

IConnect007

NOVEMBER 2021

# DESIGN007

M A G A Z I N E



“A must-read for anyone  
new to system analysis.”



Lee Ritchey  
Founder and President  
Speeding Edge

THE  
**SYSTEM DESIGNER'S GUIDE TO...**<sup>TM</sup>

**SYSTEM ANALYSIS**

Electromagnetic Interference and  
Thermal Analysis of Electronic Systems



Brad Griffin  
Cadence

100%  
Books



Download Now

# Distinctly different.

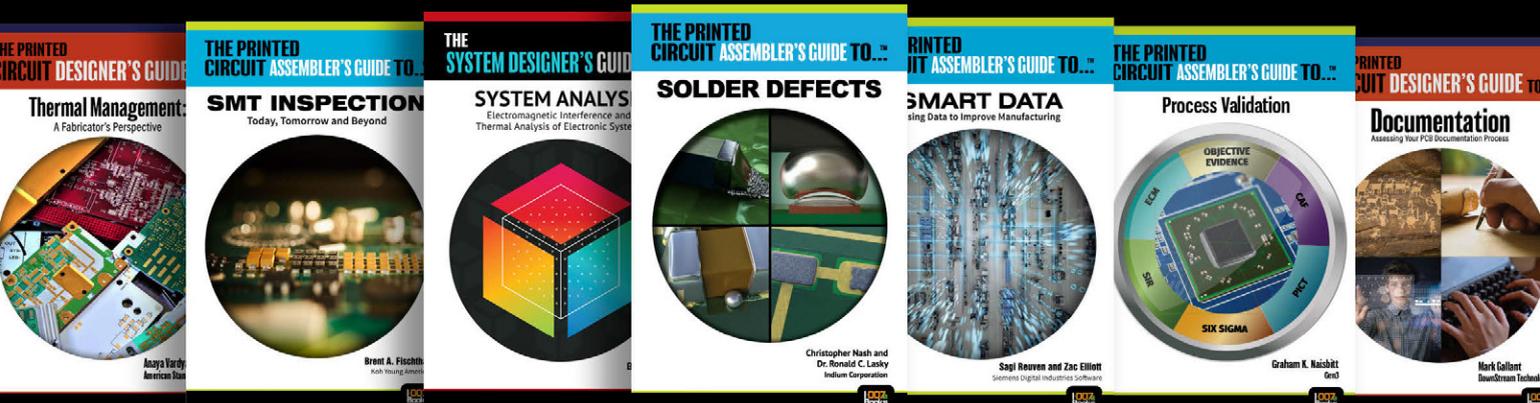
Our books are written by recognized industry experts. At around 8,000 words, they are unique in that they are able to be incredibly focused on a specific slice of technology.

“I-007ebooks are like water in the desert ...it’s up to you to drink it in order to survive!”



**Stephen V. Chavez**  
PCEA Chairman, MIT, CID+

[VISIT OUR LIBRARY](#)



# DESIGN007

M A G A Z I N E

## Herding Cats

As more than one designer said, “Managing design data is a lot like herding cats.” It’s no wonder: schematics, footprints, BOMs, netlists, fab notes, assembly notes—millions of petabits (no pun intended) of data are used to design and engineer PCBs, and readers cite mismanaged data as a constant source of heartburn.

**FEATURE ARTICLES**

**12** **A PCB Design Data Management Overview**  
Interview with John Watson, Dugan Karnazes and Patryk Akhurst



**20** **Why Good PCB Data Management is Essential**  
by Dana Korf



**26** **Manage Your Data and Document Everything**  
by Tamara Jovanovic



**56** **Solid Data Management Key to Accurate Quotes**  
by David Gronner

**FEATURE COLUMNS**

**44** **Say ‘No’ to File Hoarding: Data Management Tips**  
by Tim Haag



**62** **IP Reuse Enables a Digital Transformation**  
by David Wiens



**72** **Cost of Compliance and How Data Transfer Standards Can Help**  
by Patrick Crawford



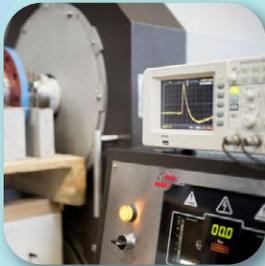


# Candor

Circuit Boards

## Innovative solutions for complex circuit board manufacturing

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



RF/Microwave  
PCBs



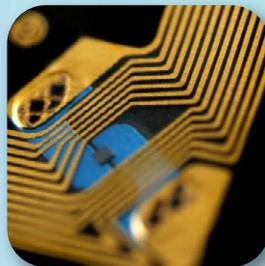
Power  
Electronics



High-Speed  
Data



Automotive  
Electronics



Lamination



Startups

Our technology produces what our customers dream!

[www.candorind.com](http://www.candorind.com) | (416) 736-6306

# DESIGN007

M A G A Z I N E



- SHORTS**
- 24** All Systems Go! Signal Integrity Signoff of 3D-IC Systems
  - 29** Elementary, Mr. Watson: First, Component Shortages, and Now Hot Dogs?
  - 42** Nanoscale Lattices Flow From 3D Printer
  - 60** Book Excerpt: *The System Designer's Guide to... System Analysis*
  - 70** A New 3D Printing Frontier: Self-Powered Wearable Devices

**DEPARTMENTS**

- 101** Career Opportunities
- 116** Educational Resource Center
- 117** Advertiser Index & Masthead



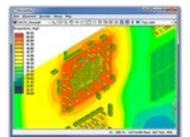
**ARTICLES**

- 38** DFT and Test Strategies Shouldn't Be an Afterthought  
Interview with William Webb
- 48** 45+ Years of PCB Manufacturing Experience  
Interview with Kim O'Neil
- 76** American Standard Circuits on the Fundamentals of RF and Microwave PCBs  
by Pete Starkey



**COLUMNS**

- 10** Data Management: It's a Lot Like Herding Cats  
by Andy Shaughnessy
- 30** Impact of Filled Vias on Thermal and Signal Integrity  
by Barry Olney
- 66** A Grand Announcement  
by Kelly Dack
- 82** Diving Into the Chemical Processes of PCB Manufacturing  
by Matt Stevenson



**HIGHLIGHTS**

- 36** PCB007
- 80** MilAero007
- 98** Top Ten Editor's Picks



# 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 construction.



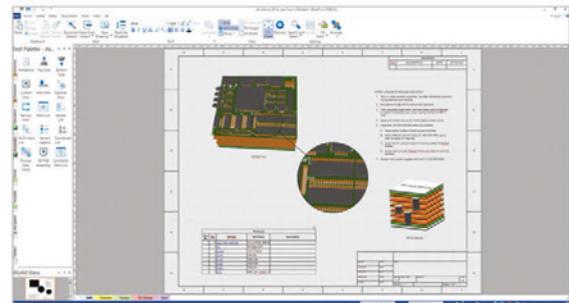
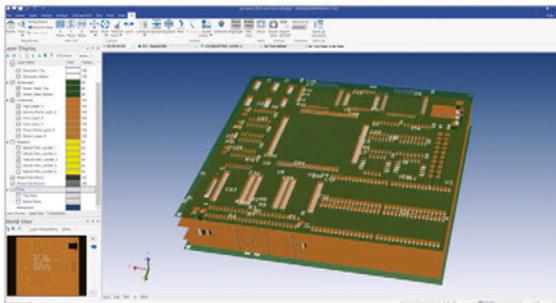
**Panelize**

Minimize costs in both PCB fabrication and assembly through maximum 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.



800-535-3226 | [downstreamtech.com](http://downstreamtech.com)

©2019 DownStream Technologies, Inc. All rights reserved.

NOW WITH **3D**

# FLEX007

## This Month in Flex

The flexible and rigid-flex segments are hotbeds of innovation right now, and OEMs that create household electronics can't seem to get enough of them. But a lot has changed since flex first appeared on the market 60 years ago. Columnist Joe Fjelstad celebrates his 50th year in the business with a retrospective of his career in electronics, and a few predictions about the electronics of the future. And Jamin Taylor of All Flex discusses the company's new flexible medical applications.

### FLEX007 FEATURE COLUMN

- 86** **A Brief Retrospective of 50 Years in the PCB Industry**  
by Joe Fjelstad
- 

### HIGHLIGHTS

- 92** **Flex007**
- 

### FLEX007 INTERVIEW

- 94** **All Flex Discusses Merger and New Medical Applications**  
Interview with Jamin Taylor
- 

### FLEX007 SHORT

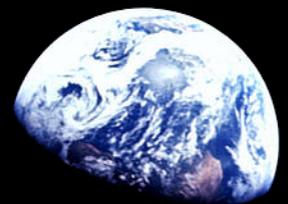
- 90** **Using Carbon Key to Decarbonizing Economy**



# 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](http://www.asc-i.com) | (630) 639-5444

# Data Management: It's a Lot Like Herding Cats

## The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

“It’s all about managing your data.”

That’s a refrain that we’ve been hearing from designers over the past few years—in surveys and conversations with designers and design engineers.

When we started planning this issue, our most recent reader surveys pointed to data management as a perpetual problem for PCB designers. It’s no wonder: schematics, footprints, BOMs, netlists, fab notes, assembly notes—millions of petabits of data are used to design and engineer PCBs, and readers cite mismanaged data as a constant source of heartburn.

Designers today must manage more data than ever before, while turning it into actionable information that they can use to do their

jobs and put food on the table. Think how much of your job revolves around data—updating, measuring, tracking, and ensuring that it’s accurate and complete, because every time another human touches that data, there’s a risk the data will be compromised.

As more than one designer said, “Managing design data is a lot like herding cats.” As a former cat owner, I have scars that can attest to cats’ opposition to being herded. Herding one cat was quite a job. (Sometimes it took me a half-hour to get my 24-pound Maine coon, Night, into a cat carrier when it was time to go to the vet.) This is how we picked our cover image for this month.

In the past, we’ve dedicated issues of *Design007 Magazine* to specific steps for han-



dling one type of design data, such as fab notes, datasheets, or footprints. But this month, we wanted to step back and look at data management in general, from a more macro level than we have in previous issues. What data management truths are self-evident, to paraphrase our Founding Fathers? Who owns the data in the first place? What are the best methods for ensuring traceability throughout the design process? Should you trust data that originates outside your company?

These are just a few of the questions we asked this month's contributors to address. We start off with a conversation with Dugan Karnazes and Patryk Akhurst of Bissell, and John Watson of Altium. They discuss the need to customize your data management process to fit your company, and why that process must be constantly improved to remain effective. Tamara Jovanovic of Happiest Baby explains how a solid data process keeps her team running smoothly, and she shares a few horror stories about designs that went off the rails due to a lack of documentation. And columnist Dana Korf explains why data management is so essential, and which stakeholders own which data.

Next, columnist Tim Haag reveals some of his secrets to tracking revision history, figuring out who owns what data, and a variety of methods for avoiding data hoarding. David Gronner of PCB Technologies discusses why accurate data is the key to an accurate quoting process. Columnist Dave Wiens explains

how the reuse of critical IP is a large part of a design team's data management strategy, and why design teams need to overcome their aversion to using circuits "not invented here." And columnist Patrick Crawford shares a variety of reasons why data transfer standards can simplify your certification compliance process, as well as streamline your company's data management framework.

We also have columns by our regular contributors Barry Olney, Kelly Dack, Matt Stevenson, and Joe Fjelstad, who is celebrating 50 years in the industry this year. We have an interview with Will Webb of ASTER Technologies, and Pete Starkey's interview with John Bushie and Anaya Vardya of American Standard Circuits, who discuss the challenges of designing and fabricating RF and microwave PCBs. And Jamin Taylor discusses All Flex's move into medical applications.

It's hard to believe that IPC APEX EXPO and the co-located AltiumLive show are less than three months away. Are you going to San Diego? If you're not, don't worry—we'll bring you complete coverage of these perennial events.

See you next month! **DESIGN007**

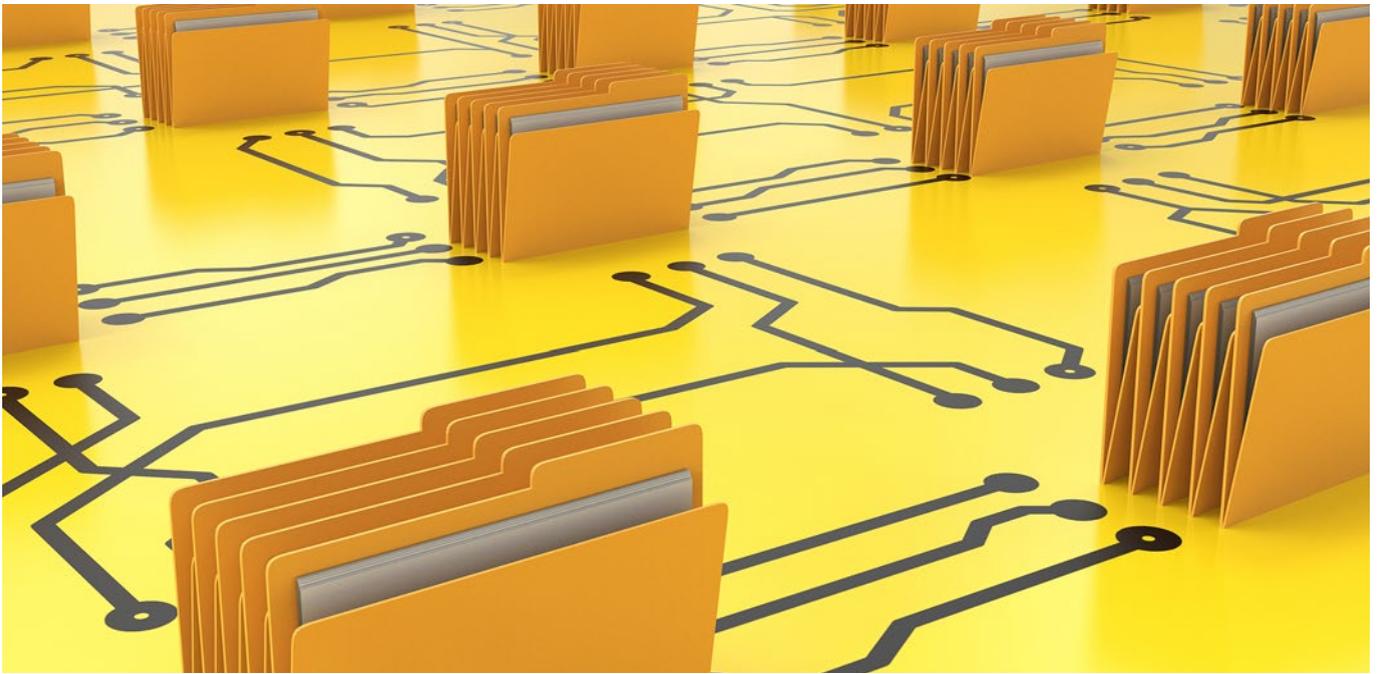


**Andy Shaughnessy** is managing editor of *Design007 Magazine*. He has been covering PCB design for 20 years. He can be reached by [clicking here](#).

**jobCONNECT007**

Companies seeking talent with circuit board industry experience post their jobs with us.

[jobconnect007.com](http://jobconnect007.com)



# A PCB Design **Data Management** Overview

Feature Interview by the I-Connect007 Editorial Team

In recent issues of *Design007 Magazine*, we've covered strategies for managing specific types of data. But in this issue, we're looking into data management techniques from a company level.

The I-Connect007 Editorial Team recently spoke with three PCB designers who have quite a bit to say about PCB data management: Altium's John Watson, and Bissell's Dugan Karnazes and Patryk Akhurst. They discussed their data management philosophies, the need for detailed processes that fit each company, and why data management techniques must undergo continuous improvement to be effective.

**Andy Shaughnessy:** Dugan, would you please share your overall thoughts on data management, and we'll go from there?

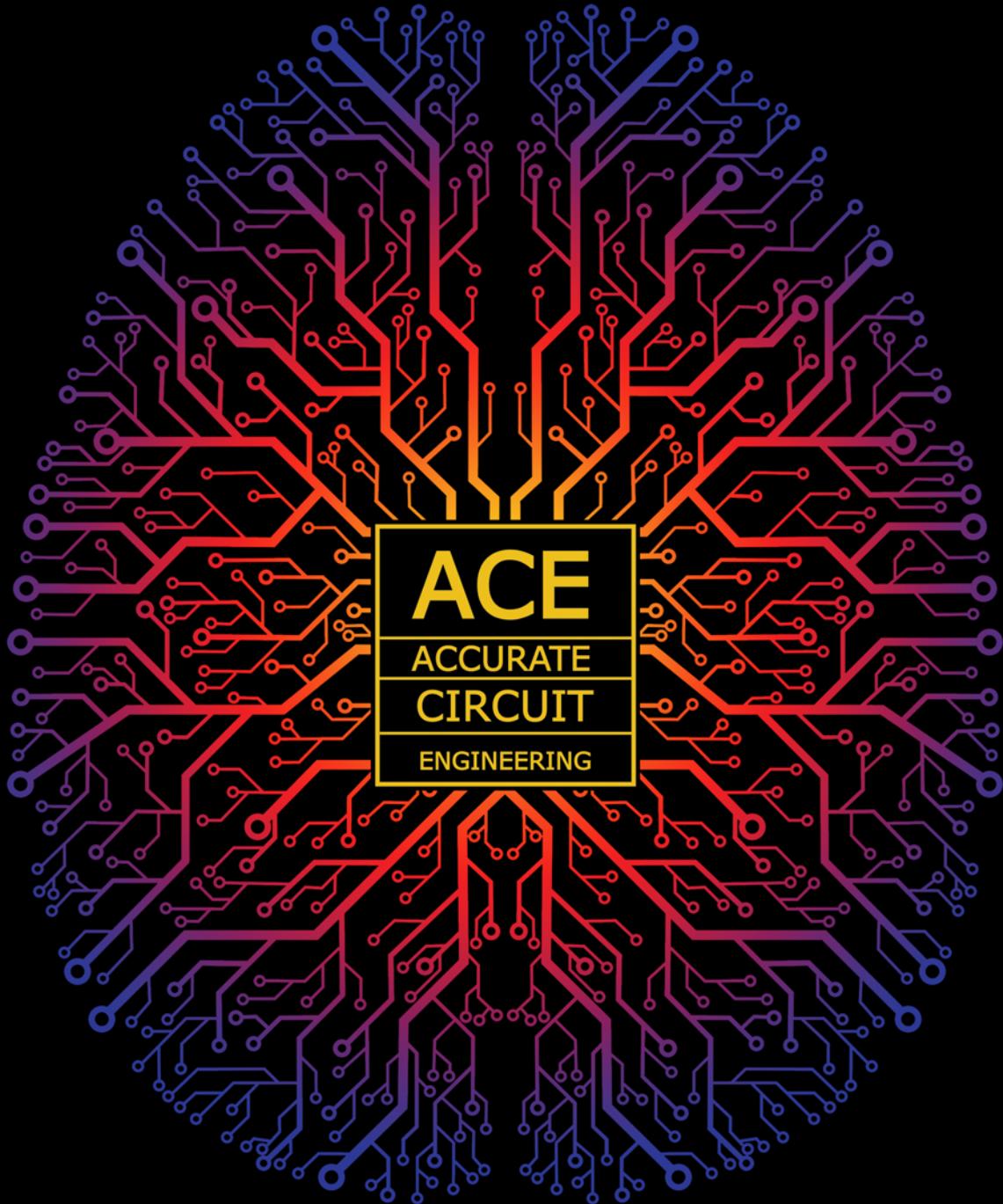
**Dugan Karnazes:** Coming from a consulting background, my first question, which shapes

who's managing the data, is "Who is your customer?" If you're managing data for a contracting situation, ultimately you must give all the files back to the customer. You hope that they have standards for how they're storing these things. The unfortunate reality that I saw is that many companies don't. Even though they specify how they want their files back (and the good ones will give you a list of everything that makes up a complete data package), often they don't even enforce that. Many times, four months down the road, they'll ask for it again. So, who's the customer? What are their requirements, and what are your own higher requirements?

When I was responsible for handing off a design to a customer, one company gave me a list of everything they wanted: "We want the PCB in this format. We want exports for the schematic libraries and a separate export for the footprint libraries." It made it easy. When I gave it to them, they were highly surprised because no one else had ever given it to them.

I had other customers who said, "I don't even know what an RC filter is. Why are you asking

# Don't Overthink Your Prototype Builds



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

RF • MICROWAVE • HIGH-SPEED • QUICK TURN EXPERTS



## Accurate Circuit Engineering

3019 Kilson Dr  
Santa Ana, CA 92707

(714) 546-2162 • [sales@ace-pcb.com](mailto:sales@ace-pcb.com)

For technical assistance, call:

**James Hofer**  
(714) 227-9540

[www.ace-pcb.com](http://www.ace-pcb.com)

me that? I just want to come to you and get the boards.” Now I need to do it all for my standards, and I need to have a system internally for tracking that. I think that ownership of the standard needs to be set at the manager’s level, but ultimately, it’s always going to come down to the engineer and the people who are in the CAD package to make sure that the data is meeting those standards. I’m curious to see how other people handle this, because every group I’ve talked to handles it differently.



Dugan Karnazes

**Shaughnessy:** Right. John, you worked at an OEM for years; you had a lot of designers working for you. Who was responsible for all that data?

**John Watson:** First, it really depended on whether things were working well with the data or not working well (laughs). But, understanding your finish line and what you’re providing to that customer is vital. Let’s say your company makes Christmas cards that play music when you open them. They’re not going to be looking for the same quality of data as a DoD company. It’s a matter of understanding everyone’s role internally in your company. Who is responsible for creating the documents or that data? Who is responsible for QC-ing it? Who’s responsible for coming up behind and making sure it’s correct?

**Dan Feinberg:** Let’s say that there’s a gross error, something that should have been caught. Who is responsible: the company, or the individual employee who made the error?

**Watson:** I think it would be both because the company is responsible for putting in place the

quality checks that are necessary to make sure it’s correct. If the process is not put in place and followed, then you’re doomed for failure.

**Feinberg:** That’s a good answer. I agree.

**Watson:** If you’re putting footprints on a board that have not been looked at and verified, according to a standard, and you’re using your fab house and assembly house to verify your footprints, forget it; you’re done as a company. I have worked

in the medical field and the DoD as a PCB layout person. I can tell you their standards are so high that they had a set procedure and process to do things with a required endpoint, a deliverable; if you broke that SOP, you were done. That’s the level you must work at today.

**Feinberg:** Especially today. I agree because we see a trend toward individuals not wanting to take responsibility.

**Watson:** Yes. When something happens, the first words out of anyone’s mouth are, “That’s not my fault. That wasn’t my area. That wasn’t my responsibility.”

I actually got to the point where I had my engineers signing documents, signing the schematic, signing the PCB, and releasing it. We would do a design review. We would pass a piece of paper around, and everybody who signed was saying, “Yes, I approve of this design.” We had gotten into a habit where six months down the road, if something was missed, everyone said, “No, I didn’t agree to that.” Then, you could pull that piece of paper out and say, “Well, there’s your signature.”

What happened? We need to do a postmortem and identify what happened here. Why did

this occur? That's where a lot of the solutions then happen in a company where a negative situation could become a positive for them. At least they learn a lesson through that.

**Shaughnessy:** Patryk, I understand that you have some ideas on this.

**Patryk Akhurst:** My viewpoint is that it's not one person owning anything. Everybody must collectively do their part for it to work, right? The person designing the footprint must design it according to a datasheet and the person producing the board must design the board according to standards, right? But you also must rely on your fab to use the right components. Like John said, everybody must be responsible for what they did. You can't be a part of that chain and then say, "Oh, at least I didn't break the chain."

But a lot of times, we don't have more than two designers, and it really falls on two designers to do just about all the product development. We do our footprints. We do our component management. We do our designs, and data management really falls on us. Every time we get something back, we've said, "Why is something not working exactly how we want it? How can we fix it to make it better?" In doing that, we need an adaptive reuse. How can we fix something on one board and put it on all the other ones that have the same problem right here?

**Karnazes:** Patryk, I want to add onto your point. John, you mentioned getting designers to sign off on the designs, and that's a trend we've been seeing for a while. I think Patryk captured it well when he mentioned that the way that we're handling that shift now is

**REALTIME** with...



**American Standard Circuits**

Creative Innovations In Flex, Digital & Microwave Circuits

## An important discussion on Flex and Rigid-Flex...



Anaya Vardya



David Lackey

**WATCH NOW** 

PRESENTED BY

**I-Connect007**  
GOOD FOR THE INDUSTRY

through facilitating reuse and really utilizing software development techniques for hardware, to improve the efficiency of a team and to make sure that something is fixed, like Dan mentioned. You might make a mistake, but you only get to make it once. And then it's fixed after that.

**Watson:** How good a designer you are is dependent on your motivation and your goals. Let's be honest, there are some designers who are way beyond their expiration date. They should have moved on, and now that we're getting over COVID, companies are turning back on again and designers are really pushed for time. I spoke about this in my [July PCBDesign007 column](#), about companies now pushing to get things done. But often when you do that, the details are skipped.

I hear that so often, "We don't have time to double-check things here. We need to get this out the door." I've said this before—many good PCB designs are sacrificed on the altar of expediency. Often, designers think, "I got it. I got a board through fabrication. I got it through assembly. It must be right."

However, that doesn't mean it's correct. It just means you got lucky. I think it takes a lot of intestinal fortitude to push back on those people who are pushing the schedule. If you asked the sales department when they would like to have something done, what do they say? Last week? The mentality is, "Just give us something to sell." When that happens, it's just like the whole process is pushed by an unrealistic time schedule.

**Shaughnessy:** What data do you share with the other stakeholders in the process? Do you



Patryk Akhurst

share everything, or is it on a case-by-case basis?

**Watson:** The one document I never put out to either a fab house or an assembly house is a schematic. That's a protected document that gives them an entire design and it's very important not to give that to them. But I've seen some companies who will put out some of those documents, and the fab house will actually change the Gerbers or certain specifics about the board. There is a little bit of a disconnect.

There is this symbiotic relationship, we'll call it, between the engineer and the fab house and what they do with their Gerbers. I think every Gerber set gets changed a little bit.

**Happy Holden:** Dugan, as a product developer, a product contractor, did people come to you for products or just for board designs?

**Karnazes:** In my last company we designed low-volume products. There's one customer who maybe needs about 20 units a year, but it involves sensors, a dedicated circuit board, RF module, and battery management. Other companies would just need a USB-C power supply reformatted because they changed the packaging, and it was a different shape. I got into it first as an electronics engineer, designing boards for people, but then added on formal engineer and mechanical engineer. I've now moved on to this role at Bissell. But managing all the data and figuring out how to create that system was definitely one of the challenges that we faced.

**Watson:** Was there anything out there that you could pick up as a model to start with? Do you have to start from scratch?

**Karnazes:** It depends on the customer. We have about 25 different companies we've worked with. Some of them were just an inventor with an idea and no assets other than an Adobe Illustrator sketch. Other companies came to us with a circuit studio board and schematic with their own libraries and they just want us to update it. So, we really got everything. There was a pretty good spread to it.

**Shaughnessy:** I was thinking back to John's idea about having the initials on the design. Are checklists used to track the chain of custody of this data?

**Watson:** It's a mishmash of different things. It depends on your company and how good your bigger team is. I mean, you may have two or three guys working from their garage right now, and they still have a great process because

they wear different hats. It's just not assigned to a specific person who sits there and does a specific task.

But you're correct; there is a set process that goes through. Often, when you divert from that process, you introduce risk. There is that hand-off. Throughout that process, there are checks and milestone events that have happened that are gatekeeping. I always referred to them as gatekeeping items because that design needs to be verified at that point to make sure that you're still on track with the original objective.

**Shaughnessy:** How much of this data is kept in the cloud?

**Watson:** Not to do a shameless plug here, but Dugan mentioned Altium 365, a platform that is getting some interest. Your whole project can be placed up there—your components, librar-

**REALTIME** with...



**American Standard Circuits**

Creative Innovations In Flex, Digital & Microwave Circuits

## An important discussion on Thermal Management...



John Bushie



Anaya Vardya



David Lackey

**WATCH NOW** 

PRESENTED BY

**I-Connect007**  
GOOD FOR THE INDUSTRY

ies, templates, and everything else in your design. That has really broadened the team that you can work with. Now you can have contractors and other individuals who come in, maybe work on your library, and they don't have to be sitting next to you. It has really expanded the paradigm for us.

**Karnazes:** Yes. I built my company on Altium 365. At the start of COVID, we were working on this small ventilator project, and we just wanted to make an open-source ventilator. We wound up sponsoring the project with some licenses and because we were managing our data in the cloud, we were able to go from one license that we were just sharing amongst our team to eight licenses being shared with 20 different designers around the world. With cloud-based storage, if you're not working on defense stuff, it works well.

**Nolan Johnson:** That's fair. What happens when you're working in a design team and the processes don't work? For instance, a major automotive company PCB designer would often use our tool to get certain things done because if he used the normal process at his employer, he was going to miss the cutoffs for getting that functionality into a particular model year. He had to go rogue. How do you manage that?

**Karnazes:** It's my philosophy that if your engineers need to go rogue, your management has a problem they need to address. If an engineer is being forced to go through an impossible process and must violate a business's standard practices—that take a lot of time to develop, a lot of manpower to create,



John Watson

and are there for a reason—that's a failure of management, absolutely. But there are some cases where there is a speed element that must happen. At Bissell, we have an advanced development group and we're released from some of the restrictions on following all those procedures, but the trade-off is that we don't touch the production stuff. All our projects are concepts. They're meant to be revisited and that allows us to explore that technology and do so in a safe sandbox area

where we're not going to be interfering with anything that affects the company's bottom line.

**Watson:** Is it the engineers' responsibility to be communicating that upward?

**Karnazes:** Absolutely. Then it comes down to the company's culture. You don't want to be the naysayer: "I've been given this impossible task, and I'm not going to be able to deliver." No one enjoys saying that. Some cultures are tolerant of it, but others are not. To John's point, a great way to tell when a company is dead in the water is when they stop listening to their engineers.

**Watson:** I would also mention the new tools. One of the biggest tools that we've added into our cloud structure here is a version control system with a Git repository or something like that. Now, that has redefined the roles of a PCB designer because you can have an entire team working on a PCB layout. They're all working on the exact PCB at the same time.

But the key is communication. A PCB design is not a static thing. It's a living, breath-

ing thing that has issues, problems, and situations. How about part procurement? That's a big one right now. You have to go through that process. Along the way, it's going to be a process of communicating to the people who are doing the task and to management about shifting goals, maybe shifting perspectives or objectives that they may be expecting. I think that's where there's not a communication process upward because the managers will come down and say, "Do it."

That's not communication. When things go wrong, there must be a serious discussion. A lot of times we deal with the effects of the results rather than the actual root cause of the problem itself. Nothing ever gets fixed when you do that. You need to dig deeper into the problem itself and ask why this happened. Sit down, be honest with

yourselves, and don't point fingers. Do not place blame but find the solutions to the problems and then implement the changes necessary to make sure that it doesn't happen again.

**Feinberg:** Gentlemen, any final thoughts on the topic?

**Karnazes:** I think Patryk really hit the nail on the head: data management is all about continuous improvement.

**Feinberg:** I would agree.

**Shaughnessy:** Thank you all for doing this. It's been very instructive.

**Karnazes:** Thank you. DESIGN007

**REALTIME** with...



**American Standard Circuits**

Creative Innovations In Flex, Digital & Microwave Circuits

## An important discussion on RF/Microwave PCBs...



**John Bushie**



**Anaya Vardya**

**WATCH NOW** 

PRESENTED BY

**I-Connect007**  
GOOD FOR THE INDUSTRY



# Why Good PCB Data Management is Essential

Feature Article by Dana Korf  
KORF CONSULTANTS

*“Data really powers everything that we do.”*  
— Jeff Weiner, LinkedIn

Manufacturing data management consists of four primary components: data transfer, data storage, revision control, and data access.

The PCB fabrication and assembly data management system and process is required to be robust to protect customer IP and ensure that the proper data is used to build the product. The entire process assumes that the provided design data package can't be built as-is. The design data must be updated/completed after it is received by the manufacturer. Powerful CAD/CAM software has made it easy for designers/companies to pass the final editing and revision control on to their manufacturers. Over time, nefarious entities have significantly improved their ability to steal/copy data in parallel with this, thus complicating data transfer and management.

Conceptually, the required design data should be sent as a single intelligent file to the

manufacturers, so production tooling is created without any human intervention. The standard industry process is still based on sending multiple files within the data package. These include the Gerber graphical data, Read.me ASCII file, PDF formatted fabrication print, etc. These files may be sent together in the same Zip/TGZ file or at separate times. This method requires humans to review these files since they often have conflicting data and/or violate the OEM's acceptability and/or manufacturing capability/preferences. Intelligent data formats, such as IPC-2581, eliminate the additional, and often conflicting, documents by incorporating the intelligence in the CAD data.

## Data Package Security and Transmission

When I went through ITAR training many years ago, the most interesting information was how people, companies, or governments would gather pieces of product data and then piece it all together so it could be duplicated. The information could be gathered by breaking into the data storage system, employees copying files and selling them, and/or by intercepting emails.



AltiumLive 2022

# CONNECT

Connect with fellow engineering professionals, thought leaders, and industry experts from every aspect of the electronics development ecosystem.

January  
**26-28**  
2022

San Diego Convention Center  
San Diego | California

**Available Tickets:**

3-Day All-Access  
2-Day Summit Pass  
Virtual All-Access

[Learn More](#)

**HYBRID EVENT** | In-person (San Diego Only) + Global Livestream

- Five Unique Technical Tracks
- Networking with Industry Peers, Experts, and Leading Electronics Companies
- Four Exciting Keynote Presentations from Industry Leaders
- In-person Full-access to the IPC APEX Expo

Join us at AltiumLive 2022: CONNECT, where you will **Learn, Connect, and Get Inspired** in a new immersive experience of technical tracks and convenient access to all that IPC Apex Expo has to offer.

Proudly Co-Located with



In one instance that I was involved in, a large OEM contacted my company and asked why we were building their boards, in volume. We checked our records and told them that we didn't have any order from them. So, they sent us a copy of their data package. We did a database query for datasets with similar characteristics; low and behold, we found a dataset that was an identical match but ordered by another company. In another case, we received an order for a consumer PCB that had a ship-to address for a jewelry dealer in a large Asian city. We informed the OEM, and they took on the investigation at that point. We obviously refused the order.

There are many methods to transmit data between companies using various security methods with complex encoding/decoding protocols. The specific method is generally specified by the ordering company. Good data management is also required within a manufacturing facility to restrict data access to only those who require it. It also denotes that individual logins/passwords, not generic logins for a team of people, are required to log who accessed the data, when it was accessed, and who modified any data. This includes going all the way down to the actual equipment.

Access to internal CAD/CAM/ERP software must also be controlled so only authorized people may approve and/or modify the specific manufacturing tooling. If I count my fingers and toes, plus the number of birthday candles on my last cake—I was born the same

year as Disneyland—it still doesn't equal the number of times someone had to modify artwork on a Sunday night, without the proper folks available to authorize the edit per procedure. The requestor will propose to edit the tool and let the ECO paperwork catch up the next day. This, of course, opens the company for unintentional scrap, rework, or customer returns because the proper ECO/document update does not properly occur or should not have been authorized.

## Quote and DFM Data Vs. Build Data Revisions

Figure 1 shows typical paths for data flow from the original OEM/ODM to the final assembled board. There are many contacts where humans touch the data package. It will not be unusual for the quoting department to receive partial data to generate a budgetary quote. An updated version of the data may be sent for the final order. For example, the designer may have version 5 of the data, the CM may have an order for version 4 and the quoting may have occurred on partial revision 1 data. From a fabricator's perspective, the only version that should be used to manufacture the board will be the version that is placed by the company/person who places the purchase order (version 4 in this case).

The current process has a minimum of six or more human touch points before the assembly is returned to the OEM. Each touch point creates potential security and revision control risks.

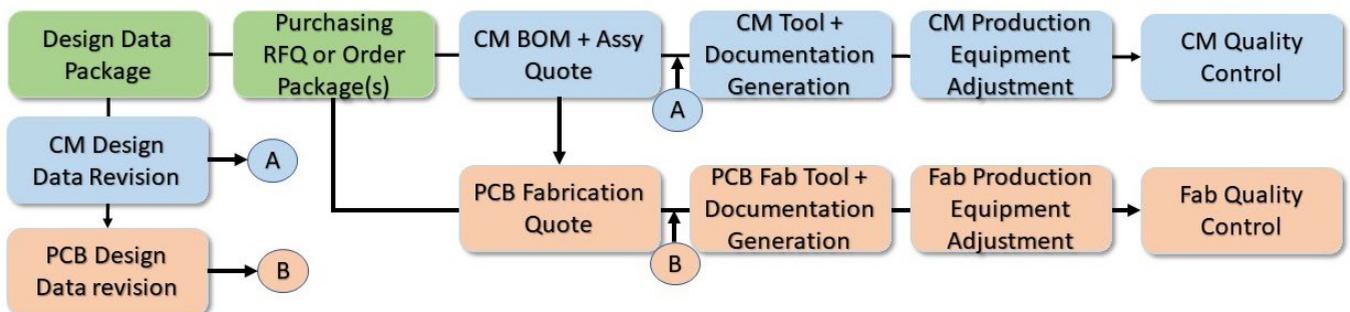


Figure 1: Generic data hand-off.

Pre-release DFM level data has the same issue as quote data. It is common to have preliminary material stackups sent back and forth between the fabricator and engineer until a final decision is made. Partial data files are also sent to get DFM requirements early in the layout process, before the routing is complete.

## Design Data Build Package Revision Control

Now let's discuss revision control. Who actually owns the data used to create a PCB? Is it the designer, the CM, or the fabricator? Let's look at examples (Table 1) of who requests a change compared to who performs and owns the final dataset.

Changes may be requested by many people within a company, or from multiple companies. It is essential that the allowed change approvers are determined when data is received to ensure proposed modifications are correctly approved. Every board manufactured by excellent companies will have an internal part number, internal revision, original part number, and original dataset revision numbers for the dataset using in manufacturing. This allows all changes to be tracked. These revisions may be made by a tool—for example, a single art-

work layer—or for an entire dataset. Change management is complicated, because a common change request may be sent to multiple suppliers who may implement the change differently.

Many OEMs/ODMs will ask the manufacturer to revise the data instead of having an updated dataset resent. They say, "It takes too long to get a change approved using our internal ECO process." It is also essential that a single point-of-contact from the ordering company be provided who will be responsible for providing responses to all technical questions. That person may be the person who generates the response or the person who coordinates/filters responses from team members.

## Manufacturing Build Data Management

Data used on/in manufacturing equipment is fundamentally the master data. The editing, storage, and transmission of the data must be retained and traceable to the individual board, ECO, specific equipment/line, operators, and/or production lot. This include the material that is shipped within the product. Traceability and defect tracing is error prone and lengthy when this information is recorded on paper or in a non-integrated work cell computer.

Change Requestor	Implementor	Changed Data Owner
Designer	Designer	Designer
Designer	Fabricator (1-n)	Designer
Designer	Fabricator (1-n)	Fabricator (1-n)
CM engineer (1-n)	Designer	Designer
CM engineer (1-n)	Fabricator (1-n)	CM engineer (1-n)
CM engineer (1-n)	Fabricator (1-n)	Fabricator (1-n)
Fabricator process engineer	Fabricator front-end engineering	Fabricator front-end engineering
Fabricator process engineer	Fabricator process engineer	Fabricator process engineer
Production equipment	Production equipment	Production equipment

Table 1: Who requests the change, who implements the change, and who owns the change.

The lowest level of traceability needs to be maintained to reduce the quantity of product that will be scrapped or reworked when an issue is discovered. Electronic records are critical for this. Information contained on paper or in cross-section slugs are easily lost. CFX/IPC-2581 is a good method to ensure the entire data integrity is maintained for assembly. CFX has not been implemented for PCB fabrication at this point. The IPC-2581 structure incorporates both PCB fabrication and assembly/test data.

## Solutions

There are many data formats and encryption formats available to securely transmit data. The IPC-2581 intelligent data package, along with the IPC-CFX collected data history, allows for a significantly better chance to hold all the revisions in a common dataset. It is essential that the data management system contain

as much data that can be readily queried. All touch points and actions must be recorded so proper root cause analysis can be quickly and accurately performed. The seemingly high cost to automate, record, control access, and restrict data is easily recovered due to process yield improvements, overhead cost avoidance, and fault analysis improvements.

The industry's existing standard data transfer process needs to evolve from manual intervention to a fully automated transfer if we ever want to reduce data management complexity. **DESIGN007**



Dana Korf is the principal consultant for Korf Consultancy.

---

## All Systems Go!

# Signal Integrity Signoff of 3D-IC Systems

by Brad Griffin

3D-ICs meet the demand for integration of disaggregated system-on-chip (SoC) architecture built from multiple chiplets and heterogeneous architectures such as analog, digital, optoelectronics, and non-volatile memory. They provide improved performance and area, low power consumption due to short interconnection length, and reduced signal delay. We can broadly classify 3D-ICs as transistor-level 3D integration, system-in-package (SiP) and system-on-package (SoP), and wafer-level through-silicon vias (TSV)-based 3D integration.

TSVs are the paramount interconnection structures in 3D-ICs. 3D-ICs with TSVs have a broad impact on applications that require ultra-light, small, and low-power devices to achieve high-throughput memory access and hybrid logic circuits. They serve as vertical channels for interplane communication, offering a wide range of granularity. TSVs increase the average number of connections between two dies up to 10 times compared to the chip-to-chip

connections on a PCB. They improve capacitance by six times, average connection length by 200 times for 3D stacking vs. side-by-side stacking, and relative interface power between the CPU and DRAM by approximately six times.

Signal integrity (SI) implies the capability of the signal to propagate without distortion, ensuring both the clock signal timing and the quality. Accurate SI analysis ensures both functionality and regulatory compliance. There are many factors to be considered in the 3D-IC structure for reliable transmissions, including loading effect and reflection of various 3D structural elements such as limitation of high-speed signaling by capacitive loading and impedance mismatching, reflection, and crosstalk between TSVs, die-to-die vertical coupling, jitter by inter-symbol interference, vertical die-to-die EMI coupling, high-frequency noise coupling and transfer, and RF sensitivity reduction by EMI.

To read this entire column, which appeared in the Design007 Week newsletter, [click here](#).

**1-007<sup>e</sup>**  
Micro Webinars

On Demand: Free 11-part Webinar Series

# Predicting Reliability in Electronics

with experts Graham Naisbitt  
and Chris Hunt

This webinar series explains how  
new ground-breaking test standards  
are helping to ensure board reliability  
throughout the world of electronics,

SPONSORED BY



[START WATCHING](#)



# Manage Your Data and Document Everything

Feature Article by Tamara Jovanovic

HAPPIEST BABY

One of my biggest nightmares involves getting a call from my boss over the weekend; there's a problem and they need documentation that I manage, but they can't find it on our shared drive.

As someone who is responsible for the files and documents related to electrical work, I always want to make sure that all the data I manage is accessible to my team and coworkers. If they need to look at anything I'm working on, they'll know exactly where to find it. It goes without saying that my teammates should always be able to find what they are looking for at any point in time.

At a small company, it's important to have rules and guidelines about managing documentation. I'm sure bigger companies have full-time employees who oversee PLM systems and make sure every piece of work is documented in a clean and comprehensive way. In a startup environment, though, this type of work usually falls on engineers and the tech-

nologists doing the actual testing. On top of our engineering tasks, we must make sure that the details of our research and testing are documented. In such a fast-paced work environment where new issues and tasks arise almost every day, there is always lots to do, and it is very easy to forget what was done a week ago, not to mention a month or a year ago.

For a while, I was the only electrical engineer at my company, and I was constantly backing up my data. However, sometimes when I'm in the rush to get things done and move on to the next task, things like documentation slip through the cracks. I always reassure myself, "I did the testing, and I will remember what I did if it's needed." But we deal with so many tasks that it is impossible to keep it all in my head. The information is in there somewhere, for sure, but no human can remember every detail of everything they've ever done, unless they have a photographic memory. Even then, a photographic memory is no use to team members looking for that previous revision.

For instance, recently I was working on some EMI issues with one of my set-ups. We

# ENABLING NEXT-GEN SYSTEMS

DOMESTIC  
MANUFACTURING  
**READY**

Advanced additive manufacturing down to 15  $\mu\text{m}$  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

did some work on this over a year ago, but I couldn't remember exactly what was done or if any changes were made to the system because there wasn't any real documentation about the testing or results. I asked the co-workers who were involved with the issue at the time, but no one could remember any of the specifics, or even the outcome. So, I had to start from scratch: everything from defining the issue, identifying why it is present, and understanding the root cause, to figuring out the solution and implementing it, as well as making sure that I documented all the information every step of the way.

---

**I would never let anything I do go undocumented, because doing it all over again from scratch proved to be a big, complicated task. And an unnecessary task at that.**

---

A few months later, someone asked me about it, and I had a hard time remembering every piece of the puzzle. I looked over my detailed documentation and I immediately remembered all the specifics of the project. Sometimes that's all we need—a quick refresher, a summary, or a few bullet points to remind us of what was done. Now, I can always go back to that document and answer any questions about the issue with certainty. This was a great lesson for me personally; now, I would never let anything I do go undocumented, because doing it all over again from scratch proved to be a big, complicated task. And an unnecessary task at that.

Some areas are problematic in the best of times. For instance, revision tracking of design files can turn into a rabbit hole very quickly.

Whether it's 2D drawings, PCB design files or CAD files, everyone has different processes and philosophies about revisions. What constitutes a revision bump? Do I have to let everyone know, even if it's a really small change?

In theory, of course, it would be best to track every single change, but as with testing documentation, this can prove to be a tedious and time-consuming task. Some people update the revision with every minor update, while others don't even start revisioning documents unless they are close to being released. In my opinion, the revision tracking process is dependent upon your team members and their needs.

As long as everyone on your team is on the same page and there is a system in place, it's easy to follow the process and guidelines and keep everything up to date. This, of course, means you need to come up with a system in the first place, and this on its own can be quite a challenging task. On one hand, you don't want to—for a lack of a better word—spam your coworkers or manufacturing partner with documents every single day. But on the other hand, you don't want to be complacent about changes either, as this can lead to issues such as modifications not being implemented, different drawings with the same revisions, and many more mistakes, missteps, and miscues.

Revision tracking also proves to be challenging when you have a lot of products and a variety of technical documentation that needs to be tracked. Certain projects can have one revision convention, while others may need something different. Some documents don't get updated as often, while for others, there aren't enough letter and number combinations to keep up with all the changes.

PLM systems and software can be helpful for tracking, but before you even get started with one of those systems, you must have all your documents with proper revisions, and set up in the first place, so that the system can track it for you. Anything we can do to keep the documentation up to date is very helpful when it comes to sharing files with contract manu-

facturers, new co-workers, or contractors; if it is managed properly, it is useful for avoiding problems in the future.

But getting to the point where everyone is used to the revision tracking system you put in place can be a long and challenging process. This is why it's best to start the tracking system from the early stages of design, even though that's not everyone's first thought while developing a new product. If you start early enough, by the time you're overwhelmed with the amount of work you must do, documentation becomes second nature. The truth is that this type of discipline and proper data management can help engineers and companies complete their goals and meet their objectives.

My philosophy is that we should leave our positions better than we found them. If we move on to another job or company, whoever comes after us can use the data and files we created to get familiarized with the job and start running with it right away. This is all achievable if you follow the rules that are set forth and make it your goal to not leave anything undocumented.

In the end, it really comes down to doing what works best for you and your design team. Develop a data management process, stick with it, and take responsibility and ownership for your portion of the work. There are many benefits when it's done properly and it's definitely better than the alternative. I'd rather struggle a bit in the beginning and take the time needed to get familiar with the system in place than start panicking at the last moment when documents are not up to date, and revisions are not matching up or files are missing.

As an anonymous engineer once said, "Control your data; don't let it control you." **DESIGN007**



**Tamara Jovanovic** is an electrical engineer with Happiest Baby, a Los Angeles-based developer of smart baby beds.

**Elementary, Mr. Watson**

# First, Component Shortages, and Now Hot Dogs?

by John Watson

When I considered the title for this column, I seriously considered calling it, "From the frying pan into the fire." I thought this because, as I'm sure you've noticed recently, the component shortage problem has only worsened; we're now seeing other supply lines breaking down. For example, the other day, I made my usual lunchtime run to a fine food establishment—the local Sonic—hoping to indulge in my customary Chicago dog. I know it's scary to consider Sonic as "fine dining," but imagine my horror when I heard that they were totally out of stock of hot dogs—of all things! Now I can handle component shortages and not bat an eye, but when something comes between my Chicago dog and me, that's where I draw the line. PCNAAlert connected to IHS Markit confirms that EOL notices for components are increasing at an alarming rate. Furthermore, IHS says, "The estimated cost to manufacturers for missing one EOL notice is \$20,000 to \$50,000." It seems everyone is feeling the pain.

Because of my position with Altium, I have the privilege of speaking with customers every day. A common theme I hear is that companies are making a considerable paradigm shift from engineering new product lines to supporting and sustaining legacy product lines. Several well-known companies informed me just this week that they have ceased all recent engineering efforts or will delay the release of new products and concentrate solely on keeping the legacy products alive. That is a direct result related to the component shortages. Keeping those legacy products lines viable is key to whether a company will survive.

To read this entire column, which appeared in the Design007 Week newsletter, [click here.](#)

# Impact of Filled Vias on Thermal and Signal Integrity

## Beyond Design

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

The growing popularity of wide bandgap (WBG) semiconductors, such as gallium nitride (GaN) and silicon carbide (SiC) has enabled components to achieve higher operating temperatures and power outputs than silicon-based technology. However, this has not eliminated the need for careful thermal management to evenly distribute the heat generated to avoid the formation of dangerous hot spots and to minimize power losses. The heat generated by integrated circuits (Figure 1) poses great challenges, especially given today's higher speeds, smaller board surface areas, and multiple devices populated on PCBs. These

demands call for solutions to effectively dissipate the heat and ensure the performance and lifetime of the electronic system product. One solution is to use copper filled vias which complement thermal management, but how do filled vias affect signal integrity?

The conventional plating process cannot close the via completely—there is always an air gap in the middle. Thicker barrel plating improves the thermal transfer and current-carrying characteristics of vias at DC and low frequencies. IPC-4761 Design Guide for Protection of Printed Board Via Structures Standard covers via fabrication and protection. There are seven categories, but just three basic types:

1. Tented via.
2. Plugged via.
3. Filled via.

All three via types can also be covered (on both sides); however, the filled via may also be capped. Capped vias are generally used for via-in-pad applications to prevent paste or resin flowing down the hole—the surface is planar and solderable. Typically, vias are plated over with electroless nickel immersion gold (ENIG). However, if via-in-pad is used, then ensure the connection to the planes uses thermal reliefs or you'll

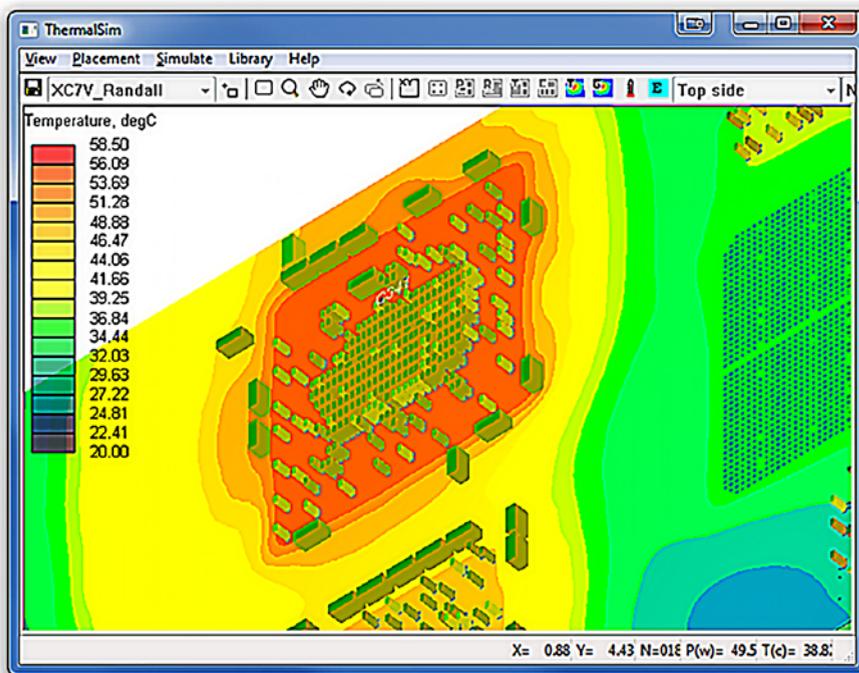
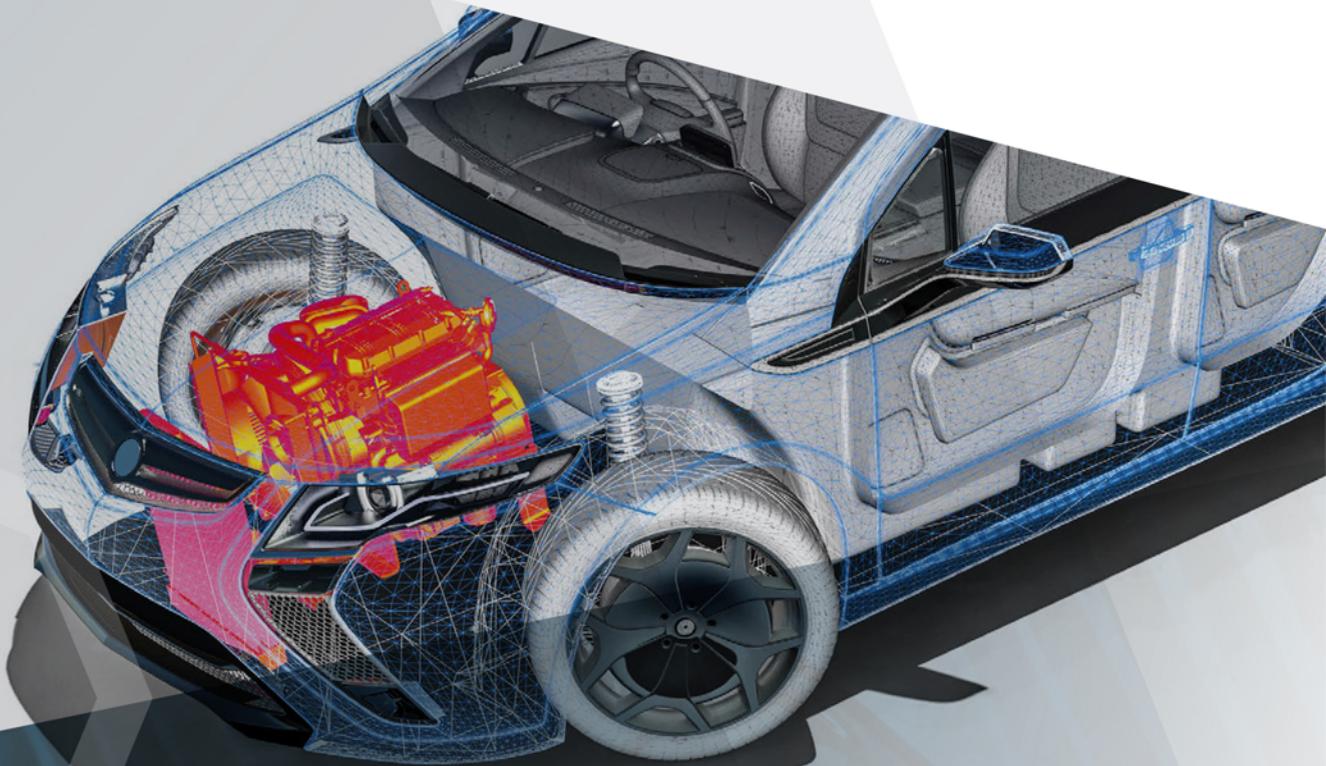


Figure 1: Thermal simulation of hot spots on the bottom of the PCB. (Source: Siemens EDA)



**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.



[ventecclaminates.com](http://ventecclaminates.com)

have a difficult time getting the device back up if it needs rework.

I have been tenting vias with either dry film or liquid photoimageable (LPI) solder mask for over 30 years. The basic reason is to seal the via hole for vacuum test fixtures and to prevent voids in the solder mask for silk-screening the component legend. Vias can be partially plugged with non-conductive paste but there is a chance of cracks from encapsulated gas during soldering. Or they can be filled with non-conductive or conductive materials.

Proper thermal management is necessary to keep each component within safe temperature limits. The junction temperature should never exceed the limit indicated in the manufacturer's datasheet (generally between 125–175°C for silicon-based devices). The heat generated by each component is transferred to the outside through the package and the pins. The two main techniques used to improve PCB thermal management consist in the creation of large ground planes and in the insertion of thermal vias. Thermal vias are used to transfer heat from one layer to another. Unfortunately, FR-4 does not offer high thermal transfer.

Copper-filled thermal vias provide an efficient heat dissipation path directly incorporated into PCBs with placement directly under a surface-mounted IC (the heat source). This allows direct surface mount bonding for maximum heat transfer using surface mount copper material. Specialized materials such as Kuprion's copper thermal via paste are capable of filling vias of at least 5 mm in diameter. When fused, the copper paste converts to solid copper without melting, which provides thermal conductivities in the range of approximately 110-180  $W^*m^{-1}*K^{-1}$  (watts per meter-Kelvin) and up to 290  $W^*m^{-1}*K^{-1}$  for microvias (up to 25 mil in diameter).

Engineers have debated the merits of hollow vias vs. solid vias for RF performance. There is a great temptation to believe that making the via solid will somehow reduce the equivalent inductance of the structure. However,

once the skin-effect kicks-in, in the megahertz range, then the current tends to only flow in the perimeter of the barrel.

Conductive epoxy-filled is the best practice for the vias to have a finished diameter between 8–18 mil. This allows the epoxy—conductive or non-conductive—to be pushed through the hole completely, but not to run out. The associated aspect ratio is best if the depth-to-diameter is less than 10:1.

A plate-shut process provides greater reliability for microvias, as opposed to filling with non-conductive ink and then plating over. Filling vias creates a solid core. This allows for a flat surface to be plated, and keeps the solder at the assembly level from leaking through and compromising the solder joint. This provides for the most reliable finished assembly.

The following materials can be used to ensure vias are sealed when filled:

1. A special plugging resin (e.g., Taiyo THP-100 DX1 thermally curable permanent hole-filling material) is suitable for plated through-hole and via-in-pad applications.
2. Copper: Classic copper via filling methods involve using pure copper to fill the hole.
3. Silver conductive epoxy resin: This is an alternative for filling vias but is expensive and copper works more effectively (e.g., DuPont CB100 or Tatsuto AE3030 screen-printed material).

PCBs that have copper-filled vias will stand up to the conditions presented by high power, radio frequency, microwave, and LED applications. The high-power integrated circuits that run these types of PCBs use currents that a copper-filled via can withstand, but not a plated through-hole.

A complete view of the current distribution can be visualized using a 3D solver. Figures 2 and 3 present three views of the conduction current at 18 GHz on a microstrip, hollow via, and solid via using Flomerics Micro-Strips, a 3D TLM solver: a) perspective view; b) top view of strip and ground plane; and c) bottom

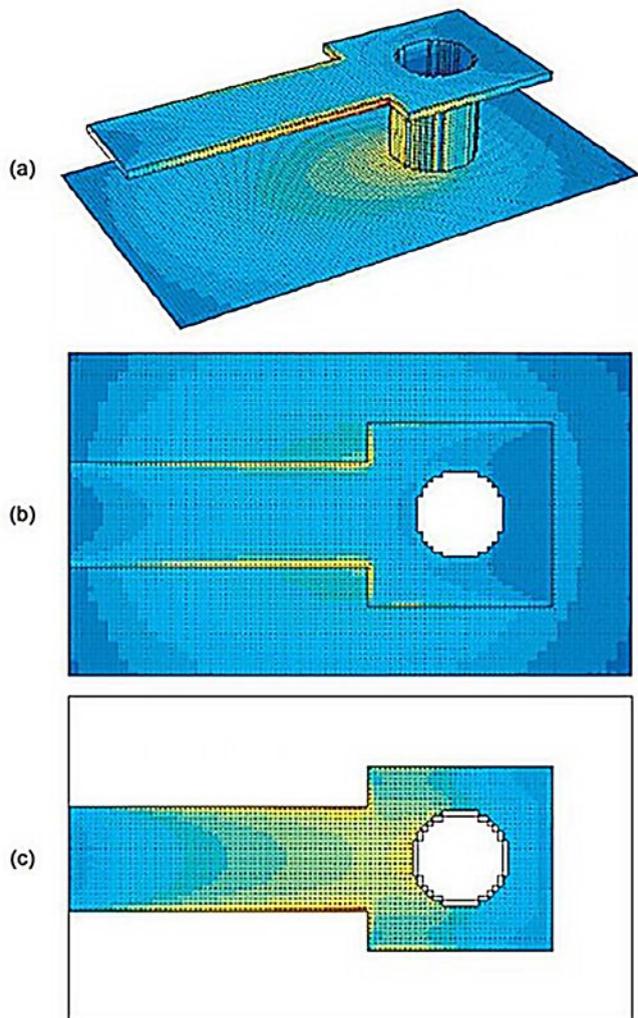


Figure 2: Via with hollow barrel.

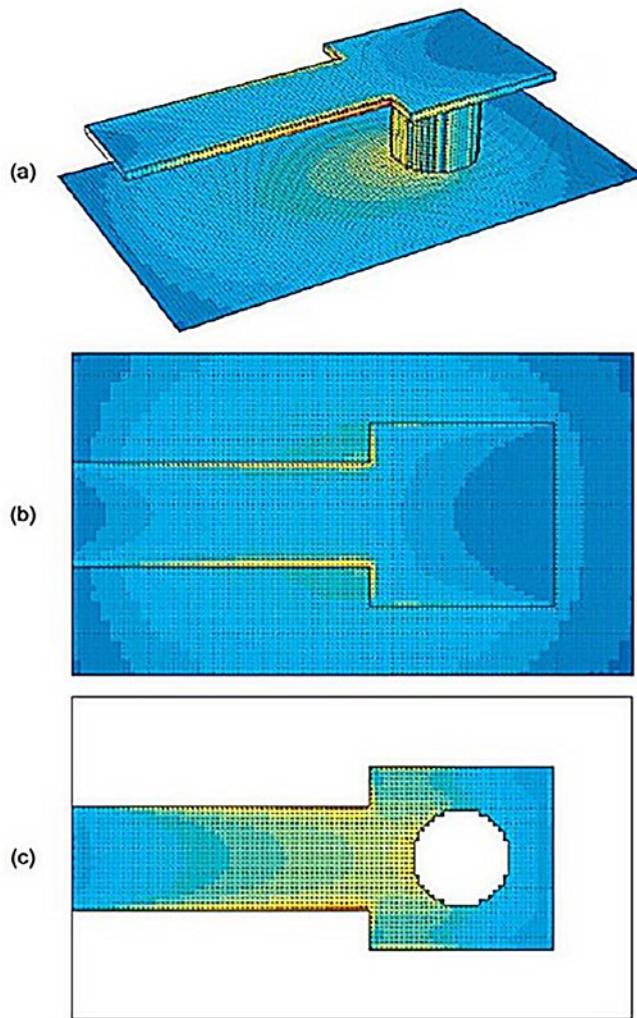


Figure 3: Via with solid barrel.

view of top strip. The scale in all three plots is 0 to 120 nA/m.

In both the hollow and solid barrel cases, the current is clearly flowing on the near side of the via barrel. There is also a small amount of current that flows down the inside surface of the via barrel. The return current in the ground plane can also be seen in Figures 2a and 3a. We tend to forget that microstrip is a two-conductor system—the return path is just as important as the signal path. Figures 2b and 3b show a top view of the complete structure. Note the non-uniform current distribution across the width of the strip. Finally, in Figures 2c and 3c, there is a view of the bottom side of the top strip.

Comparing the last two views we note that there is more current on the bottom side than

the top side of the trace. At low frequencies, the current would split more or less equally between the top and bottom surfaces of the strip. But, as frequency increases, the current distribution shifts toward the bottom side of the strip due to the proximity effect. The current distribution of the hollow and solid barrel has not changed significantly at 18 GHz, although this would be quite different at DC. The equivalent inductances for the hollow via and solid via cases are virtually identical. At 18 GHz, skin depth effects alone will force the current to the surface of the solid via.

Filled vias have little impact on signal integrity at high frequency. However, there is a definite benefit in using filled vias if thermal relief is the goal. At DC, filled vias also provide

increased current capacity but at 18 GHz both filled and hollow vias perform much the same.

## Key Points

- Copper-filled vias complement thermal management.
- The conventional plating process cannot close the via completely.
- Thicker barrel plating improves the thermal transfer and current-carrying characteristics of vias at DC and low frequencies.
- Thermal vias are used to transfer heat from one layer to another.
- FR-4 does not offer high thermal transfer.
- Solid vias do not reduce the equivalent inductance of the structure. The current tends to only flow in the perimeter of the barrel due to the skin effect.
- Conductive epoxy-filled is the best practice for the vias to have a finished diameter between 8–18 mil.
- Filling vias creates a solid core and allows for a flat surface to be plated. This keeps the solder at the assembly level from leaking through and compromising the solder joint.

- The equivalent inductances for the hollow via and solid via cases are virtually identical.
- At 18 GHz, skin depth effects alone will force the current to the surface of the solid via. **DESIGN007**

## Resources

1. [Beyond Design: The Proximity Effect](#), by Barry Olney, *Design007 Magazine*, March 2019.
2. “Microwave Circuit Modeling using Electromagnetic Field Simulation.” D. Swanson and W. Hofer, *Guide to Copper-Filled Vias, Process & Benefits*.
3. *Copper & Epoxy Filled Vias*, Cirexx International.
4. “High-Power Dissipation Copper Filled Thermal Vias by Kuprion,” *Power Electronics News*, April 23, 2021.



**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](http://www.icd.com.au). To read past columns or contact Olney, [click here](#).



We are dedicated  
to excellence through

# innovation, technology and most importantly, service.



COMMERCIAL • MILITARY • MEDICAL • BAREBOARD



**U.S.CIRCUIT**

[www.uscircuit.com](http://www.uscircuit.com)





# PCB007 Highlights



## **Trouble in Your Tank: Case Study— Interconnect Defects and a Few Other Problems** ▶

For this month's edition, we are taking a slightly different approach—that of presenting an actual case study. However, the basic principles of these columns continue.

## **LPKF Names Dr. Klaus Fiedler New CEO** ▶

Dr. Klaus Fiedler will become the new chief executive officer of LPKF Laser & Electronics AG. The Supervisory Board of the SDAX-listed technology company has appointed the top manager to join CFO Christian Witt in the Management Board, effective 1 April 2022 or earlier, with a three-year contract term.

## **SÜSS MicroTec Continues to Grow, Has Record Order Entry in 3Q** ▶

According to preliminary figures, technology company SÜSS MicroTec SE has received ca. EUR 101 million incoming orders during the third quarter of 2021.

## **Insulectro Advances Its Digital Transformation With All-New Website** ▶

Insulectro, the largest distributor of materials for use in the manufacturing of printed circuit boards and printed electronics, has launched a new, “from the ground up” website.

## **ILFA Places Multi-Million Order for UCE Machines With Viking Test Ltd.** ▶

Modernization at Hanover-based PCB manufacturer ILFA is progressing steadily. With the placement of an order worth several million euros, Viking Test Ltd. has been

contracted to supply state-of-the-art wet machines from the Asian market leader UCE for production.

## **Ventec Completes Asset Purchase Agreement With Holders Technology in Germany & UK** ▶

Ventec International Group Co., Ltd., is pleased to announce that it has finalized an asset purchase agreement with Holders Technology in Europe on October 21, 2021.

## **Luminovo Acquires PCB Platform Provider Electronic Fellows** ▶

Luminovo, Munich-based software company building products to digitize the electronics value chain, acquires the Wiesbaden-based startup Electronic Fellows.

## **Sunstone Circuits Recognized With Cigna Well-being Award** ▶

Cigna has selected Sunstone Circuits as a recipient of its annual Cigna Well-being Award for demonstrating a strong commitment to improving the health and well-being of its employees through a workplace wellness program.

## **Elmatica Announces Promotion of Robert Kurti to Chief Information Officer** ▶

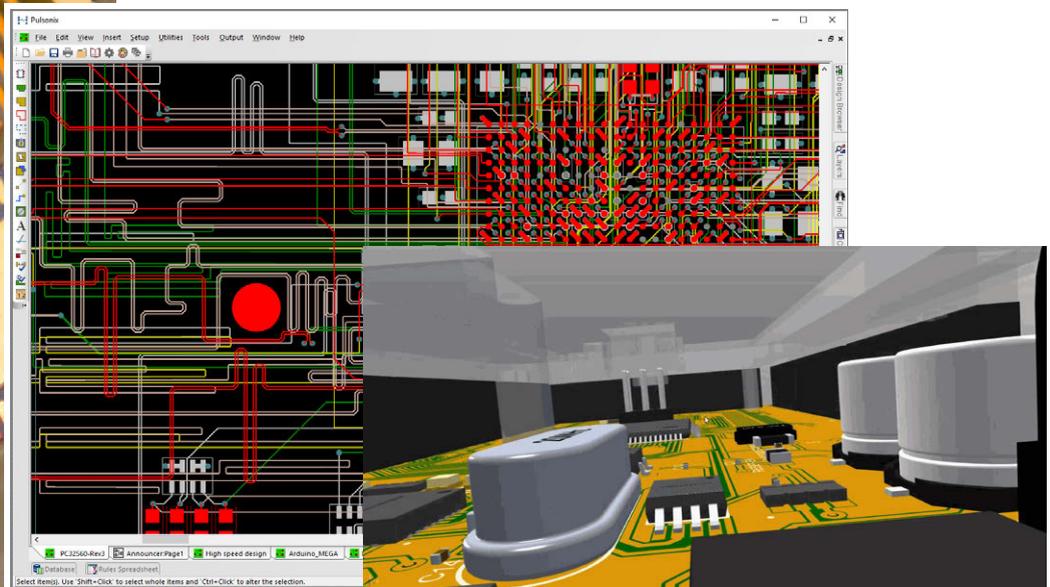
Elmatica, a global actor in the printed circuit industry, announced the promotion of Robert Kurti to chief information officer. Kurti joined Elmatica in 2016 as IT manager after supporting the U.S. and NATO coalition forces for six years.

# 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](http://pulsonix.com/trial)





# DFT and Test Strategies Shouldn't Be an **Afterthought**

Feature Interview by the I-Connect007 Editorial Team

Over the years, there has been a lot of focus on the “design for,” primarily design for manufacturing (DFM) and design for assembly (DFA). For many designers, design for test (DFT) is an afterthought, if it even enters the conversation during design planning.

The I-Connect007 Editorial Team recently asked William Webb, technical director of test software provider ASTER Technologies, to discuss the company’s approach to test, the technological advances that are driving changes in DFT, and what PCB designers need to know to plan their DFT strategies.

**Nolan Johnson:** Planning for design for test must have some influence on what happens on the manufacturing floor in test and inspection. And, of course, the manufacturers see all that equipment as the cost of doing business. It’s not a value-add. Does this match up with your perspective?

**William Webb:** Yes, it does, and you’re right. A lot of times, it can be hard to make a DFT tool sale to an EMS company for the reasons you said. We have many EMS customers, but in initial discussions they sometimes don’t see the value straight away. DFT can be seen as just a cost to them, whereas an OEM has more interest, if you will, in buying a tool like ours.

If you have a test escape that you don’t know or understand, and it’s giving you a production yield hit, then you’ve likely got product coming back as a field return, and that’s a high cost to the EMS provider. So, the EMS provider can use a tool like TestWay to identify devices that are not covered properly, or it helps you identify your DPMO (defects per million opportunities) that you weren’t aware of, which is leading to a yield hit and will ultimately be a cost for the EMS provider. Once we’ve explained this benefit to EMS companies, they usually understand the potential savings. It’s then easier for them to put a return on investment (ROI) together for management, to show that they can easily pay for the tool if

# FRUSTRATED DEALING WITH MULTIPLE SUPPLIERS?

Hundreds of customers have taken advantage of our  
**"One-Stop PCB Solution"**

**Get a FREE SMT stencil on your 1st Assembly order**  
*(when you mention this Ad)*

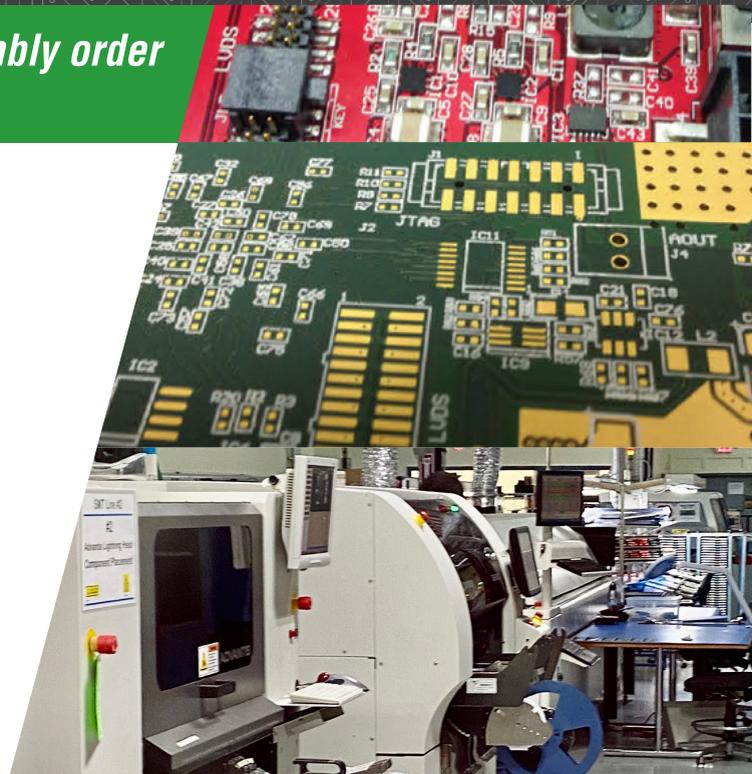
## STRATEGIC SOLUTIONS

- ✓ Avoid Communication Gaps
- ✓ Prevent Engineering Delays
- ✓ Single Point of Contact from Sales thru Delivery
- ✓ Streamlined Process-Fab to Assembly
- ✓ Consistent Quality Results

## PRECISION ADVANTAGE!

For the past 25 years, we have been helping PCB Designers and engineers avoid unintended design flaws and prevent costly errors. With our technically knowledgeable team and expertise in time-sensitive, quick-turn PCB Fabrication and Turnkey PCB Assembly services, we guarantee the highest quality and reliability of your products.

Let us help you get your products to market today!



**GET YOUR FREE INSTANT  
ONLINE QUOTE NOW!**





William Webb

they can improve the yield by some amount, and they have less product coming back as a field return. That's one way to do cost justification. I don't know if that totally answers your question, but it's a discussion that we have and we've been able to lead EMS providers to see the light with that type of a discussion, whereas an OEM is typically easier. They have an easier time understanding that they're going to get a benefit from DFT and test strategy simulation right away.

**Johnson:** Is there a point where an EMS company can require this pre-work of their customer?

**Webb:** Yes. We've seen them say, "We don't want to accept the board design to build, unless it's been qualified." If proper DFT and test coverage analysis hasn't been done, then they know that's going to cause them trouble, it's going to cause their engineers more time, and it's going to be a more difficult product for them to build. They may not necessarily be able to charge more money to their customer, just because it's more difficult for them to build.

**Andy Shaughnessy:** How does that software work?

**Webb:** Essentially, the TestWay tool allows you to close the feedback loop from your design clear through your manufacturing and production phase. In the old days, the designers operated in a silo. They would often throw the design over to the test engineers when they were done with routing/layout. Test engineers then reviewed the design, and maybe found some potential issues and possible improvements. However, in many cases it was too late to fix these problems. There was no continuous feedback loop built into the process. TestWay allows you to quickly analyze the design, even early in pre-layout phase, and give feedback to the board designers about what they could do with the design to make it more testable, and where you need (or don't need) the test points because, perhaps, you're going to test that circuitry with some other method.

Then as you go into manufacturing, you can extract and examine the failure data. And you can have a better understanding of your DPMO. It can help you the next time you look at designs so that you're continually driving improvement. Now, rather than test operating just at one point, you have a nice, continuous feedback loop clear from the design through DFT, test strategy estimation, test program creation, test program measurements, and even field returns. Because you need to understand what your field returns are, you can ask, "Are we getting these back because we're missing some test coverage? Is there some test coverage that's really not where it should be?" That helps you get better and drive improvement.

**Johnson:** Where should a software tool like this be utilized? Because both parties, OEM and EMS, benefit from this.

**Webb:** That's right. It's something we've thought about and refined so we could have those discussions when an EMS provider or

OEM comes to us. If both the OEM and EMS provider are using the tool, it's a great sharing of data. You also touched on where we fit in the design world. Typically, we sell into test groups, test engineers, and that realm, but we also have started our initial talks with the design engineers, and with the quality teams. All these groups have a potential interest in a tool like TestWay.

**Shaughnessy:** PCB designers don't mention DFT very often. What are some of the most common problems that you wish designers would do or stop doing? What advice would you give designers?

**Webb:** The typical problem is usually that they don't know how you're going to test their board, and what strategies will be used. Maybe they don't understand that you're going to test the board with boundary scan, for example. The designers may go right into the layout phase and put test points where they think they can get them in. If they don't have that understanding of how you're going to test the board and the methodologies you're going to use, they may go off and not really understand the best locations and need for the test points.

Of course, these days everybody is dealing with shrinking boards, so real estate is always at a premium, and nobody generally has 100% test point access anyway. But there are other test methods being used today that don't need as much access. We have customers who are doing more flying probe testing than ICT. Flying probes are getting more accurate and faster. Again, this means you may not need a test point if you're going to probe a component pad or something like that. It's just that lack of communication between the designer and the test guys that can lead to issues. If these groups can work more closely together, which a tool like TestWay allows them to do, they will ultimately achieve a more testable design with better coverage and less cost to produce.

**Johnson:** Realistically, where in the design process should the testing methods be defined? It sounds like the specifics need to be figured out before you start doing layout.

**Webb:** That's absolutely correct. Most of the time, when people talk about DFT they instantly think about layout and accessibility, but there's a lot you can do while looking at the design early on before layout. That was the key point that ASTER did initially before we started working with test coverage and all the other modules that we now offer.

---

**Most of the time, when people talk about DFT they instantly think about layout and accessibility, but there's a lot you can do while looking at the design early on before layout.**

---

You should start very early in the process. The test guy should be looking at the analysis when schematic capture is in process, and that's when you can make some big impact on testability. If you give quick feedback to the designer and say, "Oh, don't do it that way, do it this way," you will have a more testable board that works well.

**Johnson:** There are plenty of CAD tools out there, plenty of environments and workflows that get put together by an OEM or an EMS provider. How agnostic are you?

**Webb:** We can import almost any industry standard schematic net list file. That's a file that's available before layout. And it's the same with the layout files—whether it's ODB++, PADS, Zuken, Altium, or whatever, pretty much all

the major players—we can import the data and work with it. So, we’re not limited in that sense. Also, TestWay can actually be a plug-in for Altium. So, if somebody’s using Altium and they have TestWay installed on that same computer, they see a plug-in show up in Altium and they can run some of the electrical design checks right from Altium. So, the designer doesn’t even really need the test guy. He can click the button and get feedback right there while he’s working in that environment. From the test equipment side, we are also quite agnostic because we support the ability to read in actual test coverage from over 60 different machine types, and we also produce the output files (CAD2CAM) for 50 different machine types.

**Shaughnessy:** One hacker I interviewed said test points are always risky. He said, “If I get hold of your circuit board, I can go in through the test point and find out everything about the board and maybe the final product.” Is security really a big consideration?

**Webb:** Security is certainly becoming more of a concern to our customers. I’ve heard concerns about the boundary scan file, which would allow somebody to get into the part itself. More people are securing their boundary scan files these days. No, I haven’t heard anybody’s concern about a test point, but I can understand it. Somebody could probe that in the field and maybe they could find out some information that they otherwise wouldn’t be able to. It’s another reason test points will probably get more limited going forward on certain design types. Security is becoming more of a concern with our customers, and we’re hearing more about it. We don’t currently have a module or anything that addresses that, but it’s certainly something we’re thinking about as we hear people talking about it.

**Johnson:** Thanks for speaking with us, Will.

**Webb:** Thanks for the opportunity. I enjoyed it.

DESIGN007

---

## Nanoscale Lattices Flow From 3D Printer

Weaving intricate, microscopic patterns of crystal or glass is now possible thanks to engineers at Rice University.

Rice materials scientists are creating nanostructures of silica with a sophisticated 3D printer, demonstrating a method to make micro-scale electronic, mechanical and photonic devices from the bottom up. The products can be doped and their crystal structures tuned for various applications.

The study led by Jun Lou, a professor of materials science and nanoengineering at the George R. Brown School of Engineering, appears in *Nature Materials*.

The lab uses a two-photon polymerization process to print structures with lines only several hundred nanometers wide, smaller than the wavelength of light. Lasers “write” the lines by prompting the ink to absorb two photons, initiating free-radical polymerization of the material.

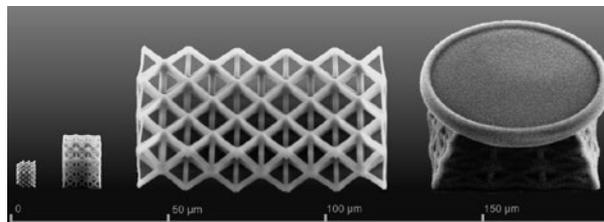
“Normal polymerization involves polymer monomers and photoinitiators, molecules that absorb light and generate free radicals,” said Rice graduate student and co-lead author Boyu Zhang of the process that commonly uses ultraviolet light in 3D printing and to cure coatings and in dental applications.

The printing process required the Rice lab to develop a unique ink. Zhang and co-lead author Xiewen Wen, a Rice alumnus, created resins containing nanospheres of silicon dioxide doped with polyethylene glycol to make them soluble.

After printing, the structure is solidified through high-temperature sintering, which eliminates all the polymer from the product, leaving amorphous glass or polycrystalline cristobalite. “When heated,

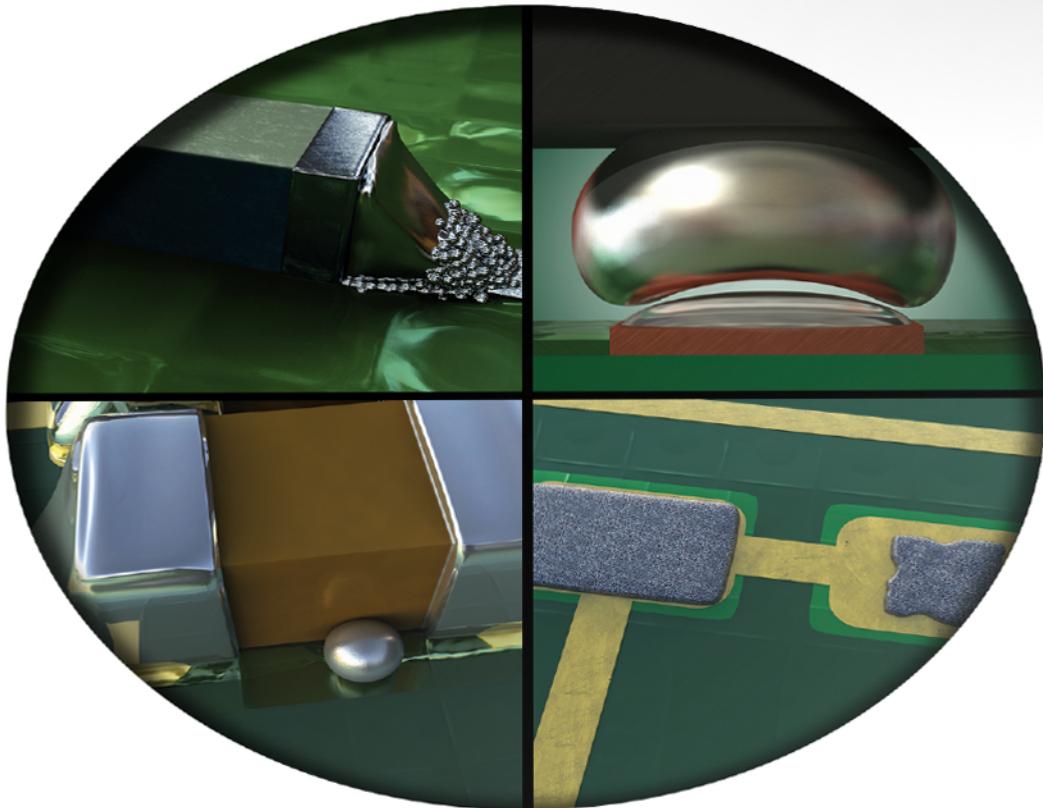
the material goes through phases from glass to crystal, and the higher the temperature, the more ordered the crystals become,” Lou said.

(Source: Rice University)



# THE PRINTED CIRCUIT ASSEMBLER'S GUIDE TO...™

## SOLDER DEFECTS



Christopher Nash and  
Dr. Ronald C. Lasky  
Indium Corporation

1007e  
Books

Available Now  Download Here

# Say ‘No’ to File Hoarding: Data Management Tips

## Tim’s Takeaways

Feature Column by Tim Haag, FIRST PAGE SAGE

According to the Internal Revenue’s online resources, you should keep your tax returns and associated financial documents for three years. This goes up to seven years if you’ve filed a claim for worthless securities or bad debt, but for most of us, three years is the max. Now, in case you are wondering why I’ve started out with such a dry and boring fact, it’s because it uncomfortably applies more to me than probably any other person in the world. Yes, I admit it, I’m a tax return hoarder.

I’ve tried to tell myself over the years that keeping those documents was an act of preserving history, and from time to time they actually have provided some entertaining moments. For instance, looking back on our incomes from 20 or 30 years ago, or finding out how little we used to pay for health insurance, has made for some interesting jaunts down memory lane. The truth, however, is keeping those old documents was just plain laziness on my part. It was a lot easier to collect those documents every year, bag ‘em up, and toss them in

the back of the upstairs closet where I didn’t have to think about them anymore. Unfortunately, my tax hoarding habits have produced some real problems:

- The closet was running out of room
- Critters tend to love piles of paper like this
- My family was getting increasingly annoyed with the mess
- All of that paper had become a potentially dangerous fire threat that kept growing each year

To summarize, I had convinced myself that the massive amount of data that I had collected over the years was essential for one reason or another. In reality, my collection was nothing more than an unmanage-

able mess that served no useful purpose. Fortunately for me, this particular problem was resolved by a trip to the recycling center and \$30 worth of paper shredding services. But there are a lot of different types of “collections” in life that need managing, and like my proliferating pile of paper publications, they all need their own





# APCT

Passion | Commitment | Trust



## Long Standing Supplier Partnerships Reassures Our Customers That APCT Can Deliver

Fears of Supply Chain Interruptions Fade Amidst Long-Invested Relationships

### PARTNERSHIP BRINGS

- Preferred Vendor Positioning
- Preferred Pricing
- Saved Inventory
- World-Class Service
- Quicker Response Times
- World-Class Cycle Times

Advanced Technology | Cycle Time Speed | Knowledgeable People

**512.548.7435 | APCT.com**

Printed Circuit Board Solutions

APCT.com

APCT Santa Clara HQ

408.727.6442

APCT Anaheim

714.921.0860

APCT Orange County

714.993.0270

APCT Wallingford

203.269.3311

APCT Global

203.284.1215

eloquent solutions to keep from getting out of control. Take, for instance, the amount of data that is generated during the design of electronics.

## PCB Design Data

The first thing to consider in our world of PCB design is just how much data there is that needs to be managed. From a casual overview it may not seem that extensive, but let's break the average design down into its four separate pieces. This gives us the schematic, circuit simulation, PCB layout, and analysis, and that is just a generalization. Designs often have more pieces than that in them, especially when you consider the depth of system level design. But starting with the basic four pieces, we can break those up even further:

- Design databases and associated system files
- Library files and sub-library element files
- Read-me, template, and example files
- Reports, standards, specifications, special instructions, etc.

The main four pieces of design data will have their own output or interfacing files. For example, let's examine the PCB layout data in greater detail. Circuit board manufacturing requires a lot of different design data including:

- Artwork, drill, apertures, and other fabrication files
- Drawings, component XY locations, artwork, and other assembly files
- Netlists, test point XY locations, and other test files

These items are just a sampling as the average design will produce much more data than what I've listed here. Of course, all this must be multiplied for design revisions, backups, or alternate part numbers and board configurations. As you can see, the amount of data in a PCB design can easily stack up and create plenty of unexpected problems along the way—just like my collection of tax documents had.

## Pitfalls of Data Storage

First, this data requires storage space. Everyone knows that, with all the advances in technology, our storage systems are larger now than they've ever been before, which is a great thing. However, what is easy to forget is how our databases and file sizes are also getting larger, and the amount of data that we are saving increases all the time. A friend once warned me, as we were moving someone else's "stuff" into a larger facility, "People are like fish, they grow to fit the tank." Ironically, this rather obscure law of nature seems to fit PCB design data as well. As soon as the data storage is upgraded, someone just fills it back up again.

Another problem to contend with when dealing with excessive amounts of data is confusion. It is not unusual for PCB designers or design groups to encounter one or more of the following problems:

- Which design is the most current?
- Incorrect databases being selected for work by designers
- Essential design data stored on personal drives where it is inaccessible to others
- Open file permissions allowing for critical data to be altered, corrupted, or over-written

Interestingly enough, in some cases, just the opposite is happening; not enough data is being saved and managed. Here are some of the problems that we've heard reported:

- Original design history and data that is permanently lost
- Missing changes or revisions creating a disconnect between design and manufacturing
- Inability to produce traceability documentation and data for regulated or controlled products
- Having to recreate databases from artwork, or completely redesign a PCB instead of simply rolling a board revision

Obviously, there can be a lot of problems associated with the storage of data generated

during the PCB design cycle. The best solution is to incorporate proper management processes to keep all the design data in check; we'll look more into how to do that next.

## Data Management

The first step of good data management is to decide who owns what. This pivotal point can actually end up becoming a sticking point for some design groups, as data ownership gets swatted back and forth between personnel like a shuttlecock in a badminton game. The important thing to remember here is that it isn't nearly as important who owns the data as it is that the data is actually owned by someone. In other words, if the design group is going to own its own data, that's fine. If, on the other hand the company's IT department is going to own the data instead, that's fine too. As long as someone has the responsibility for ensuring that the design data is available, secure, and part of the regular workflow of the design group, the actual ownership isn't as important. So don't let the little detail of ownership derail anyone from setting up proper data management processes.

Speaking of those processes, it is essential to have good data management processes in place. These processes need to clearly outline who is responsible for data ownership, as well as what each individual user's data responsibilities are. Additionally, file security protocols should be in place, and the processes should specify generation and storage procedures for users to prevent data losses or corruption. The processes should also outline what data should be generated for specific vendors, and what the standard manufacturing file sets should contain. These data management processes should also detail the company's archival and vault procedures, and where to go with additional questions.

Another important part of effective data management is to ensure that the right data is being created and collected in the first place. As we mentioned earlier, a lot of data can be

generated by the PCB design process, and you don't want to get swamped by every single sub-library or system file in your design directory. On the other hand, you don't want to casually delete a seemingly innocuous file that actually contains some key traceability data in it either. So how can this dilemma be managed efficiently? While it may seem like data hoarding is the right way to go, we also know that it can create its own mess of problems. Thankfully there appears to be an answer to CAD file organization in some new design data automation tools.

Many design teams are turning to database solutions for file generation such as the IPC-2581 file format generators. CAD systems using this format will automatically add the necessary fabrication and assembly files into an IPC-2581 formatted database, making output file generation for your manufacturers quick and simple. As Patrick Davis, a product management director for Cadence Design Systems, said in the [September 2021 issue of Design007 Magazine](#), "I don't need dumb data. I need something that allows us to actually communicate back and forth efficiently." With tools like IPC-2581, you can be sure that all the data that is needed for manufacturing is included in one easy-to-access database.

Whether you use design automation like IPC-2581, or you come up with your own data management processes and procedures, the important thing is to keep at it and not be overwhelmed by design data. With the proper data management and a clear delineation of who is responsible for what, you can keep the piles of data under control. Until next time, friends, keep your file systems neat, clean, and organized; and of course, 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](#).

# 45+ Years of PCB Manufacturing Experience



Kim O'Neil

Interview by Barry Matties  
I-CONNECT007

I-Connect007's newest columnist, Kim O'Neil of Prototron, shares his insights from more than 45 years of PCB manufacturing experience. Kim explains why it's so critical for designers to begin working with their fabricators as early in the process as possible. He also details how managing and measuring design data has cut the number of errors over the years.

**Barry Matties:** Kim, you've started writing a new column with I-Connect007 called Circuit Chronicles. Tell us about the basis of this column.

**Kim O'Neil:** I'm looking at it from the perspective of a small business, a small fab shop operation, and my experience over the 45 years that I've been involved in printed circuit boards, trying to draw on that and lay down some feelings I have, some ground rules, and what I think comes to the job.

**Matties:** What are some of the angles or the important things that customers need to know from your point of view?

**O'Neil:** They need to know that they can call me. The communication between the fabricator and the customer is critical. We deal with a lot of contract manufacturers, educating them on what we do and don't do, and talking to them on a personal level. You're not talking to an automated phone message, and this personal interaction really establishes a business relationship.

Dave Ryder, president of Prototron, had a philosophy, and we've carried it through. We have a point of contact who stays with customers for years. You won't just get the next person when you put an RFQ in and establish that relationship. There's a very good communication network between the customer, inside sales, and our operations managers—Caesar Cruz and myself—about what we're really dealing with. Can we do it or not? Those types of things. We help the customer by educating them all the way through the process.

# 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](https://www.siemens.com)

**Matties:** We often hear that the designers don't fully understand the fabrication process. I think that your column is in a great position to help them understand the critical areas they need to know.

**O'Neil:** Right. Years ago, there was a disconnect between designers and fabricators. As the big shops have closed, front-end engineering has gone away for a lot of these folks, and those who have worked in the industry have left, so there's a gap. Folks who come straight out of school and then go down the street to the big tier-one guy here in Tucson [Arizona] don't really get any kind of education about building a circuit board. So, they come up with some radical and funny ideas of how you go through this process. We encourage those folks to come to the shop or at least pick up a book, like Dan Beaulieu's PCB 101. We hand that book to new employees, platers, and anybody who walks in the door.

---

## Years ago, there was a disconnect between designers and fabricators.

---

But every time we tour either a graduate student, an EE, or a fresh-out engineer, they're always amazed at what actually happens in a printed circuit board shop. To get that basic with designers is critical so they don't have to keep coming back asking questions. We welcome questions because that gets the product out faster, but we aim to establish that base level about what we really do here and what it takes to make a circuit board.

**Matties:** They may not understand the 300 steps of complexity.

**O'Neil:** Absolutely. They come out of our tour with their eyes wide open.

**Matties:** How important is it for the designer to really understand the manufacturing process?

**O'Neil:** It's critical. Besides designing the board, we go through design for manufacturing (DFM) in the front end to determine whether we can build the board before we quote it. But if the designer had insight into what it takes, routing, drill hole size, those things would be taken into consideration right off the bat instead of designing the product and then realizing his fabricator can't do that. "Let me go over to 'A.' They can't do that. With 'B' and 'C,' there are some real restrictions, not only with shops, but the equipment they have to do that type of fabrication." So it's critical, absolutely.

**Matties:** Right. Now in your column, you will touch on some of these topics that are most critical for the designer to learn or understand.

**O'Neil:** That's correct. The number of layers has gone down because of part sizes, but real estate is very critical. How you utilize it is so critical for what happens on the shop floor.

**Matties:** So, in these 40+ years, what have been some of the best memories or the most surprising moments in the industry for you?

**O'Neil:** I have seen technology change, and the board design change. Every six months another design was coming out, and the quick turn, and how just-in-time manufacturing made it down to the circuit board level. I have worked at OEMs before, and you weren't even considered anybody in the organization, you were just there. When it went to independent shops, there were some guys, bucket shops really, that suddenly had to get with the program. Surface mount was the one that really got everybody going.

I was talking to a quality engineer the other day, and he goes back to when CAD and CAM

first started coming out. When they started going from double-sided to a multilayer, it was like, “How many layers can you do? We do up to 22 here.” He just lost his mind. He didn’t understand that. From the standpoint of real manufacturing coming out from what felt like the Stone Age, educating the shops to where we are today with TQM and our quality management system, it’s a business management system.

It’s how we run our business. And that was a flip. When someone comes out to audit us, we don’t look at it as, “Oh no, here comes an auditor.” No, we look at it as someone with knowledge who will look at our system and say, “You can improve here, you can improve there.” Because if you work in it, you might not see the forest for the trees. We are the best there can be until you have a third party come out and say, “Hey, just a minute here.” It used to be that you sweated an audit. That’s not the way we look at it. It’s improving our business system.

For a small business, it’s really critical because resources, especially today, are in short supply. Even in our supply chain, there is short supply everywhere. You had better know everything about the business; you’ve got to have a handle on it.

**Matties:** Let’s talk about process improvement. We live process improvement in our own organization because business is nothing more than one process connecting to another. It’s that critical 15%, first 15% handoff—if you don’t get that right, it’s all wrong, right? Then you have to be focusing on the critical 20, because obviously 80% of your results come from the 20% of the inputs.

**O’Neil:** That’s how I look at it here. You can’t control every little thing, but you’re right, 80% of your results come out of 20% of those decisions that you made. Same with cost: 80% of the cost, 20% of the items out there. You control those, and you’re in good shape.

**Matties:** Until you stop and question, “Why

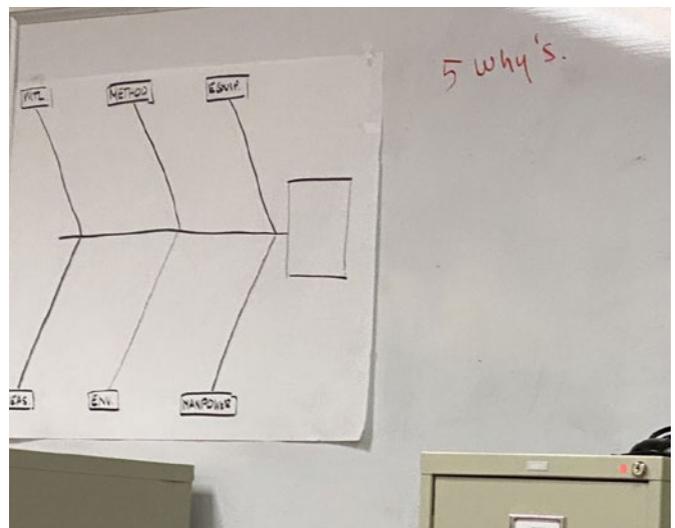
do we do it this way?” You don’t know, and all this waste is inherently built in, and you just live with it because it’s the way we’ve always done it.

**O’Neil:** Yes, very true. For example, looking at when we started to develop into the QMS. My office is situated as such that you come through the door out of final inspection, you hang a hard right, and you head into inside sales. Our lead inspector, Chito, was making that trip six to seven times a day. I could see him doing it, and I wondered why Chito was always running up to inside sales. It was simply that he had a print, the print didn’t match what was going on with the board and traveler, and he had to ask inside sales what was right. I thought, “Call the customer. That’s right at the end of the process. You’re going to ship that board that day and everybody’s scrambling around.”

**Matties:** It’s much too late.

**O’Neil:** Yes. It was way too late in a process to do that. Part of our quality objectives was to ask, “Why do we do it this way?” We got the inside sales and final inspection folks together. We did the simple fishbone analysis, which is one of the best and simplest tools to use.

**Matties:** Well, I see a fishbone on the wall right over your right shoulder. And the “five whys.”





**O'Neil:** We use that and it works. We came up with a quality objective of conflicts, which is what we call it when it doesn't match what you say it should. When we started, if you can believe this, we had 184 conflicts. Chito was making us sprint there every day—we had several little items, all the way from wrong thickness to something was wrong with the photoplot sheet, and this, that, and the other thing. Last month, we went through our TRB meeting and we had four things.

**Matties:** Now, when you calculate the savings in time, that's one thing, but it's also the wasted or lost opportunity that is hard to measure. But in terms of hours, how did you quantify that to say it was a success? People have to feel it so it can become contagious. That's probably the wrong word to use in the pandemic, but it becomes a way of life.

**O'Neil:** It was the number of footprints on the tiles from Chito; it got less. We noticed less frustration and how the two groups began to work together. Throwing it over the wall and someone had to catch it on the other side? Well, the wall came down.

**Matties:** You're making an important point that you're all one team, and inside the team, you have internal customers. If your goal is 100%

customer satisfaction, it's not just external; it's got to be internal, too. It's important for the supplier of your internal process customer to ask what the requirements are and the form that they want it in. The more we do that, the smoother the process goes, and the more we learn where the resistance is, or the problems could occur.

**O'Neil:** Right. What came out of that is not just the end of the journey. We came out with different checklists. Our photo platform was extremely detailed, and several other little things that would go down the line. Awareness was those details. When we do a quote, it's not a 15-minute quote. Maybe it would be for something we've previously done, but generally, we're going to make sure that when that quote goes out, it's as detailed and as comprehensive as we can get it.

Sometimes with specialty material, which is part of our gig here, it will take eight or nine hours because of all the people that we need to touch on the outside. But if that quotes right, obviously for the customer, that's that customer. But when it comes back in a PO, it doesn't have to sit in the front end and ask questions anymore; it gets released. That's another one of our quality objectives: New jobs have a measurement on them, and how long does it take from PO received to on the floor?

**Matties:** What sort of gains have you made since you've started making measurements?

**O'Neil:** Shipments have improved; obviously the time to get that job on the floor has vastly improved. In the details we have, mistakes have gone away; we measure those types of things, like conflicts, and what that does is create time, fewer quotes waiting at the end of the day.



**productronica**

Munich - Germany  
16th - 19th November 2021  
Hall A4 - Stand 466



# Thermal management solutions that perform when the heat is on

With a consumer requirement for ever-more diminutive devices and an expectation of improved efficiency and power, effective thermal management materials have become an increasingly essential part of product development.

From bonding and non-bonding thermal interface materials, to thermally conductive resins, our solutions offer the ultimate level of protection and heat dissipation

With an expansive product range and a strong emphasis on research and collaboration, we provide a complete electro-chemical solution to the world's leading manufacturers across a variety of industries.

Isn't it time you discovered how Electrolube can serve you? Simply call, or visit our website.



Scan the code to discover our full spectrum of superior Thermal Management and electro-chemical solutions.

Tel: 800 474 1472  
[www.electrolube.com](http://www.electrolube.com)

## ELECTROLUBE



Electronic & General Purpose Cleaning

Conformal Coatings

Encapsulation Resins

Thermal Management Solutions

Contact Lubricants

Maintenance & Service Aids

**Matties:** You were talking about a lot of points of contact in the quoting process, and one is the supply. Supply lines are challenging these days. How are you dealing with that?

**O'Neil:** It's difficult.

**Matties:** Day by day.

**O'Neil:** It really is. Our laminate vendor ships prepreg to us with dry ice because it's too expensive for refrigerator trucks from California. She called me up and told me there is no dry ice available. I said, "What? CO<sub>2</sub> gas is not available? How could that be?" So we came up with a different way to pack with regular ice, and it worked, but that's a challenge when you're talking about a huge supply chain.

**Matties:** These are the kinds of supply chain issues that you don't think of, like dry ice.

**O'Neil:** It's not really a go-to, no. But you must adapt and be ready for those types of things. Every day it's a different thing with price increases along the way. I just got one from a welding place where we get our gasses for plasma; the price has gone up. There's no CO<sub>2</sub>; there's oxygen going to the hospitals and whatever. It's just a chess game that has become that three-tier thing that you used to see on Star Trek. It's not easy at all.

**Matties:** How is that affecting the turnaround times, deliveries and such? Are customers more patient? Are they building in more time?

**O'Neil:** I think they're probably building in more time, but we have a few customers where their business model is quick turns, and they always expect five days. It's always five or fewer days, but now they have started to say that maybe five days is just fine. Some other customers are having a difficult time with our lead times going forward. They're now starting to get affected by supply chain issues in the assembly side of things. It is just a huge thing, so we still quote our five days at standard and if we have material issues, we're getting some lead times

that are six to eight weeks out. It's just a way of life today. It's not next door, it's hard to get.

**Matties:** Kim, we're excited to have the new column. What advice would you give designers today from a fabricator's point of view?

**O'Neil:** If you've got a difficult design that you're working with, maybe get in touch with your design

group (if you have an organization locally) and join that because there will be fabricators there. Look online; there are not too many fabricators out there anymore but give them a call. I've had calls cold calls from designers, if you will: "I'm this guy from here and I'm in the middle of a design, and I have this question, and this question." I get those all the time, maybe about four or five a month.

That's good marketing for us, to be able to say you can call me anytime, and we've helped folks out, whether or not you get the board to build. I feel it's part of the responsibility as we go through this because the industry has been decimated; many shops are no longer there.

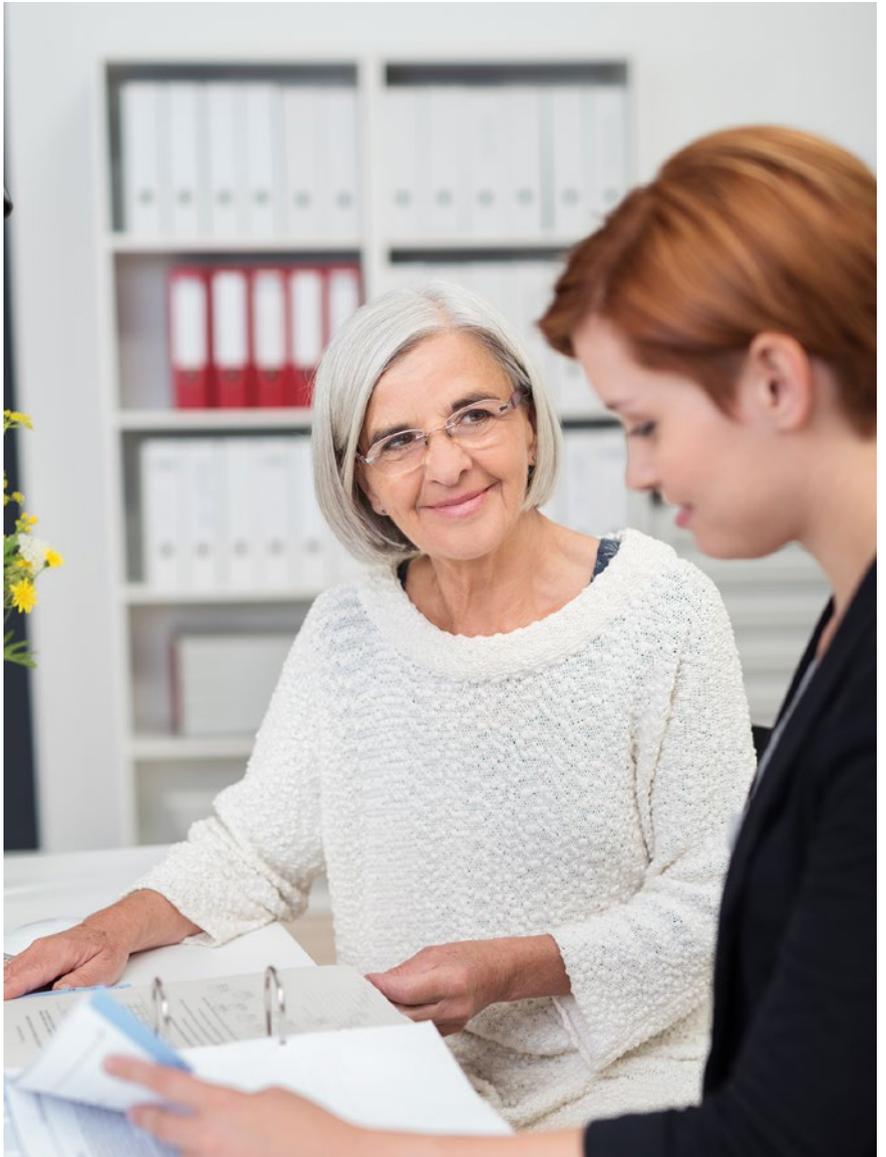


People coming up have to know and that's part of the responsibility of the industry itself too. It's educating the next group.

**Matties:** That leads me to our final area of conversation in this interview, which is tribal knowledge. You have 45 years of experience. At some point, you'll retire, and when you retire that knowledge retires as well. How important is tribal knowledge? How do we capture it? How do we move it forward?

**O'Neil:** It's very important, but I believe if you're passing it down to your ops manager and being inclusive in your group, listening to what other people say and answer questions like we're doing right here, it's getting some knowledge out there. But be inclusive of your staff and document that into your process. We tried a little bit of this. Create a library in your own shop showing when things went bad, when things went well, and what you did along the way. Resources are hard to get that done, but I think if you include your whole team into your discussions, and that means people off the floor too, it moves there.

When I got out of the Navy, I didn't know anything about circuit boards. Someone said, "Come up here and put in your application; after 20 years, you'll have a good education." Here's an example. Once, the company I was working for brought in an EE and they sat us together in the same office. He gave me an education and I gave him an education and it worked really well. So, the office bullpen type of things are wonderful, if you have that. We have an inside sales department. Does it bother



you that everybody's talking? No, because I'm picking up stuff as I'm going along and putting it in my library up here. It's important that you share, it really is.

**Matties:** I agree. The tricky part is capturing it, as you said, getting a library. But one thing you said is you're documenting it into your process and if you can get it inherently into a process, then it's being passed on formally.

Kim, I certainly appreciate your time today. This has been a fun conversation. Thank you so much.

**O'Neil:** Oh, thank you very much. DESIGN007



# Solid Data Management Key to **Accurate** Quotes

Feature Article by David Gronner  
PCB TECHNOLOGIES LTD.

Data is omnipresent. At times it goes unnoticed, just waiting there for someone to collect, analyze, and make use of it to create value. Data that seemed irrelevant at the time might come in handy when you need to come up with a solution to a new challenge.

When, for instance, you need a reliable quote that can accurately predict a product's price, you need to base it on past actual operation machine time, raw materials cost, etc. Otherwise, guessing a new product's price might result in unexpected spending, especially when it concerns sophisticated systems.

Each quote relies on utilizing data. The question that remains to be answered is how you do it and how fast. A data management strategy is an essential tool for turning a company's data assets into useful information supporting its strategic roadmap. While determining this strategy, each company must address processes aimed at storing, categorizing, controlling, and standardizing data.

The best data management practice revolves around the ability to make the necessary interfaces between various data sources flow across the company's relevant departments. When it comes to decision-making regarding financial investments concerned with the production of new systems and products, data management becomes even more crucial. To make these decisions that ultimately affect your business performance, you need to have a strong information base, constantly analyze it, and make sure it is deployed across the board in your IT infrastructure.

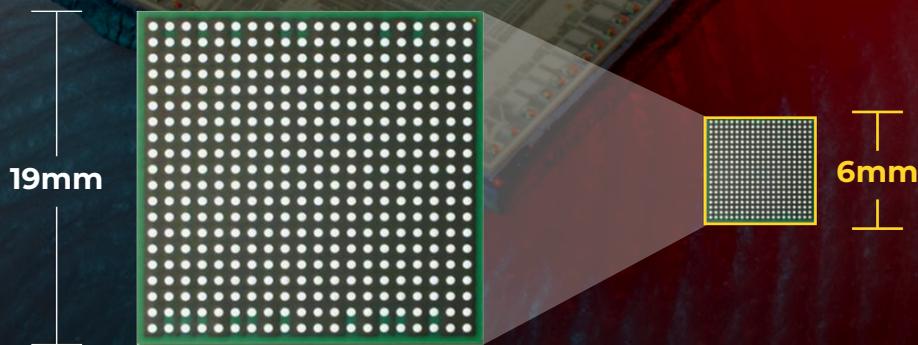
Every new product and new project needs to start with a solid plan comprised of various topics, such as manufacturability, reliability, and quality. One of the most important ones is cost. Its level of importance matches its level of complexity, since it relies on the direct or indirect outcome of all the other topics.

If you chose the wrong material, failed to come up with the optimal block diagram, designed your product relying on certain components without considering their MOQ, or

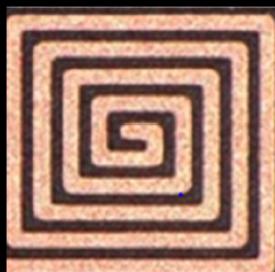
# Smaller is better.

**A-SAP™** Averatek semi-additive process offers a 90% reduction in size and weight over current state-of-the-art with 15 micron (.6 mm) trace and space

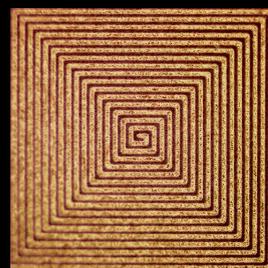
**A-SAP™** enables ultra-high electronics density



**A-SAP™** allows use of much smaller packages



Typical capability in the USA today: 75 microns



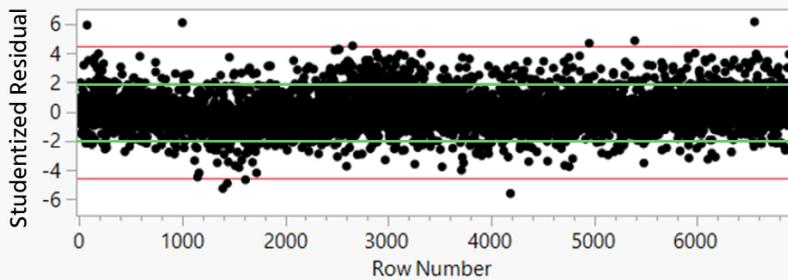
A-SAP™: 25 microns and below

[More info on A-SAP™](#)

**Averatek**

[www.averatek.com](http://www.averatek.com)

### Studentized Residuals



Externally studentized residuals with 95% simultaneous limits (Bonferroni) in red, individual limits in green.

### Summary of Fit

RSquare	0.980732
RSquare Adj.	0.97866

Figure 1: Our pricing fit model is based on more than 6,000 different part numbers that have completed the full engineering process.

any other thing you did (or didn't do), your final cost could be impacted.

How many times have you been told that, though your plan sounds brilliant, it is much too expensive to produce? How long did it take before you realized this was based on past experience or lack of data in regard to the cost of raw materials or working hours, and that the production could have been cheaper than expected?

How many times have you, on the other hand, plunged into what seemed to be the next bonanza, only to discover, too late into the project, that you have based your entire profitability prospects on an inaccurate quote which has left you no option but to stop production or go through a new design all over again? This causes you major setbacks in lead-time, not to mention the loss of the capital you have invested.

In both cases, you will not make it on time. In a competitive, fast-track market such as ours, that means you lost.

Any new design must strive for an accurate cost at the very early stages of its design. For the in-house cost, you must have the accurate data to support immediate costing, while for the outsourced part you need a speedy quote.

This information would help you to predict the full system production cost and bring you to the finish line on time.

One could say that new designs are destined for inaccuracy simply because they are new, but that is not true. And in a world built on time to market, speed, and customization, companies going through new product introduction (NPI) phases, looking for high-complexity board manufacturing, need to stay on top of their game in fear of their prod-

ucts becoming irrelevant to the market. A fast quote may become handy in these cases and shorten the lead time by 10–20%.

We decided to make use of an automated machine learning system to serve as an infrastructure endlessly collecting data on every item or part number in the production process. Every PCB's design file is uploaded once it is received from the customer to the system, initiating a classification process that, from there on, is automatically boosted with additional relevant data. In the short term, we don't always see the point of gathering all these facts but in the long run, they might be used for solving problems we couldn't have anticipated. Not only that, but the use of this accumulated data (the more the merrier) when used by big data systems can be re-evaluated and given a new perspective, simplifying complex processes. The system can hand pick the necessary information it requires from all the data to complete its mission.

We are not only interested in theoretical engineering data but also in what is practiced on the actual production floor. A database of hundreds of thousands of work orders, failure analysis, the precise production stage at which the error occurred, etc., gives us an indication of homogenous population characterization

according to which we can base a model that can predict the actual cost of a product. If, say, we know from experience, the failure rate of a certain category of products is 20%, that means the quote should rise accordingly. Failure ratio is an integral part of any future quote.

When you plan a production route based, for example, on a set of 100 actions, and in the actual process we realize there's a failure which calls for repair and adds yet one or more actions to the process, that means the quote should be higher.

That requires digitally signed and documented actions. Each action has an owner, a start, and a completion time report, and of course the indication of the machine on which it was produced. This is the only way that we can enable the system to add data, analyze it, and integrate it with other pieces of information already deployed. Data is processed daily. Some models undergo the same procedure on an hourly basis. The result is the ability to predict not only the failure ratio but also lead time, which also has bearings on the price. The system is based on the business intelligence software QlikView, which processes the information. The data is saved for a long period and can be used whenever necessary.

We have faced the same challenge with our customers over the years. Our new pricing system, JUMP, was designed especially for this purpose. The idea was to enable customers to get an accurate quote as painlessly and as fast as possible. This big-database analysis at the heart of the system runs scripts that can predict, in the early RFQ stage, the final product price and, indicate their lead-time and level of complexity. Our pricing fit model is based on more than 6,000 different part numbers that have completed the full engineering process, and includes their BOM, routings, and material classifications. The result is a unique and extremely reliable system with a very high R square of 0.98.

The unique manner in which data is managed in the system allows for statistical identification of various raw material combinations,

JMP module	
Circuit Type	RIGID
SPEC. Type	IPC 6012 CLASS3
Material Type	GLASS EPOXY
No. of Layers	17
Stackup Type	FOIL
No. of SUBs	0
Board thickness	2.155
Board type	MILITARY
Board Size - X	302.010
Board Size - Y	107.290
Array size X	323.00
Array size Y	323.00
Supply in Panel	NO
No. of PCBs in Arry	1
No diff PCBs in Arry	1
Panel Size - X	18
Panel Size - Y	24
No. Boards in Panel	4
Finishing - CS	HAL
Finishing - PS	HAL
Impedance	YES
Exist Gold Connector	N No
T.H Holes In Board	0
Laser Holes In Brd.	0
No. Laser Drill Cyc.	0
% of Assembly Scrap	11.000
Select Plating - CS	NONE
Select Plating - PS	NONE
Back Drill	N No
Etchback	NO
Exist H.S	NO
Paste Type	THROUGH
No. of Paste cycles	1
Routing PHASE	NONE
HATS Coupon Type	EXTERNAL
FAI Type	NONE
Wrap Requirement	IPC 6012 CLASS3
No. of Flex Cores	0
Cocoon type	NONE
Cocoon Depth Route	NO
Strain Relief	NO
Need Microwave	NO
Min. Line Width - CS	8.000

Figure 2: Big data analyses helped us to build a script that predicts, in early RFQ stage, the final product price based on a very short classifications questionnaire.

even those which were not introduced to it in the past. It scans the parameters detailed in the questionnaire for regularities and correlations to characterize various kinds of products. The process is performed at high speed to supply the customers with the quote they are looking for.

On another level, this system is considered “semi-AI,” meaning it constantly learns new parameters and manipulates the new data into new protocols that would be in use for the next quotes. If production utilizes a new technology that the system is not familiar with, all it takes is to manually deploy it into the system until its repeated appearances form a strong enough statistical volume.

When it comes to high-complexity boards, this short process is crucial for decision-makers trying to avoid the need to go through a renewed DFM (design for manufacturing)

process in the middle of production and can save delays and bad decision making. The procedural order is of no importance when providing a quote, much like an oven’s temperature is secondary to the cost of the cake’s ingredients and labor when determining the cake’s price.

Data is ubiquitous. We can guide, measure, and shape it to help create wonderful innovations, especially with the advent of AI and semi-AI. But every company must create—and adhere to—a solid data management process in order to turn data into actionable information. **DESIGN007**



**David Gronner** is director of OPEX and PPC for PCB Technologies.

## BOOK EXCERPT:

# *The System Designer’s Guide to... System Analysis*

## Chapter 4: A New Electronics System Design Solution

Achieving performance excellence requires a solution that can provide forward-looking estimation and predictability of downstream physical effects during the design exploration phase, overcome the bottlenecks of using multiple tools, and one that uses automation excellence. At advanced process nodes, circuit design/implementation schedule and timing convergence remain challenging tasks during the physical design phase. Additionally, the placement stage can be one of the most important and time-consuming steps in the physical design implementation flow as circuit timing and routing congestion must be considered at this stage. Therefore, to consider

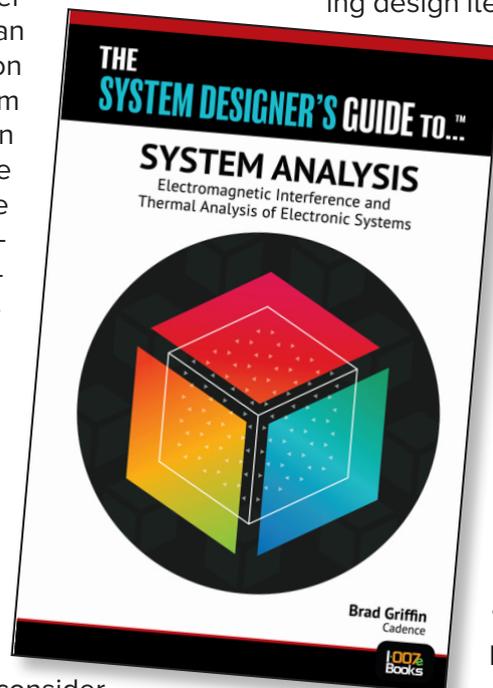
physical layout effects during the circuit synthesis phase, it is preferred to adopt a full flow, which bridges the quality gap between physical-aware circuit synthesis and physical placement for reducing design iterations. Tools for new architectures

and the complexity of advanced designs must be able to identify and leverage the strengths of computational flow algorithms. They must bring them upstream through a unification flow to mitigate the challenges.

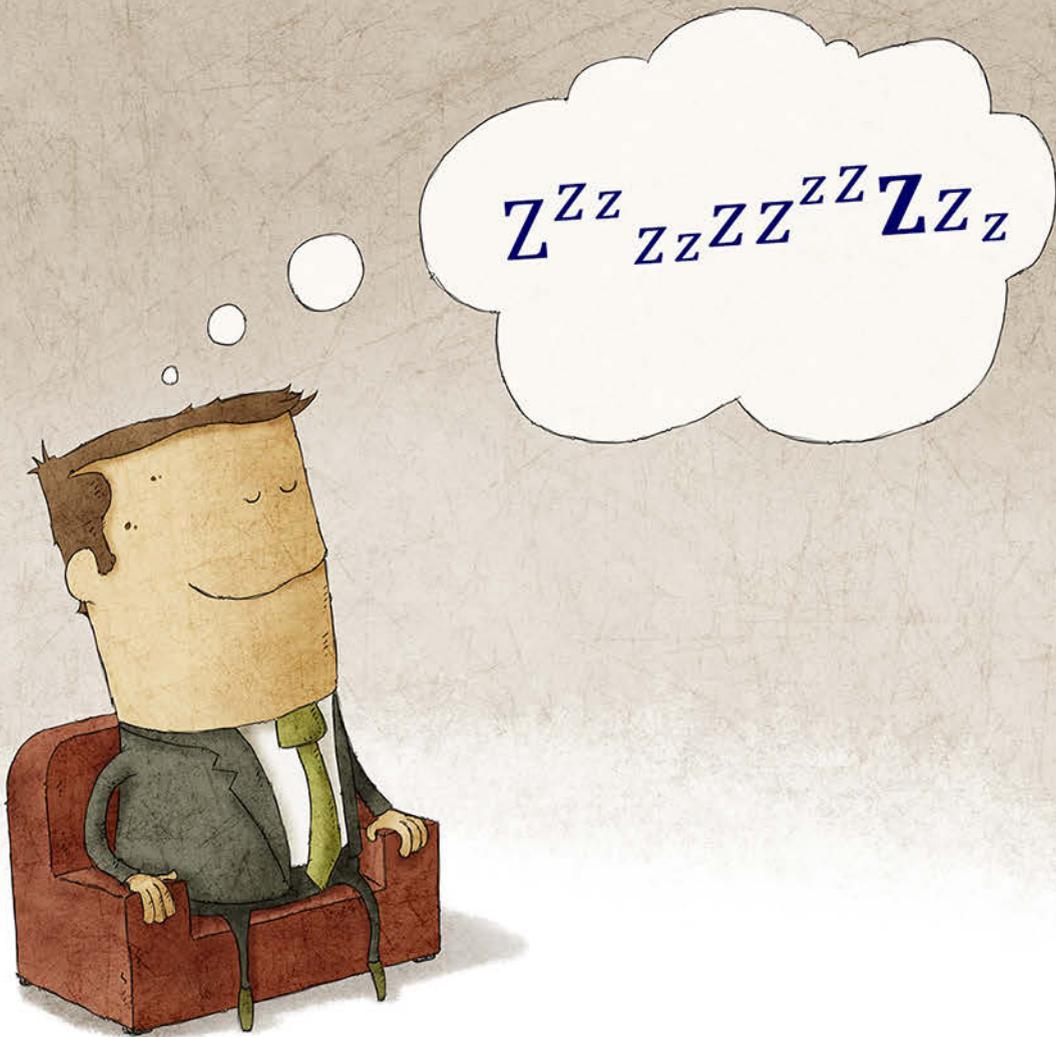
To curb the issues that delay the market entry, engineers must be able to foresee and fix issues at an early phase and not when manufacturing. There must be a system in place to perform simulation before manufacturing.

To download this free I-007eBook, [click here](#).

To view the entire I-Connect007 library, [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



# IP Reuse Enables a Digital Transformation

## Digital Transformation

Feature Column by David Wiens, SIEMENS DIGITAL INDUSTRIES SOFTWARE

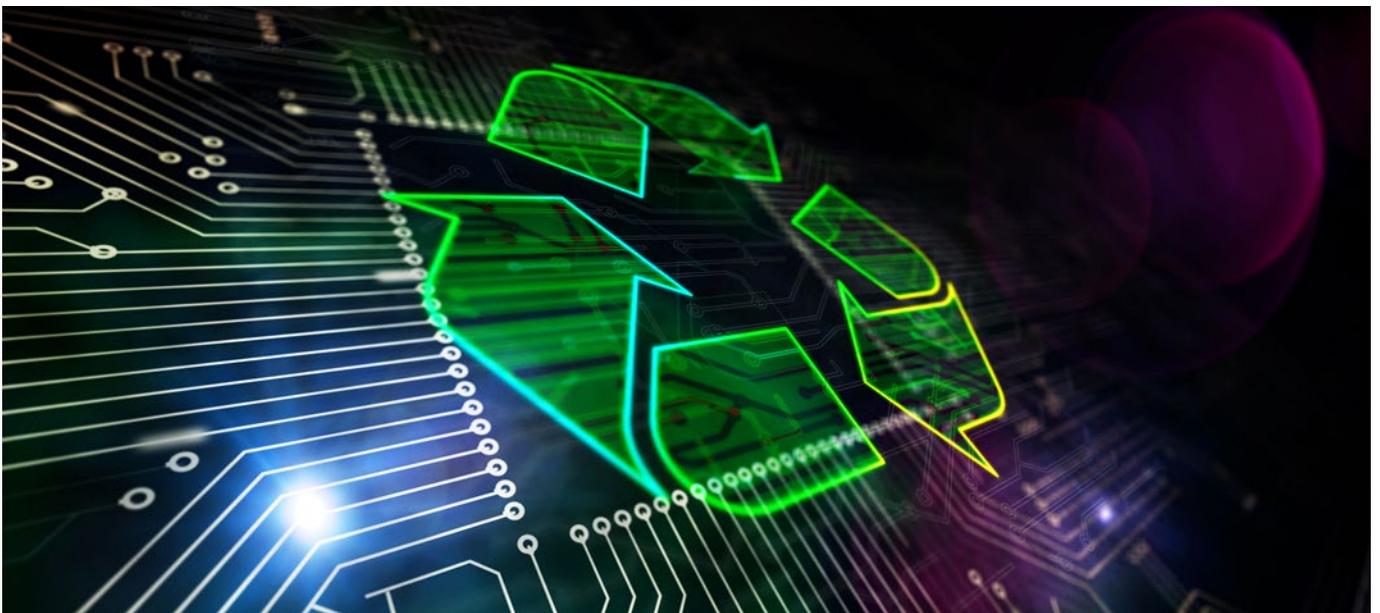
Last month's column identified the significant challenges today's electronic systems engineering teams face, and how a digital transformation of the entire design and manufacturing flow promises to resolve them by confronting product, process, and organizational complexities. One of the areas where we see opportunities for improvement is in the reuse of critical IP as part of a team's data management strategy: whereas the data is already digital, it's not always leveraged efficiently from one design to the next.

IP reuse isn't new; it's been used extensively in software and IC design. It's estimated that 68% of an IC design is built on data reused from in-house or third-party IP. In both cases, reuse was instituted to support concurrency, reduce cost, improve quality and reliability, and help manage complex IC designs and software projects.

The complexity of electronic systems is driving PCB design toward this same hierarchical "building block" approach.

Many new products are variations of existing products, so it only makes sense to reuse data from those previous products whenever possible. Leveraging "known good" verified circuits in new PCB designs accelerates new product development and eliminates repeated creation of the same circuitry. For example, once a power supply circuit is built and verified, why recreate it for every design that has the same power supply requirement?

Reuse also facilitates component consolidation and ensures continuous circuit improvement as qualified data from the consumers of the reuse blocks are incorporated into the source circuit blocks. Finally, for designs containing sensitive or restricted components or





Hmm, what is the  
difference between  
base and finished  
copper weights?

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?

**Reliable** answers. **Reliable** PCBs.

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](http://www.ncabgroup.com/PCBquestion)



circuitry, formal management of the source circuit blocks allows for proper tracking of critical information, such as the IP owner, ITAR classification, and any export control requirements.

In PCB systems, reusable IP comes in three primary forms:

- 1. Component libraries:** Most are familiar with a central library of part models—in the form of symbols, 2D/3D footprints, simulation models, etc.—that get reused across multiple designs. The current focus here is on team synchronization (ensuring everyone is always using the same library data) and library completeness (measured by the number of parts and the inclusion of newer elements like 3D representations and multi-physics simulation models).
- 2. Boards:** Teams often reuse internal or COTS boards across many multi-board systems. Incorporating these boards into new projects is critical so that the complete system can be modeled without data redundancy.
- 3. Circuit blocks:** These fit between #1 and #2 in design hierarchy and consist of

multiple components and/or the interconnects between them. The data could be logical (schematic-based), physical (layout-based), or a combination of the two. They could be developed internally (e.g., power supply, antenna, or standards-based interfaces between processor and memory), or leveraged from external sources (e.g., reference designs from IC vendors).

There are a lot of lessons to be learned from reuse in IC and software design that can now apply to PCB design. These include:

- Reuse requires a systematic, new process to verify known-good data, store the design data and associated verification results where they can be found, and then ensure that searching for existing IP is part of the process for new designs. It's a classic case where a little added process complexity resolves much more significant challenges, but at the time it's just another hurdle to overcome.
- Once IP is being reused, it becomes critical to track it in case changes are required. The impact of changes must

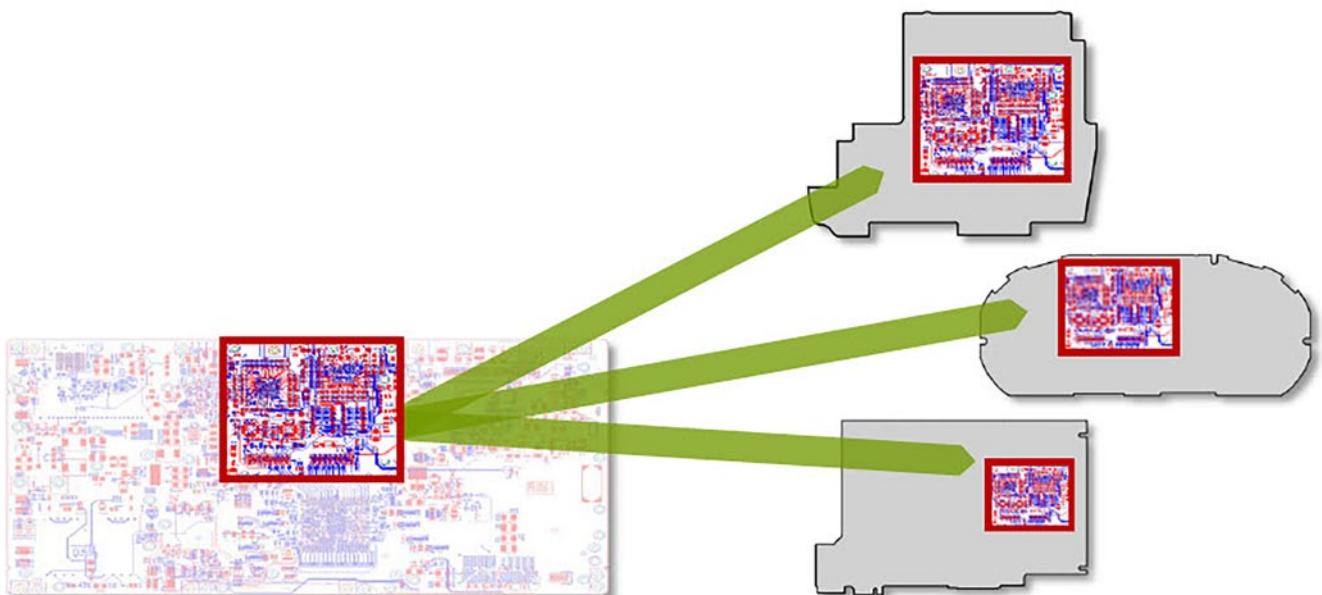


Figure 1: IP reuse manages complexity with a building-block design process that accelerates product development.

be assessed, and the appropriate teams notified if it's executed.

- Certainly, design software tools are critical to simplifying an IP reuse process, but a new methodology requires a commitment from management to ensure compliance with the new process and accept some short-term delays as it's implemented.
- There is a pervasive reluctance to use circuits designed by others because the quality is unknown, it feels like plagiarism, or is due to the classic “not-invented-here” syndrome. This reluctance needs to be overcome, first by a commitment from management and then through the benefits that teams will see from practicing reuse.

If deployed across an engineering organization, IP reuse can deliver significant benefits:

- Getting to market first by jump-starting new designs through the reuse of existing IP, thereby accelerating the product development process. Collaboration and efficiency are increased through a “building block” design process that fosters concurrent development, enables knowledge-sharing, and helps manage the increasing complexity of today's PCB systems.
- Reducing product development cost by eliminating repeated creation of the same circuitry, consolidating components to

increase purchasing power, and reducing the number of prototypes due to inherent lessons learned.

- Improving design quality by leveraging past successes—which also reduces risk in new products, reducing verification time by “certifying” known good circuits, and ensuring continuous circuit improvement.
- Ensuring compliance with corporate and industry requirements by leveraging certified IP, managing access rights for IP security, and reducing an organization's overall risk.

Effective reuse is ultimately a combination of corporate culture, efficient methodology, and enabling EDA technologies. I would encourage engineering teams to evaluate their current processes for effective IP reuse; a little process improvement pain will lead to extensive time, cost, and quality gains.

By digitally transforming the reuse of design IP, electronic systems engineering teams will reap the benefits and rewards of being the first to deliver tomorrow's designs to market today. **DESIGN007**



**David Wiens** is Xpedition product manager, System Design Division, for Siemens Digital Industries Software. To contact Wiens or read past columns, [click here](#).

An advertisement for a training video series. On the left, there is a logo for 'I.007e WORKSHOP' and 'American Standard Circuits' with a stylized lightning bolt icon. In the center is a portrait of Joe Fjelstad, a man with grey hair wearing a leather jacket. On the right, the text reads 'On Demand: Free Training Video Series' above 'FLEXIBLE CIRCUIT TECHNOLOGY' in large white letters, followed by 'with Expert Joe Fjelstad'. A green button with the text 'WATCH NOW' is in the bottom right corner.

# A Grand Announcement

## The Digital Layout

by Kelly Dack, CIT, CID+, PCEA

### Introduction

This month, I share some special news and changes regarding the PCEA. Next, I will humbly and gracefully hand this column back to its original founder, PCEA Chairman Steph Chavez, who will expound on some very exciting announcements which were made in early October at PCB West in Santa Clara, California.

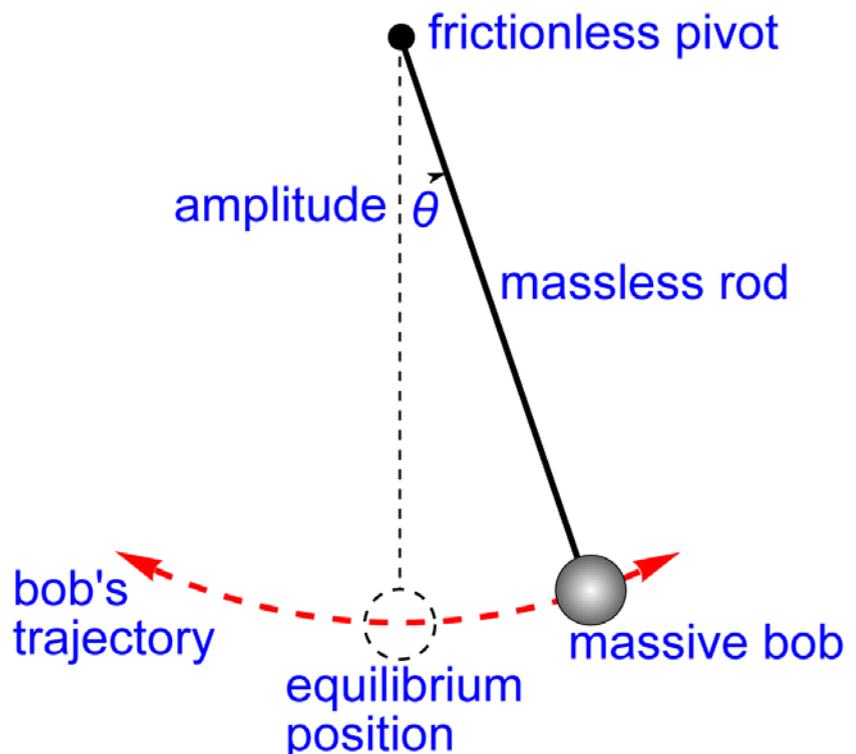
As always, we'll provide our readers with a list of events to finish out 2021 and take us into 2022.

### PCEA Updates

Who other than a dynamic organization like the PCEA could decide they want to establish a trade show footprint one month and then muster the creative talents of their executive staff to design a trade show booth to exhibit at DesignCon the next? Oh, and then exhibit at another major trade show such as PCB West 2021 only a short time later? I'll tell you, the PCEA has a momentum in this industry the likes of which we have not seen. I do not use the word momentum lightly. Because, like the shiny, spherical bob of a pendulum in a Newtonian mechanics experiment,

the leadership of this organization seems to be able to swoop down from their rightward (positive) displacement, pass their zero-position goal of achieved success, and still have enough momentum to reach their leftward displacement where they tend to set yet another, even loftier goal. The harmonic motion repeats but—unlike a pendulum—appears to gain energy rather than lose it to physical pseudo-forces.

I'm usually bad at metaphors though I've (ab)used a few in this column from time to



A simple gravity pendulum.  
Source: Wikimedia Commons



***LEAVING THE OTHERS  
IN OUR DUST BY INVESTING  
IN SPEED & TECHNOLOGY***

*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](http://www.eagle-elec.com)

time. As a graphics-minded career professional in an industry which still relies on “artwork” to create PCB hardware, it is sometimes difficult for me to describe things without using this metaphor shtick to make a point. But here I am, exceeding the physical confines of my pendulum metaphor. This metaphor is inaccurate because unlike a pendulum—which will eventually lose its kinetic energy and stop—the PCEA is growing and gaining energy. By now, you might have heard PCEA has signed a letter of intent to purchase certain assets of UP Media Group.

Talk about momentum. Wow! Does this mean that this organization went from an idea hatched between a group of PCB industry friends and colleagues at a trade show to buying a trade show booth, attending two trade shows, and then entering a deal to purchase a major trade show all in the span of less than two years? Even Star Trek’s admirable chief engineer and miracle worker, Montgomery Scott “Scotty,” would be amazed at our PCEA leadership’s ability to seemingly “defy the laws of physics.” I’d better turn it over now to Chairman Steph to fill you in.

## Message from the Chairman by Stephen Chavez, MIT, CID+

I want to personally thank Kelly for his unique flair in capturing the positive PCEA emotions we’ve built over these past two years and putting them into colorful words for the success of this column. I also want to thank him for his professionalism and tireless efforts in taking this column to the heights where it is today. Outstanding job, Kelly! Here is the big news of the day: PCEA is acquiring the assets of UP Media Group, including its publication. Words don’t adequately express how excited we are at this prospect. Let me briefly explain what this means.



Stephen Chavez



Figure 1: PCEA Chairman Steph Chavez, Chairman Emeritus Gary Ferrari, and Vice Chairman/Treasurer Mike Creeden.



Figure 2: PCEA Chairman Steph Chavez, Chairman Emeritus Gary Ferrari, Vice Chairman/Treasurer Mike Creeden, and Portland, Oregon Chapter Leader Zachariah Peterson.

The PCEA board of directors consists of 12 industry engineers. Just like you, we spend our days designing and building printed circuits. Our roles in the association are voluntary; we do this out of a passion for helping our colleagues by means of collaboration, education, and inspiration (the core mission of PCEA).

The acquisition of these assets, which also includes the PCB West and PCB East trade shows, among other properties, brings with it the first full-time staff for PCEA. The board



Figure 3: A group of excited show-goers awaiting the exciting announcement.

can henceforth focus on its traditional roles as directors, including reviewing the strategic and operational issues we need to address, and setting the goals for the organization. The board, in short, will set the strategic plan. The staff, for its part, will be responsible for the operations and executing of that plan.

This puts each of the respective board members and the PCEA staff in roles where we can individually (and as a unit) reach optimal effectiveness.

We expect to form a nominating committee for future board members, so if you are interested, please drop me a line.

I will have much more to say on this as we go forward, complete the transaction, and work toward fleshing out the strategic plan. Until then, feel free to reach out to me any time.

As always, if you have anything to do with printed circuit engineering, I highly recommend you get involved with and join the PCEA collective, if you have not already. By joining, your percentage of long-term professional development increases significantly.

Refer to our column and the PCEA website to stay up to date with the up-and-coming industry events. Take advantage of free webinar opportunities and consider joining us. Visit [pce-a.org](http://pce-a.org) for details.

I wish everyone's families health and safety. Best of success to all as 2021 wraps up.

Warmest regards, —*Steph*

## Next Month

As I mentioned, Steph Chavez will be taking over the column each month. We can look forward to the entire column being an inspiring, direct information feed regarding the plans and activities of the PCEA. If you've noticed, Steph often makes a point in his writing to "highly recommend" participating in all things PCEA. I'd like to borrow his gesture here and highly recommend that you tune into this column to keep up with how the PCEA is continuing its momentum to collaborate, educate, and inspire us all in this organization and this industry.

## Upcoming Events

Hope to see you at any or all of these.

### PCB Carolina 2021

Nov. 10, 2021

Raleigh, North Carolina

### productronica

Nov. 16–19, 2021

Munich, Germany

## IPC APEX EXPO

Jan 22–27, 2022

San Diego Convention Center  
San Diego, California

## Altium Live 2022

Jan 26–28, 2022

San Diego Convention Center  
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 details to [stephen.chavez.pcea@gmail.com](mailto:stephen.chavez.pcea@gmail.com), and we will consider adding it to the list.

## Conclusion

Back in September, I posed a question: “Could we be looking forward to any big announcements?” Like you, I could not have

imagined such an opportunity would materialize so quickly for the PCEA. Whether it has to do with the laws of physics, karma, or both, the PCEA is metamorphosing before our very eyes. Let us not only watch as a new PCEA begins to emerge, but jump in and find ways to connect, engage, learn, and grow with the membership which will certainly benefit because of all this.

It has been an honor to keep in touch with you by way of this monthly column. But for now, I’ll stop typing, and look forward to putting on my glasses to start reading about all the great things to come by way of the PCEA—of which I am a proud and grateful member.

So long for now! **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](#).

---

## A New 3D Printing Frontier: Self-Powered Wearable Devices

When most people think of wearable devices, they think of smart watches, smart glasses, fitness trackers, even smart clothing. These devices, part of a fast-growing market, have two things in common: They all need an external power source, and they all require exacting manufacturing processes. Until now.

Yanliang Zhang, associate professor of aerospace and mechanical engineering, and Ph.D. student Yipu Du have created an innovative hybrid printing method—combining multi-material aerosol jet printing and extrusion printing—that integrates both functional and structural materials into a single streamlined printing platform.

Zhang and Du, in collaboration with a team at Purdue University led by Professor Wenzhuo Wu, also have developed an all-printed piezoelectric (self-powered) wearable device.

Using their new hybrid printing process, the team demonstrated stretchable piezoelectric sensors, conformable to human skin, with

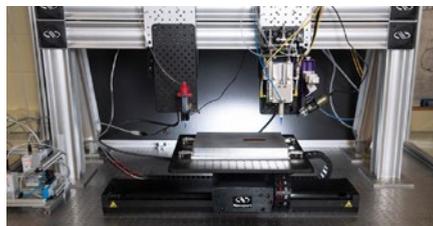
integrated tellurium nanowire piezoelectric materials, silver nanowire electrodes, and silicone films. The devices printed by the team were then attached to a human wrist, accurately detecting hand gestures, and to an individual’s neck, detecting the individual’s heartbeat. Neither of the devices used an external power source.

“The biggest advantage of our new hybrid printing method is the ability to integrate a wide range of functional and structural materials in one platform,” said Zhang. “This streamlines the processes, reducing the time and energy needed to fabricate a device, while ensuring the performance of printed devices.”

Vital to the design, said Zhang, are nanostructured materials with piezoelectric properties, which

eliminate the need for poling or sintering, and the highly stretchable silver nanowire electrodes, which are important for wearable devices attached to bodies in motion.

(Source: University of Notre Dame)



cadence®

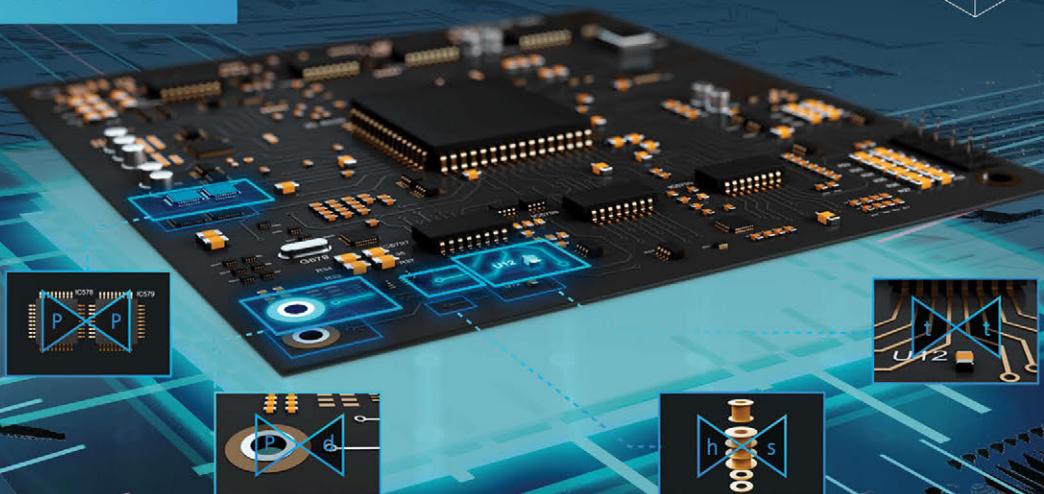
GET EBOOK



# Real-Time DFM with DesignTrue

Identify and correct DFM issues prior to releasing designs to manufacturing slashing turn time.

LEARN HOW



# Cost of Compliance and How Data Transfer Standards Can Help

## Design Circuit

Feature Column by Patrick Crawford, IPC

In the more than half-century since the rise of electronic devices, nearly every human being on the planet has, at some point in their life, come into contact with one. As this contact became an inevitability in most parts of the world, human health and safety also became a critical factor in the design of all electronics. From this need to protect users from the myriad deleterious effect of toxins in electronics there arose dozens of global regulations regarding hazardous substances.

(A necessary caveat: There are drivers for these regulations that have absolutely nothing to do with electronics. But seeing as this is a publication targeting the design of electronic circuits, I will not be spending too much time discussing leather for car seats or thermoset plastics for electric drill bodies, etc.)

One such regulation that I am sure most electronics manufacturing industry professionals are aware of is the EU Restriction of

Hazardous Substances (RoHS), the most recent version of which bans the use of 10 substances in the EU which have been proven to have negative impacts on human health and safety. Perhaps most well-known to electronics is Pb (lead) which prompted the need for (and conversion to, in most cases) Pb-free solder. Need I go on?

Of course, EU RoHS is only one example of

a restriction of substances,

and that is not the only

RoHS. However,

this column is not

intended to educate

on the various regulatory agencies

and their programs,

nor provide instruction

on which compounds

and substances should

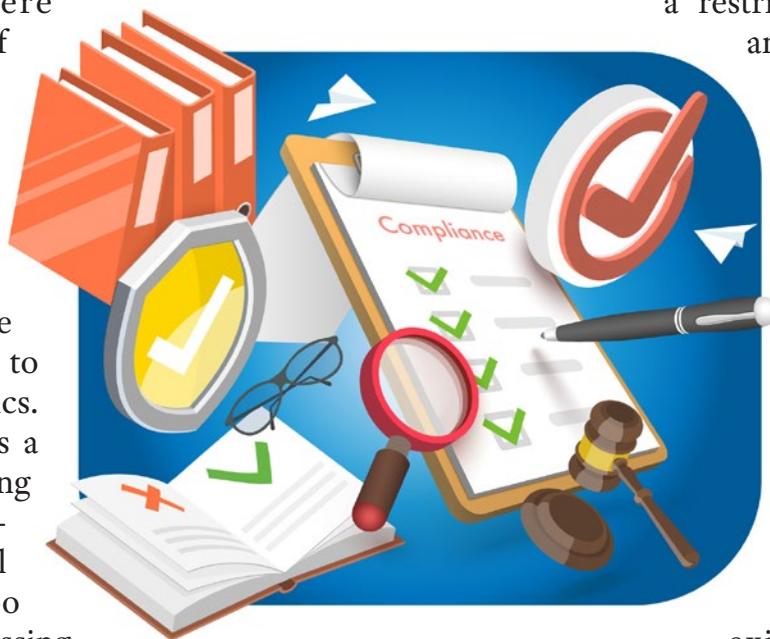
or should not be used

for fabricating boards.

Resources already

exist on such matters

which far outpace both my understanding of the duties and nuance of these regulations and of the chemistries cited therein. Furthermore, this is not an opinion piece regarding the merits of these various regulations.



*Welcome to the Summit  
Interconnect Family!*

# *WELCOME TO THE SUMMIT*

**SUMMIT ACQUIRES EAGLE ELECTRONICS**

**READ THE PRESS RELEASE HERE**



[summit-pcb.com](http://summit-pcb.com)

[eagle-elec.com](http://eagle-elec.com)

Rather, I want to look at one of the chief cost-burdens associated with complying with these regulations, specifically the cost of obtaining and communicating materials information throughout the supply chain.

As an example, let us consider the new reporting obligations associated with the European Chemicals Agency's (ECHA) substances of concern in articles as such, or in complex objects (products) (SCIP) database. In brief, the European Union (EU) Directive 2008/98/EC on Waste, commonly referred to as the EU Waste Framework Directive (WFD), entered into force in July 2018. The WFD set requirements on waste management, recycling, and recovery of products manufactured in, sup-

---

## The database is now live with more than four million submissions available for review. (Whew!)

---

plied to, or imported to the European Economic Area. As part of these requirements, the WFD mandates that any article containing substances of very high concern (SVHCs) must be declared to the ECHA so that this information can be made available to waste operators and consumers. The database is now live with more than four million submissions available for review. (Whew!)

Even more, if your product contains a candidate list substance and you would like to manufacture or sell it in the EU, you have to notify ECHA and declare whether any of those candidate list substances are below threshold.

As I indicated, almost every engineer has heard of RoHS, and most have probably heard of REACH. But if you are not a compliance manager, or in some other kind of purchasing or procurement role where you are exposed

to the immensity of your supply chain, you may not be aware of how onerous determining whether your product contains SVHCs (let alone their positions in product, percentage of total mass and whether they are above threshold, etc.) can be.

Let's look at an example specific to the SCIP database, keeping in mind that I am making this as absolutely generic as I possibly can. Consider hypothetical Company A is an OEM producing inexpensive, low-complexity phones. In fact, these phones are so low-complexity, that Company A has only two suppliers—Company B and Company C.

Now, Company A is based in Canada but would like to sell into the EU. They find a distributor in Germany who will provide to convenience stores there and in a few neighboring countries. On top of other bureaucratic necessities—too many to name here—Company A learns that they may have obligations to report to ECHA.

A representative from Company A contacts Company B and C and asks them to provide chemicals information on their products. As it turns out, Company B's supplier stream is a bit wider, and now they have to contact their suppliers—all 10 of them—to ensure their chemicals composition, and onward and upward the pipeline. Company C, feeling proud of their proactivity, provides Company A with an Excel sheet containing candidate list substances (and a few others for good measure). Of course, the information does not satisfy all requirements needed for SCIP; you get the idea.

Oh, and when Company B does get back to Company A a few months later, they hand over a PDF report.

What is critical in this overly cartoonish example is that there are real humans performing real work to support every action verb in the above paragraph. Consider the very real complexity of your supply chain, even if you are not a tier one company, and you can understand how complying with even one global regulation can add to your costs.

I will pause here to reiterate that I am not making a judgment on the merit of these regulations, but rather pointing out that there are objective costs associated with compliance thereof. For more information on IPC's official position on these regulations and how we are supporting industry going forward, please reach out to the IPC Government Relations department. You can find more information on [IPC.org](http://IPC.org).

What can you or your company do to attempt to minimize costs? Three words can certainly help: data transfer standards.

Apart from design standards, my other hat is to manage the committees that are responsible for the IPC-1750 family of supplier declaration standards. These documents all describe xml file formats which enable companies to request data from their suppliers or push data to their customers that relate to various regulatory requirements: RoHS, REACH, data sufficient for a SCIP submission, etc. The beauty of the xml format is it's standardized across submissions, so a company can eliminate the manpower associated with parsing various data formats from their suppliers or negotiating submission data formats with their customers. It's all just IPC-1752 or IPC-1754 (material declarations and materials and substances declaration for aerospace, defense, and other industries, respectively); communicating data pursuant to SCIP; IPC-1755 (responsible minerals sourcing) communicating information pursuant to EU Conflict Minerals Legislation; or IPC-1753 (lab reports) communicating analytical lab report data. There are no miscellaneous data formats that must be reckoned with.

If a company does not have the resources to create an in-house system to receive and collate these files, then they can utilize one of the many solution providers who offer compliance tools that are capable of importing and exporting IPC-1750 family compliant files. Most of them will contact suppliers and communicate with customers on your behalf to

collect and manage data. (These companies are all verified by IPC to ensure that their data is, indeed, compliant.) Of course, there are up-front costs associated with purchasing any given compliance tool or service, and you or your company should weigh this before making any decisions.

---

## What can you or your company do to attempt to minimize costs? Three words can certainly help: data transfer standards.

---

These standards help eliminate costs by "greasing the wheels" of communication between companies in a supply chain. They do not necessarily eliminate the amount of data that is required to submit to customers or regulatory agencies, but they can help ensure that your company has all the data necessary to complete a submission. In my example, Company C did not provide the correct data to Company A, which would prompt another round (or 20) of back-and-forth until the correct information was obtained.

In the end, there will likely be a cost associated with compliance. However, with the proper tools, any electronics manufacturing company can mitigate those costs to the fullest extent possible. As regulations continue to evolve, IPC standards will be revised to match those regulations and their various requirements. **DESIGN007**



**Patrick Crawford** is the manager of design programs and related industry programs at IPC. To read past columns or contact him, [click here](#) or email [Patrick-Crawford@ipc.org](mailto:Patrick-Crawford@ipc.org).

# American Standard Circuits on the Fundamentals of RF and Microwave PCBs

Article by Pete Starkey

I-CONNECT007

In the second of a series of three *RealTime with... interviews*, I-Connect007 managing editor Nolan Johnson received knowledgeable and informative answers from John Bushie, director of technology at American Standard Circuits, and Anaya Vardya, president and CEO, to his questions on the unique challenges of RF and microwave PCBs.

Asked by Johnson for a definition of what was currently considered RF, Bushie responded that there was no clear boundary. But as data rates continued to increase, high-speed and ultra-high-speed digital circuits progressively took on analogue properties so that they began to require performance characteristics traditionally associated with RF and microwave circuitry.

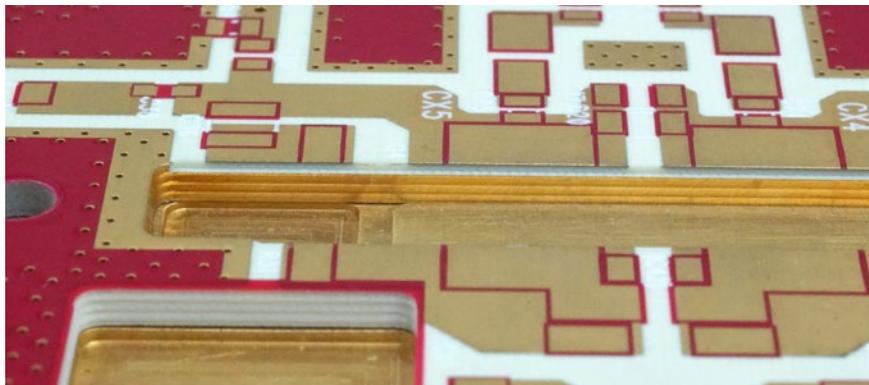
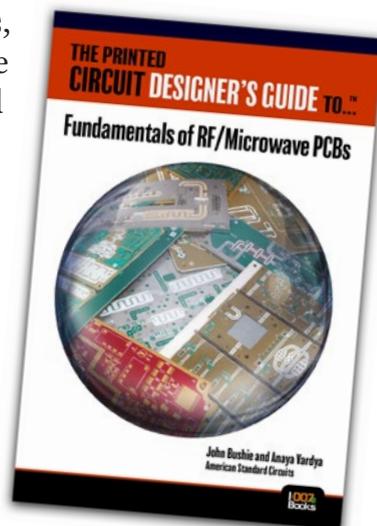
Referring to observations noted in *their recent book*, Johnson remarked that designers had appeared more concerned about material selection than considerations of manufacturability. Bushie's comment was that in reality the performance achieved resulted from a combination of material factors with fabrication tech-

niques and tolerances, and it was a challenge for designers to find a realistic balance between cost and performance. Attitudes toward material cost ranged from "performance at any price" to "cost is everything." At the latter end of this spectrum there was potential for using

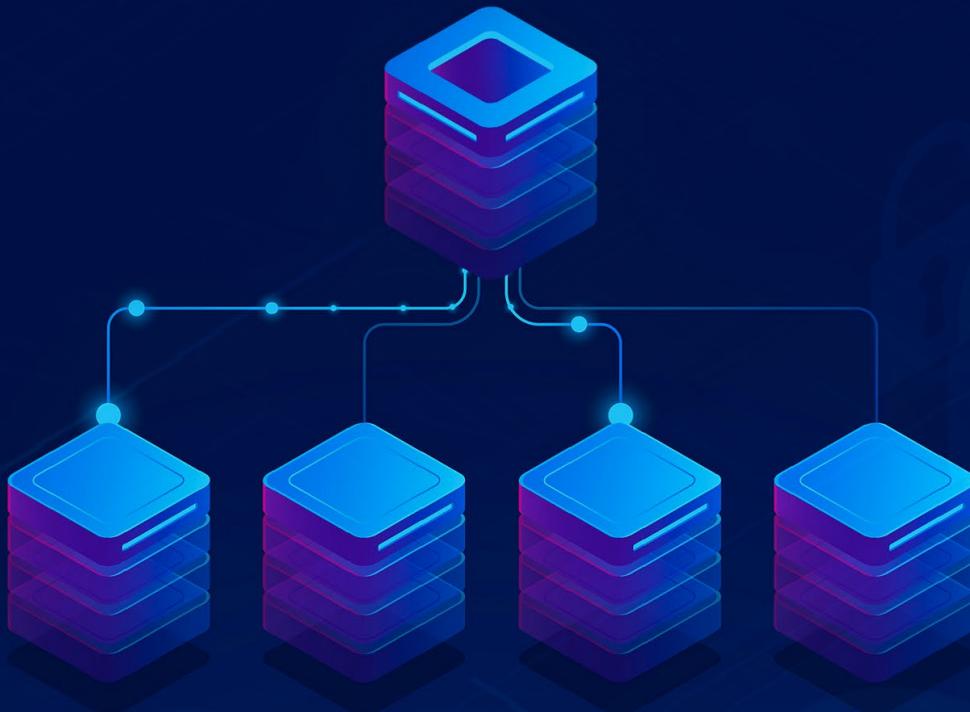
lower-loss variants of more traditional materials. And properties like mechanical strength and dimensional stability were other factors to be taken into account. Bushie stressed the importance of getting the fabricator involved as early as possible in the design process to help optimise the balance between material selection and design for manufacturability, in order to achieve cost-effective and functional designs which might combine different material types. An example was a multilayer stack-up using a core of FR-4 laminates and prepregs in combination with PTFE outer layers.

Anaya Vardya added that some of the rigid-flex designs discussed in the previous session could incorporate RF materials: constructions were becoming increasingly hybridised.

There were no actual boundaries, but it remained critical to be aware of the processing requirements of the different materi-



# Integrated Data Management, First-Pass PCB Success



## Achieve Data Clarity Today

Optimize every part of your design process with integrated access to meaningful data. Eliminate manual data collection, tracking, and sharing with our comprehensive, real-time design data management solutions and be confident knowing you'll always hit your PCB design goals.

See How >>



John Bushie

als to avoid unintentionally, but mistakenly, creating designs that were effectively unmanufacturable. It was also important to understand the consequences of specifying custom design features and selective

finishes on cost-effective manufacture.

Johnson enquired whether there was an easy way of avoiding misunderstanding, in response to which Bushie re-emphasised the importance of involving the fabricator as early as possible at the design stage, for expert guidance on materials, constructions and special features. He illustrated and discussed many examples of details to be considered, with an impressive air of calm professional experience and capability that instilled confidence in his wisdom without over-dramatising the potential pitfalls.

Johnson commented that copper surface roughness had become a much-discussed topic in the context of high-frequency performance and asked Bushie for a view. That was a more complex subject than could be explained in-depth within the timeframe of this interview but put simply, roughness caused small phase changes in signals travelling in or near the surface, resulting in increased losses. Bushie discussed the attributes of some of the smoother-finish copper foils now available, and made the point that choice of foil was yet another element to be taken into consideration when specifying the material stack-up. Fortunately, RF designers had become increasingly aware of foil effects on edge-coupled and broadside-coupled features.

Vardya reminded viewers of the amount of detail information available in the numerous webinars that ASC had published and were freely available [on their website](#).

He summed up by saying that if a designer took nothing else away from this discussion,

it was the principle of approaching a PCB fabricator as early as possible in the design phase of a complex circuit board. Doing so could add a lot of value and potentially save significant time and expense when presenting the design for manufacture. ASC has accumulated an abundance of experience over many years of assisting designers to understand the realities of PCB materials and processes and the benefits of effective design for manufacture. He gave his assurance that any discussion held with ASC would be in commercial confidence.

This [RealTime with... video](#) was a valuable reminder of the multiplicity of factors to be taken into consideration when designing PCBs for RF and microwave applications, and the generosity of American Standard Circuits in sharing their knowledge and experience is to be commended. I was particularly impressed and reassured by John Bushie's relaxed and unpretentious presentation style.

Twenty minutes well spent! **DESIGN007**

Visit [I-007eBooks](#) to download your copies of American Standard Circuits' micro eBooks today:

*[The Printed Circuit Designer's Guide to... Fundamentals of RF/ Microwave PCBs](#)*

*[The Printed Circuit Designer's Guide to... Flex and Rigid-Flex Fundamentals](#)*

*[The Printed Circuit Designer's Guide to ... Thermal Management: A Fabricator's Perspective](#)*



Anaya Vardya



**Pete Starkey** is an I-Connect007 technical editor.



Meetings & Courses  
**January 22-27**  
Conference & Exhibition  
**January 25-27**

San Diego Convention Center | CA

# Drive **Digital** **Transcendence**

Transcend your technical knowledge at North America's largest conference for electronic industry professionals. If you seek career development, collaboration, and networking opportunities that cause an impact, join us in San Diego.

**See you in San Diego  
at IPC APEX EXPO 2022!**

[IPCAPEXPO.ORG](https://www.ipcapelexpo.org) | [#IPCAPEXPO](https://twitter.com/ipcapelexpo)



# MilAero007 Highlights



## HawkEye 360's Third Satellite Cluster Begins Commercial Operations ▶

The third cluster of satellites launched by HawkEye 360 Inc., the world's first commercial company to pioneer radio frequency (RF) data and analytics from space-based satellites, has achieved initial operating capability and has begun to deliver RF data and insights to clients.

## U.S. Defense Department Awards \$8.9 Million for Further R&D on Lead-Free Electronics in Aerospace and Defense Sectors ▶

The U.S. Defense Department (DoD) is taking another step toward understanding and embracing the use of lead-free electronics in high-performance defense areas by allocating another round of funding to a public-private research effort.

## Boeing's Latest 737-9 ecoDemonstrator Testing Crane A&Es New Long-range Sensing ▶

Crane Aerospace & Electronics, a segment of Crane Co., has been selected to feature its new Long-Range Wireless Tire Pressure Sensors on Boeing's 2021 737-9 ecoDemonstrator program.

## Defense Speak Interpreted: The 'Trouble' With Obsolescence ▶

How could a simple word like obsolescence stir up so much trouble within the Defense Department? Obsolescence is defined as the process of becoming obsolete or the condition of being nearly obsolete. Dennis Fritz explains its connection to Defense.

## PCB Technologies Develops Substrate Integration for Miniaturization ▶

In the endless pursuit of miniaturization of microelectronic devices, aimed at their enhanced functionality and reliability in parallel to higher endurance, PCB Technologies has developed the ultimate substrate integration, relevant for various applications.

## Raytheon Missiles & Defense, Northrop Grumman Successfully Test Fire Hypersonic Weapon ▶

Raytheon Missiles & Defense, a Raytheon Technologies business, in partnership with Northrop Grumman, successfully completed the first flight test of a scramjet-powered Hypersonic Air-breathing Weapon Concept (HAWC) for DARPA and the U.S. Air Force.

## Calumet Electronics Supporting Aerospace Growth in Michigan ▶

Upper Michigan-based Calumet Electronics is developing next-generation technology to advance commercial aerospace, satellite, space, and DoD systems.

## L3Harris Selects Aegis' Platform for 20 Manufacturing Locations with Over 5,500 Users ▶

Aegis Software, a global provider of Manufacturing Execution Software (MES), has signed an enterprise agreement with L3Harris Technologies to support 20 manufacturing sites and more than 5,500 end users with the Aegis' FactoryLogix® platform.

[www.prototron.com](http://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*

# Diving Into the Chemical Processes of PCB Manufacturing

## Connect the Dots

by Matt Stevenson, SUNSTONE CIRCUITS

I have always been fascinated with chemistry and chemical processes. My first degree was in chemistry and my first job out of college was in the PCB manufacturing shop in the analytical chemistry lab. During my initial tour I was so surprised with just how many chemical processes there were in PCB manufacturing. I discovered that some of the most critical elements of PCB manufacturing involve chemical processes.

Chemicals clean the copper in preparation for the coating that prevents oxidation, and again to remove contaminants before solder resist application. The etching itself uses chemicals to remove excess copper. Chemicals are also used to electroplate metals such as copper, nickel, gold, and tin. In addition, there are many electroless and immersion plating chemicals that are used to protect exposed copper ahead of the soldering process.

Keeping all these chemical baths, sprays, rinses, and processes operating in a tight operating window to maintain quality and predictable results is a key component in PCB manufacturing.

## Best Practices for Chemical Process Control

There are enough methods available to test and analyze chemical processes in a factory to make your head spin. Here is a short primer on the most common, effective methods of testing chemical processes:

### *Titration*

There are quite a few chemical components that can be measured in a working chemical bath by a titration. Titration is the process of adding one chemical of a known concentration to a small sample of the bath until a target pH is obtained.



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

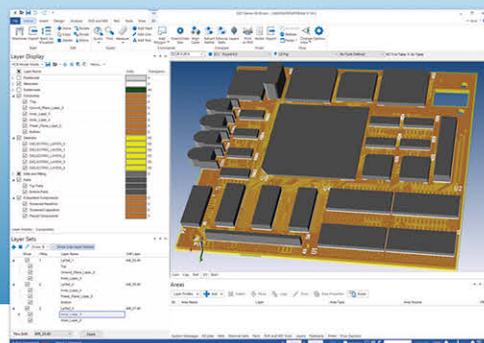
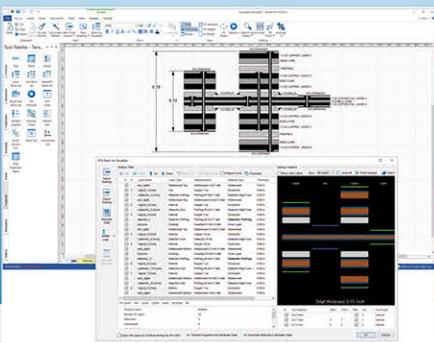
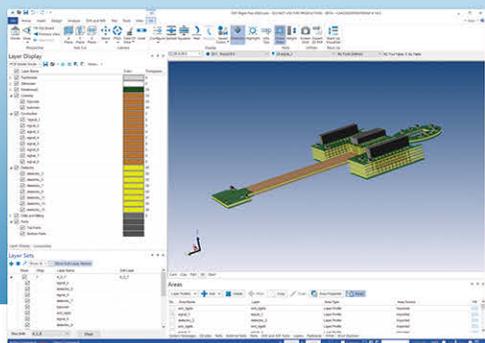


**BluePrint-PCB®**



**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



For more information visit [downstreamtech.com](http://downstreamtech.com) or call (508) 970-0670

There are several ways to catch this pH endpoint. Indicators can change color at a given endpoint. You can track pH with a meter during titration. Some reactions will provide answers by creating a precipitate (a solid) at the desired pH. If you know precisely how much chemical has been added, a simple calculation will tell you the concentration of the component in the bath.

### **Colorimetric Tests**

Another common testing method uses ultraviolet-visible (UV-Vis) spectrophotometry. We use a UV-Vis spectrophotometer to analyze a sample of a bath at a given wavelength of light. As we do, the absorption or transmittance of the light as it passes through the sample will give a reading. When this reading is applied to a calibrated curve of known concentrations, it can provide the bath concentration.

### **Elemental Analysis**

Elemental analysis is a sound method for finding concentrations of various types of metals in a chemical sample. Common elements found in PCB manufacturing include copper,

standard curve of known concentrations to determine the composition of the solution.

### **Functional Tests**

Functional testing is a type of black box testing that bases its test cases on the specifications of the component being tested. These tests are especially useful in electroplating because they can help you understand how some components are functioning. One example of a functional test is a grain refiner in a copper bath tested using a special bench top unit called a Hull cell. A sample of the plating bath is added to a Hull cell, along with an anode and a brass plating coupon. Next, current is applied. This causes an electroplating reaction to occur, and a metallic deposit will accumulate on the brass Hull cell coupon. Allowing the Hull cell to sit for a period will ensure that there is adequate metallic deposit to analyze.

This type of test can cover a variety of plating current densities. You can also add standard additions of grain refiner to the plating cell and compare it to the plating bath. This will help you to understand appropriate additives that will achieve the desired copper deposit.

All these tests and the results they produce will help you to maintain the chemical processes within their given operating ranges. You will also be better able to quickly identify problems with equipment or chemistry to ensure that an expected outcome is the result of every chemical process. **DESIGN007**

---

**Common elements found in PCB manufacturing include copper, gold, nickel, silver, tin, and lead.**

---

gold, nickel, silver, tin, and lead. One of the most accurate ways to detect metal is through flame atomic absorption analysis (FAAS). Using this method, a sample of the chemical bath is burned in an atomic absorption flame, and it measures the absorption of light at specific wavelengths to detect gas phase elements. This absorption value is compared to a stan-



**Matt Stevenson** is the VP of sales and marketing at Sunstone Circuits. To read past columns or contact Stevenson, [click here](#).



**INTERNATIONAL ELECTRONICS  
CIRCUIT EXHIBITION (SHENZHEN)**  
国际电子电路(深圳)展览会

**5G**

**CONNECTING  
THE WORLD  
万物互联**

**2021.12.8-10**

**深圳国际会展中心(宝安)  
Shenzhen World Exhibition  
& Convention Center (Bao'an)**

**[www.HKPCAShow.org](http://www.HKPCAShow.org)**

主办单位  
Organizers

支持单位  
Supporting Organization

承办单位  
Event Manager



Follow Us on WeChat  
关注展会官方微信  
获取最新资讯



Follow Us on LinkedIn  
关注展会官方领英  
获取最新资讯

# FLEX007

A SPECIAL DESIGN007 MAGAZINE SECTION

## A Brief Retrospective of 50 Years in the PCB Industry

### Flexible Thinking

by Joe Fjelstad, VERDANT ELECTRONICS

I recently reflected upon the notion that this year marks my 50th year in the printed circuit industry. It was a bit of a shock when I looked at the calendar and realized that I have been kicking around this industry for a half-century.

I was fortunate enough to find my way into the PCB industry through the analytical lab of a PCB company in Mountain View, California in the early days of Silicon Valley. The name of the company was Printex and it was one of the premier PCB fabricators in the United States.

Just prior to signing on with Printex, I had been working for Data Lab in Santa Clara, where we served as an offsite analytical lab checking the plating chemistry for many of the dozens of printed circuit fabricators that were operating in the Bay Area at the time. At Printex, my job was to analyze and maintain the numer-

ous plating and processing solutions used in the manufacture of printed circuits. I was also tasked with preparing and evaluating micro-sections of the plated through-holes of those printed circuits.

I was young, ambitious, and energetic enough to complete those tasks quickly, leaving me time to go into the manufacturing areas of the facility to learn firsthand the details of every area of processing. It was of immeasurable value to me—for the rest of my career I had learned to troubleshoot, diagnose, and correct problems in processing. I also came to appreciate

that there are few products that draw on such a varied palate of technologies to create them—composite material lamination, computer-controlled machining (i.e., drilling and routing), electroless and electrolytic plating



# PREMIER GLOBAL SUPPLIER to the **MEDICAL DEVICE INDUSTRY**

FLEXIBLE CIRCUITS | EMS/ASSEMBLY | CONTRACT MANUFACTURING

**Industry leading Flex Design Support**

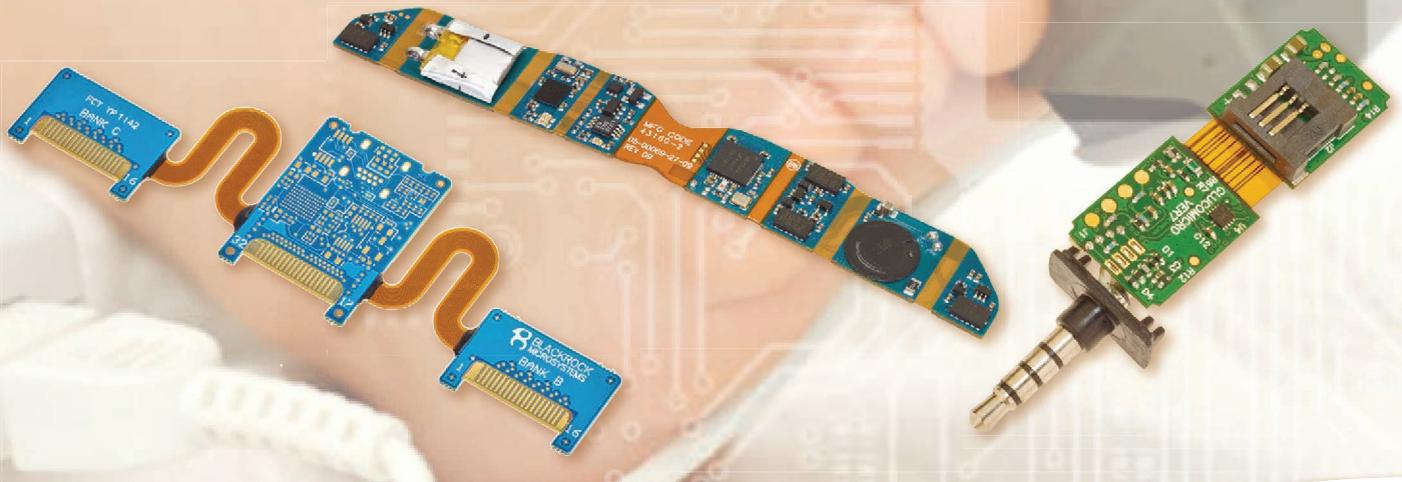
**Flexible Circuits, Rigid Flex, Flexible Heaters  
Membrane Switches, Plastic Moldings**

**Specialized EMS/Assembly Services**

**Product Module to Complete  
Product Box Builds**

**ISO 13485 – FDA registered Box Builds**

**EXPLORE the  
POSSIBILITIES!**



Flexible Circuit Technologies  
9850 51st Ave. N. | Plymouth, MN 55442  
[www.flexiblecircuit.com](http://www.flexiblecircuit.com) | +1-763-545-3333

processes, screen printing, wet and dry film imaging, developing and stripping, chemical etching, and several others. To me, it has always been one of the most attractive aspects of PCB manufacture and what makes PCB manufacturing tirelessly interesting.

## What Has Changed

Today, many of those same processes are still used, though they have been greatly improved in terms of machines, materials, and processes available. Still, they remain fundamentally unchanged except that the circuit features are now approaching or in some cases equal to those produced on semiconductor integrated circuits of that same era. One big difference is that back then, the substrates for early semiconductors were 50- and 75-mm silicon wafers and today those near-same-size features are being produced on 450 x 600 mm FR-4 panels to make printed circuits. This has been a remarkable achievement.

---

**PCB designers of the early days were largely mechanical draftsmen charged with “connecting the dots” on the schematic provided by the circuit designer.**

---

Much has also changed in the realm of PCB design. PCB designers of the early days were largely mechanical draftsmen charged with “connecting the dots” on the schematic provided by the circuit designer. Artwork was often created by taping circuit traces and pads at one to four or more times the size of the final circuit and using a large format (near room size) cameras to “shoot down” the artwork to the size needed for contact printing the circuit

image with working film. During that era, Gerber vector photoplotters were coming into use and light pens were mechanically driven using pre-programmed vector-driven information to create the artwork. In that early era, double sided printed circuits were most common and four-layer multilayer circuits were essentially state of the art, and the term “controlled impedance” was basically unheard of. Today’s PCB designers smile at such simplicity.

One unfortunate thing that has happened over the years is that, while semiconductors have grown in respect, appreciation, and even adulation, the PCB has too often remained under-appreciated and undervalued. I have, over the few decades of teaching PCB seminars and workshops, likened the semiconductor to a magician or illusionist and the PCB as the stage upon which the magician/illusionist works. My simple evaluation and statement of fact in this regard is that, without a suitable stage, the potential of magic or illusion simply will not happen. The PCB and the semiconductor must work together for the show to be a success.

## Looking Ahead

In more recent years, I have personally become ever more appreciative of the importance of PCB designers and their work. They are clearly more knowledgeable than their predecessors. Today, the choices they make are of the utmost importance to the end-product in terms of its functionality, performance, manufacturability, and ultimate reliability. Today, designers must become increasingly knowledgeable of many different design attributes to make their designs suitable for the applications intended: DFR (design for reliability), DFT (design for test), DFE (design for environment), and DFA (design for assembly) are checklists alongside perhaps the most important, which is DFM (design for manufacturing). Several months ago, I suggested in this column, that a better approach might be to design with manufacturing or DWM. This was arguably common practice in the early days of

# BENDING THE POSSIBILITIES

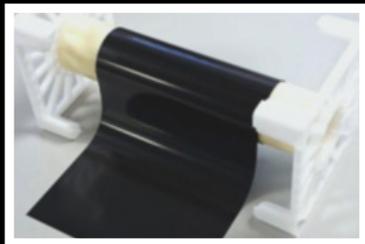


BY TAIYO

AVAILABLE 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=50/50 $\mu$ m
- 👤 Solder mask defined opening (30 $\mu$ m diameter)

**OUR BIGGEST FLEX YET!**

Contact your local sales representative today!

Visit [www.taiyo-america.com](http://www.taiyo-america.com) for more information.

 **TAIYO  
AMERICA**  
UNMATCHED INNOVATION



Follow us on social media



the industry when vertically integrated manufacturers built everything “under one roof” and ties between design and manufacturing were much closer and stronger.

Today’s IC packages and PCB substrates must work flawlessly together to meet requirements, and designers must become increasingly attentive to mechanical concerns alongside the electrical concern. Matters such as CTE (coefficient of thermal expansion) and T<sub>g</sub> (glass transition temperature) need to be part of their design calculus. So, also, will use of predictive modeling to look for prospective failures in advance and address them before they happen. Such analytical software is becoming more common as electronic products find their way into products which must perform in harsh environments.

In summary, the electronics industry at its core is a partnership between semiconductors and printed circuits and the importance of the PCB designers’ work cannot be overstated. They are the “drum majors” of the printed

circuit industry in many ways and it is incumbent upon them to be continuously learning to make certain they keep current on the latest developments in PCB technology, to keep pressing the industry forward. I hope to be around for another 50 years to see what has changed. One thing that seems certain is that the lines between semiconductors and printed circuits will continue to blur as their domains seem destined to continue to merge into the future. **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, 4<sup>th</sup> Edition*, and watch his in-depth workshop series “Flexible Circuit Technology.”

## Using Carbon Key to Decarbonizing Economy

In an opinion paper published in the Proceedings of the National Academy of Sciences, Matteo Pasquali, a professor of chemical and biomolecular engineering, and Carl Mesters, retired chief scientist for chemistry and catalysis at Shell, discuss how to transition from burning hydrocarbons to splitting them into hydrogen and solid carbon. Hydrogen is a clean-burning fuel that produces no carbon dioxide, and the solid carbon could become a cheap and plentiful source of high-performance materials used by a wide range of industries.

Pasquali said the technology already exists to both split hydrocarbons and make solid carbon materials for broad industry adoption.

“The know-how is there,” Pasquali said. “We can make nanotube fibers and composites that outperform metals, but we need to scale manufacturing processes efficiently so these materials can compete with metals on

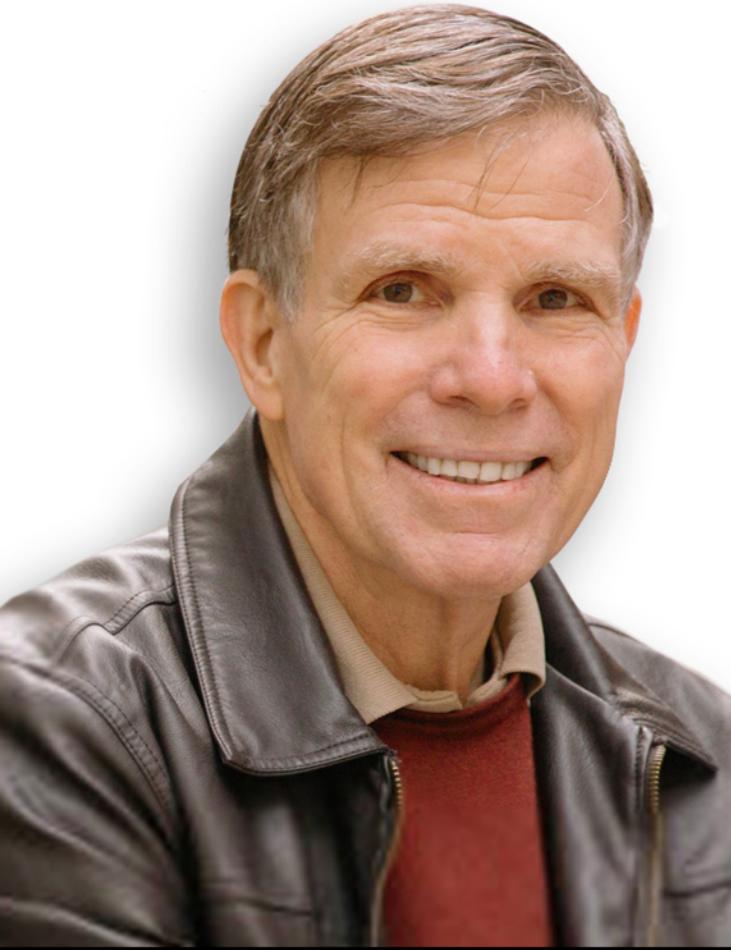
price. If high-performance carbon materials were plentiful enough to compete with metals in terms of price, market forces would take over and we could eliminate metals that today require 12% of our annual global energy budget to mine, process and refine.”

In the PNAS paper, Pasquali and Mesters say the transition to a world where hydrocarbons are split rather than burned “will generate robust growth in manufacturing jobs, most of which will stay at the local level where oil and gas are already established.”

“We’re in a position similar to solar energy a few decades ago: We know we can deliver performance, but manufacturing and scale have to improve to drive costs down,” Pasquali said. “Like solar, this is not a transitional technology: It’s a sustainable future industry. We must get there faster than solar did.”

(Source: Rice University)





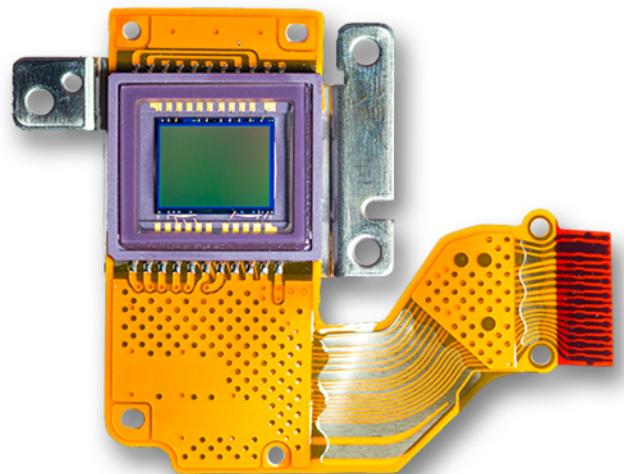
## Online Training Workshop Series:

### Flexible Circuit Technology

with Joe Fjelstad

This free on-demand workshop series is a comprehensive look into the structures, applications, materials, and manufacturing processes of flexible printed circuits.

**START WATCHING**



PRESENTED BY

**I-Connect007**  
GOOD FOR THE INDUSTRY

SPONSORED BY



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



# Flex007 Highlights



## Learn How to Avoid Solder Defects With New Book Authored by Indium Corporation ▶

*The Printed Circuit Assembler's Guide to... Solder Defects*—the latest title in the I-007 eBook library—is specifically dedicated to educating the printed circuit board assembly sector and serves as a valuable resource for people seeking the most relevant information available.

## Real Time with... American Standard Circuits: Flex and Rigid-Flex PCBs ▶

In the first of a series of three Real Time with... ASC conversations, I-Connect007 Managing Editor Nolan Johnson got some no-nonsense answers from Anaya Vardya and Dave Lackey on the topic of flex and rigid-flex PCBs. I was impressed by ASC's generosity in sharing knowledge and technology.

## From DesignCon: FCT Building Fifth Facility in China ▶

During DesignCon, Nolan Johnson spoke with Terrill Schmidt of Flexible Circuit Technologies (FCT) about their exciting facility news and the customer demand that is driving the company's growth.

## ThinFlex Reports 29% YoY Drop in August Sales ▶

Flexible copper clad laminate (FCCL) manufacturer ThinFlex Corp. has posted sales of NT\$194.8 million (\$6.98 million at \$1=NT\$27.89) for August, down by almost 28% from the previous month, and 29% lower than the previous year.

## Matrix Announces Expanded Panasonic LCP Product Line ▶

Matrix Electronics, a leading quick-turn supplier for raw materials to the North American printed circuit market, announced that the Panasonic Felios R-F705S LCP flex product line has been expanded to now include 5- and 6-mil thick LCP materials.

## Taiflex Reports Lower September Sales ▶

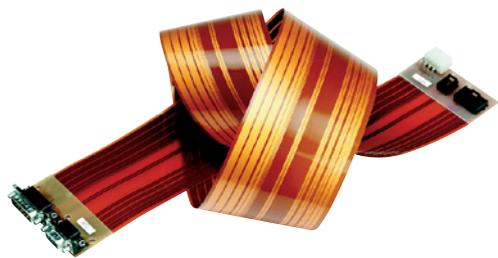
Taiwan-based Taiflex Scientific Co. Ltd, a manufacturer of flexible printed circuit materials such as flexible copper clad laminates (CCLs) and coverlays, has announced consolidated revenue of NT\$864 million (\$30.78 million at \$1:NT\$28.07) for September 2021, down by 1.4% from the previous month and by 10% year-on-year.

## Monitoring Glucose Levels, No Needles Required ▶

Noninvasive glucose monitoring devices are not currently commercially available in the United States, so people with diabetes must collect blood samples or use sensors embedded under the skin to measure their blood sugar levels.

## UCLA Bioengineers Develop New Class of Human-Powered Bioelectronics ▶

A team of bioengineers at the UCLA Samueli School of Engineering has invented a novel soft and flexible self-powered bioelectronic device. The technology converts human body motions—from bending an elbow to subtle movements such as a pulse on one's wrist—into electricity that could be used to power wearable and implantable diagnostic sensors.

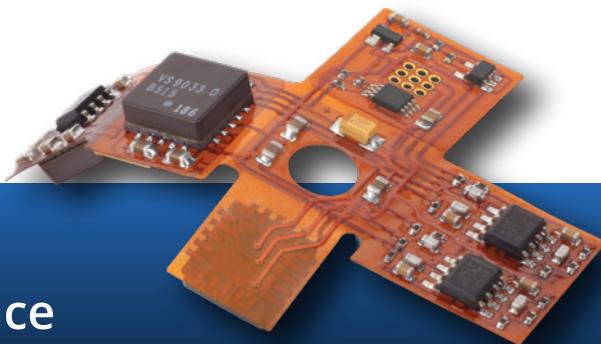


# all flex

## 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](http://allflexinc.com)



# All Flex Discusses Merger and New Medical Applications



Jamin Taylor

**Flex007 Interview by Nolan Johnson**  
I-CONNECT007

I recently spoke with Jamin Taylor, the vice president of sales and marketing at All Flex Flexible Circuits in Minnesota. We discussed the company's recent merger with Printed Circuits Inc., and some creative new flex applications, including flexible heaters and catheters for the medical field.

**Nolan Johnson:** There's been some news recently for All Flex. Why don't we start with the announcement of the merger?

**Jamin Taylor:** About nine months ago, our parent company purchased Printed Circuits Inc., out of Bloomington, Minnesota, with plans for merging the two companies. We're in the planning process right now with PCI to create a single entity and look at rebranding. The two companies have really expanded our technologies, with All Flex and our flexible circuits, and PCI and their rigid-flex capability. Now we can

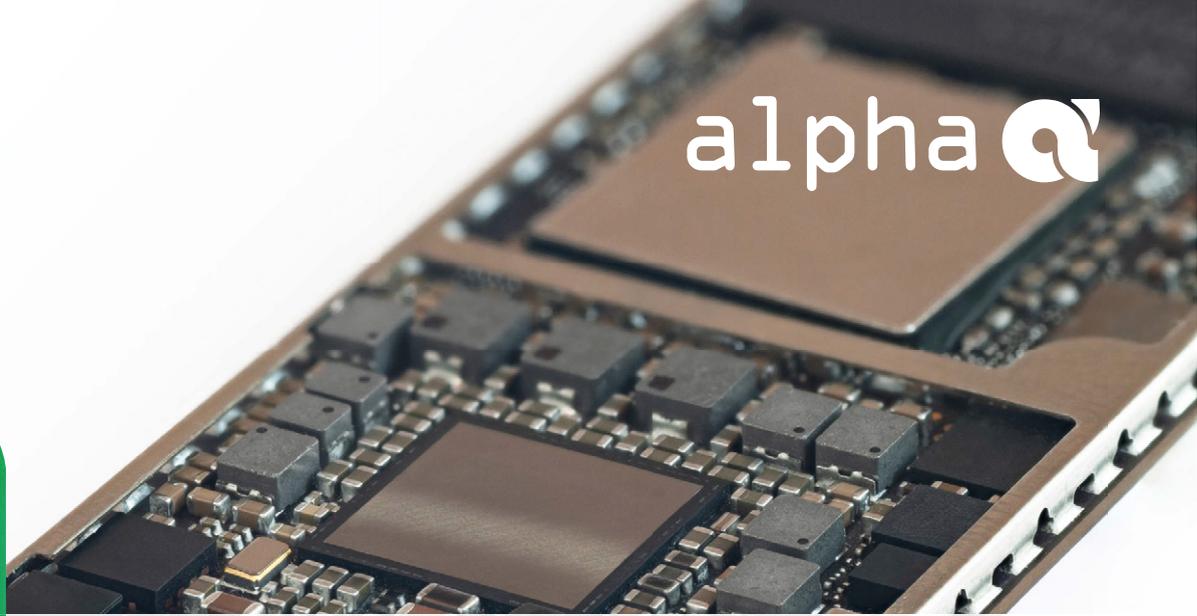
expand our product offering to our customer base in the U.S.

**Johnson:** Were there any pleasant surprises in capabilities now that you've got the two companies together?

**Taylor:** It's always interesting to see different technologies amongst different companies. We're working on a few new applications and products, like being able to do some fine lines and flex materials, but with large format, a longer format than usual. Some of their capabilities with lasering and fine features really help with our long flex capability; when we merge the two, we see a new market being created for that technology.

**Johnson:** I understand you're now developing fine-line catheters. Tell me about that.

**Taylor:** Fine line means small features, 2-mil lines and spaces; that's not new to the industry, but being able to do it in five-, six-, and seven-

The logo for MacDermid Alpha, featuring the word "alpha" in a lowercase, sans-serif font followed by a stylized circular icon containing a white shape.

# ALPHA<sup>®</sup> 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](http://www.macdermidalpha.com)

foot lengths is new. This is for catheter applications in the medical space, so we trademarked Maxi-Flex and CatheterFlex. The Medtronic and Boston Scientific of the world that are doing heart-mapping, heart ablation for AFib, for example, will now be able to do it less invasively by going through the femoral artery up into the heart.

To map or to ablate, you need electronics; using traditional wires was okay, but they're running out of space. You can't make this as large as possible. The flexible circuit allows us to get 64 or 128 channels down the length of this catheter in a much smaller, much more flexible format than traditional wires. It's really enabling the capability of mapping and providing more resolution for mapping of the heart. We're seeing a pretty significant interest amongst the big players in this type of technology.

---

**We're seeing a pretty significant interest amongst the big players in this type of technology.**

---

**Johnson:** Has there been other creative development ongoing in your R&D team?

**Taylor:** Yes, we're also looking at flexible heaters. Traditional flex circuits are made with copper—low resistance, right? We're transferring signals, or maybe some high current-carrying features. There's also the ability to use a resistive foil and create a heating element. The thin film of the traditional polyimides allows for extremely fast heat transfer; with COVID, and in some of the testing that's been done, there's a huge demand for heating elements in biomedical, bacterial, or viral testing. The flu, colds, and coronaviruses can

all be tested in a lab and that lab has to heat up the tissue sample or the DNA sample to a specific temperature and then get a positive or a negative.

The flexible heater technology is not something new. There just haven't been many players in that market. It has been beneficial to be able to do some fine line features, some long features, as well as etch resist to foils. We really looked at that market and think we can expand upon it. We can bring some of the rigid-flex capabilities that PCI has, couple that together, and broaden our market for flexible heating.

**Johnson:** I'm sensing that with this merger, you're finding some creative ways to diversify.

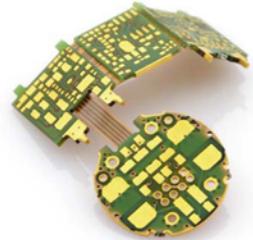
**Taylor:** Absolutely. PCI has been a high-tech company. They've focused on military applications, complexity, multiple layer counts. So, they've developed a lot of good technologies for microvias, blind vias, and fill vias, and we've brought our flex knowledge with medical applications, specifically. We're able to bring a lot of medical customers into that rigid-flex territory. And then they're able to bring a lot of the military and aerospace customers into the flex, and we also do full assembly in-house. That's something that All Flex brings to the table, being able to populate these boards, and with rigid-flex; it's like a motherboard combined with an interconnect, right? There are always components being populated and PCI didn't have that capability. Now we can do more value-add as a combined company and maybe in the future, box-build is something we could continue to look at.

**Johnson:** Jamin, thank you for the update.

**Taylor:** I appreciate it. Thank you very much.

FLEX007

# Focused on Flex



## Laser $\mu$ Flex 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



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

[burkleamerica.com](http://burkleamerica.com)

## Standard of Excellence: Great Things Accomplished With Partnerships

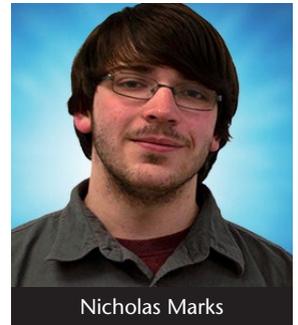


There are some very simple steps to developing and sustaining a good partnership with your PCB vendor/partners, all based

on treating your partners as you want to be treated. It's following the Golden Rule. But before you do that, you must get rid of that other golden rule, "He who has the gold makes the rules."

## Fresh PCB Concepts: HDI PCBs for New Kids in the Industry

I have been working with NCAB Group for three years now. When I began, I had no experience with PCBs and since have learned so much working with our customers' PCB designs. I would like to share the fundamentals and design features of my favorite type of PCB, high-density interconnect (HDI).



Nicholas Marks

## PCB Talk: Collaboration to Shorten the Learning Curve

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.

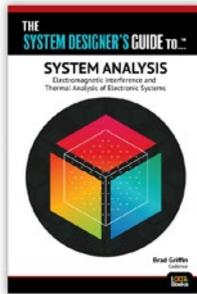


## The Digital Layout: Interaction—The Cure for Industry Contraction

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 which our Chairman Steph Chavez recently delivered to our executive staff. Again, I am happy to provide our readers with a list of events coming up in 2022.

## Excerpt: ‘The System Designer’s Guide to... System Analysis,’ Chapter 2

In Chapter 2 of this book, the subject involves the challenges in the design and development of data center systems. With the exponential growth in data center infrastructure for IT networking, numerous challenges have emerged, from limited ecosystems to high-performance computing issues.



## Dana on Data: Understanding Mechanical Drill Size Capability and Cost

Fabricator capabilities are initially provided on a one-page summary as part of the



general marketing presentation. The technical values provide the “check mark” information so the potential customer can determine if the fabricators capability is greater than the design requirements.

## Adventures in Engineering: Connectors—The Right Fit

Connectors, the physical medium our printed circuit board assemblies (PCBAs) use to purposely interface to their external environment, can make or break your design. I would like to offer my two cents on this topic, a “Penny Primer,” if you will.

## Tribal Knowledge and Design Data Formats

I recently began to investigate “tribal knowledge” in PCB and PCBA operations. My goal was to determine whether anything of value would be lost if it were not recorded or preserved for future industry technologists. In many cases, tribal knowledge seems to be composed of memories of problems solved due to trial and error, coupled with observations of resulting causes and effects.



## Zuken Accelerates Drive to Expand the Model-Based Systems Engineering Business

Zuken Inc. has formed a new R&D unit in Japan. In the U.S., it has appointed a new COO of Zuken Vitech Inc., with the aim of enhancing support for customers that intend to transform their product development process through the introduction of model-based systems engineering (MBSE).

## Elementary Mr. Watson: PCB Design—It’s a Team Sport



One of the hard lessons of this past year 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.

For the latest news and information, visit [PCBDesign007.com](http://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](http://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](mailto:barb@iconnect007.com) to get your ad posted today!**

**+1 916.365.1727**

**I-Connect007**  
GOOD FOR THE INDUSTRY



# Career Opportunities



## Technical Marketing Specialist Waterbury, CT

### JOB DESCRIPTION:

Responsible for providing technical knowledge and support to marketing communications professionals. Cross training and acting as liaison between the Innovation and the Marketing Communications teams for both Circuitry Solutions and Semiconductor Solutions.

---

## Chemist 1 Waterbury, CT

### JOB DESCRIPTION:

Perform analysis—both chemical and mechanical—of customer-supplied samples. These include both structural and chemical testing using various instruments such as SEM, Instron, ICP, and titration methods. Perform various failure analysis functions, including, but not limited to, chemical analysis, SEM analysis of customer parts, and cross-section evaluation.

---

## Applications Manager Waterbury, CT/New England Region

### JOB DESCRIPTION:

Applications Manager in the Electronics Specialties/Circuitry Solutions group to provide applications process knowledge, training and technical support of new products leading to sales revenue growth. Requires working through the existing sales and technical service organizations to leverage this knowledge globally. Experience in multilayer bonding along with dry film and solder mask adhesion processes a plus.

apply now



## Sales Manager New Business Development (m/w/d) – Sales Territory: Germany

CML Group is a global leader in the Printed Circuit Board industry, specialized in PCB manufacturing and sustainable PCB supply solutions. Our products are made to the highest quality and reliability standards, including automotive requirements.

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 responsibilities:

- Develop new customers and build long-term customer relationships
- Understand the customer requirements and acquire new contract enquiries from all market sectors
- Proactive market and customer research
- Identify new potential electronic industry sectors
- Result-oriented sales management including support and consulting on new projects
- Independent management and organization of your accounts
- Price and contract negotiations with customers and contractual partners

### Your profile:

- Several years of professional experience in sales and key account management
- Knowledge of printed circuit board production/industry would be an advantage
- Fluent in Business English and willingness to travel internationally
- Flexible and an open-minded mentality
- Strong communication skills, team player
- Self-motivated, well-organized, professional
- Your home base is in Germany

Interested? Looking forward to your application!  
Please send your application to [hr@cmit.support](mailto:hr@cmit.support).

For more information visit  
[www.cml-globalsolutions.com](http://www.cml-globalsolutions.com)

apply now

# Career Opportunities



**ventec**  
INTERNATIONAL GROUP  
騰輝電子

## Customer Service Representative, UK

We are looking to expand our UK Customer Service/Internal Sales team. As Customer Service Representative you will provide great sales and customer service support and respond to the needs of clients from industries including Aerospace, Defence, Automotive and Pharmaceutical. Duties include:

- Maintain & develop relationships with new and existing customers
- Make rapid, accurate cost calculations and provide quotations
- Accurately input customer orders through bespoke MRP System
- Liaise with colleagues at Chinese HQ and other Overseas Business Units to manage domestic and international requirements
- Assist sales team with reporting, sales analysis and other items at their request

### Skills and abilities required for the role:

The ideal candidate is a proactive self-starter with a strong customer service background. Friendly, approachable, and confident, you should have a good phone mannerism and be computer literate.

- Previous experience in a Customer Service background, ideally management or supervisor role
- Experience with MRP Systems
- Good working knowledge of Microsoft Office Tools such as Outlook, Excel etc.

### What's on Offer:

- Excellent salary & 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 [HR@ventec-europe.com](mailto:HR@ventec-europe.com)

[apply now](#)



YOUR  
JOB  
AD  
HERE

For information, please contact:

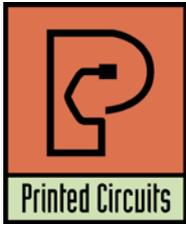
**BARB HOCKADAY**

[barb@iconnect007.com](mailto:barb@iconnect007.com)

+1 916.365.1727 (PACIFIC)

**I-Connect007**  
GOOD FOR THE INDUSTRY

# 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 3<sup>rd</sup> 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



## 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](mailto:danbbeaulieu@aol.com)

[apply now](#)

# Career Opportunities



## 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 manufactur-

ing 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](mailto: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



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](#)



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](#)

---

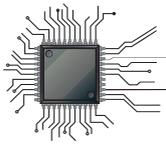
## Drilling Operator

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



**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.

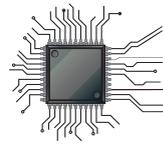
The product manager role will serve as liaison between the external market and the internal design team. Leadership level involvement in the direction of new and existing products will require a diverse skill set. Key role functions include:

- **Sales Support:** Recommend customer solutions through adaptations 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](mailto:N.Hogan@MivaTek.Global). For more information on the company see [www.MivaTek.Global](http://www.MivaTek.Global) or [www.Mivatec.com](http://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](mailto: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



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](mailto: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](http://www.arlonemd.com)

[apply now](#)



## 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



## 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](#)



## 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

## 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](mailto: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



## 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](https://www.jobvite.com))

apply now



**eptac**  
TRAIN. WORK SMARTER. SUCCEED.

## Become a Certified IPC Master Instructor

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

### Qualifications and skills

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

### Benefits

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

apply now

# Career Opportunities



## 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](http://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](mailto: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](mailto:HR@eagle-elec.com) indicating 'Process Engineer' in the subject line.

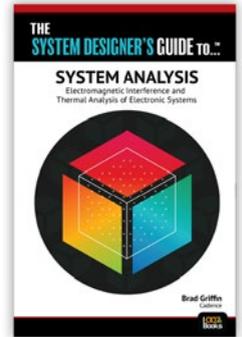
[apply now](#)

## 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!



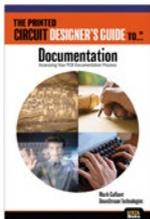
## I-007eBooks The Printed Circuit Designer's Guide to...



### **Thermal Management: A Fabricator's Perspective**

*by Anaya Vardya, American Standard Circuits*

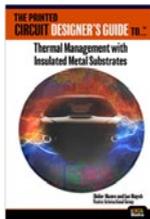
Beat the heat in your designs through thermal management design processes. This book serves as a desk reference on the most current techniques and methods from a PCB fabricator's perspective.



### **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*

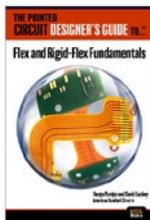
Considering thermal issues in the earliest stages of the design process is critical. This book highlights the need to dissipate heat from electronic devices.



### **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](http://I-007eBooks.com)**

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

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

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

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

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

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

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

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

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

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

TECHNOLOGY SPECIALIST: **BRYSON MATTIES**

COVER: **SHELLY STEIN**

COVER IMAGE: **ADOBE STOCK © SILA5775**

# DESIGN007

M A G A Z I N E

DESIGN007 MAGAZINE®

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

© 2021 BR Publishing, Inc. does not assume and hereby disclaims any liability to any person for loss or damage caused by errors or omissions in the material contained within this publication, regardless of whether such errors or omissions are caused accidentally, from negligence or any other cause.

November 2021, Volume 10, Number 11  
DESIGN007 MAGAZINE is published monthly,  
by BR Publishing, Inc.

## ADVERTISER INDEX

Accurate Circuit Engineering.....	13
All Flex.....	93
Altium.....	21
American Standard Circuits.....	9
APCT.....	45
Averatek.....	57
Burkle North America.....	97
Cadence.....	71
Calumet.....	27
Candor Industries.....	5
Downstream Technologies.....	7, 83
Eagle Electronics.....	67
Electrolube.....	53
EMA Design Automation.....	77
Flexible Circuit Technologies.....	87
HKPCA.....	85
I-007e Books.....	2, 3, 43
I-007e Micro Webinars.....	25
I-007e Workshop.....	91
In-Circuit Design Pty Ltd.....	61
IPC.....	79
MacDermid Alpha Assembly Solutions.....	95
NCAB.....	63
Prototron Circuits.....	81
Precision Technologies.....	39
Pulsonix.....	37
Real Time with...ASC.....	15, 17, 19
Siemens Digital Industries Software.....	49
Summit Interconnect.....	73
Taiyo America.....	89
US Circuit.....	35
Ventec International Group.....	31

# Problems solved!



**SUBSCRIBE NOW**