I-Connect007

JUNE 2022

DESIGN007 MAGAZINE

DWM

DESIGN WITH MANUFACTURING

DESIGN007 MAGAZINE

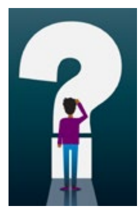
DWM: Design With Manufacturing

Our industry just loves our acronyms. The newest example is DWM, design with manufacturing, and this “design for” could do what DFM was never able to do: create a transparent communication environment for designers, fabricators, assemblers, and component and materials suppliers. This month, our contributors shine a spotlight on DWM: How to initiate it, why you should embrace it, and why DWM may succeed where DFM fell short.

FEATURE INTERVIEWS & ARTICLES

FEATURE ARTICLES

FEATURE COLUMNS



DESIGN007 MAGAZINE

COLUMNS

SHORTS

HIGHLIGHTS

DEPARTMENTS

Will DWM Unite the Product Development Family?

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

In our industry, we just love our acronyms. Especially the "design fors" such as DFM, DFA, and DFT. We now have DFX, DFR, and even DFSC—design for supply chain.

The newest example is DWM, design with manufacturing, and this "design for" could wind up having a real effect on the PCB development process. If designers and manufacturers actually embrace this concept, DWM could do what DFM was never able to do: create a transparent communication environment for designers, fabricators, assemblers, and component and materials suppliers.

Even today, after two decades of talk about the need for DFM, the designer-manufacturer relationship resembles a happy but bumpy marriage. The spouses get along well, but they often give each other the silent treatment all week. And when Friday afternoon rolls around, the accusatory phone calls start:

Fabricator: *Why didn't you mention that you wanted controlled impedance on certain layers?*

Designer: *I thought I told you in the fab notes. Sorry about that.*

Fabricator: *Yeah, well, a little late for sorry.*

Designer: *I can't even remember. I've moved on to another design.*

Fabricator: *You've moved on? Well, isn't that just peachy! Just leave me to clean up your mess. Don't worry about me. I'll be fine.*

Designer: *Here you go again…*

DFM has a positive connotation in our minds. For many companies, a "DFM check"

equals a successful design. But the DFM process itself is clearly not as effective as it could be. The average PCB design undergoes an average of 2.9 respins, according to a recent Lifecycle Insights study—stark evidence that DFM is not being embraced by all stakeholders. Designers and manufacturers often don't begin communicating until many of the most critical, expensive layout decisions have already been made.

But it doesn't have to be this way. Now, DWM promises to get every stakeholder on the same sheet of music, from the beginning of the design process through box-build. You would know your fabricator's sweet spot and preferred format for data package handoff. He would know your design intent from the word "go," help you craft the perfect layer stackup, and be aware of your impedance needs before the 11th hour. And your EMS provider would know about any tough-to-find components far before your bare board arrived.

So, this month, we asked our contributors to shine a spotlight on DWM: How do we initiate it, and what does DWM look like in action? Why should you embrace DWM? How can DWM succeed where DFM fell short, despite the industry's best-laid plans? And when is DWM not necessary and just a waste of your fabricator's time?

We started by interviewing Dana Korf, who explains the need to take the guesswork out of the designer-fabricator relationship, and how to stop making—and accepting—assumptions in the design cycle. Columnist Kelly Dack discusses the benefits of DWM's total transparency among all stakeholders in the product development cycle, and how to open lines of communication between reluctant stakeholders. We also have an excerpt from a previous interview with Happy Holden, in which he details the history of DFM strategies from the earliest days of PCBs.

Next, Siemens' Patrick McGoff takes a counterpoint position. As he points out, today's EDA tools and design techniques already enable complete transparency between designers and manufacturers under the umbrella of DFM. Is DWM just another solution looking for a problem? Then, Altium's Ted Pawela and MacroFab's Misha Govshteyn discuss their recent Altimade partnership, which has created a transparent DWM environment for designers, fabricators, distributors, and EMS suppliers. Kyle Burk of KBJ Engineering explains that there are times to use a total DWM process, but every design is not going to need that sort of focus from your partners. And Scott Miller of Freedom CAD Services discusses their DWM process from a design bureau's point of view, and how to implement such a process at your own company.

We also have stellar columns from our regular contributors: Barry Olney, Matt Stevenson, Steph Chavez, John Coonrod, Jade Bridges, and Joe Fjelstad.

Summer is here, and the design community is heating up too. 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.

What Happens When You Assume?

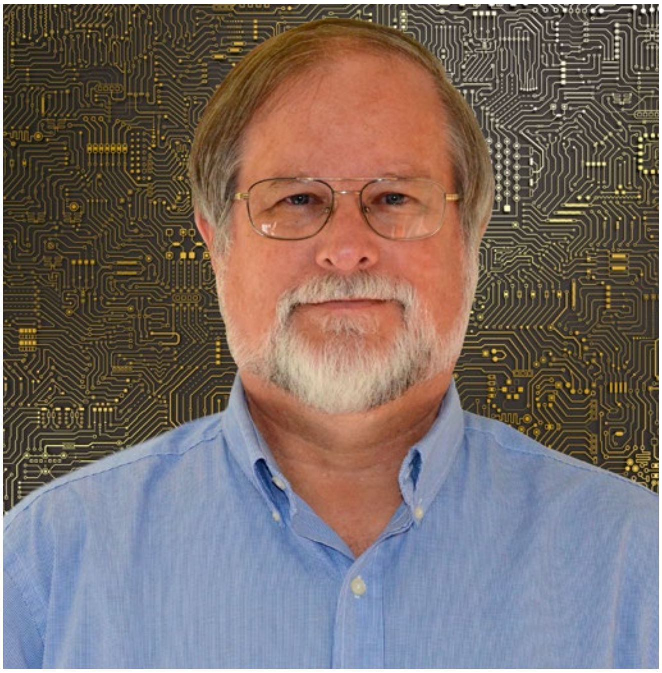
Feature Interview by the I-Connect007 Editorial Team

When we started planning this issue of design with manufacturing, or DWM, we knew we needed I-Connect007 columnist Dana Korf to weigh in on this new acronym, and what true DWM would look like in operation. In this interview, Dana explains what it will take to achieve total communication among all the stakeholders in the PCB development cycle. He also stresses the need for everyone involved in PCB design and manufacturing to stop making assumptions, even at the risk of being labeled as "that guy" who asks too many questions.

Andy Shaughnessy: Dana, we're hearing more talk about design with manufacturing, or DWM. For all the talk about DFM, communication among the designers, fabricators, and assemblers just doesn't happen as often as we might expect. In your opinion, what does true DWM look like? And how do we get designers and manufacturers to embrace it?

Dana Korf: When I first heard about DWM, I thought it implied we were currently designing without manufacturing. I'm not clear what it actually means because that's what we're doing today. I've simplified the problem into three parts. First, the fabricators don't provide all the design rules that front-end engineering uses to evaluate a design for their customers. In some companies, the process engineers won't tell the front-end engineers all the rules.

Second, the data transfer we're using is based on an N/C technique from the 1950s, with a lot of files, and people have to unintelligently guess what the data is supposed to be instead of using more intelligent formats like IPC-2581 or ODB++. Neither intelligent format is perfect, but they're definitely better than Gerber-based packages. Third, the design tool can't utilize the fabricator design rules even if they were provided.



Dana Korf

I did design review a couple of years ago with a tier one software supplier. I found that even if they were provided all the fabricator design rules electronically, they'd still have to perform a lot of scripting to incorporate most of the provided DFM rules. A common example I give is the minimum space between a conductor to the edge of a board. Is the edge scored? Is it routed? Is it punched? How are you making this? These have different sets of rules, most likely, and the design doesn't necessarily have the right rules because the design system doesn't have knowledge of how the edge will be created.

Over the years, we've just accepted this problem. In the '70s when I first got into this, we hand-taped artwork on Mylar. They photographically reduced it down to the proper size, and they built it. It was pretty simple. Then we invented CAM tools. That was where we went downhill, along with the first manufacturer who said, "I'll fix your design for free." That's when design data quality went downhill.

Happy Holden: Yes. I'm not sure that DWM is actually real, because who in manufacturing has that kind of time to devote to a customer?

Dan Feinberg: My first thought is: Do you design for manufacturing, or do you design for the performance of the device? What comes first when you have to make a choice between cost and reliability?

Korf: Right. There are two problems for me. One is that board fabricators may not be selected until after the design is done, and they're selected by procurement or the EMS company. They go out, source it, then select the suppliers at the end of the layout. You don't know who you're designing for. I teach a DFM class for IPC, and one of the things I teach is the IPC definition of DFM. It's something along the lines of, "Design for optimum cost to manufacturability," and I add, "and meets the design intent."

To your comment, Dan, you're really trying to make the board work first and then ensure it's producible. Cost is actually third, because if they all come back and they don't work, you've lost all your money anyway. I teach students that if you're going to design for a manufacturer, figure out whether it's the right one for you. Not every fabricator can build every capability, and not everyone has the same set of design rules for a given technology. That's part of the problem, too. We're chasing cost/price along with quality and performance. It's a triangle; the old saying is that you can only get two out of three of those.

Shaughnessy: It sounds like a lot of these issues arose out of outsourcing. None of this happened during the time of captives.

Korf: That's right. When you had in-house manufacturing, the factory could be tuned toward the product and everything was optimized. You had a lot less variability, a lot better chance of getting good designs out, and getting them right the first time. But, even then, they still had data errors.

Shaughnessy: Right. It sounds like a lot of this stuff is just absorbed. They budget in the money, resources, and time on respins.

Korf: I think it's just an acceptance of our industry. In the IC industry, if they design an IC, it gets built. They don't go back and forth and do three or four versions, because they can't afford it. But our industry, unfortunately, has accepted—and I hate to use the phrase—sending bad data to be built. If every manufacturer just built exactly what they were sent and the designer's company had to pay for the result, I think it would get fixed really fast.

I used to tell customers, "You're paying us for a three-day turn to make your board in three days, but we spend seven days trying to get the data right. You just spent three-day turn money on a 10-day turn." It would usually catch someone's attention and they'd say, "Oh, yeah, you're right."

Feinberg: Do you ever get asked to redesign something for lower costs, for example, or better reliability?

Korf: Yes, and it's more common than you think. The product would fail during development testing. The electrical noise was too high, something mechanical, or something thermal. "Can you adjust something in the design data a little bit?" To reduce costs, what can a fabricator do after a design is completed? It's usually adjusting the materials.

You can't go in and say, "Please re-layout this whole section or reduce two layers." We'd often say, "If you want to reduce costs, take two layers out. Take four layers out. But you have to do that; we can't." Usually, it involves just tweaking material sets. A lot of customers come in, and every three months they've got to reduce the price by X percent.

Feinberg: It's not just the design of a circuit board. When we were designing new photoresists, when we were designing new dry films, we could look at it and say, "Gee, if we can knock 20% off the thickness of that Mylar sheet, that's a significant cost reduction." We were building hundreds of millions of square feet of dry film a year. It was a big deal.

Korf: Sometimes, purchasing will require you to swap out one resist for another because it's cheaper, and the yields go south. You've got to understand the process impacts before you change.

Shaughnessy: Dana, in our last issue, we focused on designing in a vacuum, and how DWM could help designers "get out of the vacuum," as we say. Many of our design readers, and even process engineers in fab and assembly, say that they feel like they're basically working by themselves much of the time. You were working at a high-volume level in China. Did your team ever have to make do without all the information you needed?

Sometimes, purchasing will require you to swap out one resist for another because it's cheaper, and the yields go south.

Korf: That's a very good question. Actually, I'm working with a large OEM right now who wants to improve their documentation package so they will stop getting all these questions back. I was going through their prints and asking, "Do you know what this note means?" The response was, "No, I don't even know what it means. What do you think it means?"

Then I talked to the fabricator. I said, "You don't seem to ask a lot of questions about the prints. Why don't you?" He said, "Well, we assume..." If you're a front-end person, the second you assume, you assume wrong. In a generic quick-turn, customers are paying for speed. They don't have time to spend two days to fix the data or make it perfect. You've got a short amount of time to get it in and out to the floor because the board is shipping in two or three days.

To your question, in that environment, no, they may make some assumptions, fix some things, and may ignore a lot of things because they will make enough yield to ship to meet the cycle time. But in a volume environment when you may lose half a million dollars, or lose a customer if your design comes out wrong, then you're trained to not make assumptions.

I always trained my folks by saying, "If you're in doubt, ask questions." We hear, "Well, the customer hates us asking questions." I don't care. I'd rather ask too many questions. They'll eventually appreciate why we're asking so many questions. Another response is, "The last guy didn't ask that question." Well, I can't speak for the other person's process or process control.

It's our job to make sure we understand what the designer's intent of the board is functionally and then also map it against what the process engineers say they require to make the projected yield for the board.

Shaughnessy: At the SMTA Atlanta Supply Chain Roundtable, I asked a group of designers, "How many of you know where your board's going to be fabricated?" I think one guy raised his hand. A lot of them said it wasn't just a lack of communication with outside entities; it was a lack of communication in their own company.

Korf: I did an IPC presentation a few years ago. I said, "A common Gerber package problem is the fab print dimensions don't match the Gerber file." I just looked out at the audience and asked, "What would you do in that situation?" Everyone responded, "Use the Gerber." Then they all started laughing. They all realized they had the same workaround. No one ever said, "Use the print." Use the Gerber data.

It's conflicting data; even if you fix it, when it gets to the next designer, they don't know that it was bad. They'll just copy it and send it on again, and it's a kind of internal communication. Or you build revision two after you negotiated all the issues on revision one, but the poor designer's already late on the next design.

Feinberg: Anybody ever come in and ask, "We need you to do this, this and this. But by the way, would you also design for Six Sigma?" Has that ever come up?

Korf: Not design for Six Sigma. It's kind of assumed.

Feinberg: It's assumed.

Korf: Assumptions again. One of the issues, Andy, to your point, is that there are style issues in this. We had a merger of several fabrication facilities at a company I worked at previously. I was responsible for all the front end, so we said, "Okay, let's send one design out to every site and have two people at each review it. Then we're going to collect all the issues they found, put them together, and see how common they are."


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

All the reviews came back. We had raised 130 or so different issues collectively. Only one was asked by everybody: "Can I thieve copper?" Other than that, they all found different issues in the same design using the exact same tools, just with different people. There's no such thing as black and white in design. We don't have robots doing DFM reviews. There are a lot of human interpretation factors.

We don't have robots doing DFM reviews. There are a lot of human interpretation factors.

I used to have some of the front-end people go out and visit a designer because they'd say, "How can that idiot do this every time?" Then they'd see all the inputs that the layout person was having trying to lay that board out. They came back and said, "Wow, there's a lot of factors they're accounting for." We'd bring the designers back in to look at the front-end CAM people and say, "Why do those guys keep asking those questions?" Once they saw each other's constraints, they both had more appreciation for each other and it worked out better.

But I love when a customer says, "The last company didn't question that." I'll say, "It's a law of physics problem. They couldn't have built it that way. It's impossible. They edited your data and didn't tell you." Oops.

Shaughnessy: Apparently, true DWM is going to be bidirectional. The assembly folks will know enough about design intent that they can swap out a 1206 for another part and know that it will work.

Korf: That's why, conceptually, all this work in digital twin intrigues me and it's a nontrivial answer. But I wish there was a way that we could automate the transfer of just the good knowledge, not the outliers, and modify the design practices or tools or rules, or the design system. We wouldn't have to rely so much on a human to feed it back, and there's a better chance of everyone's quality improving. I think there must be true feedback for this to really make sense. Eventually, we'll have AI assisting this flow.

Holden: Yes, but we're probably not going to see anything for a while. One of these years, a PhD grad student working on some area of artificial intelligence is going to poke into the printed circuit informational arena as the focus of his artificial intelligence.

Korf: One of our grandkids will figure out AI for PCB design. What's stopping us from doing this? It's possible. Automate the front end tooling process. Well, not the whole front end, but a high percentage of it could be automated. We're not that far away from being able to do it.

That's part of the drive for IPC-2581. The drawings should not contain any design information. It should be a report or a form. If you look at 2581 now, you don't need to send a board outline drawing. You don't need to send a tolerance. You don't need to send an impedance table, a bill of materials. You don't need to send drill codes. All of that is already in the database. They don't need to call out, "These pads get immersion gold. These pads get electrolytic gold." You can label the pads on the board. To your point, Happy, we have to get rid of all this duplicate information, this hand-drawn, extracted information that's so error-prone. For some reason, people are still stuck with drawings in their heads.

So, Andy, when you talk to people, how do they define DWM vs. DFM? Do you get any kind of input on what they think the difference is?

Shaughnessy: Basically, the designer, fabricator, and assembly provider will all be on the

same sheet of music from the earliest stages of the design cycle. The designer will know which fab and assembly providers are best for this job, and the manufacturers will be aware of design intent, because they'll be involved in optimizing the design, and everyone is using ODB++ or IPC-2581. I don't think this happens very often.

Korf: I think it's simple. The designer should have rules that they need to follow to build the board when they start the layout, period. How do we get there? If they had all the rules, it would automatically be designed in the CAD tool. In CAD systems right now, we see legend on pad all the time. I keep asking the CAD vendors, "Why? Don't you folks see that there's legend on pad? Why can't you just delete it automatically? Why do you let it go through?"

Also, mask on pad, why do you let it happen? It's your design. Why would we have to fix it and get penalized when we don't catch it? The CAD tools, if they had the rules then they could automate some of these checks and balances. The defect may not ever make it on the board. It's really pretty simple. How to get there is the hard part.

I was doing a presentation at an IPC DFX meeting a couple weeks ago, and I said, "What does DFX mean to you guys, in this crowd?" We talked about it for a while. I said, "Well, for me, what's missing in IPC specifications? Most designs use them as a reference. The documents don't give people enough design rules." I've put together a document I'm willing to give to the industry to put into DFX guide and say, "Here are some rules that are pretty common throughout the world." I put these together over the years. I'm willing to give it to them to help us get improved DFX. The industry needs to help designers avoid creating bad board datasets because they don't have all the rules.

Feinberg: That's a good point. Have you brought up design rules to the IPC?

Korf: I just brought it up at our last DFX committee meeting. I think it has some attraction to people. Also, we need to improve specifications, you can't build just per IPC specifications. One example in one IPC specification: there are 40+ AABUS's, in other words, as agreed between you and the customer. That's not a specification. It's a design/producibility rule. We should talk about how to improve these documents.

Many companies have a secondary specification which says, "We also want you to do the following items," such as, "The PCB shall be RoHS compliant. We require IPC Class three via plating requirements, but Class two with everything else, etc." I've put together a generic version of that secondary acceptance specification. I'm offering that document to the industry to provide more detailed information for companies that don't have this. This specification, along with appropriate IPC specifications, should be able to provide enough information so someone could design a perfect board without any issues being noted by the manufacturer.

The industry has the IPC-2581 data format which is not perfect, but close. Get the specifications into software. Let's get the design rules out there. Maybe we'll have a better shot at doing this right. I don't know. It's just another attempt to try something, right?

Feinberg: Well, that would eliminate the designing in a vacuum. If you had an IPC class A set of design rules, which is 95% of the industry can build this with high yield and high reliability or something, and then you had design rules B, and design rules C and D as you went up, the design rules became tighter. You had some kind of test vehicle to see that a company could build to see if they meet the criteria of these higher ones. Then to be safe, well, we're always going to default to IPC design rules A. At least you can tell that this is a B, C, or a D, or something like that. If you don't answer all the questions, we'll default to that.

Korf: Yes. I usually give them, "Here's a standard in advance." In my DFM class, I actually give these documents to the students, and I say, "It's an open PDF file. I want you to send it to your vendors and have them fill out the columns. Have them edit it for you and send it back to you." If you tell them, "I want all the rules," they don't have enough time to write them all down. But if you send this, instead of starting with a white piece of paper, you start with something and edit it. You'll get something back faster.

Feinberg: Has anyone tried that?

Korf: I haven't heard back yet. I've heard back from a couple who have gotten some pushback from the suppliers.

Feinberg: It would be really interesting as a follow-up to find out if anyone has done that and gotten any response, and what the response was.

Korf: Actually, I'll agree. Let's go out and query all the students and see if these made any difference. Did the class make any difference? It's the same with brand-new PCB designers. What's the first documentation or training they need to start laying boards out? Well, we should have these sets of design rules in their face. At least they have something rather than a bunch of yellow sticky notes around the monitor, right?

Shaughnessy: Right. Dana, was there anything else you wanted to mention?

Korf: Well, if we're talking about DWM, what do we call what we're currently doing—designing without manufacturing, or DWOM?

Shaughnessy: I assume! (laughs)

Korf: We have to stop assuming. (laughs)

Shaughnessy: Thanks for your time, Dana. This has been really good.

Korf: Thank you. It's been great, guys. **DESIGN007**

Dana Korf is the principal consultant at Korf Consultancy LLC, and an I-Connect007 columnist. To read past columns or contact Korf, click here.

Designing With the 5Ws and Other Acronyms

Target Condition

Feature Column by Kelly Dack, CIT, CID+

Who, what, when, where, why, and *with*?

I've slowly become disillusioned over the past decade with the whole "design-for" schtick. Design for manufacturing or "DFM" has become extremely subjective. I am even becoming hesitant to splatter the acronym "DFM" when referencing job descriptions or when considering a layout tool's manufacturing analysis audit because the term is so cliché. I know it shouldn't be construed in such a way, but from my perspective, running a DFM check on a PCB design layout is inadequate unless the audit routine checks against the manufacturing constraint values of the volume supplier which, in most cases, it is not until it is too late.

Design for Failure

Believe it or not, there was a time years ago in PCB design when PCB designers were influenced to believe that they alone oversaw the successful outcome of a PCB design project. To them, DFM meant they had convinced themselves they had the knowledge and capability of producing "camera ready" PCB artwork and foisted their design data down from their perch for use by the nether-realms of manufacturing for fabrication, with notes which read like the "thou shalt nots" written on the tablets of stone.

"Thou shalt not modify this artwork without express written consent from company XYZ engineering."

Manufacturing for Success

In my role working for an EMS provider, I am given the opportunity to review dozens of PCB designs each week which have been created by designers from around North America and Europe. The design output can reflect a very simple two-sided PCBA or a very complex multi-layer rigid-flex PCBA utilizing ultra-fine pitch devices on one section while sporting relatively massive power supply component clusters on another. Sometimes the designs are problematic for our offshore volume suppliers, but we do not know until we begin our manufacturing workflow audit.

Sometimes the designs are problematic for our offshore volume suppliers, but we do not know until we begin our manufacturing workflow audit.

One of the first steps in setting up for manufacturing success in any manufacturing workflow is to take measurements. In the world of electronics manufacturing services (EMS) an audit of the designer's manufacturing data must be performed to make certain that all the data required to produce the bare PCB is viable. Simplified, the workflow goes something like this:

Audit → Report → Correct → Proceed

But because of various degrees of understanding regarding the subject of DFM, the PCB manufacturing workflow almost always stops after the "report" phase due to an unmanufacturable issue measured within the design data. The PCB supplier's computer automated machining (CAM) department performing the audit must issue a report outlining a suggested fix, which must be sent back to the EMS provider, who must present the issue to the original customer to modify the data or allow the supplier to modify it before the job can proceed. The process is very time consuming and must not be taken lightly as millions of dollars could be at stake if a manufacturing assumption is made and changed without approval from the customer.

Going back to the original PCB designer to inform them that a supplier has identified some "er, uh, DFM issues" is not always met with a spirit of gratitude by customers, nor a response query like, "How can we help?" An EMS provider is often put into the position of tactfully communicating a need for clarification or even modification of the existing design data because the production supplier has, in effect, "called their baby ugly."

But the problem is hardly the PCB designer's fault. It is commonly known that many designers do all but pull teeth to obtain relevant design constraints before starting a PCB layout. The May issue of *Design007 Magazine* covered the topic of designing PCBs in a vacuum and interviewed a few designers who admitted that they are often put in the position of flying blind at the beginning of a project without having a clue as to who would be fabricating the PCB in volume production. They emphasized that PCB designers are simply not given enough vision to start a PCB design with the end in sight. They are forced to do the best they can to incorporate a knowingly inadequate, cliché form of DFM without considering the five Ws in their unique contexts.

- Design for *who* in manufacturing?
- Design for *what* in manufacturing?
- Design for *when* in manufacturing?
- Design for *where* in manufacturing?
- Design for *why* in manufacturing?

If PCB designers cannot be provided enough information to gain a five Ws type of vision regarding PCB manufacturing in advance of starting a project, is it time to consider perhaps designing PCBs with a sixth W—*with*—as in "with all the other PCB design and manufacturing stakeholder counterparts."

I've contemplated a re-write of a general job summary for a PCB designer to somehow reflect this.

Job summary: Performs PCB design and CAD layout of complex printed circuit assembly products as part of a project team comprised of procurement, mechanical and electrical engineering, fabrication, assembly, and test stakeholders. Works *with* procurement to maintain awareness of key suppliers, location, logistics, capabilities, and contractual agreements. Works *with* mechanical engineering to evaluate and provide feedback on mechanical design constraints. Works *with* fabrication and assembly stakeholders to ensure layouts incorporate design for manufacturing and assembly philosophies which correspond to appropriate supplier capability. Works *with* test engineering stakeholders to ensure functional and in-circuit test strategies and requirements are well-defined at the beginning of a project and successfully implemented.

The Key to Designing PCBs *With*

But wait a minute here. One PCB designer might be thinking, "Wow, how can one PCB designer be expected to sit alongside so many project stakeholders at once while laying out a PCB? Another may exclaim, "Too many cooks in the kitchen." Still others, shaking their heads with arms folded, may say, "If Peter sees how I've robbed him to pay Paul, there's going to be a fight." Please, let's put this cozy version of designing together aside. It's not where I am going here. The key to designing with will not be found by pushing all the project stakeholders together into an office and not letting them out until a viable design is created and released to manufacturing. The key to "designing with" lies in changing the way we think about who has access, and can leverage and add rich improvement, to the source PCB design data throughout the life of a project.

Designing with PCB manufacturing stakeholders of the future will require interconnection between machines, devices, sensors, and stakeholders who will be communicating with each other using the internet of things. It will require massive collection of information and transparency so that informed decisions can be made over all parts of the manufacturing process. It will utilize systems created to help stakeholders with decision making and even utilize systems created to make decisions automatically, all based upon steady streams of gathered data.

Industry 4.0, CFX, Intelligent Data Formats, and Lights-out Manufacturing

Can we even imagine PCB layout data so powerful, so sharable and visible, that it could be viewed "live" by any and all project stakeholders 24/7 and run manufacturing systems autonomously—ordering materials, adjusting the bill of materials—even automatically updating footprints in the dynamic PCB layout correcting for supply chain issues? It's coming. And I believe this is a whole new take on the concept of "designing with" as I've heard it discussed. It is much different. You may have to make that popular brain-exploding gesture with your hands flaring out away from both sides of your head when you begin to grasp what some established electronics industry visionaries consider with regard to designing with.

These folks aren't talking about PCB designers sitting down to design with all the other stakeholders and getting their feedback to incorporate into their layout. They are envisioning all stakeholders and their automated machinery having access to the intelligent PCB design data via the internet of things—after the PCB designer completes the layout—to dynamically and automatically change a design after the designer is off onto another project. This spin on DWM spreads requirements for production out to the stakeholder ordering and manufacturing maintenance systems which will automatically Audit → Report → Correct → Proceed based upon accessible, intelligent data and monitoring sensors throughout the life of the product.

If you are a PCB designer or PCB project stakeholder of any discipline, I recommend studying subject matter regarding Industry 4.0, IPC's CFX (IPC-2591), data, sensors, the internet of things, and various articles on the topic of lights-out manufacturing. Your future designing with one another is well laid out and rather mind blowing, to say the least.

Conclusion: The Next Acronyms

New industry paradigm shifts seem to be taking a toll on the future of our old "DF" acronyms.

A few years ago, due to evolving technologies and a need to better explain what the heck designers should design for, the term DFM had to scoot over so our industry could make room for its other DF acronyms: DFA (assembly), DFC (cost), DFT (test), etc. Not long after, the industry seemed to realize that there just might be too many things to design for. We then watched as all the DFs suffered the humiliation of being lumped together into one big DF acronym with a cute little x added onto the end to represent all of them (DFX).

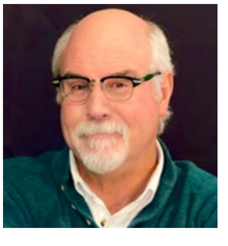
But recently, just as it seemed as though all of this DF acronym jazz had started to settle down, folks in the industry have perhaps begun to see the folly of PCB designers designing for stakeholders with whom they have no visibility, no contact, and no constraints. About a year ago, someone shouted, "How about you design with your other stakeholders!" Some clever marketeers in our midst must have thought "Well, duh!"

I don't mean to sound cocksure, but just for fun over these next couple of years, I intend to watch how our industry treats this new acronym. The way it's already being perceived differently by various stakeholders in the industry, I predict it will go through much of the same treatment cycles as DFM. We will soon see design with fab, design with assembly, design with test, etc. Then we will read about how all the DWs were merged into DWX and realize that our potential for seamless, trouble-free design and manufacturing is crippled because we continue to share non-intelligent, dumb data. Then, as the script goes, we'll need a new acronym. What's next?

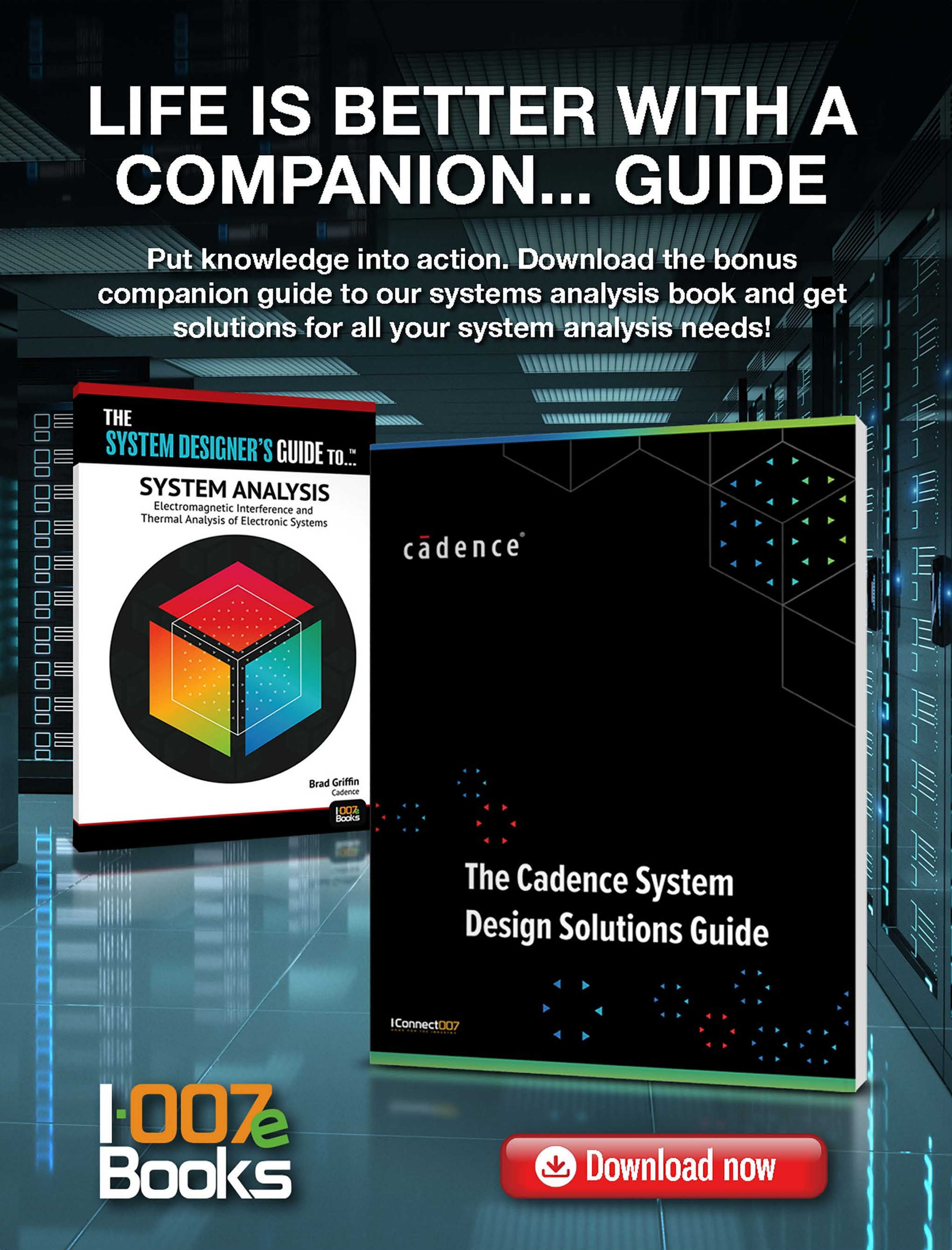
New industry paradigm shifts seem to be taking a toll on the future of our old "DF" acronyms.

With excitement we've been watching the electronics manufacturing industry begin to incorporate amazing technological advancements into their processes and machinery. The next industrial revolution, Industry 4.0, is interconnecting precision robotics, advanced materials, processes, and people which are being brought together to plan, purchase, design, manufacture, assemble, and test. The power of IPC's Connected Factory Exchange (IPC-2591) standard has been loudly on display for the past few years on the IPC APEX EXPO show floor and the industry is engaging.

Move over DFs, DWs and 5Ws, it appears that the CFX (Connected Factory Exchange) will be designing with the internet of things come Industry 4.0. DESIGN007



Kelly Dack, CIT, CID+, provides DFX-centered PCB design and manufacturing liaison expertise for a dynamic EMS provider in the Pacific Northwest while also serving as an IPC design certification instructor (CID) for EPTAC. To read past columns or contact Dack, click here.


LIFE IS BETTER WITH A COMPANION... GUIDE
Put knowledge into action. Download the bonus companion guide to our systems analysis book and get solutions for all your system analysis needs!
THE SYSTEM DESIGNER'S GUIDE TO...™
SYSTEM ANALYSIS
Electromagnetic Interference and Thermal Analysis of Electronic Systems
Brad Griffin
Cadence
I-007e Books
cādence®
The Cadence System Design Solutions Guide
IConnect007
I-007e Books
Download now

Predictive Engineering: Happy Holden Discusses True DFM

Feature Interview by the I-Connect007 Editorial Team

Note: This is an excerpt of an interview with Happy Holden that appeared appeared in the September 2017 issue of The PCB Design Magazine. In this interview, Happy traces the development of DFM over more than 50 years, providing a history of the post-war electronics industry as well.

Happy Holden has been involved in DFM for over 45 years since he first started working at HP and optimized their PCB design and manufacturing processes. Naturally, for this issue, Barry Matties and Andy Shaughnessy made it a priority to get Happy's thoughts on DFM, and what true DFM entails.

Andy Shaughnessy: Happy, why don't you start by telling us about your position on DFM, and why true DFM, to you means that PCB designers and engineers should utilize predictive engineering?

Happy Holden: If you use the term "predictive engineering," you're not going to get much recognition, because that's a term I use. What most people consider design for manufacturing, or DFM, is software that finds errors in design. Especially in CAM tooling put out by Valor, if a manufacturer puts in his design builds in a minimum, then the DFM software scans through that and finds what's cautionary and that all requirements are met. That's all done after the fact.

For me, DFM was introduced when HP took up DFM. It was kind of invented by Professors Peter Dewhurst and Geoffrey Boothroyd. Anyway, DFM and DFMA (design for manufacturing and assembly) were really invented by these two American professors in New England. They wrote a book about it, *Product Design for Manufacture & Assembly.* It later became software.

Their whole philosophy centered on ways to figure out how to do it right the first time. Their book is about the nature of performance dur-

ing design that allows you to decide if this is going to be easy to assemble. I think the more important component, that Hitachi and the Japanese took up, was whether the product could be built by automation. It's really significant what Hitachi did with the philosophy. They not only took the "do it right the first time" approach but then asked how they could do it simple enough that robots and automation could build it. The Japanese did this because they wanted to build all these first-generation products in a building next door in Japan and not ship it offshore like Americans were doing.

Happy Holden

Contrast that with Apple Inc. Apple designs products that are impossible to automate. They're so complex. They're really fanciful, but they have to be built by human hands. You've got 8,000 17-year-old teenage girls in Shenzhen building these Apple products. Foxconn put up money to install 1 million robots to replace these girls. Today only about 400 robots have been installed out of that 1 million, because they found out the products they're building for the Americans can't be automated.

I refer to DFM as being design for manufacturing the first time. We're not just designing it, running software, finding the errors, going back, redesigning it, checking again, going back and re-spinning it until it works. I'm alone in this definition unless you happen to be knowledgeable about Dewhurst and Boothroyd, which most people in printed circuits aren't.

Shaughnessy: I would imagine.

Holden: Manufacturers such as General Electric, Westinghouse, or General Motors understand Dewhurst and Boothroyd, because it's taught at universities. The automotive guys are really big on trying to simplify parts in automobiles to make them more reliable at lower costs and easier to be built up by automatic systems. But in electronics, it's never caught on.

One of the reasons is that Dewhurst and Boothroyd were mechanical engineers. They worked off the kinematics of how many motions it takes to assemble something. How much fixturing, connectors, or screws and bolts it takes to do it. That's their predicting methodology.

The simplest form of automation is the one-axis pneumatic cylinder, or air cylinder. It just goes up and down. If you're going to assemble something on a conveyor belt, with just up and down motions, that's the cheapest automation. Some of the most complex assembly is assembling flexible circuits, which I managed for Foxconn.

It's one of the reasons they kept showering me with money to automate out these 8,000 girls I had in Shenzhen who would do the final non-conformal part assembly of flex circuits, because they would work for six months on Apple products. When Apple stopped ordering after Christmas, the girls would sit around for the next four months and knit, because there were no orders. We couldn't fire them or lay them off because they were too highly skilled.

They sat around and knitted and painted walls and things like that until Apple started ordering again. Then they all went back to work. One of the highest priorities I was given, with an unlimited budget, was to automate out these girls, so that when Apple stopped ordering we'd just turn the switch off. I told them the bad news was, "Well, maybe in 10 years." **DESIGN007**

To read this entire article, which appeared in the September 2017 issue of *The PCB Design Magazine*, click here.

DWM: A Rose by Any Other Name

Feature Article by Patrick McGoff
SIEMENS EDA

We all seek new ideas, but sometimes what is proposed as new is merely a re-phrasing of something that already exists. That is the case with the phrase, "design with manufacturing."

The rationalization behind design with manufacturing is that it involves manufacturing input during the design process. I couldn't agree more. But in what form does this manufacturing input take? Does design with manufacturing imply a live meeting? If not, then what is meant to be the difference between design with manufacturing and design for manufacturing?

The genesis of providing manufacturing input into the design process in a systematic manner goes back 25 years. Since 1995, PCB fabricators have been providing their process constraints in digital form to their customers, ensuring that their designs can be fabricated without issue or delay in the initial design release. This enabled the dawn of what is widely known as "design for manufacturing (DFM)."

Design for manufacturing begins with capturing the supplier's process capabilities in a digital format that can be automatically read into the OEM designer's DFM software tool. It is important to look at DFM from a manufacturing process perspective, because how the board is constructed determines what constraints must be adhered to. There are completely different checks and clearances needed

for a mechanically drilled single-lamination PCB vs. laser-formed microvias on a board that is sequentially laminated. Accordingly, not only are multiple rule sets based on the manufacturing processes required, but you also must consider that each fabricator you work with will have different process capabilities, and therefore different constraints.

It is valuable to understand the true objective of DFM, which is to help designers optimize their designs for manufacturing during the first release. Nobody can afford to go through three or more revision spins to move from prototype to volume production. Designers want to accelerate the new product introduction (NPI) process and get to market earlier. To accomplish this, visibility is needed into the issues that will impact yield, cost, and reliability during the initial design process—not after each step of the production ramp-up. Thus, the constraint sets that are provided by the manufacturer need to include the level of detail that informs the designer of where the design could be improved to manufacture it at a lower cost or with greater reliability. Manufacturing is a process with tolerances—it's not a binary pass/fail.

It is valuable to understand the true objective of DFM, which is to help designers optimize their designs for manufacturing during the first release.

How the process constraints are communicated is also equally important. Understandably, many manufacturers treat their process capabilities as confidential information, and they want to control with whom they share this level of detail. So, it becomes critical that the digital means of sharing that information be protected and kept within the manufacturer's control. The design OEM needs to be able to benefit from access to the manufacturer's constraint set, but they don't actually need to see what's driving it. I use the analogy of rideshare programs. You derive the benefit of getting a ride from point A to point B, but you don't need to look under the hood of the car to get there. What is paramount is that your DFM analysis is aligned with your supplier's DFM analysis.

It needs to be easy for the manufacturer to maintain its constraint set for the benefit of its customers. As process capabilities improve, the manufacturer needs a platform that allows for posting its updated capabilities for customers to access immediately. Today, it's a painful "drip," inconsistently communicated.

Let's keep in mind that DFM entails all aspects of the manufacturing process—bareboard fabrication, PCB assembly, electrical test, stackup planning, and even component sourcing. A true DFM collaboration platform can arm designers with a digital twin for each of these disciplines in a manner that integrates seamlessly into their PCB flow. Collaborating on the stackup recipe before a designer is asked to lay out a board is as critical as the constraints they design to directly impact the circuit's performance. If your goal is to optimize your designs for manufacturing on the initial release, it is imperative to design and analyze using the actual stackup materials that will be used to fabricate the PCB.

As I have mentioned previously, DFM is not a binary process. Severity indicators existing within "grey areas" are a vital part of understanding and making informed decisions for manufacturing. A designer, for example, would benefit from knowing that a placement decision they made for a component will cause that component to be placed manually by the contract manufacturer (CM). The CM can build it, but the extra touches will cost the OEM more. The concept even applies to rework. A CM

will use DFM to assess the difficulty of having to rework a given component and factor that into their quote. That's information a designer would like to know before it's too late to do anything about it.

We should also remember that the commercial aspects of components add a risk to the manufacturing cycle. True DFM encompasses tools for mitigating that risk and providing visibility of choices to the design OEM for factors such as availability, cost, and obsolescence.

This is what design for manufacturing has evolved to be today. There are established platforms and applications that enable all the partners in a supply chain to collaborate with ease in a digital manner. While it is good to see other EDA companies finally realize the importance of DFM, we don't need a new name for the rose. The one we have smells quite sweet. **DESIGN007**

Patrick McGoff is market development manager, Embedded Board Solutions, Siemens EDA.

COLUMN EXCERPT: **All Systems Go**

Electronic Data Management—Can You Design Without It?

By Nitin Bhagwath

For any sizable design, PCBs are usually designed by a team of multiple design engineers (EEs) creating the schematic and multiple layout designers placing all the parts on the board and routing the traces. These teams often work with an extended team of experts in the supply chain, signal integrity, and mechanical and thermal analysis. Engineering management also has a stake in the design process, as it monitors design progress, resources, and scheduling. For a successful design, this multitude of interactions requires mandatory mechanisms to keep everyone on the same page during the design process.

Design teams worldwide are running into problems with supply chains (see All Systems Go! Supply Chain Woes—Which Comes First, the Design or the BOM?). Consequently, it's critical that designs not use obsolete parts or parts that cannot be sourced. Therefore, good communication and coordination systems must be in place between the design and the supply chain teams.

Similarly, if someone has made a change in the design and believes, erroneously, that they have communicated this to you, you could be working from bad assumptions. In contrast, if you make a change that isn't communicated to the greater team, everyone else could be working off a false premise.

Another common situation is for someone to make a perfectly valid change, which is communicated to everyone who needs to know. Still, the reason for the change is lost because it wasn't documented anywhere (or maybe it was, but no one remembers where this document resides, which amounts to the same thing). These scenarios happen far more often than most people think. At this very minute, design engineers and layout designers around the globe are looking at their computer screens, saying, "Who did this?" Or, "I know I did this, but why did I do it?" Or, "This made sense when we did it, but it's not working as planned, so why did we choose to do things this way?"

This can become overwhelming, especially if you don't know whether you have access to the latest information. Teams often develop workarounds (aka Band-aids) to address these problems.

Nitin Bhagwath is director of product management, PCB front end, at Cadence Design Systems.

To read this entire column, click here.

Copper Pours in High-speed Design

Beyond Design

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

The most common question I get asked by PCB designers is, "Do you need copper ground pours on digital multilayer PCBs?" The short answer is, "It depends." Unfortunately, the myth of copper pours is fueled by reference designs that seem to persistently use this old RF design technique. Copper pours are sometimes used, incorrectly, simply to fill in the unused space on a board. However, in some cases ground pours may be an advantage. In this month's column, I will look at where to use and where not to use ground pours.

There are three reasons for using copper pours on multilayer PCBs:

1. **PCB fabrication:** The primary reason is to provide a uniform copper distribution across a layer to make etching and plating more predictable, and to reduce the amount of etching fluid used (hence cost) during PCB manufacturing.

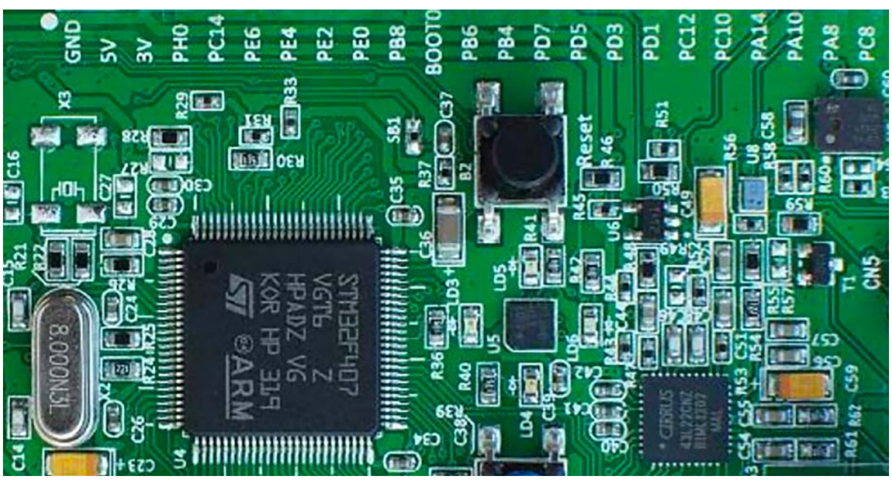


Figure 1: Unnecessary copper GND pour on outer layers.

2. **RF and microwave design:** Ground pours contain the electromagnetic (EM) radiation in a localized area thus reducing spurious coupling, radiation, and dispersion.

3. **Digital design:** A ground plane improves noise immunity, gives much greater grounding uniformity, and provides a low inductance current path to devices.

Please note the distinction between copper pours and ground (GND) pours/planes. A copper pour per se is an isolated copper plate. This is not a good idea since it buys you little benefit, whereas a ground plane is connected to the distributed ground net.

During the PCB pre-fabrication process, the CAM engineer will generally add copper pours to individual layers of a multilayer PCB. This is referred to as copper thieving and involves placing a pattern of repeated simple geometrical shapes or cross-hatched regions into large void areas of a PCB. But this is not ground—it is floating copper. Copper thieving is best placed outside the board outline to avoid influencing the functionality of the design.

Some designers advocate leaving the copper thieving decisions up to the PCB fabrication shop, but this may not

necessarily be the best design decision. While the fab shop will create thieving that will best fit the plating needs of the board, they will not necessarily have all the tools available to determine the electrical impact of adding metal to the design. Floating (unconnected copper) is a real hazard for signal integrity. All copper pours must be connected to ground or power net. A ground pour that is not accompanied by grounding vias can become a conduit for crosstalk between the traces on either side of the ground shape. The CAM engineer could inadvertently be creating antennae that radiate with the application of short wavelength signals. It is crucial that any metal changes to the board be done by the PCB designer so that all the potential effects can be taken into account. You can ensure this by stipulating in the PCB specification that no copper features be added or modified without permission.

PCB designers also can use these copper pours as more than just thieving patterns. By creating the copper thieving as part of the ground net, and configuring it as a solid plane, you can assist with providing a well-defined return current path. The caveat, however, is that this must be done in conjunction with the overall signal performance requirements of the design. Any additional planes must be positioned and spaced correctly, so they do not have a negative effect on the signal integrity of the board. The processor board (Figure 1), for instance, has what I call random copper GND pours on the outer layers. Many of these pours are totally unnecessary.

Applying this concept in RF circuits, grounded metal is sometimes used as fill between the signal traces on outer layers. The grounded areas should be connected to the ground plane with vias (stitched) at less than one-quarter wavelength (Figure 2). This technique changes the transmission lines from microstrip to grounded coplanar waveguide. The characteristic and differential impedance of the transmission line is lowered by using this technique because the copper pours increase the capacitance between the signal trace and ground.

The coplanar waveguide (CPW) is the transmission medium of choice above 20GHz. CPW

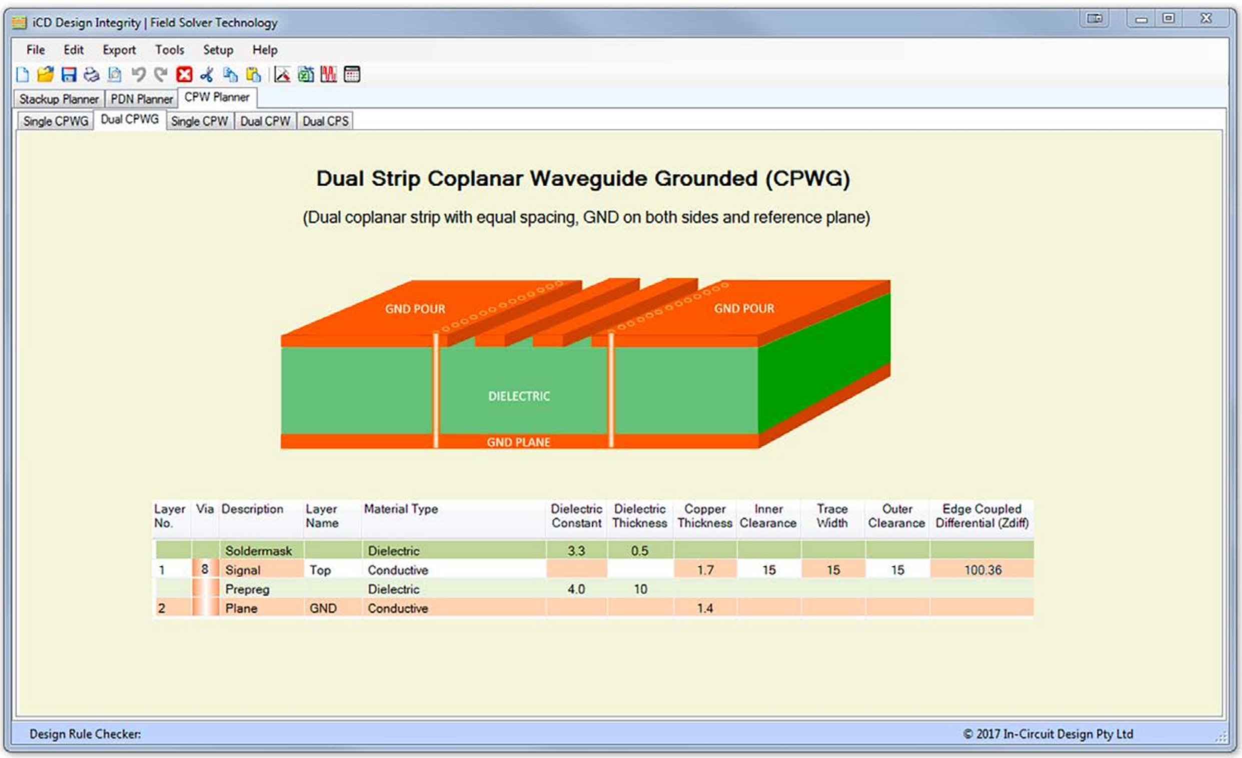


Figure 2: Dual strip coplanar waveguide grounded. (Source: iCD CPW Planner)

combines the EM fields in a more localized manner than does microstrip, thus reducing spurious coupling, radiation, and dispersion. Also, the CPW provides a precisely defined signal return path.

Internal stripline configuration has an advantage over outer layer microstrip. Microstrip can radiate. However, with the fields confined between ground planes, stripline does not. Signals can be shielded by placing the traces on internal layers between copper planes. These form stripline transmission lines. Such techniques are recommended for signals above a few hundred megahertz.

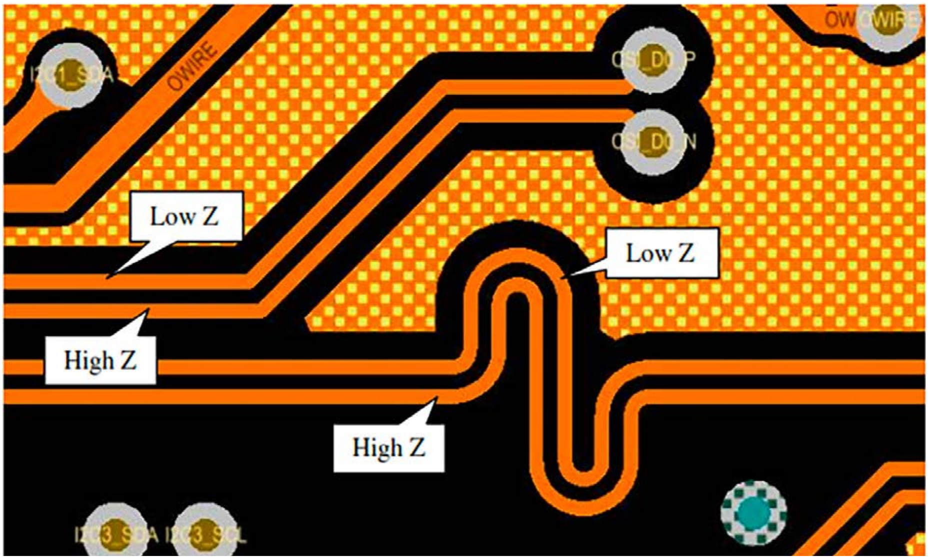


Figure 3: Copper pours lower the impedance of nearby traces.

Grounded (guard) traces are sometimes used to surround digital clock signals, which tend to have fast edge rates. In addition to having a ground plane below the signal trace, copper traces are placed on either side of the trace and are stitched to the ground plane in multiple locations. For digital systems, this technique is only useful for traces on outer layers. On inner layers, it makes more sense to just leave some empty space around the trace. However, if microstrip is very closely coupled to the reference plane (<3 mil) then radiation is minimal and providing ground shielding on outer layers has little benefit.

When copper pours come into proximity with critical signal traces (Figure 3), the impedance is reduced by 2–3 ohms. So, on the copper side of the differential pair, the impedance will be lower than the other side free of copper. This will convert differential mode signals into common mode signals at these points, impacting signal integrity. If a differential pair is well balanced, then tight coupling will achieve an effective degree of field cancellation. However, if they are not perfectly balanced, then the degree of cancellation is not determined by the spacing but rather by the common mode balance of the differential pair. This copper pour needs to be bordered by ground stitching vias and pulled back from the signal traces by at least 20 mils. Better still, it's left out altogether.

Electromagnetic emissions from digital circuits can occur as either differential mode or common mode radiation. Differential mode is typically equal and opposite and therefore any radiating fields will cancel. Conversely, common mode radiation from two coupled conductors is identical. It does not cancel but rather reinforces. Unfortunately, differential mode propagation can be converted to common mode by parasitic capacitance, any imbalance caused by signal skew, rise/fall time mismatch, or asymmetry in the channel.

My recommendations for pours:

- All copper pours should be connected by stitching vias to the distributed ground net
- Do not use isolated copper pours
- Critical signals should be routed in the inner stripline layers with no pours
- Closely coupled differential signals can be routed on the outer microstrip layers without GND pours

- Critical single-ended signals can be routed on the outer microstrip layers if converted to CPW
- However, if microstrip signals are very closely coupled to the reference plane (<3 mil) then radiation is minimal and providing ground shielding on outer layers has little benefit

PCB designs that are generally well-grounded (that is, there are ground vias connecting the planes in multiple places) create a Faraday cage that functions to contain internal resonances and energy that might be available to radiate from the edge of the board. Power distribution networks that are well-bypassed across the entire area of the planes also help suppress resonance. Copper pour and via stitching are unnecessary in many digital designs. Copper pour is not the magic bullet that will solve all your EMC problems. Pouring copper randomly to fill up space may create additional issues. Proper design practice should be applied rather than just random copper.

Key Points

- The myth of copper pours is fueled by reference designs that seem to persistently use this old RF design technique.
- Copper pours are sometimes used, incorrectly, simply to fill in the unused space on a board.
- The CAM engineer may add copper thieving patterns to balance the copper distribution on a layer.
- Floating (unconnected copper) is a real hazard for signal integrity. All copper pours must be connected to ground or power net.
- A ground pour that is not accompanied by grounding vias can become a conduit for crosstalk between the traces on either side of the ground shape.
- By creating the copper thieving as part of the ground net and configuring it as a solid plane, you can assist with providing a well-defined return current path.
- Any additional planes must be positioned and spaced correctly, so they do not have a negative effect on the signal integrity of the board.
- A ground pour on a microstrip layer changes the transmission line from microstrip to grounded coplanar waveguide.
- The characteristic and differential impedance of the transmission line is lowered by using this technique.
- CPW combines the EM fields in a more localized manner than does microstrip, thus reducing spurious coupling, radiation, and dispersion.
- Microstrip can radiate. However, with the fields confined between ground planes, stripline does not.
- When copper pours come into proximity of critical signal traces (Figure 3) the impedance is reduced by 2–3 ohms.
- This will convert differential mode signals into common mode signals at these points, impacting signal integrity. DESIGN007

Resources

1. Beyond Design: "To Pour or Not to Pour," "Common Symptoms of Common Mode Radiation," by Barry Olney.
2. An Altium Designer blog, "Copper Pour and Via Stitching: Do You Need Them in a PCB Layout?"

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

PCB007 Highlights

Summit Interconnect Acquires Royal Circuit Solutions and Affiliates

The acquisition of Royal Circuit Solutions expands Summit's PCB offering in rigid, flex, rigid-flex and ATE PCBs, significantly strengthens engineering and service resources providing CAM, DFM/A, PCB design/layout, and also adds quick-turn, prototype SMT assembly services.

EIPC Technical Snapshot: Supporting Autonomous Driving

EIPC's 17th Technical Snapshot webinar focused on developments in automotive electronics, particularly on advances in the technologies required to support the evolution of autonomous driving.

PCB Technologies' InPack to Focus on Miniaturization, Packaging

PCB Technologies' Jeff De Serrano, Yaniv Maydar, and Alon Menache explain their plans to focus on advanced packaging, miniaturization, and other high-end technology, with much faster time to market, and they offer a view of the global market as well.

Addressing the Gap in Process Performance

The first steps in process improvement are to determine what the gap is and why it happens. Having a process is not sufficient; the process needs to be effective as well. For those responsible for creating and maintaining processes, the ultimate goal is to create a procedure that becomes self-perpetuating, that seeps into the fabric of the company's culture.

Additive Reality: Let's Drop a Line About PCB Cross Section

My article in the April 2022 issue of *Design007 Magazine*, titled "Additive Manufacturing Requires Additive Design Techniques," presented several cross sections of solder mask coated with an inkjet technology.

The New Chapter: Prepping for an Internship? Three Tips to Shore Up Your Skills

When I first logged onto my computer in summer 2021, I was beyond nervous. I had just accepted the role of corporate intern at Caterpillar Inc. That first day made me realize that I wasn't expected to know everything. I was there to learn.

Testing Todd: Optimize Your Training Time

Today's training has become an essential part of any operation, especially because most Quality Management Systems (QMS) require this. To be compliant with ISO9001 you must maintain a competence and training system.

Happy's Tech Talk #7: Next Generation Application Specific Modules

In 1965, Gordon Moore predicted that the number of transistors that could be packaged into a square inch of space would double every year for the near future. Although his projection was later revised to every 18 months, Moore's Law has withstood the test of time for five decades.

Altimade Puts Designers and Manufacturers Together

Feature Interview by Andy Shaughnessy
I-CONNECT007

Despite all of the talk about the need for communication between designers and manufacturers, many PCB designers still do not talk with their manufacturers for a variety of reasons.

Altium and MacroFab aim to change this dynamic. In this interview, Ted Pawela, chief ecosystem officer of Altium and head of Altium's Nexar Business Unit, and MacroFab CEO Misha Govshteyn, discuss the new Altimade manufacturing service that Altium is introducing in partnership with MacroFab. Ted and Misha provide an overview of the Altimade process, how it links designers to fabricators, assembly providers, and component distributors, and they explain how it could pave the way for true design with manufacturing, or DWM.

Andy Shaughnessy: Ted, tell us about Altimade. It sounds like a step closer to design with manufacturing, with designers and manufacturers having total transparency and sharing knowledge of design intent.

Ted Pawela: Yes, that's right. A few months ago, we launched Altimade. For many years, we've talked about transforming the industry, breaking down the barriers that exist between the people who do design and the people who source and manufacture those designs. Altimade is intended to address these challenges, which are largely about communication.

Shaughnessy: What are some of the challenges that keep designers and fabricators from communicating?

Ted Pawela

Pawela: The problem is that each of these groups of people—designers, supply chain management, and manufacturers—have fundamentally different concerns. Yes, everybody wants to get PCBs and products built, but their requirements are different. You complicate that with the fact that they're generally not next-door neighbors. Manufacturing could be on a different continent, or at least in a different part of the country. Time, language, and communication style all get in the way, and if that's not enough, now in the last year or so, we've had these incredible supply chain and component shortages.

Shaughnessy: Can you tell me what Altium is doing to address this?

Pawela: Altium's mission is to help address these challenges through software innovation, and in doing so, to transform the way our industry works. In the last couple of years we've taken three major steps toward making this idea a reality. The first is Altium 365, our cloud platform for PCB design. Nearly 20,000 Altium users are using this platform to collaborate more easily than ever before with their colleagues throughout the design-to-realization process, across time zones, geographies, and departments.

The second big project has been the introduction of Nexar. This is the partner-facing side of the Altium 365 cloud platform, comprised of an open API that connects to all things PCB, including the design tools but also the unbelievably rich data we have for the electronics supply chain. The Nexar API has enabled us to work with MacroFab and others to begin building an ecosystem of partners and solutions that will make PCB design more connected to overall product design and manufacturing. Nexar provides a standard and sustainable way for partners to integrate with Altium 365, and as they do so, Altium 365 is becoming relevant to mechanical design, purchasing, program management, and all the functional domains that are required to go from concept to product. It's one easy-to-use cloud platform, enabling the entire team to collaborate simply and naturally.

This leads me to the third major step we've made toward industry transformation. You'll recall that one of the important collaboration use cases that we've talked about before is between designers and manufacturers, with the intent of bringing manufacturing considerations into the design process much earlier. As you said, to go from design for manufacturing to design with manufacturing.

Shaughnessy: These are important steps and I bet this is making a difference. Tell me about Altimade.

Pawela: Altimade is an application that we've built on Altium 365, in partnership with MacroFab, that makes it much easier and more convenient for an engineer or designer to not only design but actually get prototype printed circuit boards manufactured. Designers can get pricing and place an order for a fully assembled board directly inside the Altium design environment. It also enables the manufacturer to provide feedback to board designers directly on the original design, eliminating the need to resolve issues through a series of emails and phone calls that typically add days or even weeks to the process.

It's a much smoother, cleaner user experience, and it eliminates many of those challenges we talked about. The key is it's connecting our users to a modern manufacturer—MacroFab—which can source the parts, build the board, and ultimately deliver it to spec to the end customer. Altimade is available through the combination of Altium Designer and Altium 365.

Shaughnessy: What does this look like?

Pawela: We want to give the designer signals as he designs so he doesn't have to wait until it gets to the end of his design, but as he chooses components and places those components, at any point he can pull up a new manufacturing panel inside of Altium Designer and get a rapid prototype quote. It tells the designer that, based on what you've designed so far, here's what it would look like in terms of the amount of time to get that board and what the cost envelope looks like as well. So, it is based on the BOM, but it's also based on the board itself and being able to manufacture it.

The key point is to inform designers of any issues well ahead of finishing the design so that they can make different decisions with respect to sourcing their bill of materials and laying out their board. For example, we're not just telling them, "Yes, this component is available," only to find out that there were only three in stock, and they're gone by the time they are actually ready to order. We're continuously providing real-time information on pricing, availability, and even more advanced things like the lifecycle status of that component. Will it be obsolete two years, a year, or even six months from now, when we move from prototyping into production? Now, you can make smarter decisions.

Shaughnessy: It sounds like you're literally connected to the manufacturer, so the manufacturer can see exactly what you see.

Pawela: Yes. You both see the same bill of materials. Both the designer and the manufacturer can see, for example, what the substitute to functional equivalence that I could put into that design is to get to do the same thing, and they're not emailing that back and forth; they're seeing that directly inside of the application. And the manufacturer doesn't need to know how to use Altium Designer, by the way; they only need a web browser, in which case they will see all the same things from within the Altium 365 web browser interface. These capabilities allow us to move from design for manufacturing to designing with manufacturing, and I think that's a big part of what we're bringing to market now.

And the other part of it is that the designer won't have to sit and wonder, "Well, is it getting close to being done? Do they have my bill of materials yet? If they source that, is it even on the line yet? Are they doing something with it?" I want to hear something. I want to know something without having to pick up the phone every day.

We are creating an experience that is very much analogous to Uber in that from the time you place that order, you see everything that happens with it. You don't have to call and ask for it; it's right there in Altium 365 and Altium Designer. You can see, yes, the order has been accepted, the bare board has been ordered, and all the other components have been received. This board is now moving from manufacturing into testing, for example.

And this, of course, is really valuable when, inevitably, if we're successful with a product, we're moving beyond prototyping and moving to production. This would be a good time for me to let Misha talk a little bit about MacroFab.

Misha Govshteyn: I'd be happy to tell you a bit more about MacroFab. We've built the service over the years to be a cloud manufacturing platform for electronics. We have a network of factories that work in our digital ecosystem and we can build products all the way from proto-


Misha Govshteyn

type to full production stage. This includes full product assembly, sourcing all the materials—including mechanical parts, programming, testing, and virtual warehousing—because, for a lot of people, prototypes are only one of the first few steps in the product journey.

In fact, your product often goes through significant changes when it hits the factory floor, so for a number of years now, MacroFab has been in the business of building the digital thread from the prototype stage all the way to the factory stage. We are exclusively in North America, which gives us access to over 75 factories in the U.S., Canada, and Mexico.

That includes low-cost manufacturing regions, so we see a lot of customers moving from Asia to 100% North American manufacturing where they can seamlessly move between low volume U.S. factories which operate at a very fast speed and can deliver very quickly. The service can be done as quickly as 10 days, and we have factories in Mexico that operate on a very high-volume scale.

In fact, the way that we usually go to market and engage with our customers is through engineers first. But it's never been a seamless experience. First, the engineer has to output the design files, either in a native design format or an ODB++ format or—at worst—in Gerber. Ultimately, we interpret that input into our platform and identify a starting point for your design: "Is this what you really want us to build?"

What's really exciting about our work with Altium is that Altimade is an end-to-end toolchain, so the translation layer is unnecessary. Customers have a much faster on-ramp to a true manufacturing company. We can go straight from Altium Designer all the way to hundreds of thousands of units produced over multiple years in a high-volume factory. MacroFab has been at this for a number of years, and we have a tremendous amount of experience.

Pawela: Another key thing about MacroFab is that they have scaled beyond their own capacity, in that it is a factory network, not simply a single line or factory that MacroFab operates.

On top of that, there are other partners who have come into the Nexar ecosystem as well, not for manufacturing, but to bring in things like signal integrity and EMI prediction. We're even making the API open so that competitors such as Cadence can come in and use this. In addition to the big footprint that we have on the design side, with over 55,000 licenses, MacroFab likewise has a large active user community and, most importantly, they've got the scale in terms of the factory network to support our users. We think that we're not just making a subtle evolution to this designed-to-realization process, but we're actually revolutionizing it.

Shaughnessy: It sounds like you're trying to recreate the benefits of the captive companies when everything was under one roof, and you could just walk your design down the hall to manufacturing.

Pawela: Yeah, I think the key you're talking about is like the benefits of having a vertically integrated supply chain, and we're trying to do that in a way where it doesn't have to be one company that does it all. We don't believe that one company can pull that off, and we don't believe that you can create a walled garden around your ecosystem and say that you'll get everything from us. We are absolutely trying to be open to bring and to make all of that vertically integrate from the customer experience perspective, but not by requiring a single company to have every piece of it.

Shaughnessy: What would this look like from a fabricator's viewpoint?

Pawela: For now, we're really focused on solving the full problem of assembly, but the fabricator is absolutely connected in the same way to the collaboration platform. When they have questions or issues that need to be resolved that likewise is being communicated through the platform to the end customer.

Govshteyn: Yes, and to add a little bit of color to what Ted mentioned, he made an analogy to the Uber-like ordering experience on the customer side. It's also an Uber-like experience on the fabricator and assembler side as well. We are very aware of the capabilities for each assembly factory, and based on the volume that we need and our requirements for each particular board, we will essentially show that board designed to X number of factories.

We literally have a database of their pick-and-place machines and what kind of secondary assembly capabilities they have, as well as whether they can do conformal coating or programming instructions. We also know about their quality rating, so the algorithm is automatically optimized for the board to get built at the right price, with the right quality level, and as soon as possible.

Shaughnessy: And this is all visible to the designer before he begins the design?

Govshteyn: That's exactly right, and at that scale, you could actually end up in a situation where you may be building your design with multiple boards involved in a product. They might actually get built in different factories, but from an end-user perspective, all they care about is how quickly they can get it. How much is it going to cost to meet my requirements? This really abstracts the factory floor from the end-user and, in all honesty, people usually don't have a very good sense of what a factory can and cannot do, so we're letting the software drive that decision.

Shaughnessy: Very cool. This is great. Anything that makes it simpler for our readers and gives them more power earlier in the design cycle is a good thing.

Govshteyn: Ted made an analogy to Uber, but for me, it's always been more like travel agents. I think you would have to force me to use a travel agent these days. It's just so much easier for me to figure out what my travel options are by going through one of the well-known booking sites. This is very similar, and there are details in Altium Design that really don't fit into the Gerber format, so we get much more information about what the customer ultimately wants. I think one of the coolest things about it is that designers input all their requirements into Altium Designer and that's really the starting point for the design. So much of it gets lost in translation when it gets output into offline files and starts to get emailed around. That's the reason why there are so many phone calls and the reason why there are so many emails.

Engineers do not want to be on conference calls explaining what their design is really meant to do, and I think this really gets them out of the business of having to talk to people. In my experience, an engineer's main goal in life is to never just speak to another salesperson or human being. I think that's the ultimate objective.

Shaughnessy: That sounds like quite a few engineers I know. Ted and Misha, thanks for speaking with me.

Pawela: Thank you, Andy. DESIGN007

Mastering the Art of Communication With Manufacturers

Feature Article by Kyle Burk
KBJ ENGINEERING

Design with manufacturing (DWM) emphasizes the important aspect of the true intent of design for manufacturing (DFM). The intent of DFM is to consider the manufacturing process during the entire printed circuit design process starting at the earliest stages of the design cycle, when the project first begins, and continuing to the end of the product lifecycle.

As mentioned in the May issue of *Design007 Magazine,* design is performed, at times, in a vacuum. But it doesn't have to be that way. Whenever circumstances allow, design should be performed by communicating with all stakeholders throughout the design process, hence the emphasis on the word with in DWM. Communication can occur through personal correspondence such as email and voice conversations or through more formal design meetings—in person or through videoconferencing. No matter which means of communication you prefer, it's important to communicate early and often with stakeholders involved in the downstream processes as you bring your project to realization.

There's No 'I' in Team

Unlike the word "team," there is an "I" in design with manufacturing, but that doesn't make it a singular process. Rather, DWM requires involvement of a cross-functional team with the designer as the key stakeholder who communicates regularly with the team throughout the design process. This communication includes receiving feedback early from

Kyle Burk

other stakeholders and implementing any improvements they suggest.

The higher-level structure of DWM takes effort but yields superior results because it helps the designer avoid delays while lowering manufacturing costs and maintaining a high-quality finished product. For example, deadlines are more likely to be met by communicating early with the assembly team on bill of materials component availability. For those designers using revision control, communicating early can help avoid the costly process of rolling a revision for minor last-minute design changes. Communicating with the fabrication team before beginning assembly can help lower manufacturing costs and turn times by discovering cost reduction opportunities, such as combining similar drill sizes.

DWM seeks to unify all stakeholders of the circuit board design and development process, including the designer, fabricator, and assembler. For components with specific requirements, DWM may also include involving the component manufacturer. The final complete data package sent to the assembler includes drawings, specifications, requirements, ratings, a bill of materials, Gerber, and drill data, etc.

DWM thus requires contributions and feedback from members of the team most familiar with the different documents of the data package. The designer should review preliminary Gerber and drill data with the fabricator to ensure the design is manufacturable. They should also discuss the bill of materials with the assembler to make certain that all components are available for purchase. Each document of the data package should be reviewed by an expert in the corresponding process.

Designers embracing DWM should take advantage of the latest in technology. While the hand-drawn graphics of circuit board design are far in the past, some methods for integrating separate documents into the data package can be archaic at times. If you are still sending your data package in a compressed file through email, get creative about how you can improve that process. Think about how you are sharing other important files within your organization—can you use those same methods for your data package? Think of ways you can incorporate the cloud into your process.

Workplace-specific messaging programs are modern tools you can integrate into your workflow as you communicate with fabrication and assembly teams through a workspace. List-making applications are powerful resources used to assign and manage tasks, and track their status. What resources do you use in other aspects of your work that you can apply to data package management and distribution?

Counting the Advantages

The advantage of consistent collaboration with all stakeholders in the manufacturing process is that, as a designer, you can improve your design efficiency over time by building up a knowledge base of the processes involved in circuit board fabrication and assembly. Coordinating with fabricators and assemblers may seem overwhelming and time consuming at first but give it some time. As you become familiar with the nuances of fabrication and assembly you will find yourself becoming more independent as you instinctively integrate their feedback into your daily workflow. In the long term, coordination will, in fact, reduce your workload and open more time for you to spend on other tasks.

If you don't know who exactly will be fabricating and assembling your design, try to gather a list of potential partners. You could then work with those partners to pool their different requirements and capabilities. Once this data is gathered and combined, you can choose the most stringent requirements for each different category. This approach allows you to design a circuit board that will meet all potential partners' requirements, thus avoiding delays. If your circumstance does not allow you to compile a list of potential manufacturing and assembly partners, start somewhere by collaborating with the partners of your choice and go from there.

Design with manufacturing goes above and beyond design for manufacturing by integrating all key stakeholders on a project into a collaborative team. When using DWM, each document of the design package is reviewed by a relative team member who is an expert in the relevant field throughout the design process. Integrating modernized team approaches to your team workflow can help make DWM more efficient than ever. If successfully applying DWM sounds overwhelming or seems unlikely to you, give it a try by starting small. There is a good chance it will improve data package uptake. **DESIGN007**

Kyle Burk, PhD., is director of engineering at KBJ Engineering.

BOOK EXCERPT

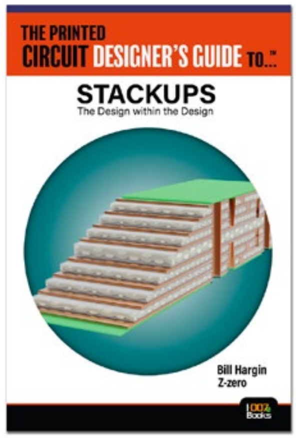
The Printed Circuit Designer's Guide to... Stackups— The Design within the Design

by Bill Hargin

Introduction: Mechanical vs. Electrical Worlds

Another book about stackups?

If you're asking this question, I'd like to know the book you're thinking of, as I was looking for it a few years back. I have a pretty good PCB signal integrity (SI) library, and I've only found one chapter on stackup design so far.



Whenever I talk in person with SI consultants—people who do SI consulting for a living—I ask them, "Of the smoke-jumping projects you've been brought in for where there were serious SI problems, how many of those projects have stackup issues?" So far, the only answer I've gotten back is, "100 percent."

The difference between a high-speed PCB design that can be built, and a design that should be built, depends upon the backbone of the design itself: the stackup. The stackup touches every single high-speed signal and yet has had surprisingly little written about it.

In my work, quite a number of PCB stackups cross my desk, and depending on who or what tools were involved in a given design, there are manufacturing parameters that affect both impedance and signal loss that design teams can improve upon.

This book is by no means the last word on the subject, but rather a place to kick off a broader discussion about stackup planning and material selection, to reach the understanding of what I call "the design within the design."

Download this book today!

Bringing PCB Quoting into the 21st Century

Connect the Dots

by **Matt Stevenson**, SUNSTONE CIRCUITS

Here we are in the 21st century, technology abounds across all sectors, and it continues to grow. Advances in wearable technology, vehicles with driver assistance features, and integrated smart home electronic devices continue to drive demand and innovation in the PCB industry.

The product development teams tasked with taking these technologies from design into reality are often stuck using procedures from the last century. Case in point: PCB quoting. To access quotes and purchase PCBs, designers often have to call or email the PCB manufacturer to request a quote. After that, the product development team must wait, usually hours (but sometimes days) for the quote to be completed and returned.

With today's rate of technological advancement, the waiting game for a quote can become the most difficult part of PCB design.

If the manufacturer can't produce the design to specifications or the price is too high for the project's budget, the designer is back at square one. This creates distinct advantages for large companies who may have established relationships with reliable manufacturers or enough budget to absorb unexpected costs. Others can dedicate a resource who sends the design to multiple suppliers and greatly increases the chances of receiving a quote that satisfies the team's requirements. Even with those advantages, this antiquated process for PCB quoting can create serious delay. For a small shop or an inexperienced designer, the difficulties can mean missing important deadlines or losing a contract.

This difficulty is magnified by today's challenges associated with finding a quality PCB manufacturer. Most shops ask designers to request quotes via

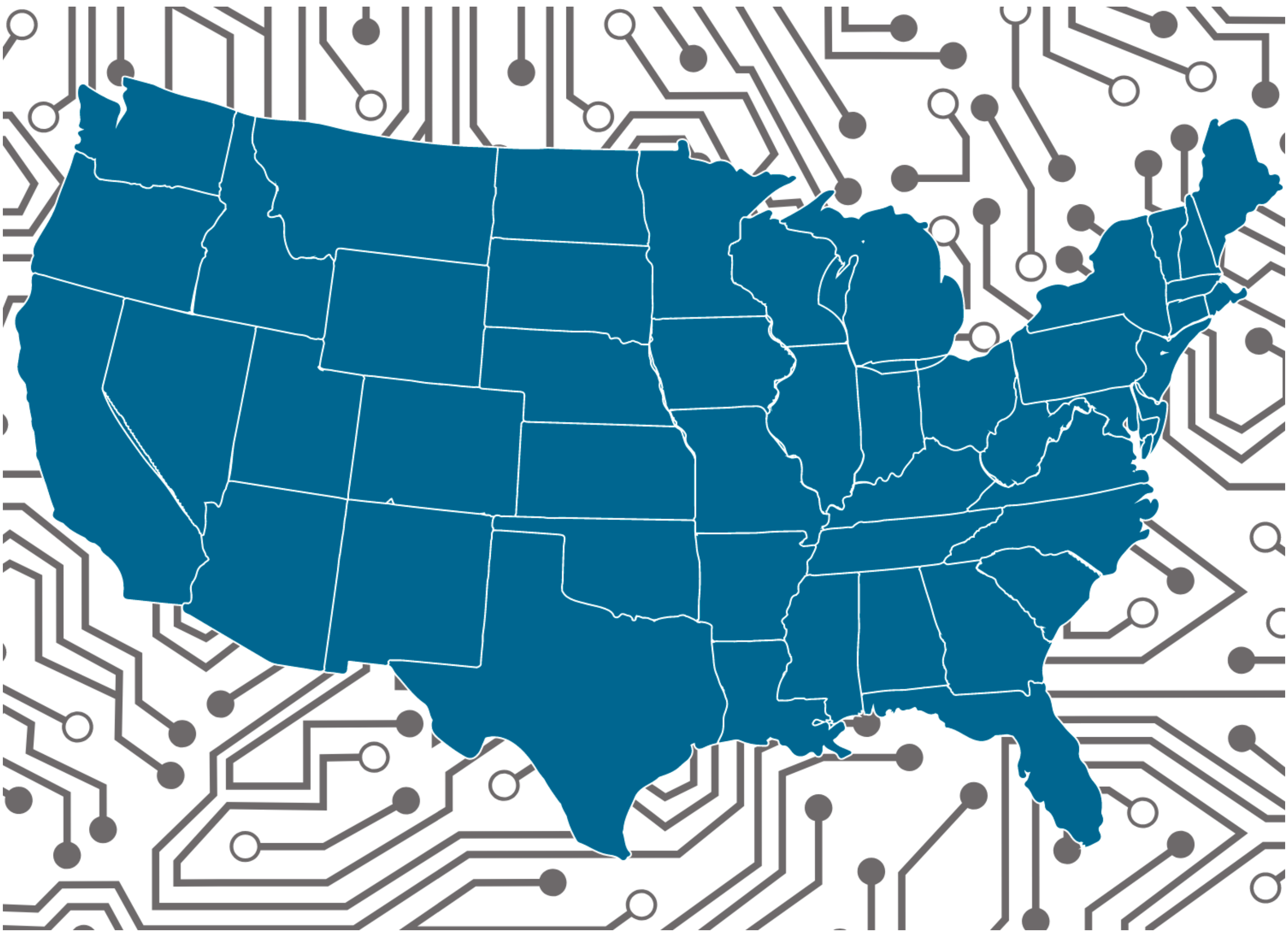
phone or email. This often does not offer the opportunity for collaboration about the design or useful guidance about how to best meet product requirements. To compound matters, it is not always obvious who you are actually working with.

According to PCBdirectory.com, there are roughly 550 listings for U.S.-based PCB manufacturers. A deeper dive into that list reveals that plenty of companies offering PCB manufacturing services are not actually PCB manufacturers. Some listings are brokers or assembly houses, and many companies have multiple listings on the directory. This makes the number of listings for actual PCB manufacturers based in the U.S. probably closer to 300.

Designers can often find themselves dealing with a middle person and not directly with the manufacturer. Working through a broker or assembler can cause several issues, any of which can create havoc for a product development cycle. Common issues associated with working through a broker include:

- There will be slower delivery in general because of the logistical realities of a go-between. Even longer delays can happen if the broker orders the PCB from overseas.
- Brokers and assemblers don't work for free and will mark up the cost of the PCB and often pass unexpected cost overruns on to the buyer.
- Indirect communication with the manufacturer limits the ability to perform collaborative cost/benefit analysis on the design and reduces visibility into the manufacturer's quality assurance program.

If a designer does select one of the 300 or so PCB manufacturers based in the U.S., most



of them have reached the late 20th century in terms e-commerce and do have a website. Unfortunately, in many cases, manufacturer websites' calls to action for requesting a quote still involve the phone or an email. This process ranges in length from a few hours to a few days, requires back and forth between the manufacturer and designer regarding cost and production requirements, and consumes valuable time.

A handful of manufacturers do have RFQ forms on their websites that allow for inputting some of the information about design requirements. These forms may speed up the turn time for a quote and potentially increase the accuracy, but not by large amounts.

A few PCB manufacturers now offer online quoting for a faster, more streamlined process. A design engineer can input the needs and design parameters of a project and get near-real-time pricing with the click of a button. The turnaround time for quotes shortens from hours or days to just minutes. Today, a surprisingly small number of manufacturers have implemented online quoting, and some are deploying systems more effectively than others. That said, this advancement is definitely a step forward for designers in the grand scheme of things.

The next evolution of the quoting process will involve real-time dynamic pricing. Designers will see how each selection they make impacts the quote price, immediately demonstrating where the design's cost drivers are and empowering them to make design decisions that will keep the project on budget.

The transparency of dynamic pricing will allow even inexperienced PCB designers to learn and assess cost and benefit with confidence. The nimbleness and transparency of this new approach has the potential to change the game for product development. As teams are empowered to provide precise requirements to manufacturers, the manufacturers, in turn, will be better equipped to provide consultation that results in a better product. This future offering will make getting a quick, cost-effective quote accessible to PCB designers and small product development teams, leveling the playing field for those making the technology of tomorrow. **DESIGN007**

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

MilAero007 Highlights

Electronics Industry Welcomes Bipartisan Congressional Proposal to Boost U.S. PCB Sector

The electronics manufacturing industry is welcoming a new, bipartisan proposal in the U.S. Congress that would help bring back the country's printed circuit board (PCB) sector.

Jahr Turchan Discusses Blackfox's Training Scholarships for Veterans

Nolan Johnson recently interviewed Jahr Turchan, director of Veteran Services & Advanced Manufacturing Programs for Blackfox Training Institute. They discussed some of the new programs at Blackfox, including the Veteran Advanced Manufacturing Certification program.

American Made Advocacy: DoD Unarmed Without the PCB

It's been more than 800 days since the global COVID-19 pandemic upended the supply chains of almost every industry. One sector that sometimes escapes the attention of everyday Americans is aerospace and defense, where high-tech platforms and equipment are essential to mission success.

Boeing Unveils First T-7A Red Hawk Advanced Trainer Jet to Be Delivered to the U.S. Air Force

Boeing has unveiled the first T-7A Red Hawk advanced trainer jet to be delivered to the U.S. Air Force. The T-7A Red Hawk incorporates a red-tailed livery in honor of the Tuskegee Airmen of World War II, who made up the first African American aviation unit to serve in the U.S. military.

TTM Technologies Reports Fiscal Q1 2022 Results

TTM Technologies, Inc., a leading global manufacturer of printed circuit boards (PCB), radio frequency (RF) components and RF microwave/microelectronic assemblies, reported results for the first quarter, which ended on April 4, 2022.

Ventec to Host Live Webinar on Thermal Management with IMS

Ventec International Group Co., Ltd. (6672 TT) is pleased to announce that it will be hosting a live webinar on "Top design techniques for thermal management with IMS" on Thursday, June 23 at 3 p.m. CET/9 a.m. EST

Jabil Strengthens Additive Manufacturing Offerings with New PK 5000

Jabil Inc. launched PK 5000, an eco-friendly, powder-based additive material engineered to deliver improved strength, chemical resistance, and resilience.

NASA Selects Satellite-based Space Launch Tracking, Command Systems from Inmarsat

NASA's Communications Services Project (CSP) partners with commercial satellite companies to develop and enhance communications services for launch support, mission contingencies.

Murray Percival Co. Brings Ersa Rework System to L3Harris

The Murray Percival Company, an award-winning leading supplier to the Midwest's electronics industry, is pleased to announce the sale of an Ersa HR600/2 rework system to L3Harris.

Is DWM Just Another Buzzword?

PCB Talk

Feature Column by Tara Dunn, AVERATEK

Designing *with* manufacturing, rather than for manufacturing is much more than a buzz word. New materials and processes, and new and developing design rules make collaborating with your PCB fabricator a necessity more than ever before. I sat down with Meredith LaBeau, CTO of Calumet, to learn her thoughts about bringing a new technology to commercialization and the importance of PCB designers working closely with their preferred fabricators to be most successful when implementing designs with feature sizes previously not available.

Tara Dunn: Meredith, you and the Calumet team are leading the North American market in next-generation PCB fabrication technologies, enabling a much-needed capability for your customers, and I am excited to dig into that for our readers. First, please provide a little background on yourself and Calumet Electronics for those who are not yet familiar.

Meredith LaBeau: Thanks, Tara. Calumet Electronics is a domestic printed circuit board fabricator located in Upper Michigan. Started in 1968, Calumet Electronics has a long history of manufacturing PCBs, along with developing solutions that are needed by customers in defense, medical, aerospace, and industrial controls. In addition, for the past nine years, we have been on a journey to rapidly improve technology to provide a beacon

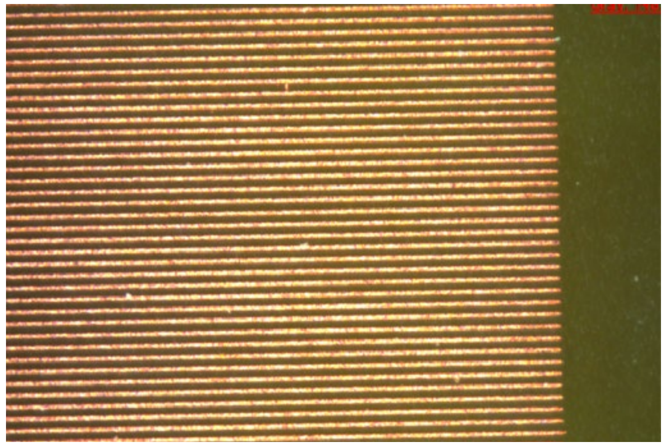
of light for domestic advanced technology PCBs and advanced packaging.

This drive has led us to industrializing next-generation additive manufacturing: the Averatek Semi-Additive Process, or A-SAP™. With this technology and state of the art equipment, Calumet Electronics is now providing next-generation capability and capacity for tightly packed BGAs and other features on the most challenging circuit boards, realizing features below 25 microns. Calumet is an NADCAP, AS9100D, MIL PRF-55110 and IPC-1791 certified facility.

I have the distinct pleasure of representing Calumet Electronics as its chief technology officer, charged with strategically developing and advancing the most complicated and needed PCBs and ultra-high-density PCBs for the domestic market. The past nine years have been a rewarding experience, but one of the most significant technologies to allow breakthrough success is the use of the A-SAP processes.

Dunn: Calumet Electronics was the first licensee of Averatek's A-SAP process. You have navigated not just the learning curve for fabrication, but also the design learning curve, and how to best apply this technology. Because this technology is still new to many, can you talk about the process and its benefits for PCB designers?

LaBeau: It is always difficult to usher in new technologies within an industry which has struggled with offshoring, stunted innovation, and lack of adoption for new processes. With that said, industrializing Averatek's process has been a fulfilling experience that has opened many new doors of opportunities, ones we never knew were available. It's a semi-additive process for feature realization, allowing the United States to break through 60 microns. It allows for reaching down to limits of the photolithography equipment, around 12–15 microns currently, but testing at less than 10 microns. This technology allows for leapfrog opportunities to unite chips (semiconductors) with printed circuit boards, all here in North America.



10 micron plated 1x.

For the designer, this is a true game changer. The designer can now use finer lines and spaces to effectively fan out traces from BGAs and other devices without adding more layers or introducing increasing complicated stacked or staggered microvia structures. Designing with finer lines may result in re-setting the manufacturing curve of complexity. Imagine reducing the number of sequential laminations, and not worrying about multiple stacked features; the opportunities are virtually endless.

Now we just need to adopt this new technology that has been industrialized and qualified.

Dunn: Meredith, where are you in the process development cycle? I know readers will be curious about reliability testing and data. Would you share your experience?

LaBeau: We have finished all the process development over the past two years, moving the manufacturing readiness level from a five to nine (low-rate production). Through this development process, we have done significant testing for reliability including peel strengths, thermal stress and cycling with microvias, as well as staggered structures. Additionally, we have processed over 1,000 panels, tested with electrical continuity and microsection analysis.

Through the development phases of A-SAP, we have used the technology on all traditional and many non-traditional substrates with success, as well as fabricating all traditional PCB features with passing reliability.

The A-SAP process is a proven and tested additive fabrication method to achieve next generation technological advancements.

Dunn: Switching gears just a bit, you and I have spoken many times about the importance of designing *with* manufacturing rather than *for* manufacturing. In fact, this is a direct quote from a video we worked on together quite a while ago:

"The use of this innovative and transformative manufacturing method requires a new approach to design: with manufacturing instead of for manufacturing. Together, the designer and manufacturer can develop a collaborative approach, to "Drop the SWAP," while increasing the reliability and robustness of the PCB for next-generation electronics systems."

I often hear you impress on people that collaboration is critical to utilize the full potential of the A-SAP technology. How do you facilitate this high-quality communication, and what is your advice for designers who want all the advantages of A-SAP capabilities?

LaBeau: When utilizing a transformative manufacturing process, one must fully understand its advantages as it applies to both design and the product requirements. With a market-changing technology, the manufacturer and designer must work in collaboration to gain all the benefits—while not increasing the cost.

The Averatek process allows a designer to simplify designs by using finer traces and spaces, greater line width control, and impedance control. If the designer understands this, you can re-set the technology curve: simplifying designs to make the process and end-product more reliable and robust, while reducing risks of lead time or yield delays.

The most important takeaway is this: collaboration between the designer and manufacturer is critical and must happen to fully utilize this technology and create the intended benefits. As we transition from a subtractive to a semi-additive approach, there are new rules that we must apply, at times understanding that not all IPC-6012 standards work for fine lines, especially wrap plating, plating thickness and others. When an innovative technology comes forth, we must design and test, then adapt new standards, which are currently in the works.

This design with manufacturing is key to the success and will open avenues to design and manufacture some of the most advanced technologies for all our customers.

Dunn: Not only has Calumet Electronics invested in Averatek's A-SAP process, but you have also been actively investing in equipment, new materials, and a facilities expansion. As the CTO, I am sure that keeps you extremely busy. What other technologies do you have in development that the electronics community will be excited about?

LaBeau: Calumet Electronics is driving to solve the next generation for unrivaled electronics systems, which includes utilizing Averatek's A-SAP, sintering technologies, and build-ups films. We are excited about the future, with a strong R&D and engineering team.

Dunn: Meredith, thanks for taking time for this short interview.

LaBeau: Thank you, Tara. DESIGN007

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

Producing Diverse Designs in Concert With Manufacturing

Feature Article by Scott Miller
FREEDOM CAD SERVICES

There is a new acronym bubbling up in the design world: DWM, which stands for "design with manufacturing." Why is this different than design for manufacturing, or DFM? With DWM, the emphasis is on integration between the design team and the manufacturers during the design process. DWM is much more than that.

As an engineering service company, we provide design services to companies that require additional capacity to meet time-to-market objectives or technical expertise that may be beyond the experience or core competency of the company we are supporting.

Whatever the reason, we are tasked with producing designs that meet various technical requirements, yet are cost-effective and manufacturable. We provide this service to hundreds of customers who have varying degrees of processes, tools, and manufacturing partners. Given this diversity, we have recognized the importance of designing with manufacturing to achieve the product development goals of manufacturability and technical excellence.

Our business revolves around printed circuit board design services. We are not an OEM and we don't develop products; therefore we start with customer-supplied data. The depth of the data varies depending upon the customer and the project, but we are receiving inputs from the customer to start the process. This is when the most important part of the process starts: communication. It's important to understand the technical and tactical requirements.

Besides the typical technical product development questions, there are several questions that we will also flush out early in the process:

- What are the product specifications that must be met?
- Are there geographic or ITAR restrictions?
- What are the projected prototype and production volumes?
- Do you have an approved vendor list (AVL), incumbent board fabricators, and/or contract manufacturers?
- Does the product require RoHS compliance?
- Do you have any single source components or custom components on your bill of materials (BOM)?
- What is your project's target completion date?

Addressing these questions helps identify all the DWM stakeholders so that we can begin communications to assure that we are in sync. The goal is to validate the decisions early in the design process rather than later, or during fabrication or assembly when problems are much more expensive to resolve. It is important to match the technical requirements with manufacturers whose process capabilities are well aligned with the technical requirements.

Communication: Saving Money, Resources

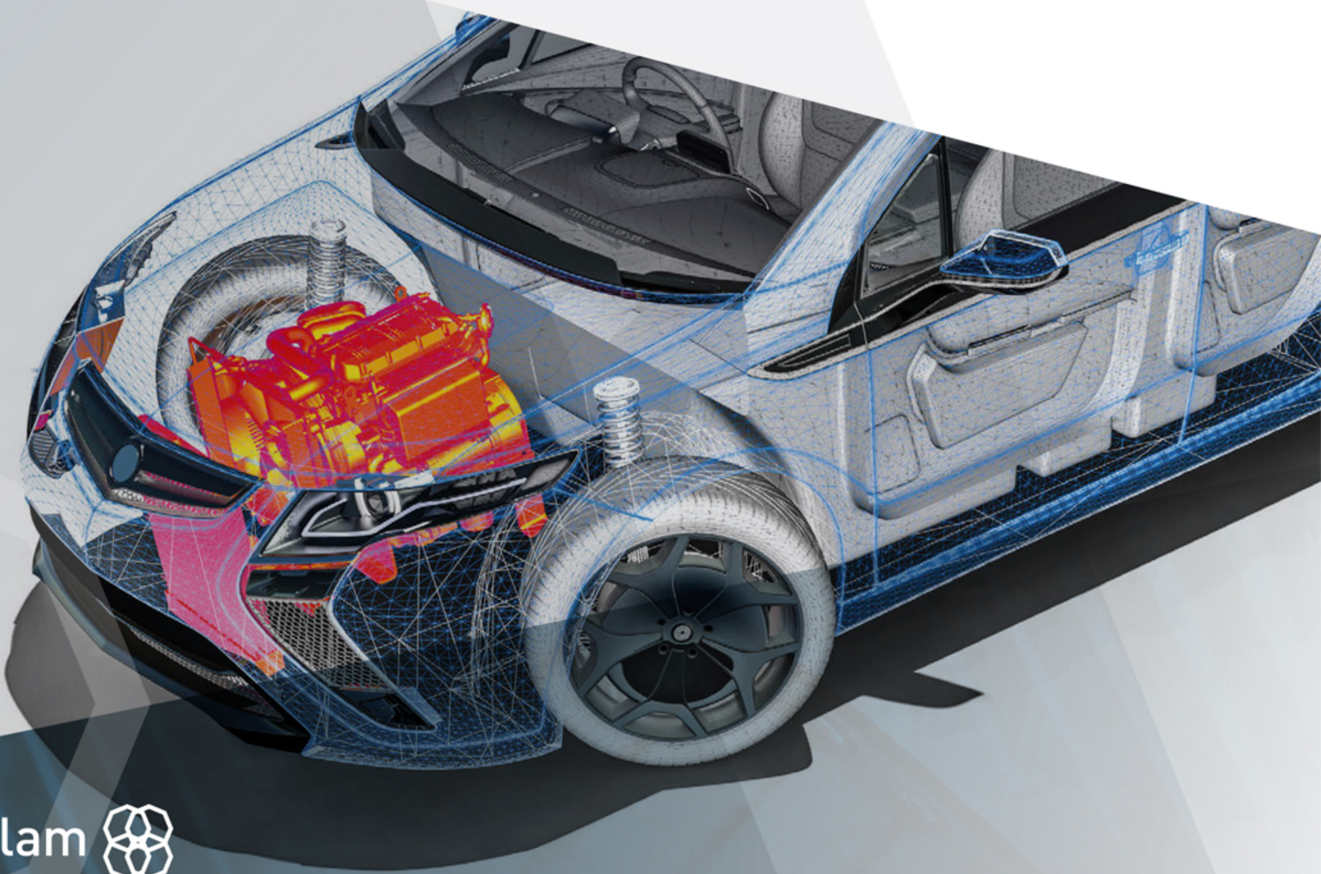
For instance, receiving guidance from your PCB fabricator on laminate material options, costs, and availability issues is very beneficial. Establishing a board stackup that meets the technical requirements and is manufacturable is a prerequisite for successful high-performance or high-volume designs.

Identifying coupon requirements, testing, panelization, thieving, and assembly rail requirements helps to streamline the PCB fabrication process and promote efficient assembly of the printed circuit board. Designing with blind and buried vias can increase the fab processing time but it may enable a PCB to be designed in fewer layers, which can more than offset the increased processing time. This could be very significant for high-volume products. Addressing these requirements upfront enables the design to move smoothly through the fab and assembly processes.

Circuit-to-panel optimization should also be addressed during the design phase, particularly in higher-volume requirements. Panel utilization is dependent upon the panel sizes supported by the fabricator. Copper thieving is often added to the layers of the PCB to provide a better balance of copper to prevent the boards from warping. Non-functional plated through-holes may be eliminated to reduce the drill time and potential for fallout due to broken drills. All these issues are addressed between the design engineer, the layout designer, and the fabricator.

Circuit-to-panel optimization should also be addressed during the design phase, particularly in higher-volume requirements.

Particular diligence is required to manage the lifecycle analysis during the component selection process. Component availability issues are crippling product development and manufacturing around the world. We highly recommend that the design process should include the engineer providing alternate components on the AVL and BOM. Providing component alternates can give the contract manufacturers the ability to manage availability issues without involving the OEM for approvals, which can slow down the process.



At the beginning of the design process, we should have built the footprints with a component courtyard size that has been agreed upon with the assembler, to prevent parts from being placed too close to each other. The software will also have the capability of adding rules to check the component spacing using the courtyards for the same reason. Courtyards are an important part of the footprint—they not only prevent parts from butting up against each other but will also prevent placing short parts too close to tall parts, which makes it impossible to rework the shorter parts. We also add heights to the footprints to enable us to check that the parts are within a particular height envelope and prevent cards that mount next to each other from colliding.

As PCB placement nears completion, we like to engage with the assembler to have them review the placement data for manufacturability. Components that are too close to one another or to a board edge may not meet their DFM requirements or may inhibit the parts from being reworked should that become necessary.

Model Citizens

Designers can generate 3D models of the PCB to provide analysis of the placement to assure that components fit appropriately and cables have the required room to be installed. Checking the CAD models of the component placements of adjacent PCBs in a card cage is also recommended to assure they can be inserted and removed without issue. Some PCBAs require conformal coating to protect them from harsh environments. This will require input and review by the conformal coating provider and may require additional documentation to identify keep-out areas.

Circuit reuse has become an increasingly popular practice. Circuit reuse enables a designer to use existing, "proven" circuits, such as a power supply or memory circuit, which can be huge time-savers when the designs have the space and same stackups to use them identically, as intended. However, they can become less proven and less cost-effective when they can't be used identically due to space constraints or stackup variations.

We use Siemens' Valor NPI software to perform a DFM analysis to assure that the design will release to PCB fabrication seamlessly. Other software tools can perform similar DFM analyses. We generate the "deliverables" file from Valor to assure the deliverables match the Valor analysis.

Circuit reuse has become an increasingly popular practice.

Some PCBs are intended to be assembled one-up and others are intended to be multiple-up in arrays. It's important to identify this during the design process so that the process is supported. In some cases, rails are added to the PCB to facilitate the assembly process and then they will be removed afterward. Communications between the design engineer, layout designer, and assembler are critical to streamlining the assembly processes.

Early and regular communication is the common theme in our approach to DWM. While we've focused this article on the processes related to PCB design, you can easily adapt a DWM approach to other design activities. Integrating with all major stakeholders to understand the impact of your process on their processes is critical to designing with manufacturing. DESIGN007

Scott Miller is chief operating officer for Freedom CAD Services, Inc.

Supply Chain Resilience, Part One: The Supply Chain Problem

Digital Transformation

by Stephen V. Chavez, SIEMENS EDA

In part one of this two-part series on supply chain resilience, I'll first address the problem being witnessed throughout the industry regarding supply chain disruptions and its negative effects.

For more than two years, supply chain issues have headlined the news, identifying a worldwide vulnerability magnified by the global pandemic. This vulnerability had been problematic long before the pandemic, quietly growing, and ignored until the pandemic hit with full force. Then, this sleeping dog raised its ugly head and now present worries in today's headlines. From our local grocery and department stores to our local auto dealerships, empty shelves and empty car lots have been the negative effect of sporadic supplies causing serious consequences and disruptions as we struggle to emerge from this pandemic.

Within the electronics industry specifically, supply chain issues have become a major factor that need to be addressed. It is one of the most concerning issues at hand as companies look for ways to move forward and get products to market as quickly as possible. Unfortunately, the cost of doing business is higher than ever and increasing, and is influenced by a global electronics value chain already facing rapidly dynamic market forces that are being amplified by the pandemic and global chip shortages.

The simultaneous collisions of today's rapid dynamic forces have created the perfect storm in the electronics industry. Sadly, there is no immediate end in sight, nor is there any temporary relief in the near future. If anything, there is real potential for matters to get worse, especially as an indirect result of the global eco-

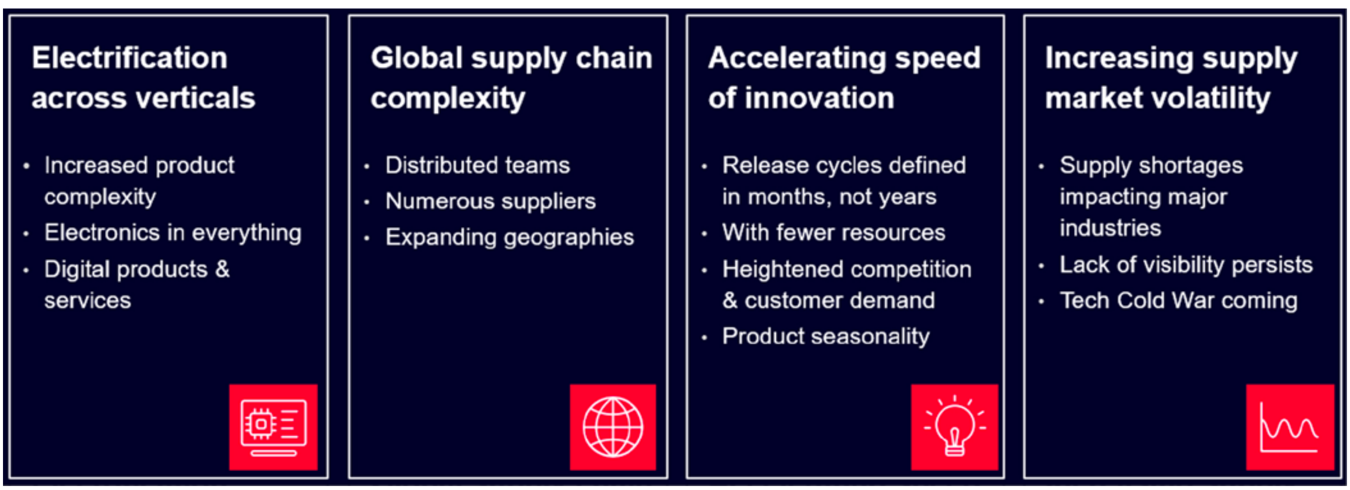


Figure 1: Today's rapid dynamic forces.

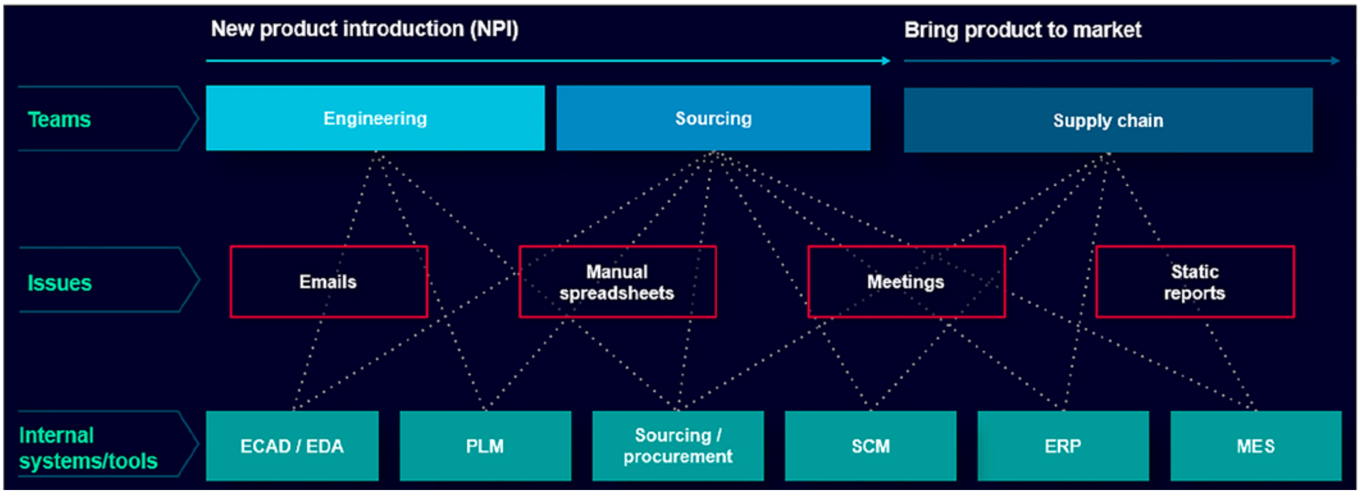


Figure 2: Challenges within the enterprise, across functional silos, disparate systems, and fragmented cross-functional decision-making processes.

nomic turmoil arising from the war between Russia and Ukraine.

With all these external issues already making matters difficult, and observing many companies operate today, we see a very complex, spiderweb-like network of informal processes that add even more challenges within the enterprise, across functional silos, disparate systems, and fragmented cross-functional decision-making processes. Based on my recent experience as a principal engineer of PCB design at a leading mil/aero company over the past 12 years, I can attest to this complex network of communication and functions inherent to larger corporations that are typically a conglomerate of absorbed smaller companies spread across the globe.

Two recent surveys[1] found that 81% of respondents reported that commodity availability has forced expensive spot buys, while 91% observed that sourcing issues have caused delays in product launches. In the same surveys, 79% of the respondents stated that collaboration issues have caused delays in new product introductions, while 63% expressed that the numbers of BOM iterations have increased significantly. More recently, Goldman Sachs forecast an average 20% shortfall in computer chips, which is expected to last through 2022.

As we dive deeper into the problem specifically regarding today's design and sourcing processes, it's clear that supply chain resilience is not inherent in these processes. Therefore, many design organizations are highly vulner-

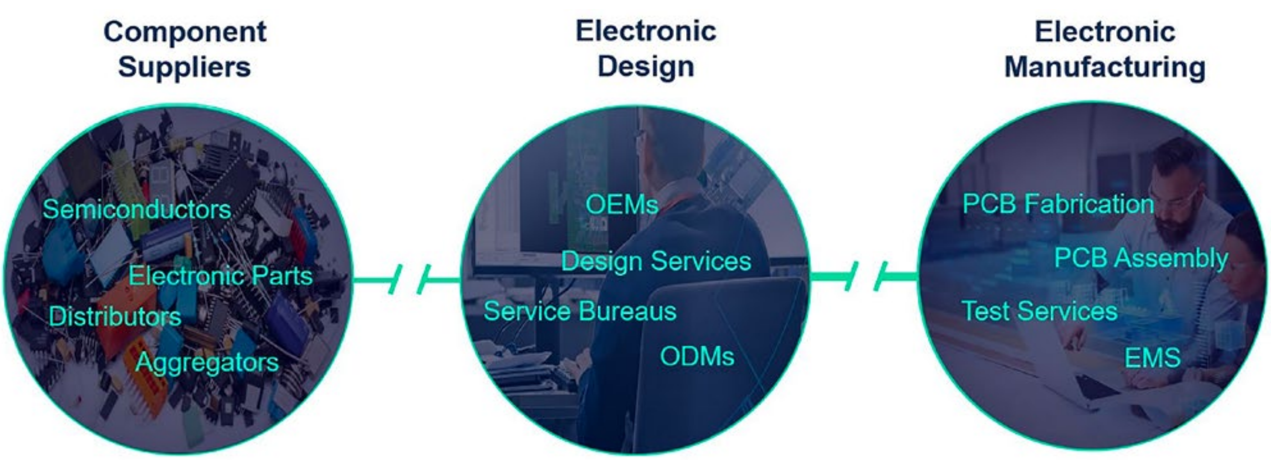


Figure 3: Broken digital threads, inhibiting cross-functional collaborations in the electronics value chain.

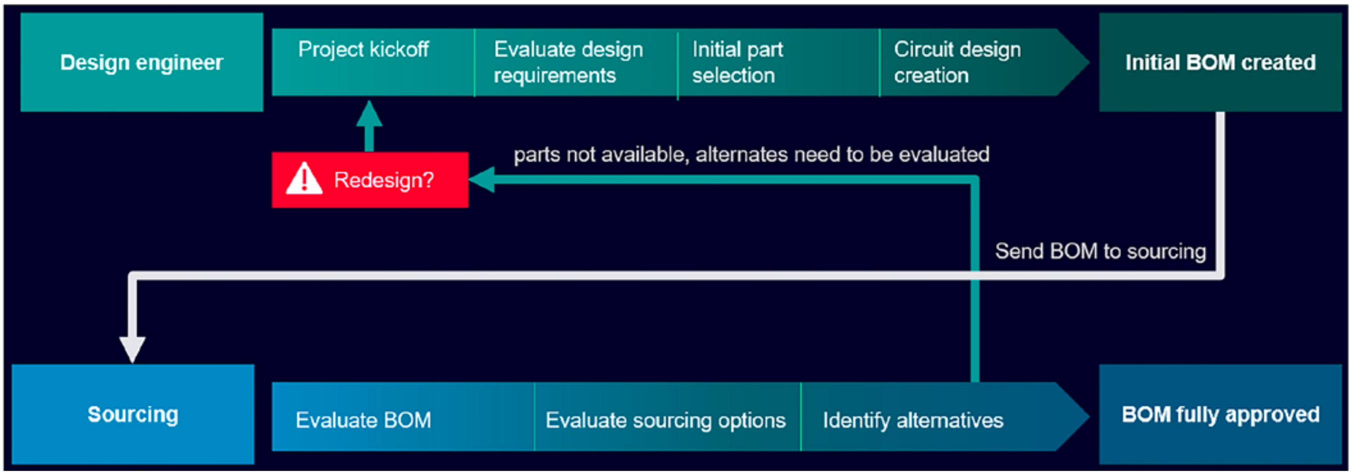


Figure 4: High-level view of the general design and sourcing process.

able to supply-chain volatility since the engineering and sourcing hand off is highly linear and not built with resilience in mind. Designs are seeing multiple cycles of BOM review and analysis of alternatives resulting in re-design events and/or outright extensions of schedules.

On a recent project I worked on, it seemed that at every daily project meeting, there was always something new to address regarding supply-chain issues. Whether the lack of available components meant that the EE had to stop his or her current activities and go back and manually research potential alternate parts not existent in the master library, submit a new library part request, create new circuitry, design a new section of the circuitry, or whether it forced me (the PCB layout designer) to redo a section of the board layout, the iterations and delays became extremely frustrating for the

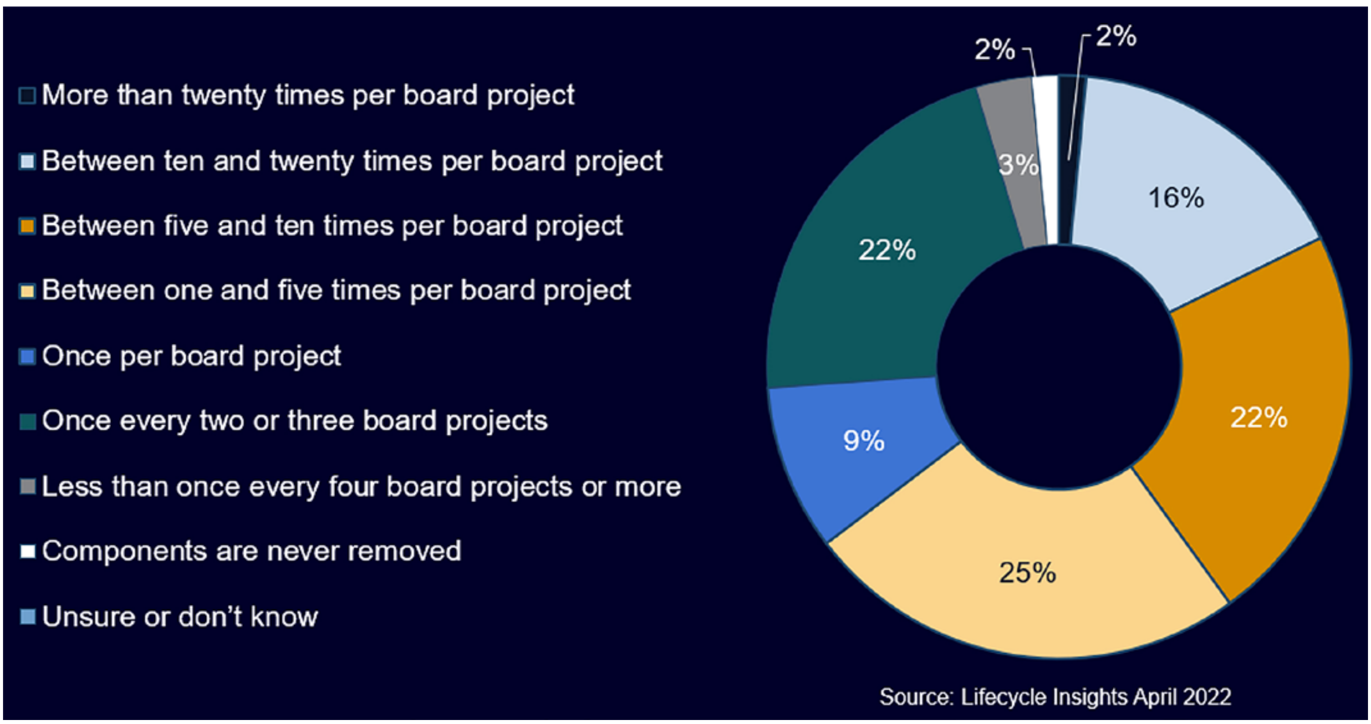


Figure 5: The average number of times engineers must remove an electronic component from their board design due to availability, lifecycle, or compliance.

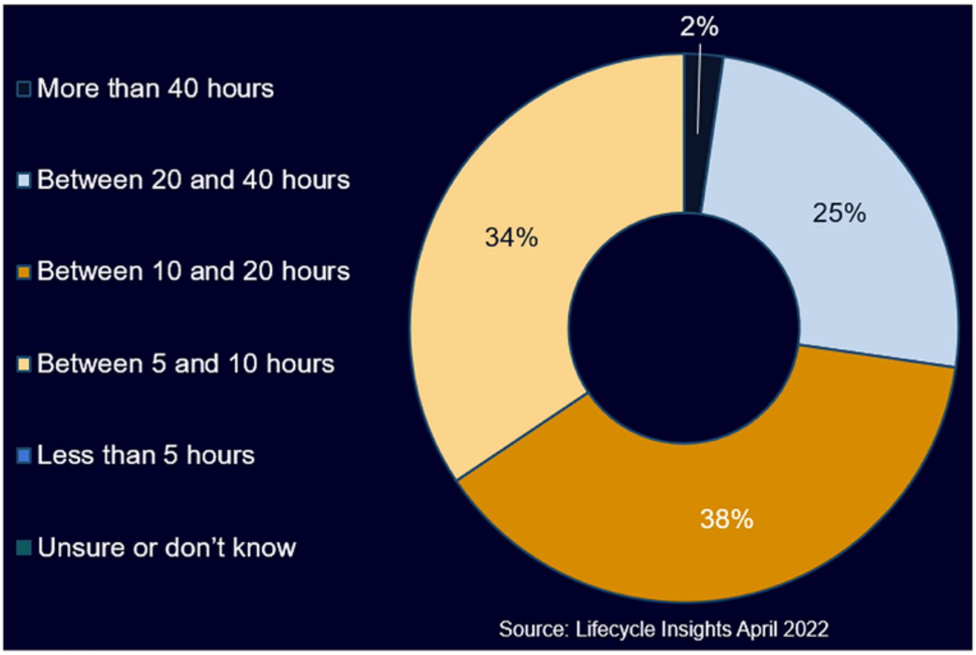


Figure 6: The average amount of time required to update a design after replacing an electronic component on a board due to availability, lifecycle, or compliance.

entire design team. The ripple effect of these supply-chain issues felt by the design team was painful and led to about 12 churn layout process loops. In my three decades of experience, I have never seen supply chain issues this bad. Talk about feeling the pain. It was brutal.

According to a survey published by Lifecycle Insights in April 2022, the negative impacts from supply chain disruption on these linear processes are significant and can be detrimental to a company's overall success. Whether it is due to increases in the average time engineers spend replacing components or the extra time it takes to update board designs after replacing an electronic component, availability, lifecycle, and compliance issues continue to lead to longer schedules and budget overruns. The metrics captured in industry surveys and from anecdotal evidence speak for themselves.

In the current state, operations combined with today's sourcing chaos in the supply chain, engineering teams are finding it harder and harder to be successful. From the perspective of many engineering teams, there are four key areas where supply chain disruptions have impacted their projects, with the net result

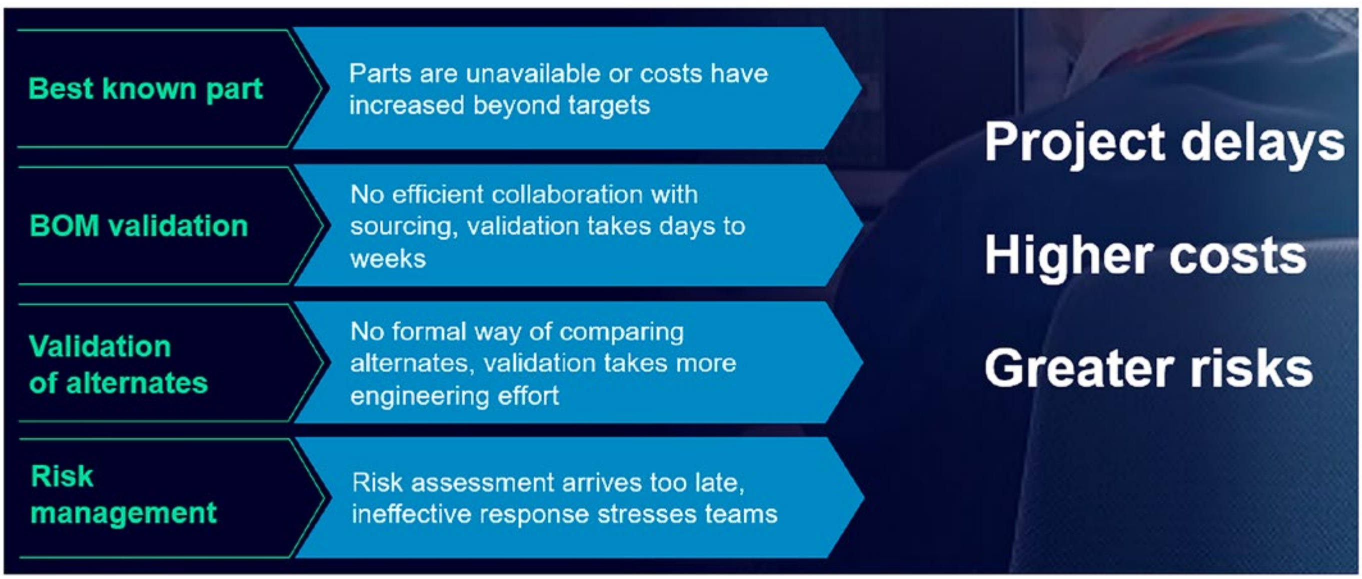


Figure 7: Four key areas where supply chain disruptions impact engineering projects.

IPC EDGE | ELECTRONICS WORKFORCE TRAINING

BUILD A BETTER WORKFORCE

- Electronics Assembly for Engineers
- Electronics Assembly for Operators
  - IPC-A-610 for Operators
  - IPC-J-STD-001 for Operators
- Wire Harness Assembly for Operators

2/3 of electronic industry companies have difficulty finding production workers.[1]

VS

Upskill Your Employees with IPC Electronics Workforce Training

IPC Electronics Workforce Training delivers the fundamental electronics and wire harness manufacturing knowledge critical to the success of engineers and operators. Scale consistent training across your organization.

See our current course listing on training.ipc.org.

Courses can be offered directly to employees or integrated into your training programs.

[1] IPC. (2017). Findings on the Skills Gap in U.S. Electronics Manufacturing.

being delays, high costs, and greater risks. The four key areas are best known part, BOM validation, validation of alternates, and risk management.

Supply chain instability is further compounded by other factors amplified by the pandemic: the rise of counterfeit parts impacting trust in the component supply chain, differing geopolitical interests resulting from globalization, labor disruptions arising across all industries, and increasing demands to design for environmental sustainability.

All this volatility highlights the dependency electronic systems companies have on the other members of the ecosystem they rely on to bring a successful product to market. These include component suppliers, product engineers, and product manufacturers. The reality is, before the pandemic, although we understood the value of integrating this ecosystem, recent events have clearly demonstrated its fragility and the critical need to bridge the chasms that result in lost innovation, budget hits, and time-to-market delays.

We see the electronics value chain as ripe for a digital transformation, given the persistent chasms that stifle innovation and product development execution. If the uncertainty of the pandemic demonstrated how companies employ digital transformation to move forward and excel, it's a relevant proof point to what's possible and to how new operational paradigms can be adopted quickly.

So, what can companies, and you, do about "The Problem?" In part two of this series, I'll share some of the promising and effective solutions that are key to supply chain resilience. This three-phased approach will shift supply chain resilience to the point of design and allow companies to optimize not only their systems design process but also every link to the stakeholders in the global electronics value chain. By uniting this value chain with the engineer's desktop, system design companies will see higher levels of digital transformation and the greater profitability that will result as they are empowered to realize tomorrow's designs today. **DESIGN007**

References

1. The Dimensional Research, 2020 State of Electronics Sourcing Decision-Makers Survey; and *Lifecycle Insights, 2021 NPI and Sourcing study.*
2. *Lifecycle Insights,* April 2022.

Stephen Chavez is senior product marketing manager with Siemens EDA. To contact Chavez or read past columns, click here.

An Overview of Copper Foils

Lightning Speed Laminates

by John Coonrod, ROGERS CORPORATION

Copper foils used in the PCB industry are deceptively complex. Copper is an excellent electrical conductor and thermal conductor, which makes copper foil ideal for the conductive layers of most PCB applications. There are many other copper foil properties which are important for an engineer to understand.

The copper foils used in the PCB industry are typically made as rolled wrought copper (rolled annealed copper or RA) or as electrodeposited (ED) copper. Rolled copper is made by starting off with a copper billet and then going through successive rolling processes to reduce the copper billet to a copper foil at the desired thickness. ED copper is made with an electrolytic plating process, where copper is plated on a large drum and peeled off as copper foil while the drum rotates. The speed of the drum rotation will dictate the thickness of the copper foil. After the copper foils are made, both RA and ED copper go through several processes to apply treatments to the copper.

There are many varieties of treatments applied to the copper and for many different reasons. Some treatments are a passivation treatment to ensure the copper does not oxidize until the consumer uses it. Other treatments are applied to the copper to help with good chemical bonding with certain resin systems (e.g., PPE vs. PTFE). Due to differences in the resin system, the materials react differently to the various treatments. Since each resin system has different bonding characteristics, some treatment/resin combinations

will be better than others for achieving a good bond. There are also copper treatments used to ensure proper bonding when exposed to high temperatures. Additionally, there are other treatments that can be applied to the copper foil to support a reliable bond interface when considering long-term elevated temperature exposures.

RA copper is often not used unless there are specific properties of rolled copper that can be beneficial to the application.

For most rigid PCB applications, ED copper is the most popular, however RA copper is used as well. Generally, RA copper is typically more expensive than ED copper. RA copper is often not used unless there are specific properties of rolled copper that can be beneficial to the application. Due to the nature of how RA copper is made, it has a very smooth surface. A smooth copper surface is advantageous for high frequency and very high-speed digital applications, where having lower insertion loss is desired. Another property of rolled copper that is innate to the copper manufacturing process is the in-plane grain structure that is very good for applications where bending the circuit is necessary. One technical drawback to RA copper, which is also related to the grain structure, is that the etching of fine circuit features can be problematic. This can be overcome to some degree, with specific etching conditions tailored to RA copper.

ED copper is widely used throughout the PCB industry. There are many different types of ED copper and typically they are classified by surface roughness and/or treatment. IPC has different classifications for ED copper based on copper roughness. Examples of a few of the classifications are LP (low profile), VLP (very low profile), and HVLP (hyper very low profile).

Copper roughness measurements can be confusing due to different methodologies. Generally, there are two categories for roughness measurements: contact measurements and non-contact measurements. The contact measurements use a physical probe, a stylus, to measure the peaks and valleys of the copper surface. The non-contact measurements are typically using reflected light or laser measurements to determine the peaks and valleys of the surface.

The contact profilometers can be less accurate due to the stylus tip size, where the stylus may not be able to resolve the depth of a very narrow valley. Also, the stylus can “plow” through the peaks and not get an accurate measurement. Those who are experienced with this technique understand these issues and there are some adjustments that can be done to mitigate the accuracy concern. However, as a general statement, the non-contact profilometers are typically more accurate than the contact profilometers for fine-grain copper types.

There are many ways to describe the surface roughness profile. Engineers working directly with the PCB fabrication will typically want to use the Rz profile, which is the peak-to-valley measurement over a sample area, defined by multiple line measurements. If the Rz profile is measured as an area (not a line measurement), the Rz profile would be designated as Sz. The engineers working with electromagnetic modeling software for high frequency or high-speed digital (HSD) will often want the Rq or Sq profile of the surface. The electromagnetic modeling industry has found good correlations between the Rq or Sq profiles and the surface impact on RF or HSD performance. The Rq and Sq are basically the root mean square of the surface roughness when measured with many samples.

There is also another property of surface roughness which the modeling industry is interested in: the surface area index (SAI), sometimes called the surface area ratio. This roughness number can be used for a specific electromagnetic model which has shown good accuracy across very wide bandwidths. The SAI is basically the roughness of a scan area as compared to that same area with an ideal flat surface.

This column gives a simple introduction to copper foils for PCBs. For more information on the copper foils used in the PCB industry, visit the Rogers Technology Support Hub on the Rogers' website, and view the Application Note, "Copper Foils for High Frequency Materials." DESIGN007

John Coonrod is technical marketing manager at Rogers Corporation. To read past columns or contact Coonrod, click here.

SEMI Europe Honors Outstanding Leadership in Microelectronics

Imec President and CEO Luc Van den hove received the 2021 European SEMI Award and Soitec CEO Paul Boudre the 2021 Special Service Award at the SEMI Industry Strategy Symposium Europe (ISS Europe 2022). Established more than 30 years ago, the European SEMI Award recognizes key players in the global manufacturing supply chain, highlighting their leadership excellence and strategic contributions that led to critical advances in the semiconductor industry.

"We applaud Luc Van den hove and Paul Boudre for their excellent work and outstanding research and development leadership and contributions to the European semiconductor community," said Laith Altimime, president of SEMI Europe.

Van den hove received his Ph.D. in electrical engineering from the KU Leuven in Belgium. In January 2007, he was appointed imec EVP and COO and has served as president and CEO of imec since 2009.

Boudre holds a graduate degree in chemistry from France's Ecole Nationale Supérieure de Chimie de Toulouse. He joined Soitec in 2007 as Director of Sales, Marketing and Business Development. He has been the CEO since 2015 and a member of the Board of Directors. Boudre has been active in the semiconductor industry for 30 years.

Nominations for the 2022 SEMI European Award are open. Please see award guidelines. Prior European SEMI Award recipients hailed from companies including CEA-Leti, Technical University of Dresden, Catholic University of Leuven, STMicroelectronics, EV Group, Infineon, and the Fraunhofer Institute. See the list of past SEMI European Award recipients.

(Source: SEMI)

Luc Van den hove

Paul Boudre

Optimize Your Thermal Management

Sensible Design

by Jade Bridges, ELECTROLUBE

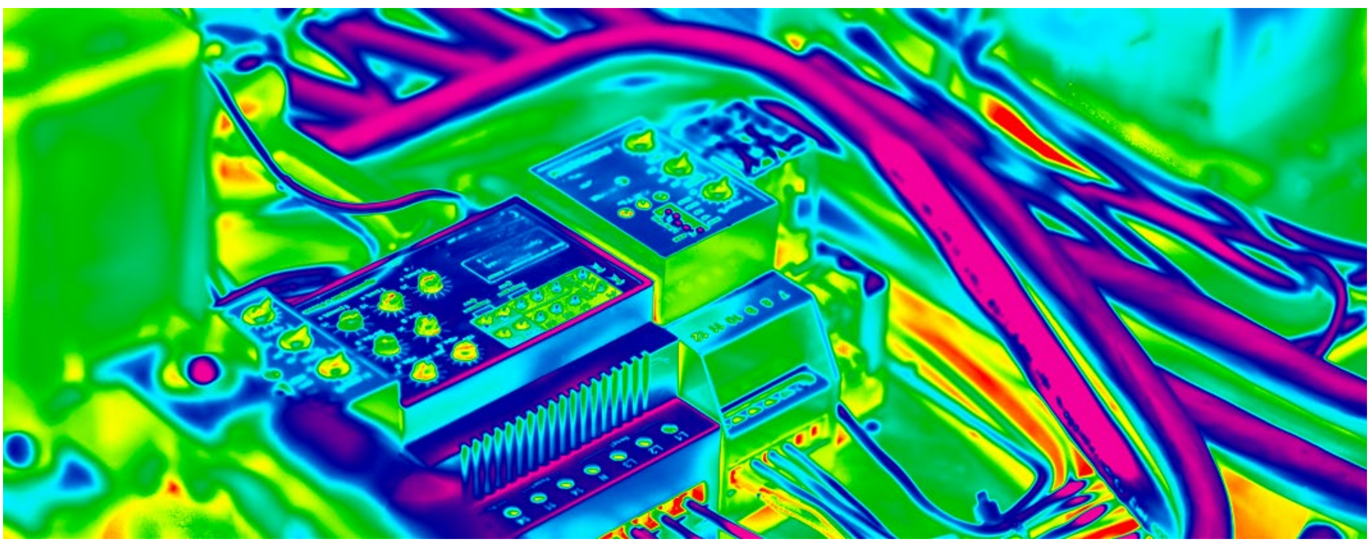
In this column, I'll be looking at how to help you improve reliability through thermal management. Poor reliability arising from thermally induced circuit failures might prove detrimental to brand reputation, but if the application served a critical role, the outcome would be unthinkable. Such applications might include:

- A safety-critical device upon which the safety of personnel working in a hazardous environment might depend
- One that simply would not function without proper thermal management procedures in place
- A device that has a defined working temperature range when in use, or a piece of equipment designed to work in harsh or extreme conditions, which must work reliably, regardless of those conditions

There are a variety of thermal management materials to choose from, and they serve different purposes depending upon the physical constraints of the application, component layout and assembly geometry, the environment in which the assembly will be placed, the severity of duty, and so on. Then there are more specific questions to ask, such as what's essential for efficient heat transfer, or should I use a bonding or non-bonding material? So without further ado, lets address this in our five-point format.

1. What key issues must be considered when trying to achieve efficient heat transfer?

Again, think variability of environmental conditions. Just because the heat transfer is efficient under standard ambient conditions, doesn't mean that it will remain stable over



the lifetime of the product. Accelerated testing might be able to reveal related design issues; however, in-application testing is more likely to provide definitive information about the long term performance of heat transfer materials.

The application method and applied thickness of heat transfer compounds will depend on the type of interface or gap filling material used and both are crucial to achieving efficient heat transfer. These compounds provide a medium for improving the conditions under which heat transfer takes place and thereby maximise efficiency. Compounds should not be applied in excess amounts in the belief that they will achieve the thermal conductivity of a solid metal heat sink, and remember, high thermal conductivity doesn't necessarily mean high heat transfer. Measuring thermal resistance at the interface is the most effective way to ensure the thermal compound is improving the efficiency of heat transfer and these results can differ depending on the thickness and quality of application.

These compounds provide a medium for improving the conditions under which heat transfer takes place and thereby maximise efficiency.

Throughout our discussions, you will see both points come up time and again. They are crucial to ensuring effective and efficient heat transfer with reliable performance from each device as well as over the lifetime of the product.

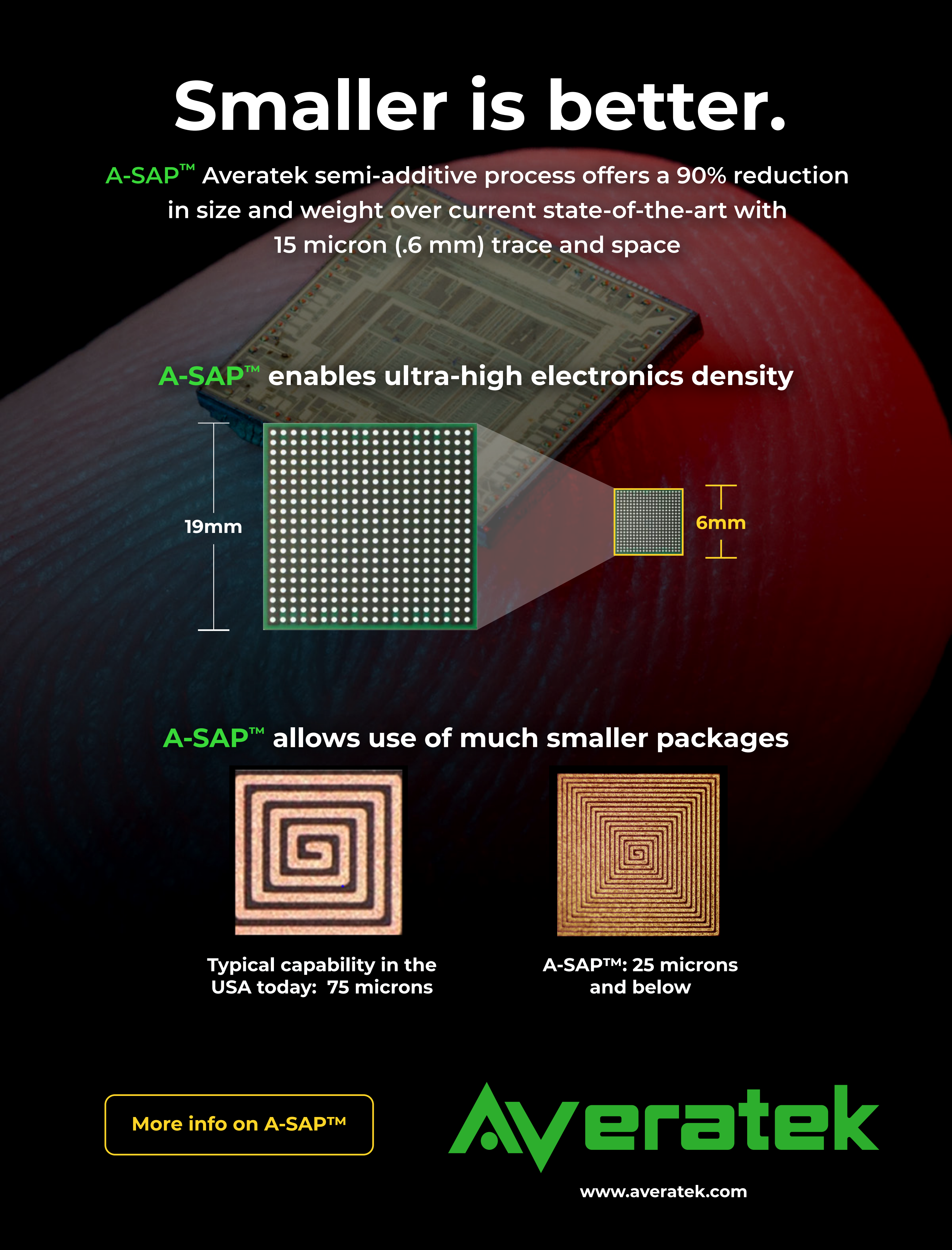
2. Bonding or non-bonding?

There are many different types of thermally conductive materials, and choosing between them will be dictated by production requirements and application design, as well as critical performance factors that must be achieved. Due to the variety of products available, this question is often one of the first to arise and can sometimes be answered very simply. For example, choosing between a bonding or non-bonding material may depend on whether the heat sink needs to be held in place by the interface material, in which case a bonding compound is the better choice. There are also many other factors that can affect this choice and it is often better to review the requirements as a whole first. This allows a holistic view of the requirements from application through to performance in use. Usually by taking this approach, the decision of bonding or non-bonding often becomes clear.

3. Increasing efficiency across a wide temperature range

Thermal changes are common within heat dissipation applications because most devices are switched on and off, or have varying power requirements in use. In addition, environmental temperature changes can lead to extremes within the device. Automotive applications are a good example, as these must also operate after being powered down in conditions well above and below what we would consider a standard ambient temperature.

It is therefore critical that the chosen thermal dissipation media operates within the temperature limits defined for the device, while maintaining performance during changeable conditions. A typical problem is "pump-out" whereby the stresses exerted by the minute changes in dimensions of the interface substrates can cause a non-curing interface material to move over time. The ability of a thermal interface material (TIM) to resist these stresses will improve the performance of the device over its lifetime and will be dependent upon the interfacial spacing, as well as the type and amount of TIM applied.


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
19mm
6mm
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

4. Know your dimensions

Understanding the dimensions of your application is critical to the selection of appropriate thermal management materials. A thermal interface is the space between a component and its heat sink, and the thermally conductive media used in this space are referred to as thermal interface materials. This space is usually very small, i.e., on the micron scale. A gap filling application, on the other hand, is more to do with the distance between a component and the metal housing that encloses an electronics assembly and is usually measured in millimetres. For example, a thermally conductive material is placed to help minimise the chances of hot spots within the unit itself while the casing is used as the heat sink.

The difference between a few microns and a few millimetres could be critical to the performance of the thermal medium chosen. For example, if you place a TIM in a gap filling application, it is likely to be unstable in a thicker layer; with vibration or following a period of temperature cycling, it could easily be displaced. And if a gap filling material is used in a thermal interface application, it will be very difficult to achieve a thin, even film, creating a higher thermal resistance at the interface and consequently reduced heat transfer efficiency.

5. Tips for material selection

As discussed, for an interface material, the bond line thickness needs to be as thin as possible to minimise thermal resistance. For a gap filling material, the dimensions of the gap need to be considered and environmental factors evaluated to decide whether a curing or non-curing product would be best in terms of the stability of the material under the application conditions as well as considering the possible effect on physical stresses within the unit.

In all cases, the operating temperature range and environmental conditions of the application need to be reviewed. If very high temperatures are expected, a silicone may be required, and if the assembly is subject to rework, then a non-bonding, non-curing product should be used. If thermal protection is localised to one component, a curing product is the better choice as it will avoid migration of the material to neighbouring components. However, if that component or area is sensitive to changes in coefficient of expansion, it must be ensured that the cured product used does not cause any negative impact over the application temperature range. With consideration of the application requirements in unison, the optimum thermal management product can be defined, achieving increased performance and reliability of the design.

There's a lot to consider for successful thermal management.

There's a lot to consider for successful thermal management. Getting it wrong could compromise the reliability of an electronic assembly and shorten its life expectancy or even prove critical. It's strongly advised to do your calculations; consider the application requirements as well as operational and environmental conditions, and test, test, test. Experimenting with the product in the actual design is the only way to truly understand the performance that can be achieved and conclude with the correct selection of thermal management products. First and foremost, seek some expert advice, discuss your application, and identify the most suitable products for review. DESIGN007

Jade Bridges is global technical support manager at Electrolube. To read past columns from Electrolube, click here. Download your free copy of Electrolube's book, *The Printed Circuit Assembler's Guide to... Conformal Coatings for Harsh Environments,* and watch the micro webinar series "Coatings Uncoated!"

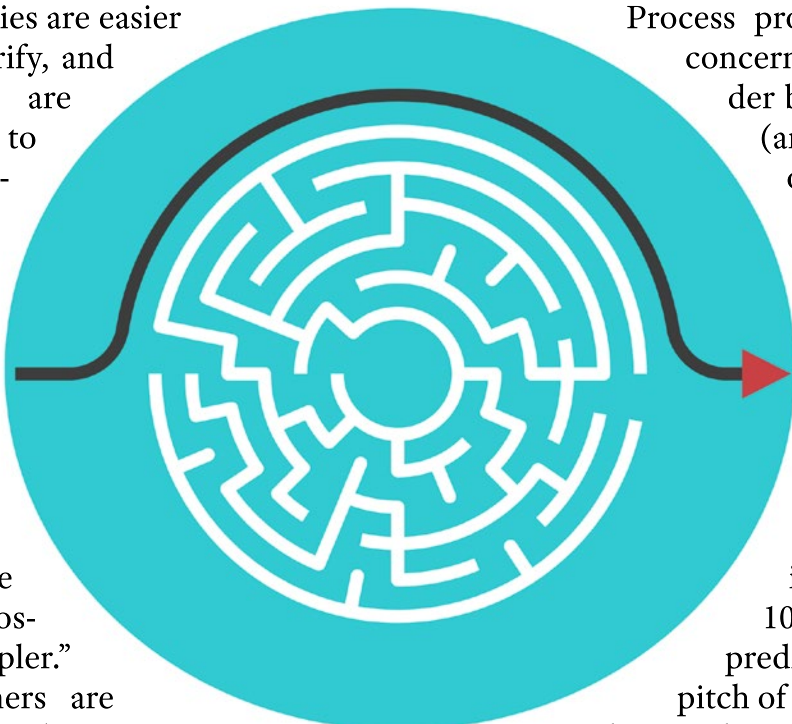
The Many Benefits of Eliminating (Most) Solder

Flexible Thinking

by Joe Fjelstad, VERDANT ELECTRONICS

"It is vanity to do with more that which can be done with less." —William of Occam, 14th century monk, logician, and philosopher

Many scientists are familiar with the concept of *Occam's Razor* (also sometimes called the Law of Parsimony). The basic idea postulates that when attempting to explain any given observed phenomena, the simplest explanation is preferable to those which are more complex. The reasoning behind the concept is that simple theories are easier to examine and verify, and not coincidently, are more often found to be true. Simplicity is a theme that has been recommended by some of the greatest minds in history. "Simplify, simplify," urged Thoreau. Einstein advised, "Everything should be made as simple as possible, but not simpler." These great teachers are both instructional and inspirational; while echoed throughout history, their words are not always followed because it is, ironically and unfortunately, easier to make things more complex. Yet these touchstone words have held sway over me for most of my life.



With that in mind, I will postulate that some long-time readers of my musings may be familiar with the Occam Process, which I first proposed in 2007 on the heels of the EU ban of lead in electronics solder. The Occam Process proposed to eliminate concerns about lead in solder by eliminating solder (and the root cause of perhaps as much as 70% of failures, according to a relatively recent scholarly paper[1]) from most (if not all) electronics manufacturing.

Having been influenced by the industry's roots in 100-mil-center designs predicated on the lead pitch of a ubiquitous dual-in-line package (DIP), the idea of a

standard grid was something that just made sense to me. Thus, the idea of solderless assembly was tied together and accompanied by a notion that all SMT IC packaging industry (which had allowed lead pitches and formats to explode in the conversion to SMT) in the future could/should minimize the number of lead pitches, preferably to a single, standard base grid pitch akin to the beauty and simplicity of the earlier 100 mil pitch. From a single base, the package developer could depopulate to meet the majority of IC package needs. The suggestion was 0.5 mm and that all package designs have terminations on that grid—making design layout much simpler. Further, it was suggested that all terminations be simply copper (i.e., no solderable finishes such as solder, nickel-gold, or the like). Because no solder would be used in assembly (all connections would be made by copper plating), the devices would not be subjected to high temperature processes associated with first attaching solder balls at high temperature, then mounting them to a PCB with one or more additional thermal cycles, which cumulatively reduce device and substrate reliability.

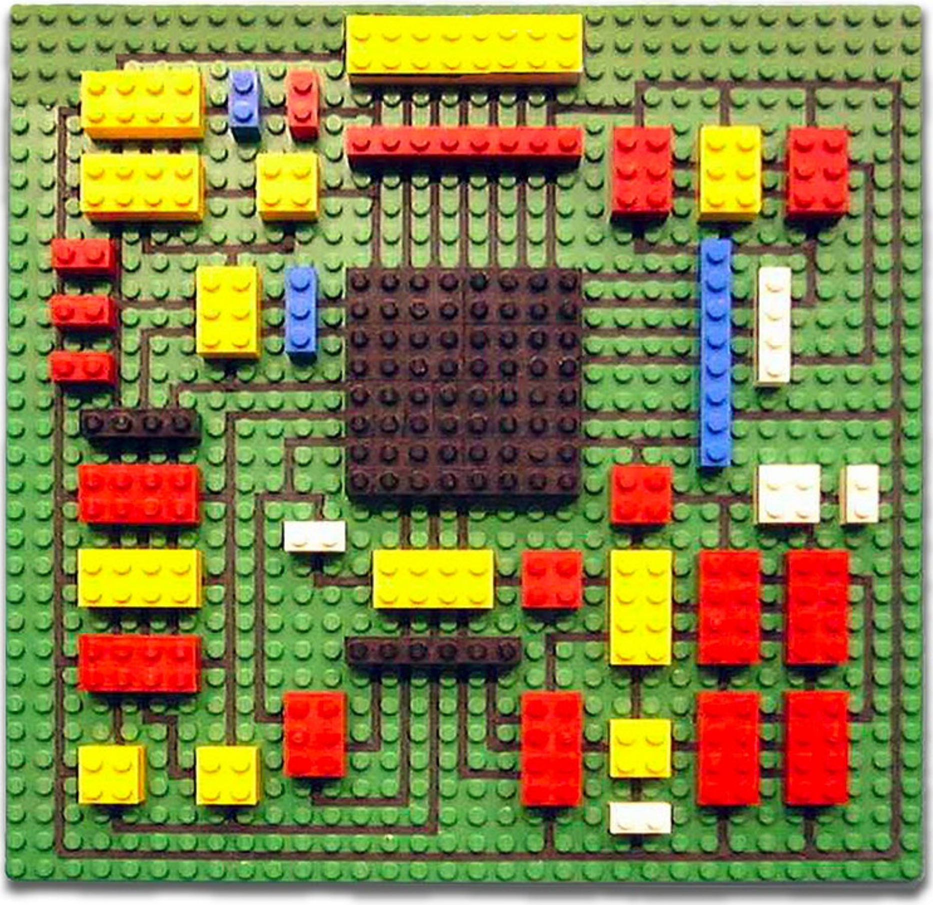


Figure 1: Photo of a mock electronic module created using LEGO bricks in 1995 to visually illustrate the compelling advantage of using components all having standard grid pitch for terminations—making the potential for design layout and routing much simpler.

Several years ago, I wrote a short, simple book on the Occam Process titled *Solderless Assembly for Electronics—the SAFE Approach*[2]. While, it did not make the New York Times best-seller list, it has nevertheless enjoyed decent circulation for a technical book. But it occurred to me that I could simplify my message further and set my sights on creating a one-page comparison of Occam with traditional PCB manufacture and assembly that anyone might be able to understand. I am providing that list (Table 1) along with a picture of a LEGO "module" that Vern Solberg, my long-time friend and colleague, created at my request with the LEGOs and my simple instructions to visually illustrate the idea in a way anyone might understand. The purpose was to help the customers of Tessera, the pioneering chip-scale packaging company where we both worked in the mid-1990s, to visualize and appreciate the huge potential value of using a common pitch for all devices. Darren Smith, the brilliant electronic interconnect designer and colleague, went a step further to demonstrate the potential[3] by reducing his

Table 1: Occam/SAFE processing comparison with traditional PCBA manufacturing.

Project phase	Traditional Manufacturing	Occam/SAFE Processing
Design		
Interconnection Design	Complex due to lack of fixed grid	Simple Manhattan grid route
Interconnection Layers	Nominal	~30-50% less (standard grid)
Design software	Complex (multiple lead pitches)	Simpler (standard grid route)
Drill and pad size	Varies	Single option possible
Component formats	Numerous types (legacy carryover)	Fewer when standard grid used
Equivalent design size	Greater and more layers required	Smaller up to 50-70% smaller
Component spacing	Further apart (rework and cleaning)	Closer (encapsulated assembly)
Part to part shielding	Post assembly	Integral (designed in carrier)
Stacked assembly	Difficult	Easier to plan and execute
Procurement		
PCB (circuit) production	Required	None (direct component connect)
Material options	Limited	Many (limited thermal exposure)
Material cost	Higher	Lower (lower temperature processes)
Time to market	Longer	Shorter (fewer steps in process)
Supply Chain	Longer	Shorter—fewer materials required
Infrastructure needed	Greater	Less (less special equipment)
Conflict materials	Requires verification	None required
Assembly issues		
Assembly complexity	Greater—huge component variety	Simpler due to more standardization
Solder stencil maintenance	Routine and required	None
Thermal excursions	Several high temp soldering steps	Fewer, lower temp steps
Component storage	Environmental control required	Limited to no concern
Solderability	Important for components and PCB	Unimportant
Solder related problems	Endemic	Obviated
Lead-free solder issues	Yes	None
Inspection requirements	Multiple places	Fewer (reduced process steps)
Test requirements	Many	Few (component burn in)
Post assembly cleaning	Varied difficulty component dependent	Easier (intrinsic/in process)
Component/PCB warpage	Solder temp related concern	Limited
Performance		
Thermal management	More challenging	Integral metal core
ESD/EMI management	Post assembly with formed metal shields	Integral w/plated metal exterior)
Hermeticity	Box level or conformal coat	Integral w/plated metal exterior)
Signal integrity	Challenging due to mixed grid pitch	Easier
Reliability	Nominal (solder quality dependent)	Better (fewer interconnections)
CTE Control	Nominal to challenging	Better and easier
CAF concern	Nominal (worse at fine via pitch)	Obviated by process and structure
Microvia reliability	Nominal (assume staggered vias)	Better – stacked or staggered
Strategic		
Design security	Limited	More secure (encapsulated BOM)
Environmental impact	Nominal to worse	Lower
Packaged chiplet amenable	Varies	Chiplet friendly—LEGO model
Familiarity/experience	Established	Limited presently but logical

Note: The value judgements cited above are subjective but based on decades of objective observations and analysis of industry technical literature which routinely describe the problems related to the manufacturing processes which can manifest defects and failures in PCB assemblies.

Premise: Better to first do the right things right than to do the wrong things perfectly.

original design effort by roughly 70% in size while taking the original layer count down from 12 layers to six without reverting to the use of extreme fine lines.

So, to keep things simple, a list is shown in Table 1 as it stands now. I expect the list will get longer as more distinct benefits of solder-free assembly are recognized.

Conclusion

Soldering is a complex process where many things can and do routinely go wrong. It is far from perfect in both yield and reliability; however, I have frequently mentioned in my presentations over the past decade that it will continue to be used for assembly well into the foreseeable future. In all honesty, I can see important places where solder can and will work in the future of electronic manufacturing.

Unfortunately, familiarity with a process does not make it infallible. I am reading with increasing frequency the myriad problems of solder along with the proposals for the next lead-free solder alloy (78 alloys at last count), the next best flux, the best way to characterize the reflow process, the most recent problem identified, and the best piece of equipment to identify that problem and tool to fix it. I graphically characterized the soldering industry in my SAFE book as a modern-day Sisyphus; instead of a boulder, it's characterized as pushing a solder ball up a hill only to have it roll down again for another push to the top the nest day. Job security, for sure, but at what cost to the soul?

There are better ways to execute most routine activities, but they typically require an appetite for change, and though inevitable, can be frightening. The Wright brothers did not make their first flight off a cliff; they did it on a sandy beach. They were not fools, but they did understand the potential benefits of flight and so met their fear head on. Today, it is safer to be at 40,000 feet than it is to be on the ground and air travel is the safest possible way to get around. I believe that future designers will eventually see the benefit of designing products using the principles which Occam gifted the world eight centuries ago. Again, to make things simple.

This inspirational quote on the topic of design simplicity is from esteemed French writer Antoine de Saint-Exupéry in his autobiography, Wind, Sand and Stars, "A (good) designer knows that he has achieved perfection in design not when there is nothing left to add but when there is nothing left to take away."

Personal note: It is with great sadness that I share news that my colleague Darren Smith died by his own hand in April after decades of physical and mental suffering. We spoke of his suffering on numerous occasions but unfortunately his depression won out. Darren's great enthusiasm for the Occam idea (which included his getting a personalized OCCAM license plate for his car) was immense but insufficient to keep him in life. If the reader is having such problems, please seek professional help through a suicide prevention service. The industry needs you to stick around. **DESIGN007**

References

1. "Reliability issues of lead-free solder joints in electronic devices," *Science and Technology of Advanced Materials,* Volume 20, 2019, printed at Taylor and Francis Online.
2. *Solderless Assembly for Electronics: The SAFE Approach,* by Joe Fjelstad.
3. Ibid, p. 51.

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

Flex007 Highlights

DuPont Opens New Kapton, Pyralux Production Line in Circleville, Ohio ▸

DuPont Interconnect Solutions held a ribbon-cutting ceremony with elected officials and business leaders to formally mark the completion of its $250 million capital project to expand production of Kapton® polyimide film and Pyralux flexible circuit materials at the Circleville manufacturing site.

Altix Receives Bundle Machine Order from Major Chinese FPC & PCB Player ▸

Altix is delighted to announce a significant order for both Direct Imaging and Contact Printer equipment. The bundle encompasses both panel and RtR solutions to be installed at a new plant in Jiangxi province.

Flexible Thinking: Blue Skying It With Aluminum Rigid-Flex ▸

Aluminum is an amazingly versatile metal and has found its way into countless products since its discovery during the reign of Napoleon III of France. At the time, it was more valuable than gold. It took some time for scientists to calculate that it was by far the most abundant metal found in the earth's crust at 8.3%.

FLEX Conference 2022 to Highlight Latest Flexible Hybrid Electronics Innovations ▸

FLEX Conference will gather industry experts July 11-14, 2022, at the Moscone Center in San Francisco for keynotes, panel discussions, technical sessions and product-based demonstrations highlighting the latest innovations in flexible and printed electronics.

Rogers Corporation Reports Q1 2022 Results ▸

Rogers Corporation announced financial results for the first quarter of 2022.

Nano Dimension's Preliminary Q1 2022 Revenues: Approximately $10.5 Million ▸

Nano Dimension Ltd. announced a preview of its financial results for the first quarter ended March 31, 2022.

Eltek Reports 2022 First Quarter Financial Results ▸

Eltek Ltd., a global manufacturer and supplier of technologically advanced solutions in the field of printed circuit boards (PCBs), announced its financial results for the quarter ended March 31, 2022.

Trackwise's Space, Cost-Saving FPC Technology Enables New Generations of Medical Instruments ▸

Trackwise is seeing a significant increase in medical applications using its flexible printed circuits (FPCs) including the company's patented Improved Harness Technology (IHT) that uniquely enables multilayer FPCs of unlimited length to be constructed.

Flexium YTD Sales Up 17.8% ▸

Taiwan-based flexible printed circuit (FPC) manufacturer Flexium Interconnect Inc. has posted sales of NT$1.72 billion ($57.97 million at $1=NT$29.72) in April, down by 28% year-on-year, and by 49.37% from the previous month's NT$3.4 billion ($110 million).

TOP TEN

EDITOR'S PICKS

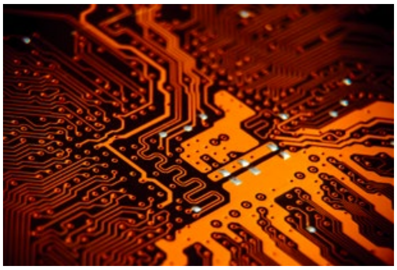
Connect the Dots: Leaning into Lean Manufacturing

The worst part of the global COVID pandemic brought unpredictability and uncertainty to an otherwise stable PCB Industry. Like many in the board business, Sunstone faced increasing demand from essential businesses while also dealing with inconsistent employee availability and social distancing guidelines that slowed the manufacturing process.

Lightning Speed Laminates: The Importance of Circuit Features for mmWave Applications

Over the past several years there has been a steady increase in millimeter-wave (mmWave) applications. As those applications increased, most aspects of the electronics industry, including the PCB industry, were forced into a steep learning curve.

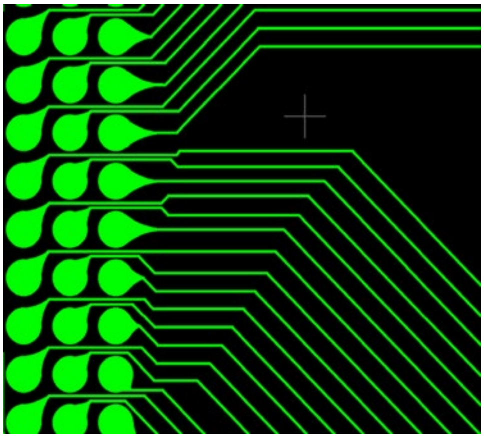
Designing PCBs With Additive Traces



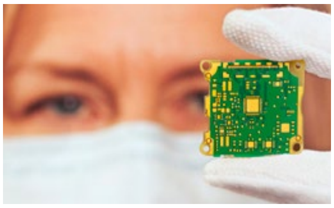
Advances in technology have been clear within the component packaging industry, as the ball grid array (BGA) package sizes reduce from 1.0 mm pitch to 0.8 mm, 0.4 mm, and even beyond. Currently, the high-density interconnect (HDI) method typically used for the breakout of such parts has been to create the smallest possible subtractive-etched traces with microvias to allow for connections and escapes on the inner layers of your PCB.

Designing Additive and Semi-Additive PCBs

With components getting smaller and electronic devices becoming more compact, we are reaching the physical limits of the typical etched fabrication processes. To address these limits, new additive and semi-additive processes are being developed to fit into the current fabricators' production lines without too much disruption or extra cost.



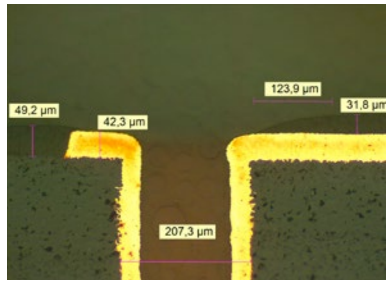
Fresh PCB Concepts: Sustainability in PCB Design



In order to meet the increasing demand for smaller, more powerful, and complex electronics, PCB designers are under pressure to create boards that are not only reliable but also sustainable.

Additive Manufacturing Requires Additive Design Techniques

I want to share my thoughts on what additive manufacturing means for designers, especially how it relates to solder mask.



Altium Focusing on Educating Designers of Today and Tomorrow

Rea Callender, Altium's vice president of education, and Zach Peterson, founder of Northwest Engineering Solutions and a technical consultant for Altium's educational programs discuss Altium's curriculum—what drives the content development, the goals of their programs, and why there's never been a better time to continue your PCB design education.

Sensible Design: Let's Take Down the Heat on Resins

Electrolube's Beth Turner picks up the thread of her previous columns to describe where resins are currently playing a vital role in the modern world, while also offering an insight into their thermal conductivity, suitability for RF applications, and more.

A Textbook Look: Signal Integrity and Impedance

Believing that I knew a bit about signal integrity and controlled impedance, I was pleased to connect with an educational webinar that I hoped would extend my knowledge. In the event I was surprised at how little I actually knew, and the webinar was an excellent learning opportunity.

Elementary, Mr. Watson: The Five Pillars of Your Library, Part 5—Traceability

We have reached the end of this series regarding the five pillars of the component library. We now have a robust library that provides the required resources for the ever-changing industry.

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

Career Opportunities

Field Sales Engineer, North America

Location: New Hartford, NY

JOB SUMMARY:

The Field Sales Engineer, North America, is responsible for serving as Indium Corporation's lead sales contact and customer advocate to maintain existing sales and to drive new qualifications and sales of Indium Corporation products and services through effective account management and coordination of efforts throughout Indium Corporation's Metals, Compounds, Solar and Reclaim (MCSR) organization.

REQUIREMENTS:

- Associate's degree in a business or technical discipline
- Minimum 2 years related sales or technical field experience
- Technical aptitude
- Personable individual, with excellent oral and written communication skills
- Strong organizational skills
- Able to travel upon short notice
- Proficient in Word, Excel, PowerPoint

apply now

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

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

Responsibilities:

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

Qualifications:

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

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

apply now

Career Opportunities

Technical Support Applications Engineer

Full-Time — Duluth, GA

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

Responsibilities

- Provide timely, complete helpdesk support for Koh Young users
- Train users on proper operation, maintenance, programming, and best practices
- Recommend and oversee operational, process, or other performance improvements
- Effectively troubleshoot and resolve machine, system, and process issues

Skills and Qualifications

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

Benefits

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

apply now

Sales Technical Engineer

ALTIX, a French company, designs, manufactures, markets and services exposure equipment for the printed circuit board, flexible circuit, metal etching, touch panel and other industries. The U.S. subsidiary, focused on the sale and service of Altix equipment in North America, is looking for a sales technical engineer to support their growth.

Responsibilities

- Promote Altix's products by visiting customers
- Serve as a technical lead & product expert to provide technical recommendations to customers
- Gather on-the-ground market intelligence through customer contact
- Ensure sustainable growth in sales, profits, and market presence
- Develop new business and achieve targets for market penetration, sales and profit
- Manage sales partners

Skills & Qualifications

- Minimum 2 to 5 years' experience in sales for capital equipment in the PCB market or related industries
- Business development and marketing background preferred
- 5+ years' North American business leadership experience in related field
- Strong leadership, decision-making and communication skills.
- Proficiency in standard computer software applications such as Microsoft Office
- Excellent written and oral communication skills
- Willingness to travel within the US, Canada and to France for training

Email contact: sylvain.dromaint@altix.us

apply now

Career Opportunities

CHEMCUT

Director of Operations

State College, PA

Chemcut Corp., a world leader in wet processing equipment for the manufacture of printed circuit boards and chemical etching of various metals, is seeking a Director of Operations.

Objectives of the Role:

- Collaborate with the CEO in setting and driving organizational vision, operational strategy, and hiring needs.
- Oversee manufacturing operations and employee productivity, building a highly inclusive culture ensuring team members thrive and organizational outcomes are met.
- Directly oversee manufacturing operations, production planning, purchasing, maintenance & customer service (product support) and partner with the CEO and controller on sales management to budget for sufficient investment capital to achieve growth targets.
- Aggressively manage capital investment and expenses to ensure the company achieves investor targets relative to growth and profitability.

Qualifications:

- Bachelor's degree in mechanical, electrical, or related fields
- 5+ years' experience in leadership positions
- Leadership skills, with steadfast resolve and personal integrity
- Understanding of advanced business planning and regulatory issues
- A solid grasp of data analysis and performance metrics
- Ability to diagnose problems quickly and have foresight into potential issues

Preferred Qualifications:

- Master's degree in business or related field
- International business experience

To apply, please submit a cover letter and resume to hr@chemcut.net

apply now

INSULECTRO

Are You Our Next Superstar?!

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

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

apply now

Career Opportunities

ventec INTERNATIONAL GROUP 騰輝電子

European Product Manager
Taiyo Inks, Germany

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

Duties include:

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

Skills and abilities required:

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

What's on offer:

- Salary & sales commission--competitive and commensurate with experience
- Pension and health insurance following satisfactory probation
- Company car or car allowance

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

apply now

MacDermid Alpha ELECTRONICS SOLUTIONS

R&D Scientist III
Orange, CT

Job Description: The scientist will be a leader in technology for plating chemistry development, electrolytes, and additives. The position is hands-on, where the ideal candidate will enjoy creating and testing new aqueous plating processes and materials to meet the most demanding semiconductor applications related to Wafer-Level Packaging and Damascene. The qualified candidate will work as part of the R&D team while interacting with scientists, product management, and application engineers to commercialize new products for the advanced electronic solution business.

apply now

Technical Marketing Specialist
Waterbury, CT

This position provides information from the product team to the marketing communications team. It is a multifunctional role that requires some experience within electronics manufacturing supply chain or knowledge of how electronic devices are manufactured, specifically PCBs, semiconductors, and the chemical processes utilized therein. The primary function of this role is to help in the generation of product marketing collateral, but also includes assisting in tradeshow content development, advertising, and launches.

apply now

Career Opportunities

Career Opportunities

Career Opportunities

Manncorp™

SMT Field Technician
Hatboro, PA

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

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

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

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

apply now

Prototron Circuits

Sales Representatives

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

Reasons you should work with Prototron:

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

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

apply now

Career Opportunities

Career Opportunities

Career Opportunities

The Test Connection, Inc.

Rewarding Careers

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

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

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

Associate Electronics Technician/ Engineer (ATE-MD)

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

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

apply now

Test Engineer (TE-MD)

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

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

apply now

Sr. Test Engineer (STE-MD)

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

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

We proudly serve customers nationwide and around the world.

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

apply now

Career Opportunities

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

ARLON
ELECTRONIC MATERIALS

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

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

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

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

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

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

apply now

Career Opportunities

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

APCT
Passion | Commitment | Trust

APCT, Printed Circuit Board Solutions: Opportunities Await

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

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

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

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

apply now

Career Opportunities

I-007e EDUCATIONAL RESOURCE CENTER

Latest I-007eBooks

The Printed Circuit Designer's Guide to... High Performance Materials

by Michael Gay, Isola

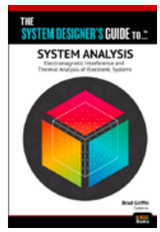
This book provides the reader with a clearer picture of what to know when selecting which material is most desirable for their upcoming products and a solid base for making material selection decisions. Get your copy now!

The Printed Circuit Designer's Guide to... Stackups: The Design within the Design

by Bill Hargin, Z-zero

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

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



by Brad Griffin, Cadence

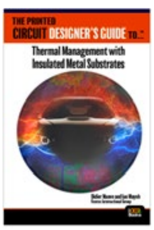
In this book, the author, Brad Griffin of Cadence, focuses on EM and thermal analysis in the context of data center electronics systems. Be sure to also download the companion guide for end-to-end solutions to today's design challenges.

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

Thermal Management: A Fabricator's Perspective

by Anaya Vardya, American Standard Circuits

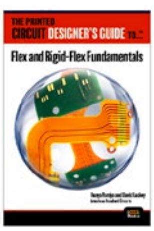
Beat the heat in your designs through thermal management design processes. This book serves as a desk reference on the most current techniques and methods from a PCB fabricator's perspective.



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.



Flex and Rigid-Flex Fundamentals

by Anaya Vardya and David Lackey, American Standard Circuits

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

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

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

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

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

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

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

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

SALES MANAGER: **BARB HOCKADAY**
(916) 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 © NUR**

DESIGN007 MAGAZINE

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

June 2022, Volume 11, Number 6
DESIGN007 MAGAZINE is published monthly,
by BR Publishing, Inc.

ADVERTISER INDEX