plan.

Duties and Responsibilities:

- Ensures that process engineering meets the business needs of the company as they relate to capabilities, processes, technologies, and capacity.
- Stays current with related manufacturing trends. Develops and enforces a culture of strong engineering discipline, including robust process definition, testing prior to production implementation, change management processes, clear manufacturing instructions, statistical process monitoring and control, proactive error proofing, etc.
- Ensures metrics are in place to monitor performance against the goals and takes appropriate corrective actions as required. Ensures that structured problem-solving techniques are used and that adequate validation is performed for any issues being address or changes being made. Develops and validates new processes prior to incorporating into the manufacturing operations

Education and Experience:

- Bachelor's degree in chemical engineering or engineering is preferred.
- 7+ years process engineering experience in an electronics manufacturing environment, including 3 years in the PCB or similar manufacturing environment.
- 5+ years of process engineering management experience, including 3 years of experience with direct responsibility for meeting production throughput and quality goals.

apply now

Career Opportunities

INSULECTRO

Are You Our Next Superstar?!

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

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

View our opportunities at Insulectro Careers (jobvite.com)

apply now

Innovative Circuits, Inc.
Real Quality Is Innovative!

Multiple Positions

Innovative Circuits, a quick-turn, high mix, low-volume PCB manufacturer located in Alpharetta, Georgia, is growing and looking for talented individuals to join the team.

Front End Engineering Manager
Oversee CAM, programming/production engineering and quoting departments. Ideal candidates will have 15 years' experience working in a printed circuit board front-end department with flex and rigid flex circuit board construction.

Process Engineer
Responsible for the implementation and maintenance of chemical and/or mechanical processes used to produce flex circuits, rigid flex and rigid printed circuit boards.

Third Shift Production Manager
Oversee third shift productions workers, product schedule and reporting.

Wet Lab Tech
Perform all lab analysis using burettes, pipettes, pH/ion meters, atomic absorption spectrophotometer, laboratory balance, hydrometers, hull cells, CVS, and all other lab-related equipment.

CAM Operator
Inspect, modify, and contribute to the initial development of producing flex circuits, rigid flex and rigid printed circuit boards based upon customer requirements and data files.

Quality Inspector
Responsible for verifying that the product meets customer requirements prior to shipping.

Wastewater Technician
Operate, monitor, maintain and troubleshoot the wastewater treatment facility and its processes.

Production Worker
Machine operator and light chemistry in a PCB manufacturing environment.

Please visit the link below to view our opportunities and apply.

apply now

Career Opportunities

Manncorp™

SMT Operator
Hatboro, PA

Manncorp, a leader in the electronics assembly industry, is looking for a **surface-mount technology (SMT) operator** to join their growing team in Hatboro, PA!

The **SMT operator** will be part of a collaborative team and operate the latest Manncorp equipment in our brand-new demonstration center.

Duties and Responsibilities:

- Set up and operate automated SMT assembly equipment
- Prepare component kits for manufacturing
- Perform visual inspection of SMT assembly
- Participate in directing the expansion and further development of our SMT capabilities
- Some mechanical assembly of lighting fixtures
- Assist Manncorp sales with customer demos

Requirements and Qualifications:

- Prior experience with SMT equipment or equivalent technical degree preferred; will consider recent graduates or those new to the industry
- Windows computer knowledge required
- Strong mechanical and electrical troubleshooting skills
- Experience programming machinery or demonstrated willingness to learn
- Positive self-starter attitude with a good work ethic
- Ability to work with minimal supervision
- Ability to lift up to 50 lbs. repetitively

We Offer:

- Competitive pay
- Medical and dental insurance
- Retirement fund matching
- Continued training as the industry develops

apply now

Manncorp™

SMT Field Technician
Hatboro, PA

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

Duties and Responsibilities:

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

Requirements and Qualifications:

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

We Offer:

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

apply now

Career Opportunities

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

MANUFACTURERS OF QUALITY PRINTED CIRCUIT BOARDS

Pre-CAM Engineer

Illinois-based PCB fabricator Eagle Electronics is seeking a pre-CAM engineer specific to the printed circuit board manufacturing industry. The pre-CAM Engineer will facilitate creation of the job shop travelers used in the manufacturing process. Candidate will have a minimum of two years of pre-CAM experience and have a minimum education level of an associate degree. This is a first-shift position at our Schaumburg, Illinois, facility. This is not a remote or offsite position.

If interested, please submit your resume to HR@eagle-elec.com indicating 'Pre-CAM Engineer' in the subject line.

apply now

Process Engineer

We are also seeking a process engineer with experience specific to the printed circuit board manufacturing industry. The process engineer will be assigned to specific processes within the manufacturing plant and be given ownership of those processes. The expectation is to make improvements, track and quantify process data, and add new capabilities where applicable. The right candidate will have a minimum of two years of process engineering experience, and a minimum education of bachelor's degree in an engineering field (chemical engineering preferred but not required). This is a first shift position at our Schaumburg, Illinois, facility. This is not a remote or offsite position.

If interested, please submit your resume to HR@eagle-elec.com indicating 'Process Engineer' in the subject line.

apply now

Career Opportunities

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

APL

JOHNS HOPKINS

CAM / Process Engineer

The JHU/APL PCB Fabrication team is seeking a Computer Aided Manufacturing Engineer to support front-end data processing of APL manufactured hardware. You will directly contribute to hardware fabrication in support of National Security, Military Readiness, Space Exploration, National Health, and Research related to fundamental scientific advancement. This position includes a variable mix of core CAM work scope with additional opportunities for hands-on support such as bare board electrical testing, laser drilling, and mechanical CNC drilling and routing.

Responsibilities:

1. Computer Aided Manufacturing for rigid PCB, rigid-flex, and flexible circuits
   a) Perform design checks, panel layout, coupon generation, file generation, stackups
   b) Support manufacturability reviews with internal APL engineers (customers)
   c) Generate work travelers
   d) Communicate status to supervisors and internal customers
2. Support transition of software tools (Genesis 2000 to InCAM Pro)
   a) Edit design rules checks and generate automation scripts
   b) Develop new ideas to further the technical progress of our product
   c) Develop CAM area through continuous improvement initiatives
3. Interface and inform APL Engineers on PCB design for manufacturing guidelines
4. Operate bare board electrical tester
5. Backup operator for CNC drilling, routing, laser drilling (on-site training)

For more details and to apply:
www.jhuapl.edu/careers and search for CAM.

apply now

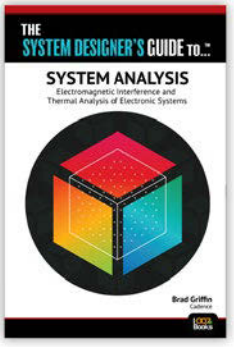
I-007e EDUCATIONAL RESOURCE CENTER

Introducing:

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

Electromagnetic Interference and Thermal Analysis of Electronic Systems

In this latest title from I-007eBooks, readers will learn how system-level analysis of complex and high-speed electronic designs is critical to solve electromagnetic, electrothermal, and electromechanical simulation challenges and to ensure that the system works under wide-ranging operating conditions. Get your copy now!

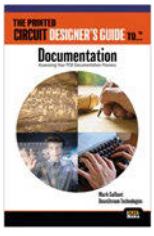


I-007eBooks The Printed Circuit Designer's Guide to...

Thermal Management: A Fabricator's Perspective

by Anaya Vardya, American Standard Circuits

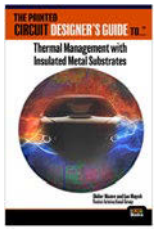
Beat the heat in your designs through thermal management design processes. This book serves as a desk reference on the most current techniques and methods from a PCB fabricator's perspective.



Documentation

by Mark Gallant, Downstream Technologies

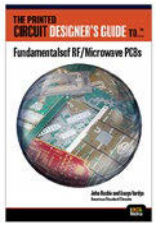
When the PCB layout is finished, the designer is still not quite done. The designer's intent must still be communicated to the fabricator through accurate PCB documentation.



Thermal Management with Insulated Metal Substrates

by Didier Mauve and Ian Mayoh, Ventec International Group

Considering thermal issues in the earliest stages of the design process is critical. This book highlights the need to dissipate heat from electronic devices.



Fundamentals of RF/Microwave PCBs

by John Bushie and Anaya Vardya, American Standard Circuits

Today's designers are challenged more than ever with the task of finding the optimal balance between cost and performance when designing radio frequency/microwave PCBs. This micro eBook provides information needed to understand the unique challenges of RF PCBs.

Flex and Rigid-Flex Fundamentals

by Anaya Vardya and David Lackey, American Standard Circuits

Flexible circuits are rapidly becoming a preferred interconnection technology for electronic products. By their intrinsic nature, FPCBs require a good deal more understanding and planning than their rigid PCB counterparts to be assured of first-pass success.

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

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

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

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

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

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

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

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

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

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

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

TECHNOLOGY SPECIALIST: **BRYSON MATTIES**

COVER: **SHELLY STEIN**

COVER IMAGE: **PROSTOCK-STUDIO**

DESIGN007 MAGAZINE

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

September 2021, Volume 10, Number 9
DESIGN007 MAGAZINE is published monthly,
by BR Publishing, Inc.

ADVERTISER INDEX

Problems solved!

SUBSCRIBE NOW