I-Connect007

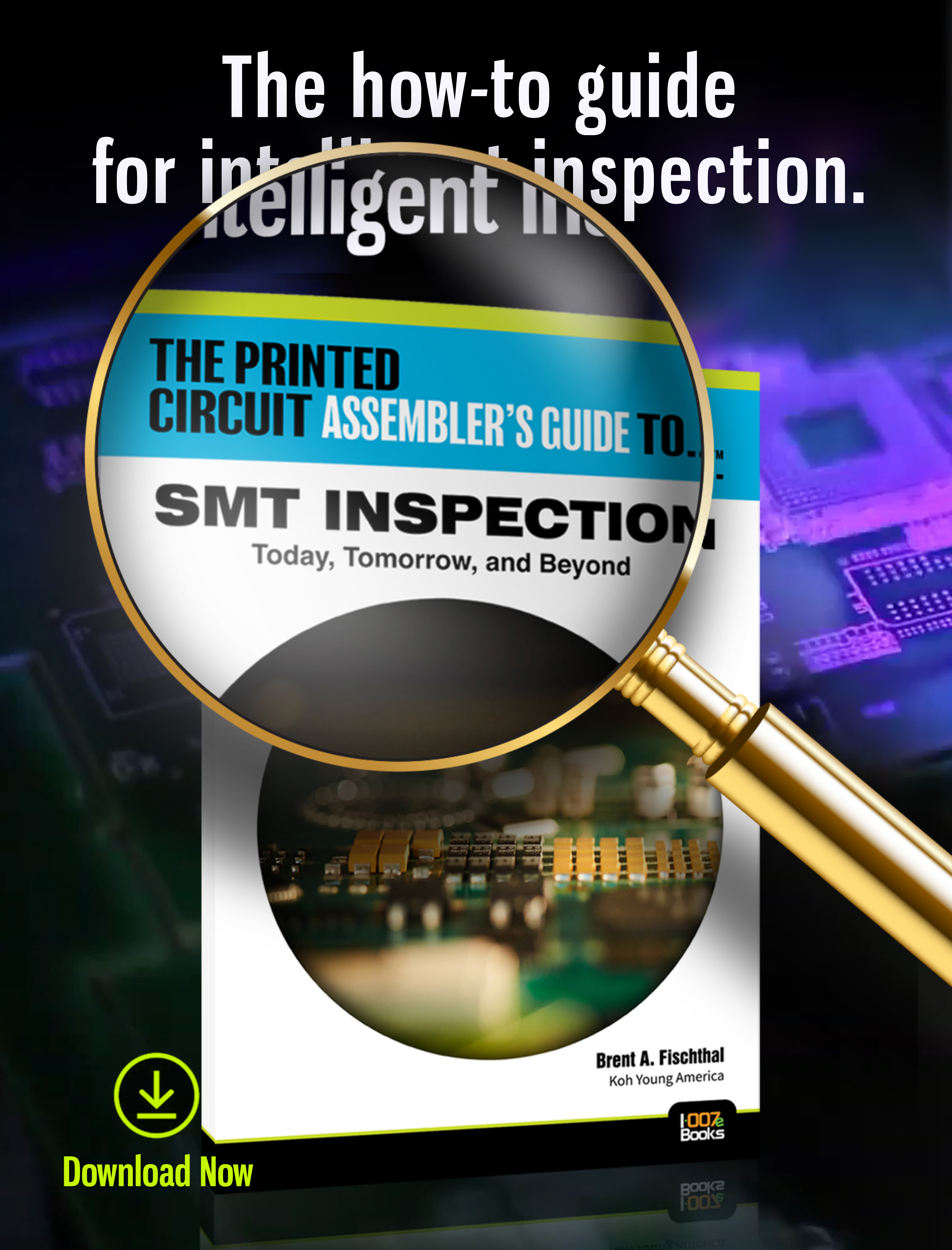
MARCH 2021

DESIGN007 MAGAZINE

DATA

don't trust...

verify

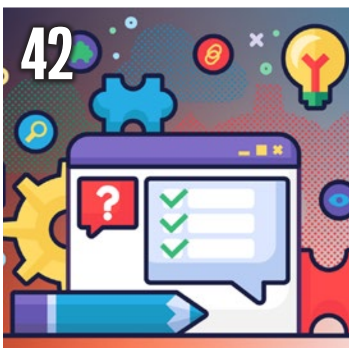


The how-to guide
for intelligent inspection.
THE PRINTED CIRCUIT ASSEMBLER'S GUIDE TO...™
SMT INSPECTION
Today, Tomorrow, and Beyond
Brent A. Fischthal
Koh Young America
I-007e Books
Download Now

DESIGN007 MAGAZINE

Validating Your Data Supply

Keeping bad data out of your design involves a lot of detective work. Designers and design engineers need to assess the accuracy and completeness of data at every step of the design process. If one small error in a footprint or schematic goes unnoticed, the CAM engineer may be calling with bad news on Friday afternoon. This month, we asked our expert contributors to discuss their special operations and tactics for keeping bad data out of their designs.

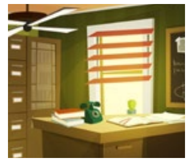


FEATURES

FEATURE COLUMN

DESIGN007 MAGAZINE

COLUMNS



HIGHLIGHTS

DEPARTMENTS

SHORTS

FLEX007

This Month in Flex

Keeping bad data out of your supply chain is no simple task with flexible circuits, and it's even trickier with rigid-flex circuits. This month, we look at how IPC-2581 Revision C takes the guesswork out of conveying build intent by digitizing much of the rigid-flex design data handoff, and what this means to flex designers.

Validating Your Data Supply

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

A few months ago. we decided to focus this issue on eliminating the "garbage in, garbage out." So many of our recent survey respondents pointed to bad or missing data as the primary cause of their design woes that we knew we had a good topic. We wanted to touch base with some people in the industry who have their processes dialed in, pick their brains about ways to keep bad data out of the design, and determine our path forward.

We started off by talking with Jen Kolar and Mark Thompson of Monsoon Solutions, a company that is constantly improving its processes by following the tenets of TQM. Monsoon is both a design bureau and provider of manufacturing services, working with fabricators and assemblers around the world.

Jen and Mark described the dozens of ways that a PCB design can go off the rails, almost all of them revolving around bad data finding its way into the design process. When they receive a design from a new customer, they become akin to PCB detectives. "Can we trust this designer's data?" "Is this their first controlled impedance design?" "Do we have to check every single aspect of this design?"

I used to be a crime reporter for a daily newspaper, and Jen and Mark reminded me of some of the detectives I once interviewed, with one big difference: They're trying to prevent a crime from happening, much like Tom Cruise's character in *Minority Report.*

Think about it: Your job involves a lot of detective work. You need to assess the accuracy and completeness of data at every step of the design process. Our surveys show that undocumented changes to CAD library parts pose a big stumbling block for designers. Do you need to check every part in your library?

At least OEM designers have some control of their data. If you're a designer at a service bureau, unless you're dealing with a long-time customer, everything that comes through the door is suspect. And the CAM engineer might as well be wearing Lieutenant Columbo's stained trench coat. It's the CAM folks' job to find faults in the design data package you've handed off—and they almost always find faults.

We have multiple reasons why bad data is missed, but I imagine time to market is the primary culprit. With all the potential areas where incorrect data can enter the design cycle, it's easy to miss one little mistake, which then propagates downstream, often necessitating a call from the fabricator at 6 p.m. on a Friday. Even the best detectives miss clues now and then.

So, this month, we asked our expert contributors to discuss their special operations and tactics for keeping bad data out of their designs. Jen and Mark at Monsoon not only discuss the challenges, they also explain why it's best to "trust but verify" data every step of the way. Dana Korf discusses how to gauge the experience level of a design team, and a few sure giveaways that designers may need more handholding on high-tech boards. Tim Haag explains how to interpret information that just doesn't make sense. Tamara Jovanovic shares a variety of examples to keep bad data out of your design cycle. Nick Barbin shows designers how to use checks and balances to stop the "garbage in, garbage out" cycle. And Ed Acheson provides an update on IPC-2581C, which allows rigid-flex designers to share build intent digitally.

We also have an article by Tony Plemel, and columns by Barry Olney, Kelly Dack, Phil Kinner, Patrick Crawford, Tara Dunn, John Talbot, and Matt Stevenson.

IPC APEX EXPO takes place March 8–12, and we'll be there, bringing you interviews with the industry's top technologists and managers. Maybe we can chat—even virtually! 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.

The Key to Eliminating Bad Design Data: Constant Vigilance

Feature Interview by the I-Connect007 Editorial Team

The I-Connect007 editorial team recently met with Jen Kolar and Mark Thompson of Monsoon Solutions to discuss ways to eliminate bad data from the design process, whether that be from CAD libraries, parts vendors, chip makers, or customers themselves. They key in on some problems and obstacles that allow incorrect data into the design cycle, and then highlight possible solutions.

Barry Matties: There are just so many places where you can introduce bad data into the design cycle. And as you both know, one of the biggest challenges is to get the data that you really need to be accurate right from the start. And, if it's wrong at the beginning, it's wrong all the way through.

Jen Kolar: It's like any additive process, where the errors are added along the way. You have garbage in, garbage out in each step along the way. Maybe you have a thoughtful front-end designer who does a thoughtful design and gives really good input, but you have a PCB designer who's sloppy and leaves half the detail out of the fab notes. Maybe it was good data into them, but they didn't translate all that to the fab shop. Or maybe they had good data from one point, but you have those additive errors. Maybe there was a partial error at the beginning, but it just builds and builds, and gets worse and worse.

Matties: The thing is, if it's good at one point, you have so many different points where it can go off the rails. It could be a combination of good data here, bad data there.

Kolar: I couldn't agree more, and I think part of it is dependent on who's driving the schedule in a project; a lot of times, the people driving the schedule just care that it's started. You have management saying, "Start," so whoever is designing the project is saying, "Okay. I'm still figuring out my electrical design, but I've been told to put it in schematic." You can end up with a project that takes a lot longer,

costs more, and works poorer than if you'd just waited and started two months later.

Matties: And you'll probably be doing multiple re-spins on it as well.

Kolar: Exactly. There are a lot of different sources of poor data: There's just lack of knowledge, schedule pressure, sloppiness, and there is misinformation. There have been a number of times that we have engineers send us reference designs and say, "Just do what the reference design says." Then we get to argue that the reference designs have nothing to do with manufacturability, and you repeat that cycle over and over.

From our perspective, when you're getting input data, ideally, you're getting enough variants of it that you have something to validate against. Maybe I get a board file that has some of the parts loaded. Then, maybe I also have a STEP file or mechanical data that I can sanity-check against that, or the schematic, which is also going to have some callouts.

Jen Kolar

Matties: You keyed in on some of the problems to get good data. One you mentioned was timing. People want this start date. What are some of the other problems that would lead us to the situation where the data's not there?

Kolar: Product intellectual property rights can be one. It may be that it's a proprietary datasheet that the engineer has access to, but you, as the designer, don't necessarily have, or they're not sure what you have access to. It can also be confidential projects and you're only getting a tiny subset of the information, so you're not understanding the big picture of how it might be used, and that's leading you to make false assumptions. We don't know what it's being used for, and we don't ask.

Matties: What other obstacles prevent the data coming to you?

Kolar: I think it's this constant mix: Some people are just sloppy, some just don't have time or the knowledge, and some people are just pulled in so many directions that they're just throwing something over the wall to check it off their list. I see a lot of errors get introduced when inheriting a previous design and leaving in the leftover "cruft." We talked a lot about this with fab notes and assembly notes in a previous issue: "I'm going to piece together these parts of the schematic and move forward, or I'm going to grab this PCB design and start with this." Well, I might set up my new layers, but I don't remove all the old ones that I don't care about. I just leave a bit of cruft, and that cruft piles up, and you go to the next person that borrows it, and the next person borrows it, and you just get all this cruft over time.

Matties: You covered a lot of problems. Mark, do you have any additional problems that you may see?

Mark Thompson: Well, a couple of different things. Jen touched on stackups—things that change, controlled impedances that change from design to design, from Rev. A to Rev. B to Rev. C. Frequently, we'll get a stackup from customers dictating the dielectrics, copper weights, effective dielectric constants on subsections. It doesn't do us any good to get controlled impedance changes and a previous stackup because all of that stuff now has to change. You'd have to redo a new stackup.

It's important to query the fabricator ahead of time, and they'll give the information for the trace and space, as well as the stackup. Now, bear in mind, if the fabricator comes back and says, "Hey, I'm going to have to change your copper weight," that's a red flag. The designer has to say, "Wait a minute. Now, if you're going to reduce my two-ounce to a half-ounce, you're digging into my current-carrying capacity of the design." That ends up being very problematic.

Mark Thompson

Netlists are another area where things can happen. Parts will change, and you'll have to re-annotate the design. You've got to bring in the new parts and update the schematics, if we don't get an updated schematic. If they just send us something on the proverbial paper napkin saying, "Here's what we're going to do, and we need to update these particular parts," and the designer doesn't re-output the IPC netlist or properly call out intentional shorts, it's going to get a stop at the fabricator.

Matties: So, what you're talking about, again, is just sloppy workmanship?

Thompson: That's generally the whole thing, yes. We always talk about the four DFs: DFM, DFA, DFT, and, last, DFX, design for excellence. If you're always keeping those in mind, you're always going to have a decent design, but the design is not everything. It's like Jen said, if you don't have a good kickoff meeting, you may get sent down an avenue that you don't want to be going down.

Matties: But the right questions are based on years of experience, I would think.

Kolar: Yes, they really are, and I think there's the nature of the design process. So, one of the most common sources of error for us is that, from the engineer's perspective, they're working the schematic. They say, "I'm just going to change the value of this part." The problem is they don't actually update the full symbol. They may update just one thing in the schematic such as the value, but they don't actually update the footprint name. They don't update the part number.

Then, that doesn't end up getting propagated into the board where we rely on the footprint name. Then we're guessing, and then they're having to go back to the PCB designer and make sure, "Hey, are you actually using the right package, and, by the way, is that package available?" Or you end up cobbling together a BOM manually based on changes in the schematic, and those aren't properly making it through the design.

Matties: Have you taken a job, looked at the time that you spend start to finish and found that you spend, say, 80% of your time doing detective work and planning, and about 10% of the time designing the layout; then maybe 10% of the time wondering what the hell just happened?

Kolar: I have not done the actual analysis on that. But it really varies by customer and by type of project, and some projects are very complicated, and you're going to necessarily need to do a fair amount of upfront planning. Others are straightforward, and some things can be figured out along the way, but I would definitely say, without fail, if you start doing placement, especially routing before the schematic is pretty locked, or at least before the schematic's fairly mature and mechanical is locked, then you're guaranteed to have trouble.

It's a matter of knowing which customer you need to factor in that you're going to have poor information. We used to joke that we should put a statement on the wall that says, "As an hourly company, we benefit from your inefficiencies."

We used to joke that we should put a statement on the wall that says, "As an hourly company, we benefit from your inefficiencies."

Matties: Happy, at HP, did the same thing.

Happy Holden: Yes, the VP called me in one time because the project managers were complaining about what I was estimating in terms of time, and what I was charging them, and I just showed him the data. I said, "Look at this particular project engineer or project manager and how many times he switched components on me; even when we thought we were done, he changed the schematic. Now, look at this project manager who does his simulation, and gets his ducks in a row. Everybody knows their jobs. They supply us, and it doesn't conflict. We race right through it, so I always give him a discount because I'm giving him the money back that I charge these other project managers as a surcharge." You don't just throw things over the wall, because getting additional costs to re-spin hurts our time to market.

Kolar: Or very carefully manage who you pair with a customer. We know our customers, and we know our employees, so I know that if I have a particularly squirrelly customer, I need to put them with an engineer or designer who's really good at managing their customer and controlling that squirrel, versus putting them with a designer who just follows them and says, "Okay. You want to do this? Okay." Because three months later, they're still going to be working on that 20-hour board.

Matties: What's the greatest obstacle from getting this to a smooth process, if you will? What is that remaining hurdle we have to get over to make this shift happen?

Kolar: It's agreeing about what quality is, and that it's not just connecting the dots. The board should look pretty. Everything should be aligned. Everything should be easy to review, and I think, "Will it work just as well if all your caps are slightly off?" Maybe, but how do you know that the person who was sloppy doing that layout wasn't sloppy with something else that actually matters? It's stepping back and defining, "What is a finished product? What is quality?"

Part of it can be helped with checklists, and we do that. "Hey, here are our input guidelines, and here's the input we need from the customer. Here's a review checklist you go through before you go to the fabricator. Here's a form you can send to your fabricator plus a stackup." You can have best practices, but, ultimately, you need people to care about that quality and be proud of what they do.

Matties: To your point, it speaks to whether you're sloppy, if you care, and what's important.

Kolar: Sure. We always have constraints, so is cost our top constraint? Is it time? Where is that constraint? It's important to be very upfront: I know I really need three months for this design, but you're telling me I have two, so we're making compromises, or we're going to be sacrificing somewhere and acknowledging, "Okay, just upfront, this is getting sacrificed. This isn't." And we document that so that the next time you pick it back up, you know that the reason this whole section looks like crap is it wasn't very critical.

We had to do this recently with a project where we were working over the holiday to get it out. We were very transparent, saying, "Okay, this is not going to be pretty. Bring us the most important parts of your project. That's what we're putting the most time on."

Matties: When they come to you with the constraints, do you go back and, especially if it's a time constraint, clarify your expectations of them?

Kolar: Yes, you have to, and we did that a lot on this particular customer. We said, "Here's our expectation of response times. You're specifying this. You don't get to make changes after this certain date. You have these reviews lined up." If you're on a time schedule like that, then everybody has to be on board.

Matties: Is that a good practice to do as a matter of routine for every customer? Define what their role is and what you expect of them, as much as what they expect from you.

Kolar: I think that's really important, and the experienced, front-end designers know that. There are some customers that I just look forward to working with. I know I'm going to get a wonderful package that's completely clear. The netlist is going to come in right the first time. Then there are customers who I guarantee I'm going to be iterating with 10 times before I can even get a proper netlist, much less to get started.

Matties: How will the industry overcome the most critical component with what you do: experience? When are the young designers going to plug in, and how are they going to manage when what you're talking about is experience based, and a lot of it, I'm sure, is tribal knowledge?

Kolar: I think there's experience, but there's also best practice, and ideally they are learning with the help of those guidelines and best practices. We have people with decades of experience in our company, and we have a whole process where we mentor people and work together as a team to bring them up, but it is a lot harder for a designer in a one- or two-person shop to get that skill. We bring in somebody who's been at a large company, and might be institutionalized in doing things in one way on one certain type of project for a very long time. They may have a hard time catching up with current technology as they start shifting, but I think good management is part of it.

Matties: So, one of the first things is to define the process, or at least go back, revisit the process that you have and make sure it's still valid, because, oftentimes, a process that's written doesn't match reality.

Kolar: Right, and where does this process tend to fall down? If I'm always trying to get this data, but I know that the engineer almost never has it, what could I work with instead that would be easier to get? If I'm in Altium, I'd love to get a STEP file. If I'm in Cadence, I want a DXF. We work with medical device companies where it's absolutely critical that not a dot changes on a note without approval because they're FDA-approved designs.

Matties: Mark, what are your thoughts here?

Thompson: I can confirm what Jen's talking about. We talked about incoming problems.

The other side of the coin are the problems that are created by the board designer, for instance. So, if I can get all the questions that I have on the design answered up front in that kickoff meeting, so much the better. If I can't, and I have to keep going back and forth, the customer might start saying, "I don't know if we've got the right designer on this project."

Matties: Right, because they would expect you to know all your questions up front.

Thompson: Another thing that Jen touched on is a situation when you're talking to a customer, and they only give you a limited amount of information. What is it that I can work on, on that project, to keep going? That's something that I have been conditioning myself to be doing. If I'm stalled on one aspect, it doesn't mean I'm done with the job. It means I can work on drawings or prepare for outputs. There's a whole plethora of stuff that I can do to keep that job going.

Matties: One of the things I always hear is that designers don't understand the total manufacturing process as well as they should.

Kolar: That is very true. One of the most common things we find with new designers is that they may do a good job with the copper and design, but they design something that's absolutely impossible to manufacture. It's a really common mistake of new designers.

Matties: But that goes to not having all the data. If you're going to a manufacturing facility, you have to ask the right questions, and if you don't know what you don't know, you have to go ask somebody who does know what you should know.

Kolar: Right, and then everything is going to vary depending on the process. This really comes into play now with rigid-flex boards, how they are manufactured, and whether it's a milling process or a routing process. Depending on how they're being done, you have very different requirements for your layout. Some of it is people knowing that just because this worked at this one shop doesn't mean it's going to work here.

Matties: Your data input supply line is from multiple sources. You have your parts vendor. You mentioned footprints. There's a data input. You have your fabricator. You have your assembler. You have all the testing data input and you have to just, as you say, check the boxes to make sure you've covered each and every input.

Kolar: One of the greatest "garbage in, garbage out" sources is footprints, and typically companies have their most junior people doing libraries because they're boring. "Oops, I didn't put in the clearances. I didn't recognize how to read the datasheet properly, that it was from top-down view vs. bottom-up view." That's a huge source of error.

Matties: Yes, and much too late at that point. Is there any way to validate that prior?

Kolar: Having good checks. Having a two- or three-part check, especially for critical parts that are hard to rework, or for any of your connectors, for any of your BGAs, QFNs, things like that. You need to really make sure that you've gone in and both checked against the schematic symbol, and then gone and checked against the datasheet, making sure you have the fully designated part number.

Andy Shaughnessy: In one of our surveys, a few designers said things like, "Changes in schematic are not communicated to the entire team," and that the board is about to go out the door. Then they realize the schematic has been changed.

Kolar: Especially on big projects where there are multiple people working on just one section of the schematic, that's a super common problem. This is, again, usually with more complex projects: "Oh, mechanical made a change to the enclosure, and didn't bother to tell the engineer, and, by the way, they can't use one of the core components they planned on using."

Matties: Where's the accountability factor there? This must cost companies millions and millions of dollars.

Kolar: We're not trying to cover our butts completely, but it's that communication. A common issue in service bureaus is that you have a front-end engineer and designer working together, neither of whom are really paying attention to the hours being spent. The front-end engineer just wants their perfect project, and the designer is trying to make him happy. Nowhere along the way do they go back to management and say, "By the way, I need an extension, or I need approval to spend two more weeks on this."

A common issue in service bureaus is that you have a front-end engineer and designer working together, neither of whom are really paying attention to the hours being spent.

Matties: It sounds like one of the most common mistakes occurs when a change is made in a large team, and it's not communicated throughout the team; it's not that that person didn't care necessarily, but they may just have a broken process within their own facility.

Kolar: Exactly. Everybody's busy and stretched on multiple projects, so, I think part of it is just figuring out how do we get people to slow down? That's the fine line and the irony of slowing down enough so you can actually be faster.

Matties: But what if you're reaching into a large corporation with multiple layers and trying to ask them to do something that's not coming from their management, for example?

Kolar: Part of it, from our end at the service bureau, is learning the styles of each of those customers, knowing what to expect and learning how we manage and work within that. Some of them, just flat out, know they're going to be inefficient, and they're going to ask us to re-output the same package 50 times, and that's just how it is.

Matties: That's great, but for new customers, if you don't have the luxury of that intelligence, it's just trial and error?

Kolar: You figure out pretty quickly after the first meeting or two. You figure out really, really quickly how well the engineer knows their processes—or not.

Matties: What are some of the indicators in that first meeting that you would look for to tell you that there's a flag going up?

Kolar: Some of it can be how much the front-end engineer is trying to specify to the designer to do their job, and they're specifying in a way that makes no sense. So, they've read something in a book, and they have this great idea. Sometimes, it can be realizing that they have a lot of theory, but they don't have a lot of practical knowledge. "Oh, is this the first time you've actually done a board with a BGA? What are your timing requirements?" And if they don't know the answer...

Matties: Those are red flags.

Kolar: Yes, or they can have such limited information that you're not getting all the pieces. You're not getting the mechanical requirements. You're not getting things and then they say, "Oh, I don't know. Just make it a board. I don't know."

Matties: When you go into a meeting and you hear somebody who raises these flags, or you walk away with those concerns, what do you do about that?

Kolar: On our end, we usually try to end up putting more effort into getting the right information out, so, we essentially do part of their job for them.

Matties: You question everything, I would think. You're like a detective, basically.

Kolar: Yes, we question everything. "Hey, who else can we talk to on your team? Who else can we bring in that might have this knowledge?" We try to approach it in a way that doesn't make them look bad. We say something like, "Hey, where can we get you more support or who else on your team might have this knowledge?" We're going to be making some assumptions, documenting those, helping them reach out to the fabricator. Essentially, we're filling in the gaps for them.

Matties: Happy, did you have any thoughts here?

Holden: I've been listening, and thinking, boy, don't we live in an exciting time in an interesting industry in which complexity keeps multiplying on us? But we need to get some manufacturing curriculum at the university or community college level so that electrical engineering students get at least a little bit of introduction to the world of manufacturing. If you don't understand some of the basics of manufacturing, those people who do will shoot past you. Everybody wants to move up, but the education is not going to start after you have your diploma.

Kolar: Very true, yes. Putting more of that application into the theory, into the programs.

Matties: Mark, did you have final thoughts you want to share?

Thompson: I just wanted to recap: Ask the right questions. Sometimes it requires extracting that information from the customer. Sometimes it requires leading the customer in the right direction. And know your tools. If you know your tools and set up the rules correctly, you'll be able to build it.

Kolar: Time and communication. Which steps have to be done first? Which ones can you do in parallel, and just how do you document that communication and change?

Matties: This has been very, very useful. We greatly appreciate your expertise and your time and willingness to share with us. It's so appreciated.

Kolar: You bet. It's fun to have these conversations.

Matties: Well, we keep trying to be a voice of change and help companies improve, and you guys are a big part of that, so we appreciate it.

Thompson: Thank you, Barry. DESIGN007

Dampening Plane Resonance with Termination

Beyond Design

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

Today's high-speed multilayer PCBs have multiple planes. The ground planes are used for shielding and to provide return current continuity. Whereas, closely coupled power/ground plane pairs provide low inductance power to the ICs and reduce the AC impedance and plane resonance of the power distribution network (PDN). However, these parallel-plate waveguides (plane pairs) also form a radial transmission line. That is, they form a transmission line that propagates electromagnetic (EM) energy within a plane cavity emanating from a feed point, within the plane, outward in all directions. And, like all transmission lines, it will reflect if not terminated.

The parallel-plate waveguide gets excited by currents flowing through the power/ground plane cavity which can lead to simultaneous switching noise. The current flowing through vias, connecting signal traces, can also cause a similar excitation at harmonic frequencies. In this month's column, I will look at how to dampen plane resonance, radiating from the fringing fields of the board, with RC termination to match the plane's characteristic impedance.

The first line of defense against power supply transients is filtering, bypassing and decoupling. However, in the critical 100 MHz-1 GHz band, the effectiveness of a typical decoupling capacitor is determined almost entirely by its series inductance. This is the frequency band now being used increasingly by digital logic. These strategies lower the AC impedance but do not suppress PDN transients in a system where a radial transmission line exists between the planes.

A termination that matches the transmission line's characteristic impedance reduces reflections. But, in the case of parallel-plate resonance, termination also dampens the standing wave of propagating EM energy that can be built-up within the cavity and radiate from the edge fringing fields (Figure 1). EM energy can be dissipated or absorbed by terminating the edges of the PCB, in its characteristic impedance.

As frequency and edge rates continue to increase, the impact of intrinsic electrical characteristics become more pronounced. AC switching currents in the power/ground planes can be very large. Under these circumstances,

Figure 1: A standing wave built up by plane cavity resonance.

a plane pair acts more like a radial transmission line rather than a planar capacitor.

A region under a large BGA densely populated with vias also appears as a discontinuity due to the large array of anti-pads eating a hole in the plane. A discontinuity reflects propagating energy because it represents a mismatch with the characteristic impedance of the transmission line. The edges of the board cause the greatest amount of reflection since an edge is a totally abrupt (open circuit) surrounding the board. Reflected energy is accompanied by phase reversals in its components, and combined reflections from the open circuit at the edge of the board can cause a phenomenon known as voltage doubling creating a standing wave.

Parallel planes in multilayer PCBs exhibit multiple resonances, which increase the impedance and the EM radiation. A typical FR-4 laminate, of 4-mil thickness, produces a characteristic impedance of about 3 to 5Ω for adjacent planes. The larger the plane area, the lower the impedance. This is a good reason to make planes as large as possible.

How Do We Dampen the Plane Resonance?

Resistive termination along the board edges reduces the resonance peaks. But in practice, this means approximating a continuous structure with resistors spaced around the perimeter. Obviously, multiple low-value resistors cannot be placed directly between the power supply and ground as it would needlessly dissipate a huge amount of DC power. To prevent this, they should be AC coupled with a ceramic capacitor of sufficient capacitance to allow the resulting impedance to appear predominately resistive at and above the lowest frequency of interest. A 10nF, X7R ceramic capacitor is a typical value. The addition of loss to dampen modal resonances is more important than the exact termination value and distribution. Notice how the 20 RC terminations (blue) dampen the plane resonance pushing the effective Zpdn down below the resonance, above 300MHz in Figure 2 (right).

A standing wave can be generated at switching locations within the interior of the PCB. Therefore, there may be hot spots within that cavity that would benefit from the placement of an additional AC-coupled load at or near the source.

Reflections occur in a transmission line only when there is a discontinuity. If the edge of the board is terminated, in the characteristic impedance of the radial transmission line, then there will be no reflections. This will not

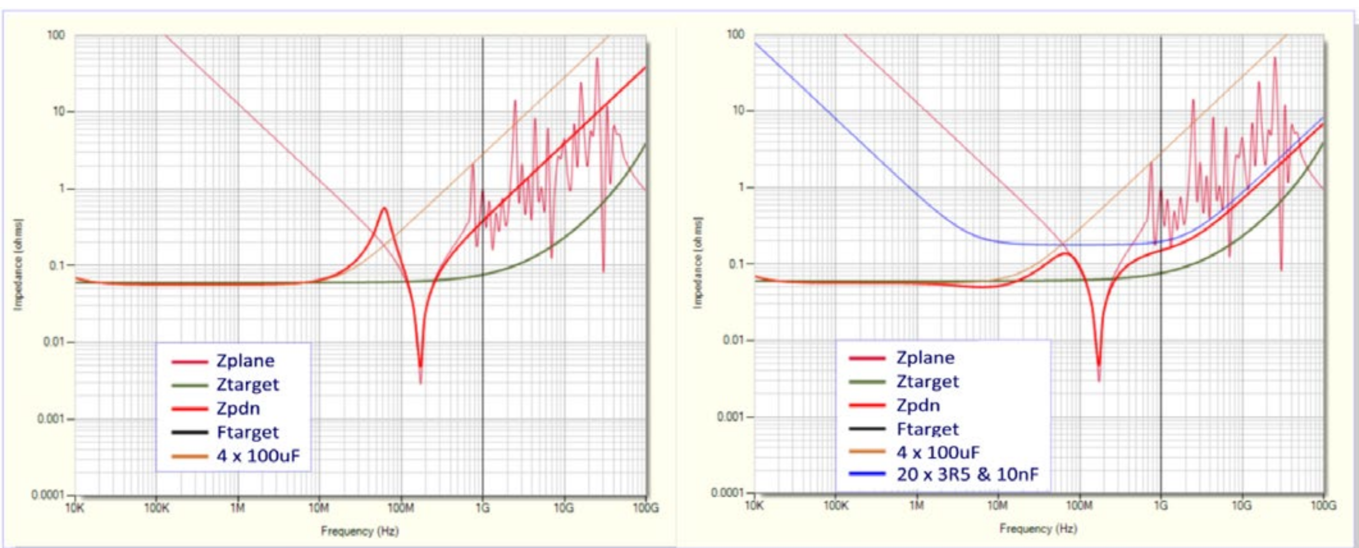


Figure 2: Zpdn with 4x100uF (left), Zpdn with added RC damping (right). (Source: iCD PDN Planner)

eliminate the initial transient but does, however, prevent it from being compounded. Edge effects can be particularly problematic since it is the board edges that are in such close proximity to the chassis and hence the radiating fields can induce currents into the chassis frame. Termination elements inhibit standing waves from developing between the planes and protect peripheral signals and electronics from radiation.

Where Do the Adjacent RC Terminations Need to be Placed?

That highest effective frequency of concern is typically the fifth harmonic of the fundamental frequency, corresponding to the rise time. Assuming that the fastest rise time is one nanosecond, then we are concerned with frequencies up to 5 GHz. The wavelength, in FR-4 at 5 GHz, is about 1.1 inches. So, in this case, AC-coupled terminations spaced one inch apart is a good approximation of a continuous termination. At 2.5 GHz bandwidth (500 MHz clock), two-inch spacing would suffice. The resistor, capacitor and power/ground vias should be placed closely together with wide connecting traces to minimize inductance as in Figure 3. The added losses from the RC terminations are helpful in the overall response of the PDN as they dampen impedance peaks arising from connecting the devices to the board.

The optimization of the PDN is a trial-and-error process that needs to be done in conjunction with the stackup configuration, dielectric materials, decoupling and AC-coupled terminations to fully exploit all avenues. Adding the terminations to the periphery of the board will result in new anti-resonance peaks that need to be dampened. The iCD PDN Planner shows the impact of RC terminations on the cavity resonance, allowing the designer to level out the peaks with a selection of decaps. Suppressing the plane resonance peaks to provide a low impedance profile at higher frequencies helps to minimize electromagnetic emissions.

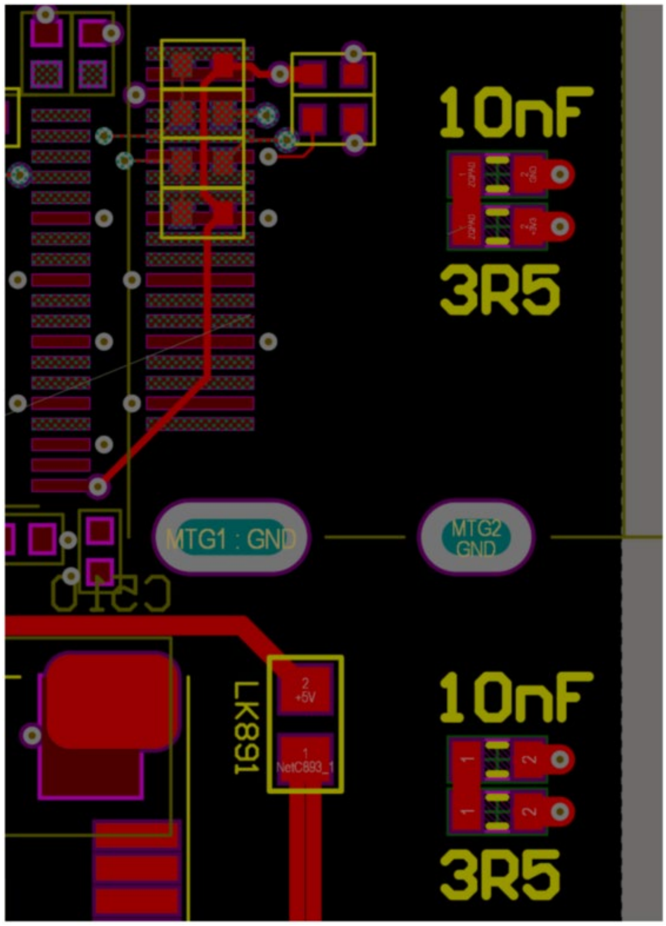


Figure 3: RC terminations (3R5 and 10nF) are spaced around the periphery of the PCB.

Key Points:

- Parallel-plate waveguides (plane pairs) form a radial transmission line that propagates electromagnetic (EM) energy within a plane cavity. The parallel-plate waveguide also gets excited by currents flowing through the power/ground plane cavity.
- Decoupling does not suppress PDN transients in a system where a radial transmission line exists.
- EM energy can be dissipated or absorbed by terminating the edges of the PCB in its characteristic impedance.
- A region under a large BGA densely populated with vias also appears as a discontinuity due to the large array of anti-pads eating a hole in the plane.

- The edges of the board cause the greatest amount of reflection since an edge is a totally abrupt open circuit.
- Reflected energy is accompanied by phase reversals in its components, and combined reflections from the open circuit at the edge of the board can cause a phenomenon known as voltage doubling creating a standing wave.
- The larger the plane area, the lower the impedance.
- RC terminations of 3R5, in series with 10nF spaced along the board edges, are generally sufficient to reduce the resonance peaks.
- The addition of loss to dampen modal resonances is more important than the exact termination value and distribution.
- The added losses from the RC terminations are helpful in the overall response of the PDN as they dampen impedance peaks arising from connecting the devices to the board. **DESIGN007**

Resources

1. Beyond Design: PDN—Decoupling Capacitor Placement, Learning the Curve, Fringing Fields, by Barry Olney, January 2016.
2. Optimized Power Delivery Performance Using Plane Terminations by Istvan Novak, Samtec.
3. US5708400A—AC coupled termination of a printed circuit board power plane in its characteristic impedance—Google Patents, Terrel Morris, Hewlett Packard.

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.

New I-007eBook Highlights SMT Inspection: Today, Tomorrow, and Beyond

Increasing board complexity is driving the need for accurate and speedy inspection systems.

In *The Printed Circuit Assembler's Guide to SMT Inspection: Today, Tomorrow, and Beyond,* author Brent Fischthal takes readers through a brief history of SMT inspection before discussing the benefits of data-driven analytics and how intelligent software solutions can help companies analyze and optimize the production process.

In this latest title from I-007eBooks, readers will learn how artificial intelligence has demonstrated promising potential in this field and has far-reaching applications within the manufacturing sector.

According to Michael Ford, Senior Director of Emerging Industry Strategy for Aegis Software, "This book provides unprecedented visibility of SMT processes, asserting inspection technology as a key active contributor to zero-defect quality initiatives, rather than being limited to simple defect detection."

Download your free copy today!

Detecting PCB Design Teams' Experience Level

Feature Article by Dana Korf

Beauty lies in the eye of the beholder. Well, what is a perfect "beautiful" PCB design? Is it defined as the data that is released per the aggressive schedule? Is it defined as data received by the fabricator as one that does not require any engineering questions or manufacturability and design edits?

I do not believe that myself or my teams have ever received a design to fabricate that was intentionally designed to fail, except for test-to-fail capability test vehicles. But when reviewing incoming data packages, we have often wondered, "What was that person thinking?" When the data is initially reviewed, the guessing game begins.

As Lt. Columbo, in his TV detective series of the same name, famously said, "Oh, there's just one more thing..." A few thoughts to consider:

- How many suggestions does the designer want to hear? Some don't want any corrections to the data. Others want all the feedback possible to improve the data.
- Asian cultures tend to treat their customers with high regard and tend to not provide much negative feedback. They will just edit the data to make it manufacturable or to meet the lead time.
- The amount and type of data issues can often indicate how much experience the design team has. A less experienced designer may not know a solution to the DFM feedback issue that has been posed. They may give an answer that inadvertently and negatively affects the design performance.

Part of a fabricator's job involves playing detective to assess each designer's skill. So, let's put on our Columbo trench coats and review a few ways that a design team's experience level can be gauged.

Violating the Laws of Physics

This is a common issue with initial stack-ups with impedance constraints. The fabricator will model the incoming impedance value and tolerance and discover that the answer

is 50% away from requested nominal value. Inexperienced designers, or SI engineers, will have good impedance simulation tools but are loaded with bad material specifications. Or the model assumes that there was a reference plane for a microstrip line that was one layer away when the actual design has sections of the trace referencing a plane three or more layers away. Or the trace width needs to be reduced so far that it puts the required width outside of the fabricator's capability.

One design was received that required the trace impedance to match the free space impedance of 377 ohms. We notified the customer that all PCBs are made with dielectric materials of which none had a dielectric constant that matched a vacuum. We could not get a reason for the specification, so this was a no-bid.

5G IoT designs have increased the quantity of first-time flex and rigid-flex designs. Rigid board designers who attempt their first flex or rigid-flex design often violate flexure rules, which will result in damaged boards during use. Many of the flex rules are different than those for rigid boards and must be learned.

5G IoT designs have increased the quantity of first-time flex and rigid-flex designs.

Regional Quick-turn vs. Asian High-Volume Capability

Inexperienced designers often have only used local quick-turn production companies to get their boards back to engineering quickly, for example, in one to three days. The quick-turn shop does not have sufficient time to run an extensive DFM review and may change materials and design features without notifying the designer of these changes. They may also have a wider process capability to ensure that boards do not scrap out. A significant amount of DFM issues may be raised when the design is transferred to a volume shop because of narrower technical capability, tighter quality requirements, and small modifications that will significantly improve the yield.

Common yield enhancements will not be included. These include edits such as non-functional pad removal, thieving open areas, filling low-pressure areas, and tear-dropping pads. Experienced designers will generally modify their design rules to match the volume capability from prior builds.

'Don't Change Anything'

Instructions included with a design may indicate that nothing can be changed at all. All requests for manufacturability improvements and design issues fixes are rejected. Much of the time, this response is because the layout person and their team does not fully understand how the changes may impact the design performance.

A new engineer is commonly assigned the task of adding additional sources prior to or just after the product is released for volume manufacturing. Manufacturing facilities often have differing capabilities and design rules, and may request slight modifications to enhance the yield and/or meet product performance requirements. An experienced engineer will have a good understanding of what variations may affect the design and what will not. The inexperienced engineer will generally require that nothing be changed.

'Build It Exactly the Same as Our Current Supplier'

This is a quite common guidance from companies that design mobile phones. The cellphone NPI cycle is measured in weeks and the product lifetime is measured in months. Some of the PCB sections are copied from other designs and merged without spending a lot of time in design verification. They will send the design to multiple companies and the PCB that is provided goes into the design validation, tuning, and qualification stage. The debugged design then gets sent to the other potential sources who are told to build it exactly the same, even if it does not fit well into their manufacturing process, because they just know it works but not necessarily why.

Nonsensical Requirements

Once, a data package was brought to me for review, complete with documentation requiring a 5-ohm impedance specification on a signal trace. I looked at the salesperson and asked if the designer was under 22 years old. He said, "Yes, how did you know?" We sat down with the engineer and I asked why he specified 5 ohms. The response was that the simulation program results yielded that answer to make the circuit work. After we reviewed the model setup, we discovered that the simulation model had been set up wrong. The young engineer didn't realize that the answer didn't make sense.

Mechanical tolerances used in sheet metal production will often be called out on the fabrication print. Examples are +0/-1-mil tolerance for hole diameters of ±1 mil for locational tolerances. Sheet metal does not expand or contract during production like polymers used for PCBs. This is generally found when an inexperienced mechanical designer is involved in their initial PCB layout.

Violating Company Specifications

New companies or third-party design companies may not have a detailed fabrication acceptance specification. They will just reference IPC specifications. More experienced companies may have a detailed acceptance specification that has been updated as fabrication and design issues were encountered over time. Many new designers will not read or understand their internal acceptance specification and provide designs that violate these documents.

Larger PCB fabrication companies have regionally located application engineers who can assist new and experienced designers in selecting proper design rules and materials. It is also common for 25–50% of a fabricator's front-end engineering capacity to be used to perform pre-design and incoming design DFM reviews as well as required adjustments.

New companies or third-party design companies may not have a detailed fabrication acceptance specification.

Conclusion

The industry needs to change the NPI process to build designs as they are provided instead of assuming that every design needs to be reviewed and modified. This will happen when layout software tools can incorporate all the design rules that front-end engineers use, fabricators provide these rules to their customers, and the industry completes its conversion to using intelligent data, like IPC-2581, instead of non-intelligent Gerber-based design packages.

Education can also play a big role here. There are a variety of DFM courses available which can provide good information on how designers can alleviate or eliminate these kinds of mistakes. One silver lining of the past year: Many of the PCB design classes that were once only available at conferences are now online.

But in the meantime, don't be surprised if your fabricator's CAM engineer tells you, "Oh, there's just one more thing..." DESIGN007

Dana Korf is the principal consultant with Korf Consultancy.

Turning 'Garbage In, Garbage Out' into 'Good In, Good Out'

Feature Article by Tamara Jovanovic
HAPPIEST BABY

We have all been there: updating our PCB design last minute before releasing manufacturing files to a fabrication house. This can be a particularly daunting task on its own and even more so if you are limited by time. While you are working on the design and making sure that you did not make any mistakes in the schematic or layout, you must also be aware of all the parts and information that you are working with.

It is probably best to create your own parts and footprints and that way keep track of the information you are putting in your system and what your outputs will be. However, that is not always the easiest task, especially if you are using complicated parts or if you simply do not have enough time to create every part from scratch. Sometimes, it is easier and faster to find a part online that already has all the information you need. If you had a system of where you store your part details, you must make sure to go back and verify that the specs for this new part from an external library do not have important details stored in different locations. Additionally, you need to make sure that the part itself is indeed the one you are looking for, as mistakes happen and sometimes the part you need is linked to wrong design files.

In this part of the design cycle, it is so easy to unintentionally introduce "garbage" into your system. Unless you have time to extensively check everything you bring in from an external source, it is very likely that something will not match up with your design data. In the end, this means you'll have to put more work into your design and basically reverse-engineer a part that was supposed to save you time and effort.

Introducing bad data is never intentional. No designer or engineer purposely puts files into their system that they know will haunt them somewhere down the road. We all assume that if the basics are covered, we can fix a couple of errors manually, or go back later and fix any remaining issues while the boards are being fabricated.

But life gets in the way, and other tasks and issues may arise in your job; you might not have the time to go back and thoroughly check everything right away, or you simply forget.

Then, a few months later, you must go through the same process again, which takes away valuable time, and you probably find yourself thinking, "I should have just done it right from the beginning." There comes a time when correcting previous mistakes becomes such a long, complicated task, and the best way to deal with this is to eliminate it from the start. A little bit of extra work in the beginning of the design process saves a lot of time and effort in the long run.

A little bit of extra work in the beginning of the design process saves a lot of time and effort in the long run.

I acquired a big part of my knowledge about keeping bad data out of my design cycle by trial and error. This can be a painful process with a steep learning curve. However, I think it is important to make those mistakes early on, and this is the best way to learn. Experience for yourself what bad data can do to your design and I can guarantee that you will do everything you can to never make the same mistake again.

Engineers know what is needed in an output package when it is being sent for fabrication, whether it is for a printed circuit board, a mechanical structure, or a code. Introducing garbage into your system from unverified external sources will unequivocally result in having garbage in your outputs. Therefore, it is most reliable to create parts on your own. That way, you control exactly what you are putting into your system and what is coming out of it. You can create and follow a checklist that contains details about part information stored on each layer. It might be time-consuming in the beginning, but when you are finished with your design and all you do to be done is click Generate Outputs to get exactly what you are looking for, it will be worth all the initial effort.

Another process you can follow to make sure that bad data does not make it into your documentation is to work with templates. This also requires a little bit of work initially, but it makes every subsequent design easier and faster. Create a template for your OutJob file. Prior to making this template, communicate with your fabrication house and contract manufacturer about their capabilities and what kind of information they require to make the boards properly. It sounds trivial but having this template will ensure you always export exactly what your manufacturer needs, and more importantly, it will guarantee that you will get your design made correctly. We are all human—sometimes we forget to include important information even if we have done it properly hundreds of times before. There is way too much information to keep track of on your own, all the time, and having this sort of system in place will help with every design down the road.

These two practices have recently had a tremendously positive impact on how quickly and accurately I can deliver manufacturing documentation for a design. It sounds simple: Have a system and stick to it. In design, the system is everything and the accuracy of your work will depend on how seriously you take this task and stick to the system. Use all the resources available to you to come up with the best method that works for you, because it is no fun dealing with issues in your output documentation after putting so many hours into the design itself.

If you do end up implementing a system that works for you, instead of "garbage in, garbage out," a better GIGO acronym would be "good in, good out." **DESIGN007**

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


Need help finding the right people for your team?
DB Management knows that finding the right people can be like looking for a needle in a haystack. We can streamline your hiring process with pre-qualified candidates.
Engineering · Management · Sales & Marketing
db MANAGEMENT GROUP
Click here to get started
207-649-0879
danbbeaulieu@aol.com

Mysteries of Incoherent Information

Tim's Takeaways

Feature Column by Tim Haag, FIRST PAGE SAGE

To keep ourselves entertained during the lockdowns of the pandemic, my wife and I have been watching some older detective shows that we never fully explored before. Character-driven shows like *Monk* and *Psych* are not only a lot of fun to watch, but they often present some great mysteries for the viewer to participate in. I have always loved a good detective story and have read many cases from the pages of the Hardy Boys to Sherlock Holmes. Thankfully, this isn't the waste of time that it may sound like; it has actually been good training for one of the most difficult challenges in life—raising children.

These scenarios may sound familiar: "Where are my shoes?" "I don't remember bringing home my homework." "No, I don't know how the lamp got broken." And my personal favorite, "I don't know how my brother got a black eye; my hand must have slipped." Mysteries like these take a lot of sleuthing skills, even if we didn't successfully solve every puzzle. Case in point: Once when moving an old, large TV away from the wall we found a nauseating pile of degrading bio-material behind it. Ewwwww. We had no idea what this mess was or why it was there until years later when our adult children finally came clean about it. When my wife would give them their vitamins in the morning, they would wait until her back was turned, pull the pills out of their mouths, and then chuck them behind the TV.

Even though we failed to solve the mystery of the mucilaginous, malodorous mound, our detective training did help us solve many other riddles and enigmas that are common when raising children. I have also found that this art of deduction has served me well in the work

world, especially in the design of printed circuit boards. There were so many times where the inaccuracies of the data that I was given to work with caused problems, that I eventually learned to double check and ask questions of everything first. Here are some of the problems that I've encountered.

Design Database

When I first started laying out boards many years ago, some designers found the automated features of the tools too restrictive and would disable them. When one of these designs later required an ECO, it was a nightmare to work on because it had been routed on a 1-mil grid without any online DRCs. After having to completely re-route a few of these designs to make a simple change, I learned to work with these designers before they started to ensure that database was set up correctly.

When I first started laying out boards many years ago, some designers found the automated features of the tools too restrictive and would disable them.

Another lesson I learned was to make sure that the design rules were correctly set up before I started working on an existing design. Many designers wouldn't take the time to set up their constraints and would alter trace widths manually for nets such as power and ground. If you weren't paying attention to what you were editing, those traces would snap back to their default skinny width, and the DRCs would be none the wiser.

I also learned to look at those designs for areas of metal that were altered using simple graphical shapes instead of intelligent copper pours. Although those planes and thermal pads looked okay, there were often vias or pins going through them, causing a short to a different plane on another layer that the DRCs wouldn't catch.

Design Libraries

Laying out boards at a service bureau back in the day often meant re-creating library parts for each new customer. In some cases, the designers would cut corners and use the incorrect method for creating a pad or shape in their footprint that either wouldn't photoplot well, or it would register a DRC incorrectly. Checking the integrity of new library parts became part of my regular routine.

On the flip side, it also paid to make sure that a designer was using the correct part rather than merely copying an existing footprint without checking it first. We often found problems with polarity or pin-outs that could have resulted in an unexpected re-design of the board and an angry customer.

I also learned to keep an eye out for the odd kind of library mistakes, like a surface mount pin being created with a through-hole drill attribute. Although this would eventually light up a design's DRCs like a Christmas tree, it would still take time to correct and was always better to catch it before it got into layout.

Design Information

A good PCB layout depends on getting complete and correct data to work with. In the service bureau we would work from a netlist that the customer gave us, and we got used to having to add bypass caps and test points, as well as to double check net names. Thankfully, with the integrated design systems we work with now, you typically don't see these kinds of problems anymore.

I also learned to check the components in the netlist against the parts listed in the bill of materials. It was amazing how many times this didn't match, and we had to go back to the customer to get clarification. Many of these designers were working with a standard set of parts in their schematic capture tools, and as long as they had the same number of pins, they wouldn't worry if the pinouts were the same.

In general, I learned to spend time with my customer to find out exactly what it was that they were expecting with the design. Some customers were new to circuit board design and didn't understand the process. Sadly, there were also some that just didn't care, and once they "threw their design over the wall" to us, we were on our own to make it all work.

As it ultimately turns out, what I discovered was the key to success in all of this: finding out what I needed to do to make it all work, and then doing it. Here are some ideas that I've discovered over the years that may be helpful in your electronic sleuthing efforts.

'I Say, Holmes, What Was That?' and Other Clue-Gathering Tips

The worst position that you can be put in when laying out a circuit board is to not understand what your customer is looking for. Your customer may be another company, department, engineer, or the dude that is sitting next to you. No matter who it is, though, you need as much information as possible to complete your work correctly and on time. These ideas may help:

- **Ask questions.** Yes, it can be really tough to admit that you don't know something, but you need to ask questions in order to succeed. Is there information missing from your design database or documentation? Find it before you proceed. Is something not clear in your instructions? Get some answers now before you get stuck.
- **Get it in writing.** I've said this so many times in this column, and here it is again: document, document, document. Not only will answers in writing give you something to refer to other than your memory, but it also gives you something to fall back on in case you are ever challenged about what you have been working on.
- **Cultivate good working relationships.** Many times, miscommunications are due to a lack of relationship and trust. The more that we can build up and strengthen our working relationships, the more likely that our communication will improve.
- **Learn from all your experiences.** Some assignments, designs, and even people, will present the same or similar problems repeatedly. Use these experiences to empower you, not to drag you down. If someone is known for not giving you a complete data package in the past, check the information carefully the next time they hand you a design to keep yourself from being derailed again.
- **Makin' a list and checkin' it thrice.** Checklists are a great thing, especially if you are constantly working with information coming to you from outside sources. A checklist that you can easily refer to can help you to remember to look for even the smallest details that often get overlooked and hurt you in the end. Hey, there's a reason that pilots use a checklist every time they fly an airplane, so there's no reason why you shouldn't use one.
- **Use the full capabilities of the tools, that's what they're there for.** There are so many features in PCB design tools today that can help ensure the complete data integrity of a design—simulators, analyzers, checking, 3D viewing, team design, and much, much more. One way to ensure the accurate transfer of electronic data between design and manufacturing, is the electronic data exchange features built on the IPC-2581 data format. This system allows a manufacturer to electronically send a circuit board stackup configuration

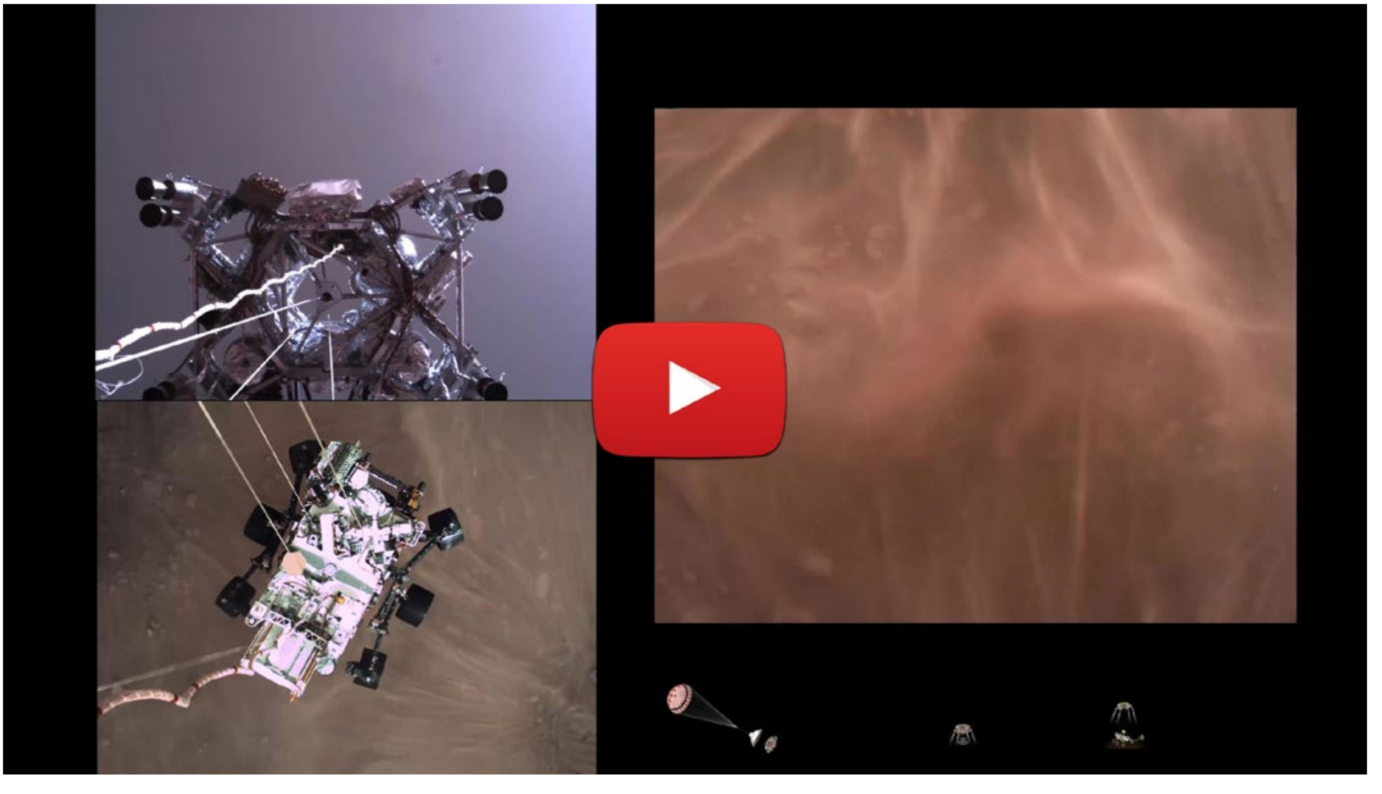
into your design tools, while you can send back the completed design files for manufacturing. No longer do you have to be concerned if all the necessary data has been compiled and transferred, because the tools are automatically doing that for you.

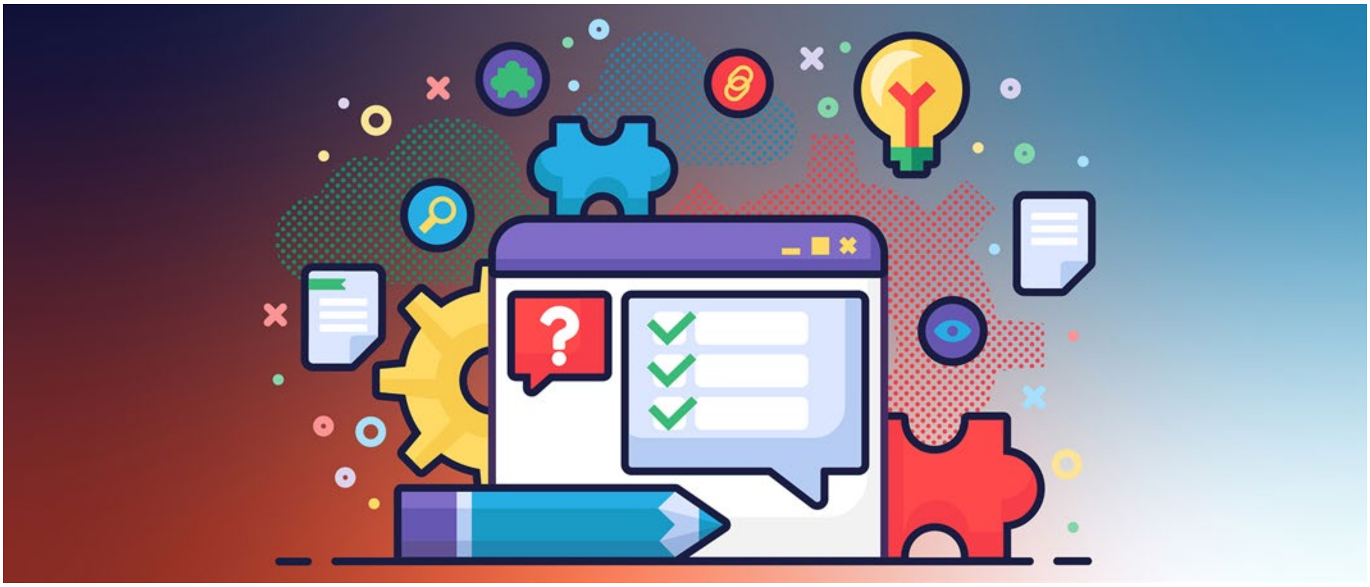
We've been talking about different ways to manage the data supply line to ensure that the data you receive is accurate and ready to be used. But it is perhaps even more important to make sure that you are sending out information and data that is just as pristine. It never hurts to look in the mirror and make sure that the standards we expect from others are the same as what we are applying to ourselves. I know I've discovered that when I keep up my end of the data supply line, those that I work with tend to do the same. Until next time, then, my fellow mystery lovers, detectives, and sleuths... keep on designing. DESIGN007

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

NASA's Mars Perseverance Rover Provides Front-Row Seat to Landing, First Audio Recording of Red Planet

New video from NASA's Mars 2020 Perseverance rover chronicles major milestones during the final minutes of its entry, descent, and landing (EDL) on the Red Planet on Feb. 18 as the spacecraft plummeted, parachuted, and rocketed toward the surface of Mars. A microphone on the rover also has provided the first audio recording of sounds from Mars.





Eliminating 'Garbage In, Garbage Out' With Checks and Balances

Feature Article by Nick Barbin
OPTIMUM DESIGN ASSOCIATES

The proverbial saying "garbage in, garbage out" holds true in the electronic product development world. PCB designers stand squarely in the middle of a busy information intersection flowing with inputs and outputs. Missing or bad information at the beginning of a design project will undoubtedly lead to board re-spins, increased costs, and most importantly, a delayed product release. The same can be said about the PCB designer who doesn't provide a fully checked and comprehensive data package to the downstream manufacturers, i.e., "throwing it over the fence."

Certainly, there is only so much a person can do with wrong or missing information, but to help ensure the success of a design project, it is incumbent upon the PCB designer to have systems in place to manage the flow of information through documented standards and guidelines, effective communication, and a series of checks and balances.

It is important to understand that most electrical engineers are not PCB designers, and many do not know exactly what information is required. Therefore, it is up to the PCB designer to be proactive and to request all the information and specifications needed for the project beyond the obvious items (schematic, mechanical, and BOM). This can be accomplished with a design specification document as we employ at Optimum or just a document checklist of items, such as the items listed below:

- Component datasheets and application notes
- Placement floor plan
- Stack-up, copper weight, material, via spans, etc.
- ICT or flying probe requirements
- Copper constraints that address impedance, timing, topology, current, etc.
- Board nomenclature details/special notes

As with most things in life, effective communication is key to ensuring information is not missed. At Optimum, every design project begins with a kick-off meeting (via Zoom these days), preferably with all the stakeholders present, but at a minimum between the

electrical engineer and PCB designer to go over the details regarding the project. This is a great time, if not already provided, to document items such as from the example list above into a design specification.

Once a design project begins, the PCB designer will inevitably have questions or concerns about something in the input package—an obvious schematic error, mechanical conflict, etc. Although a phone call is great, we have found sometimes engineers would rather communicate via email. We find it better to keep emails short and concise. Lengthy emails with too many questions (more than three) will generally result in some questions going unanswered.

On extra-large design projects that may span many months, there can be literally hundreds of emails, most with very important information from a variety of stakeholders. In these cases, it is very easy, due to a variety of reasons, for an instruction to be missed or forgotten. We find it best to organize these emails by moving them into a Word or Excel document where they can be tracked through a typical color-coding system (red, yellow, and green) to ensure nothing gets missed.

Finally, as part of our ISO process, we rely on a series of checks and balances to ensure we haven't missed any critical information from getting into the design database, as well as to ensure we are not "throwing incomplete data over the fence" to one of our downstream manufacturing friends. First, we insist with our designers that all the mechanical, electrical, and manufacturing constraints/rules are embedded into the design database. I'm always amazed when we receive a customer's database that requires edits, or the previous designer did not enter the routing constraints into the tool. Without doing this, it is virtually impossible to say for certain that all the rules have been met.

Also, as an iterative process and discipline throughout the design project, our designers have a formal checklist of 90+ items that must be reviewed and/or considered at the different phases of the project, such as the following items:

- Pre-layout review with the engineer
- Assembly breakaway rails
- Tooling holes and global fiducials placed
- ICT or flying probe test points placed
- Reference designator renumbering
- Fabrication drawing complete with correct dimension, notes, and stackup
- Valor DFM complete
- Database archived

As if we haven't done enough checks to this point, once we have the final approval from the engineer to output the manufacturing files, we run a series of design for manufacturing (DFM) checks by our separate manufacturing DFM specialists. It is important to note: We believe strongly that this check is to be completely unbiased (not performed by one of our PCB designers) to help ensure the integrity of the final data. With all the money and time at stake with what follows, this final check really allows our team (especially me) to sleep well at night.

In conclusion, likely the most important way to help eliminate "garbage in, garbage out" is to have a detail-oriented, experienced designer at the helm that understands today's electrical and manufacturing technologies. It's not bad or missing data if the PCB designer receiving the information doesn't recognize it as such. Although, some things may be beyond PCB designers' control.

But by standing in that busy information intersection, experienced designers who utilize documented standards, effective and proactive communication, and a system of checks and balances can do their part to reduce or even eliminate bad or missing data. **DESIGN007**

Nick Barbin is the co-founder and president of Optimum Design Associates, a provider of EMS and PCB design services based in Pleasanton, California.

PCB007 Highlights

Testing Todd: Getting 'Lean' in 2021 ▶

Many companies and individuals had to make life-altering adjustments in 2020 because of the pandemic, including reduced hours, telecommuting and examining how we do things in this "new normal." Although the circumstances causing these changes are tragic, it forced us into becoming lean.

North American PCB Industry Sales up 4.5% in December ▶

IPC announced the December 2020 findings from its North American Printed Circuit Board (PCB) Statistical Program. The book-to-bill ratio stands at 1.05.

Catching up With Nano Dimension ▶

Dan Feinberg spoke with Valentin Storz, Nano Dimension's general manager of EMEA and director of marketing, about how the pandemic has affected their business this past year and what they have planned moving forward.

Joe Fjelstad's Book Review: The Innovators ▶

The Innovators: How a Group of Hackers, Geniuses, and Geeks Created the Digital Revolution by Walter Isaacson is the best technology history book I have ever read, and at the same time one of the most engaging and entertaining. It is a forte of Isaacson to write biographies of great people. I have read his other books on DaVinci, Steve Jobs, Ben Franklin and Albert Einstein and found them equally brilliant. Isaacson has a number of other titles I have yet to get to in the future. He is a singularly great storyteller.

IPC's Alicia Balonek Talks Trade Show News ▶

Nolan Johnson discusses IPC APEX EXPO 2021 with Alicia Balonek, senior director of trade shows and events at IPC. Alicia provides an update on how IPC APEX EXPO will be structured in a virtual format, overviews the programming changes, and new additions designed to make IPC APEX EXPO the best virtual event possible.

The Right Approach: Leadership 101—Leadership is Hard ▶

Good leadership always makes a difference; unfortunately, so does bad leadership. Leadership is not easy to learn; if it was, everyone would be a leader. This Leadership 101 series will provide practical leadership tools and principles that can be applied immediately with your team.

Kuprion Introduces ActiveCopper Filled Thermal Vias ▶

Kuprion, Inc., a spinout of Lockheed Martin, has introduced ActiveCopper Filled Thermal Vias, leveraging a patented technology breakthrough that addresses the increased reliability demands of heat and power dissipation for complex, advanced high-performance systems.

Kemmer Praezision Partners with Insulectro ▶

In this interview, Nolan Johnson and Gregor Dutkiewicz discuss Kemmer Praezision, their new working partnership with Insulectro, and some of the recent market challenges in mechanical drill bit technology.

Messages from the Chairmen

The Digital Layout

by Kelly Dack, CIT, CID+, PCEA

Introduction

In this month's column, I interview the PCEA chairman and chairman emeritus, who provide their shared viewpoints on the importance of organizing. And as always, I provide you with a list of upcoming events.

PCEA Updates

This month I am excited to bring you an inspiring message from not one, but two of the PCEA's chairmen.

It is quite possible that many of you do not know that our PCEA board has two chairmen by design. From the beginning, our idea was to preserve the experience from our past organizational associations and use that experience as our compass needle for moving forward.

Stephen Chavez serves as PCEA's chairman. His primary interest is to lead this organization into a future which respects the ideas and efforts—the legacy—of those who have served the electronics industry so well. To fulfill that interest, Steph relies upon his counterpart, Chairman Emeritus Gary Ferrari, to offer his wisdom and experience. Gary's guiding compass needle was magnetized by a lifetime career of serving the electronics industry, as well as bringing together and leading electronics industry professionals.

So, for this month's column, please trek along with us and enjoy this special "Messages from the Chairmen" interview we've transcribed from our Zoom interview.

Messages from the Chairmen

Kelly Dack: Thank you for joining us gentlemen! We have Steph Chavez, the officiating chairman of the PCEA. And we also have Gary Ferrari, who serves as the chairman emeritus for the PCEA. I'd like to start by asking Gary for some perspective on the past and why you feel it's important that PCB designers are connected in some way by an organization.

Gary Ferrari: Way back in 1991, Dieter Bergman and I started the Designers Council. The main

Figure 1: Kelly Dack (center) interviews PCEA Chairman Stephen Chavez, MIT, CID+ (left), and PCEA Chairman Emeritus Gary Ferrari, MIT, CID+.

reason was to respond to the feedback we had from designers. PCB designers had expressed that the engineers have IEEE and other organizations to participate in. They mentioned that all they had were sore eyes from staring at their computer screens working all day long (laughs). They wanted to be represented. They wanted to have opportunities to learn from each other and to learn from the industry. That was the main thing. They inferred, "Everyone else has different kinds of credentials and we don't have any official credentials at all." That is when we started the exams and certification programs. The CID and CID+ were created "by designers, for designers." We had this as our mantra and even a logo. At one time we had T-shirts and other different swag—all kinds of stuff. It grew rapidly.

Dack: Steph, why is it important that designers connect with the other stakeholders? Gary mentioned some of the other stakeholders. What is your take on why it's so important—especially nowadays?

Stephen Chavez: When I think about our profession in the industry, to evolve we've got to collaborate. We've got to get knowledge because design is more than just someone sitting in a cube or behind a computer by themselves alone. It's a total collaboration between engineering, fabrication and assembly—all of the professions. And when we think of design as a team concept, we truly need all the stakeholders involved. With stakeholder collaboration, cross-pollination of the designer, manufacturer, and test engineering ideas meld together to become the holistic makeup of the new hybrid designer. As designers collaborate, not just within their own company ecosystem, but outward toward the industry, PCEA then strives to add synergy within this collaboration. We want to help printed circuit engineering professionals take it to the next level with all the different industry associations. I think that's the next evolution.

Dack: Can either of you share some examples of how the PCEA membership is creating that collaboration? How are they feeding off of and supporting one another?

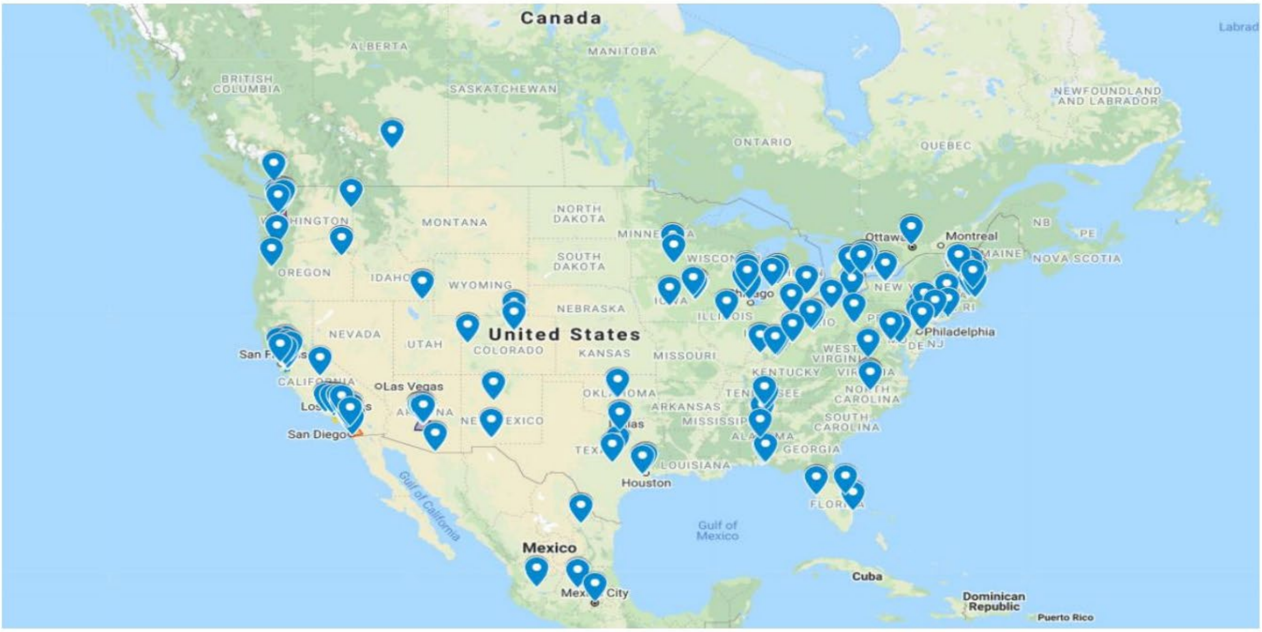
Ferrari: In the past, the chapters were not actually working together. Now, they're meeting together and sharing. One chapter is on one side of the country and there is another on the other side of the country. They are spreading out beyond the chapter itself with their own programs. I think this is outstanding because I remember years ago, some of the chapters in the Designers Council felt isolated and they lost their enthusiasm. It was the same people all the time and there weren't any fresh ideas. However, Steph took a different approach. I'll let Steph describe it, because it's beautiful.

Chavez: Sure. With the pandemic in late 2019 and 2020, the silver lining is the ability to collaborate virtually no matter where you are in the world. So, while our physical footprints are smaller, our virtual footprints are that much larger. We can effectively collaborate with each other across our platform. Our PCEA chapters are doing this and we see the chapters growing astronomically. Our multi-chapter events had never happened before Zoom. Now, they're sharing, "Hey, what did you do that was successful? Let's take some of that and do that here in my region or in my state." In states that don't have a local chapter, they can still affiliate, and gain some of that perspective and knowledge by being part of the overall PCEA collective. That's what we see as a huge benefit, and it is spreading like wildfire!

Dack: Now, I don't want to pass up the opportunity to ask you both about your perspectives on Gary's role as emeritus chairman. Steph, can you explain?

Chavez: Sure. Gary and Dieter Bergman were the two founders—the visionaries—for the Designers Council. They were instrumental in mapping out the educational and certifica-

North America PCEA members



Global PCEA members

Figure 2-3: These two maps show the reach of PCEA chapters in North America and globally.

tion programs for PCB designers. Gary is an IPC Hall-of-Famer and has championed so much of what the trade organizations stand for throughout his career. He has been involved in IPC subcommittees and has chaired the IPC Designers Council executive staff. I strongly feel we should always remember our roots to keep us grounded as we move forward. Gary's role within the PCEA is as an honorary chairman because we value his input and decades of lifelong lessons to help guide us. We feel a strong urge to embrace the past while we can. We want to carry those values and experience with us as we move into the future.

Dack: Well said, Steph. Gary, do you have anything to add to that?

Ferrari: Everything he said is 100% correct. The main difference I'm seeing is that the chapters that we had, as Steph mentioned, were separated. The only time they had an opportunity to share anything was twice a year at the regular meetings and not all the chapters would show up. But now, they are sharing, even with relatively short notice. The movement is just growing astronomically. I'm so proud of the team we have, including you, Kelly. When we started, our group was small. We now have a vastly more diverse and beautiful organization. I have to give kudos to Stephen and the whole team for that.

Chavez: Let me just add one more thing. Gary is truly at the essence of really taking it forward and making a difference. Let's call it what it is—Gary is our esprit de corps! He refuels us as we look at what he has done and how he continues in the second half of his career. His example gives us that extra bounce in our step moving forward.

Dack: Very good, right on. We're going to move forward now. I'd like to open it up to either of you for some visionary comments. It's been just a little over a year since PCEA hit the ground running. Can you review what has been done over the past year and what we're doing now? Then, please, tell us where we're going. We need to talk about the future.

Ferrari: One of the main items that has taken a tremendous amount of work over the past year is our website. When we started, we had no website either for PCEA or our previous organization. In the '90s, we came out with "Route," a newsletter for communicating with our members. Now, in the PCEA, it feels great to be leveraging the power of the internet and social media. We have sponsors and so many ways to reach out that did not exist before.

Chavez: Another difference is we now have the ability for internet collaboration. We evolve and our media team adapts on the fly. We have activities which are happening constantly. For example, we had our kickoff meeting—our initial grand opening—last year. That was followed by several chapter events. We even held an international tri-chapter event. As a result of the shutdowns during the pandemic, we have evolved into this virtual world. Eventually we'll go back to face-to-face meetings, but that doesn't mean we're going to walk away from a virtual format. We'll continue to embrace and do both. It is amazing to see new chapters coming on board or coming to life in areas where we didn't have chapters before. As far as a footprint, we've got the greater Michigan area; St. Paul, Minnesota; and Portland, Oregon, just to name a few. And we have chapters coming up in Chicago; the Wisconsin area; the New England, New Hampshire and Massachusetts area; Albuquerque, New Mexico; and in Texas: Houston, Dallas, and Austin. Those are all in the early stages of their formation, but it's just amazing how it's taken off. It's all happening so fast. And we have those growing pains too. When you think about what we have done—starting a new industry association and successfully getting it off the ground—it's been a challenge, but a worthy one; it's supplying the adrenaline of a start-up experience, so to speak.

Dack: What is PCEA looking forward to on the 2021 roadmap?

Chavez: Okay, so when we think about PCEA and where we're going, we definitely want to see the continued growth that we have been

experiencing with our chapter activities. We also want our chapters to grow internationally as well. We do see lots of activity going on there and an excitement for engagement. There are a lot of us wearing multiple hats right now so getting more people involved will be a critical part of our evolution. We must make sure to have the right team evolve together. We must all share the same vision. With so much activity and eagerness about so many ideas, we must be certain that we're doing the right thing in the PCEA that will be best for the industry. This is key, and that's where I see us in our 2021 evolution.

Dack: That's tremendous. Steph, Gary, thank you both for sharing your insights.

Next Month

There are more positive PCEA announcements pending which are awaiting official release. I plan to focus in on these exciting topics in our next column.

Upcoming Events

Below is a list of upcoming events which may lead you to mutate your thought process or at least provide you with antibodies to help ward off career stagnation.

- March 8–12: **IPC APEX EXPO** (Virtual)
- May 11–13, 2021: **IPC High-Reliability Forum 2021** (Baltimore, Maryland)
- June 15–17: **PCB East** (Marlborough, Massachusetts)
- August 16–18, 2021: **DesignCon** (San Jose, California)
- November 10, 2021: **PCB Carolina** (Raleigh, North Carolina)

Spread the word. If you have a significant electronics industry event that you would like to announce, please send me the details and we will consider adding it to the list (click here).

Refer to our column and the PCEA website to stay up to date with the upcoming industry events. If you have not yet joined the PCEA collective please visit us online and find out how to become a PCEA member (click here).

Conclusion

Never forget your roots. Carry them forward into the future, but be willing to change, grow and evolve. These are wise and profound messages from the Chairmen. See you next month or sooner! DESIGN007

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

The Role of Resins and Conformal Coatings

Sensible Design

by Phil Kinner, ELECTROLUBE

This month, I examine some of the key differences between conformal coatings, encapsulation resins, and potting compounds to help designers make more informed decisions, and ultimately help increase the performance, reliability and lifetime of your electronic circuitry. I also look at issues that some of you may have with mixing resin packs and air bubbles, what can go wrong and why.

No one wants a product recall, or worse still, a product failure. Therefore, protecting circuitry for its intended purpose and end-use environment is imperative to extend product lifetime and maintain your brand's reputation.

Resins are an excellent way of protecting electronic circuitry, however, there are some excellent conformal coating alternatives on the market as well. For instance, we have created a two-part 2K coatings range, which provides more resin-like protection, with the application ease of a coating. This has been a great success with applications in aerospace and automotive, where protection in harsh environments is absolutely critical, whilst minimising the increase in product weight. I will be exploring coatings vs. resins in a little more detail, using my usual five-point Q&A format.

1. What are the main differences between resins and coatings?

The first difference between resins and coatings regards the various methods of application. For instance, there are aerosols, conformal coating spray equipment, manual spray guns, and brushing for coatings, compared to mixing and dispensing equipment, and resin

packs for resins. Then there is the thickness of the application to consider: <100 microns for conformal coatings, <500 microns for thick coatings, and >500 microns for resins, as well as approval ratings (coatings are generally approved to UL746, while for resins it is very much dependent upon the application).

Due to the coating thickness, coatings occupy less volume and give a much smaller increase in overall weight compared to resins. There are both coatings and resins based on epoxy, polyurethane, and silicone chemistries, but there are also acrylate, acrylic, and parylene coatings which do not have a direct resin equivalent. Most resins are 100% solids systems, so that few or zero VOCs are released during curing with minimal shrinkage, while many coatings are solvent-based, although there are two-component (2K), UV-curable acrylate systems, and solvent-less silicone coatings that are also 100% solid.

Due to the coating thickness, coatings occupy less volume and give a much smaller increase in overall weight compared to resins.

In particular, 2K coatings are designed to be applied more thickly, without cracking during thermal shock testing, and enable a greater degree of component lead coverage to be achieved, resulting in improved performance during thermal shock, powered salt-spray testing, MFG testing and condensation testing (traditionally gruelling test regimes, commonly used during automotive qualification campaigns). The 2K series is also VOC-free, solvent-free, and fast-curing, whether by thermal or the revolutionary UV/chemical dual-cure mechanism.

2. What are the main considerations for choosing a resin instead of a coating?

The main considerations for choosing between a resin and a conformal coating are normally down to the specifics of the application. If the housing is designed to be the primary protection against the environment, then a conformal coating is generally a more appropriate choice. However, if the housing is not the primary environmental protection or if the unit involved is to be subject to long-term immersion in various chemicals, then a resin is generally a preferred, more appropriate choice. Also, if there are many tall and heavy components on a PCB that is expected to encounter significant vibrational loads during use, then an encapsulation resin might be a more appropriate choice. Further consideration should also be given if the unit is to be used in a situation where it is not easily accessible or if a long continuous service life is required, in which case a resin might be recommended to provide the extra protection and durability needed.

3. What is the best way to mix a resin pack and what can go wrong?

Remove the resin pack from the outer packaging. In the case of polyurethane and silicone resins, don't remove it from the foil pack until ready to use. Lay the pack out onto a flat surface and remove the centre clip. Use the clip to push the resin from one half of the pack to the other, then pick the pack up in both hands and mix in a circular motion for a couple of minutes. Place the pack back on the flat surface and use the clip to push the resin from the corners into the centre. Pick the pack up in both hands and continue mixing for a further minute. Repeat, pushing the resin from the corners into the centre of the pack, and then continue to mix for another minute to ensure that the material is fully mixed and is a uniform colour. Use the clip to push the mixed resin to the side of the pack with the angled seam. Roll the pack film up so that the pack fits into the hand. Cut the corner of the pack off and then tilt the pack to dispense the resin, applying slight pressure as required to maintain the flow.

If the material from the corners of the pack is not pushed into the centre of the pack then unmixed material can be dispensed. If the resin is not mixed long enough, then the resin might not cure, or have a patchy cure. In the case of filled resin systems, some sedimentation might have taken place over time, so it might take a little more mixing to ensure that the fillers are correctly distributed throughout the resin. With the optically clear resins, when first mixed, the resin may appear hazy. This is perfectly normal, and the haziness will disappear as the material cures.

4. What happens when air bubbles get trapped in encapsulation resins and how can this affect performance?

There are numerous effects that air bubbles can have on the performance of the cured resin. Depending upon the number and distribution of the bubbles, the actual thickness of the polymer layer applied will be decreased, hence the level of protection will also be reduced, particularly against chemical attack. If the air bubbles are next to components, wiring or tracks, and bridge conductors, then the insulation between those points is compromised and design rules around spacing and clearance may be broken. Ultimately, this can result in premature failures by creepage and clearance breakdowns. Voids also act as a weak point under thermal and physical shock, which can lead to the resin cracking in service.

5. What are the differences between encapsulation resins and potting compounds?

In terms of the resin chemistry used, there's little difference, but there is a difference in how the resin is applied and the performance expected from it. An encapsulation resin will totally cover the PCB and the components and can act as the protective support structure, while a potting compound is used to fill a housing or enclosure containing the PCB and components.

So, an encapsulation resin will adhere to the PCB and the components, and its outer faces will act as the primary barrier to protect the unit. The encapsulation process may or may not be performed with the assistance of a mould. Without a mould, the flow characteristics of the material will be required to control the deposition of material, and a dam and fill process using two different resin chemistries may be used to ensure correct material deposition.

A potting compound must adhere not just to the PCB and components, but to all materials within the device, including the housing. In this case, the differences in CTE (coefficient of thermal expansion) between all the materials used are often a critical factor, as all the materials within the housing will be subjected to different rates of expansion and contraction due to the differences in the materials' CTE values. This can put the resin, components, circuit board, and even the housing under extreme stress, and lead to failure over time.

If in doubt, it is always advisable to discuss with your suppliers which material is best suited to your application. The technical support teams of reputable suppliers have a wealth of experience to call upon and, should it become necessary, they have the expertise to modify chemical formulations to meet your application needs. I hope the points covered this month have been informative. Please do look out for my next column, where I'll be covering more issues on getting the most out of conformal coatings. **DESIGN007**

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

MilAero007 Highlights

USPAE Launches $42M DoD Consortium ▶

The I-Connect007 editorial team recently interviewed Chris Peters, Kevin Sweeney and Shane Whiteside, members of the U.S. Partnership for Assured Electronics (USPAE), about the award the association received from the Department of Defense to create the Defense Electronics Consortium. In this conversation, they discuss the objectives of the consortium, which was created to help the government identify and address potential risks in the electronics industry.

Defense Speak Interpreted: So, What's a JADC2? ▶

The term JADC2 was prevalent in the late 2020 debate about the National Defense Authorization Act. It is a new way defense is using electronics to shape battle strategy. JADC2 is Defense Speak for "Joint All Domain Command and Control." Sounds impressive, doesn't it, but what does that mean?

X-Rayted Files: The Dark Side of the Chip Shortage—Counterfeits ▶

It's February 2021, and as the world slowly recovers from the COVID-19 pandemic, another problem plagues the global economy: the electronic component shortage. What some economists have deemed to be a decade of immense prosperity and growth, the "roaring '20s" started with a hiccup.

Ventec UK Continues to Maintain Highest AS9100 D Quality Compliance ▶

Ventec International Group Co., Ltd. is pleased to announce that the company's European headquarters in Leamington Spa, UK continues to maintain highest AS9100 Revision D compliance in accordance with the Aerospace Supplier Quality System Certification Scheme following successful completion of its surveillance audit.

Zero Defects International Renews ITAR Registration ▶

Zero Defects International [ZDI] has recently received notice of compliance with and renewed registration for ITAR, the International Traffic in Arms Regulations.

New Defense Electronics Group Invites Industry Participation ▶

The U.S. Partnership for Assured Electronics is inviting electronics manufacturers and related companies to participate in its programs, highlighting the opportunities to collaborate with industry peers and the U.S. government. The USPAE was established in 2020 with a mission of ensuring the U.S. government (USG) has access to resilient and trusted electronics supply chains. The USG has many electronics needs, especially for defense- and security-related missions, and the USPAE is lining up funding and collaboration opportunities to address those needs.

L3Harris Technologies' Darkwing Terminal Certified to Operate on Inmarsat's Global Xpress Network ▶

L3Harris Technologies' DarkWing terminal has received Category 1 and Category 4 approval to operate on Inmarsat's Global Xpress (GX) network in both commercial and military Ka (Mil Ka) globally.

IoT is Changing How We Design PCBs

Connect the Dots

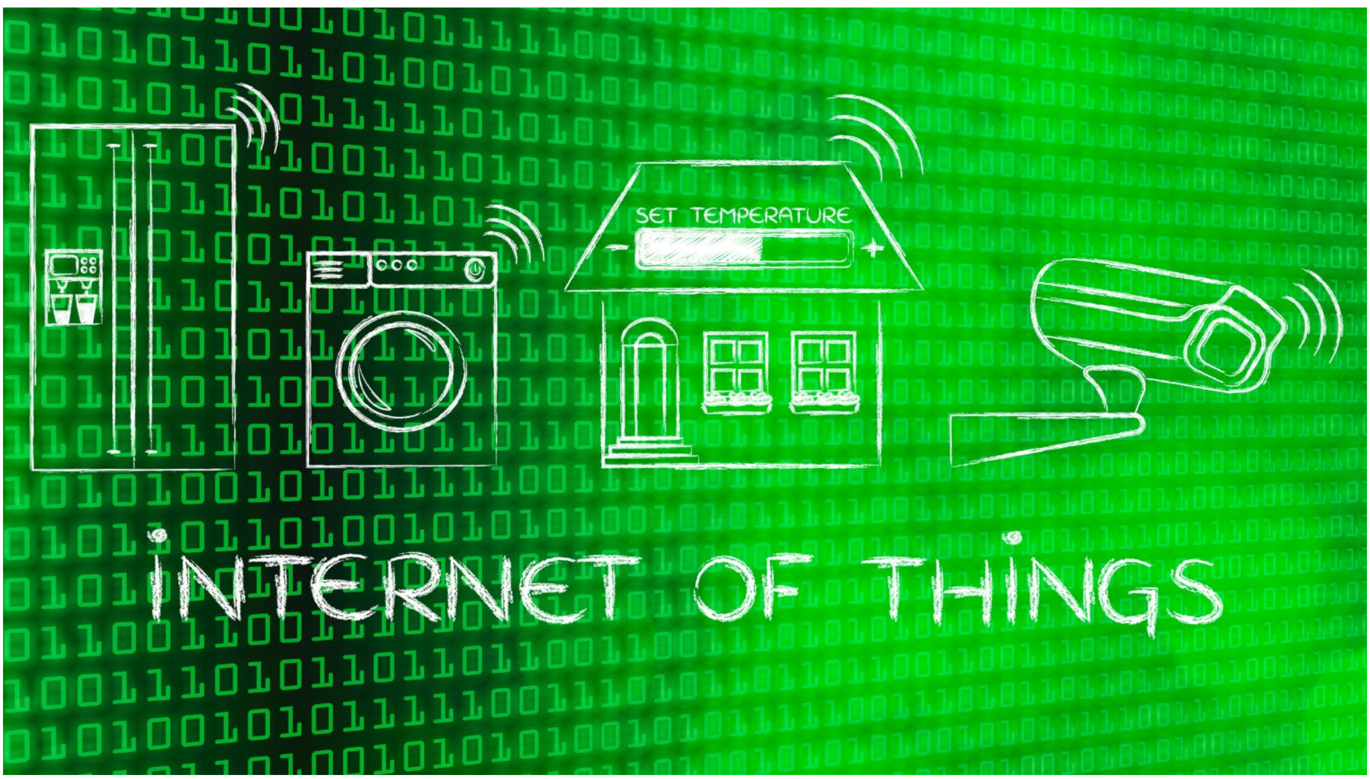
by Matt Stevenson, SUNSTONE CIRCUITS

"It's technology that helps you sleep, not keeps you up. It tells you when you've had enough. It gives you space to create or draw or write or learn, not refresh just one more time."
— Tim Cook, CEO of Apple

The Internet of Things (IoT) is everywhere now—purpose-driven technology that improves lives by making technology work for people, not the other way around. With nationwide 5G wireless capacity rapidly expanding, the demand for and deployment of IoT products is exploding. Households increasingly rely on networked smart appliances. Traditional watches have largely been replaced by fitness trackers, and more of us get our weather reports by asking Alexa than watching the local TV weatherperson.

Demand growth is fueled by business as well as consumers, with pandemic-accelerated healthcare and industrial machinery applications leading the way. IoT devices of every stripe will continue to improve and add functionality while also becoming smaller, lighter, and faster.

Pressure to create innovative solutions for these devices puts the PCB designer at the epicenter of IoT.



It's Time to Broaden our PCB Design Horizons

Designing boards for these devices will take a lot of skill on the part of the PCB designer, as well as a willingness to learn new tricks. Device memory, CPU, and wireless circuitry can often fit onto a system-on-a-chip (SoC) component in the assembly, but chip designs will have to accommodate other critical components packed into a small layout.

Using the same old tech to design boards for IoT will get tougher and tougher. There are packaging technologies available to help designers create boards small enough in thickness and diameter to accommodate the needs of smaller IoT devices. Three-dimensional integrated circuits can help reduce power consumption on a smaller design footprint. Multichip modules allow multiple integrated circuits to function as one—potentially reducing cost of the board while improving performance.

Designing PCBs for IoT also requires thinking about how the end-product is manufactured and put to use. For many IoT devices, PCBs are often embedded in other materials and must be flexible. That means your board design will likely feature newer materials like plastic, mesh, and flexible copper.

To move ahead, designers need to adhere to sound, established design methodology as well as tools that elevate their performance. Using more advanced PCB design tools will help designers respond to the demand for IoT devices.

Best Practices for PCB Designers on IoT Projects

John McMillian's 2017 whitepaper[1] remains the gold standard for IoT board design. Keep the seven design aspects laid out in that paper top of mind and it will help meet the challenges associated with IoT devices. We encourage you to download the paper and keep it on hand for reference. In the meantime, here is our take on McMillan's seven keys to IoT PCB design:

1. Know your design domains. IoT designs with multi-chip modules integrate analog-to-digital, micro-electrical mechanical systems (MEMS), and radio to make them function as one. The more experience you have with these types of circuit design, the more easily you can create a PCB for IoT that meets your functionality requirements.

2. Stay focused on design constraints. Often, the nature of IoT devices puts limits on size and weight that trickle down to your design. If you are adding connectivity to a well-established product, for example, that can really be a challenge for the PCB layout. New versions of established IoT devices need to do everything the last model did, plus more, without getting any larger or heavier.

3. Broaden your component vocabulary. Before you begin to design in earnest, make sure you are familiar with all the components that create functionality for data flow, device display, and Wi-Fi connectivity. You will be building your design around these components, so the more familiar you are with them, the better.

4. Your schematic needs to convey the design's intent. This includes issues such as physical constraints, cost considerations, and component availability. Keep this in mind as you create the bill of materials (BOM). Remember to verify part footprint sizes and height above the board as part of that investigation, especially on a highly constrained design.

5. Test early and test often using all the tools available to you. Model-based design and testing tools will allow you to test, simulate, and verify the design's functionality. Use them. This is how you catch problems and make changes before a prototype build.

6. Learn the critical elements of board layout. Your CAD tool should allow you to visualize the board in its enclosure before you deal with routing and tracing. Using both 2D and 3D views for verification of things like flex circuit bends, you can more easily design within the device constraints.

7. Validate your design. Use your relationship with a PCB manufacturer and assembly partner to make sure your design will work. They will be able to provide feedback that will increase yields and reliability, as well as improve cost competitiveness.

The future of IoT is just around the corner. Expand and hone your design skills today, and perhaps be part of creating the next Alexa or Fitbit tomorrow. DESIGN007

References

1. 7 Design Aspects of IoT PCB Designs—Siemens EDA (pads.com)

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

Putting Graphene in a Spin

Graphene is incredibly strong, lightweight, conductive ... the list of its superlative properties goes on. It is not, however, magnetic—a shortcoming that has stunted its usefulness in spintronics, an emerging field that scientists say could eventually rewrite the rules of electronics, leading to more powerful semiconductors, computers, and other devices.

Now, an international research team led by the University at Buffalo is reporting an advancement that could help overcome this obstacle.

In a study published in the journal *Physical Review Letters,* researchers describe how they paired a magnet with graphene, and induced what they describe as "artificial magnetic texture" in the nonmagnetic wonder material.

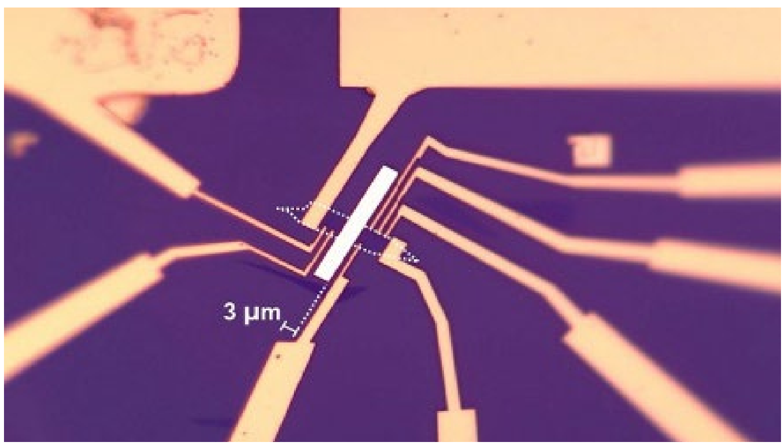
"Independent of each other, graphene and spintronics each possess incredible potential to fundamentally change many aspects of business and society. But if you can blend the two together, the synergistic effects are likely to be something this world hasn't yet seen," says lead author Nargess Arabchigavkani.

For their experiments, researchers placed a 20-nanometer-thick magnet in direct contact with a sheet of graphene, which is a single layer of carbon atoms arranged in a two-dimensional honeycomb lattice that is less than 1 nanometer thick.

"To give you a sense of the size difference, it's a bit like putting a brick on a sheet of paper," says the study's senior author Jonathan Bird, PhD, professor and chair of electrical engineering at the UB School of Engineering and Applied Sciences.

Researchers then placed eight electrodes in different spots around the graphene and magnet to measure their conductivity. The electrodes revealed a surprise—the magnet induced an artificial magnetic texture in the graphene that persisted even in areas of the graphene away from the magnet. Put simply, the intimate contact between the two objects caused the normally nonmagnetic carbon to behave differently, exhibiting magnetic properties similar to common magnetic materials like iron or cobalt.

(Source: University at Buffalo)



The image shows eight electrodes around a 20-nanometer-thick magnet (white rectangle) and graphene (white dotted line).

IPC-2231A—Insights from the IPC 1-14 DFX Subcommittee

Design Circuit

by Patrick Crawford, IPC

In mid-2019, IPC released IPC-2231 DFX Guidelines, a comprehensive guide for establishing best practice methodology in developing a formal DFX (design for excellence) process for laying out printed boards and assemblies. This process can be established at a systemic level—integrated into the workflow of departments of companies—or adhered to as an individual designer. DFX is multifold, and as defined by the document, includes design for manufacturing, fabrication, assembly, testability, cost, reliability, environment, and reuse. The sections pertaining to these major theme areas discuss how they impact the overall performance and cost of the final board assembly.

Of course, no document is perfect, and as is the case with nearly every industry standard, certain elements of the IPC-2231 were identified as needing to be changed shortly after its publication. Now, two years after its initial release, IPC-2231 is close to receiving an upgrade with a new revision—IPC-2231A. The standard is currently in Final Draft for Industry Review and the standard's subcommittee is in the process of forming a ballot group. (For those who are unfamiliar with IPC standardization processes: All IPC documents are subject to a final vote for publication via an open ballot.)

While it can be dangerous to speak of the features of a document in the middle of its development cycle, it is safe to say that nearly every section of IPC-2231 has received some kind of upgrade or addition, and an entirely new section regarding the impact of design choices on fabrication processes has been included. Speaking personally (not as a member of IPC staff), the new revision is a substantial upgrade; it's more focused and yet more rich in content, more easily digestible, and more valuable as a resource.

But enough of what I have to say about the document. After all, other than a few minor


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

editorial changes and tweaks, I am not the author; IPC documents are created "in the open" by industry volunteers. In fact, IPC-2231A is built from more than a century's worth of collective experience through the participants of the IPC 1-14 DFX Subcommittee. For this column, I thought that it would be interesting to highlight some of the 1-14's dedicated volunteers who have helped IPC-2231A come to fruition. Their insight and experience have helped to build the A Rev (and in some cases, the vanilla document) and I cannot think of any better individuals to speak on its need and promise. I spoke with a few of these supporters and asked them about their involvement in IPC-2231A's development as well as why they feel the document is needed.


Russ Steiner

Russ Steiner is a team leader in the ECAD Operations Department at CASCO Automotive, an Amphenol Company. He has been involved with the IPC 1-14 Subcommittee for the last two years. Russ manages a global team of engineering professionals who design robust products, and therefore has a critical need for a global DFX process that ensures complete compliance and integrity. When asked about the need for 2231A in industry, he responded, "Global operations and design portability is a key consideration for hedging against prospective manufacturing venue catastrophes, manufacturing redundancies, and predictable AQL/PPM."


Joe Clark

Others find value in how a guideline like IPC-2231A can help single employees work cohesively inside of a huge team, either as part of large companies or within manufacturing supply chains. Joe Clark of Lockheed Marin Missiles and Fire Control, PWB Fabrication and Design, has been involved with 1-14 for about a year. Referring to IPC-2231A, Joe said, "Knowledge continuity in the electronics industry is an issue at many companies since the design and manufacturing processes span many functions and roles. The IPC-2231 document helps bridge the gap in knowledge for those who may play a single part in the overall process." Indeed, by capturing the knowledge and needs of the complex chain of requirements imposed by manufacturing, testing, and lifecycle considerations, guidelines like IPC-2231A can become the leverage that designers can use to justify design decisions. Patrick Phillips of Northrop Grumman put that more succinctly by replying that IPC-2231A contained "tools for people to use for building a case to drive product improvements."


Pietro Vergine

Others still found that the general mindset of DFX will not only help individual companies cohere their processes internally or the processes of their contractors and manufacturing verticals, but also to create some alignment between companies in general—even competitors—who all would benefit from best practices offered by a consensus document. Pietro Vergine, president of Leading Edge, provided an excellent summary of this point: "I think that most companies are already using internally developed rules/methodologies that are DFX oriented, but maybe call them something different (i.e., designing documents with design rules, or rules for the assembly, or design rules for manufacturing, or electrical design rules). As with all other IPC standards, I think that having [IPC-2231A] that has a general consensus from the electronics industries with the collaboration of several companies may have a great impact on all of their activities. It might not be perfect, but it will let us think about the global picture, and not just a single problem."

Many found that certain sections of the document contained knowledge that is critical to help designers understand the effects of their design choices on downstream cost drivers, most notably manufacturing. Jasbir Bath has been integral in the development of the IPC-2231A, where he employed his experience as a support advisory engineer at Koki Solder to enhance its information regarding process engineering. “Manufacturing yield needs to be improved and documents such as 2231A [can] help designers understand more about the challenges in manufacturing to help design products with improved manufacturing yield,” Jasbir said, adding that he was most excited about “generally updated information on soldering materials, processing temperatures, board finishes, and reliability concerns” within IPC-2231A.

Jasbir Bath

Jasbir was far from alone in feeling the need for more understanding of DFM within industry. Scott Vorhies, a primary contributor for all DFX-related information concerning the assembly of PCBAs within SpaceX, hopes that IPC-2231A will “provide a source of information helping manufacturing engineers troubleshoot assembly issues from start to finish,” noting that “this document, if allowed to be a source of truth, would greatly enhance the ability of designs to be manufactured at a lower cost and higher reliability, and help bridge the gulf between design and manufacturing.” Geok Ang Tan of DSO Laboratories agreed, and similar to Patrick’s sentiments above, said that “with this, staff can share with management that there are a lot of details in the manufacturing process to produce electronics hardware; new staff will gain a lot from it.”

Jon Bruer of Creation Technologies Inc. is most excited about the Design for Testability (DFT) section of IPC-2231A, which he has helped shape for the new revision by channeling his more than 32 years of experience in test engineering and DFT. Regarding the need of a DFX guideline in industry, Jon said that “certainly from a test perspective there is a huge need for a stronger understanding of DFT and test requirements for production test, mostly with respect to OEM engineering teams. That said, as a contract manufacturer, my company helps to provide input and helps to fill the gaps for our customers. I think that the more awareness there is in the industry for DFT thinking to occur at the earliest stage in product design, the more testable product designs will be as they move into the production area.”

Jon Bruer

The overall outcome of IPC-2231A, according to Karen McConnell of Northrop Grumman and a current co-chair of the IPC 1-14 DFX Subcommittee, is to help companies that “do not have a DFX champion to facilitate the insertion of best practices into processes.” Moreover, she expects that the practices described in IPC-2231A should and will be rolled down to subcontractors, so that excellence can be maintained throughout the supply chain.

Karen McConnell

Regardless of which “X” in DFX is being considered, all members of the 1-14 DFX Subcommittee agree that no company is perfect, and any design process can be enhanced. IPC-2231A DFX Guidelines is expected to be available in early Q3 2021, and until then, I will leave you with one of my favorite quotes from my poll of the 1-14 Subcommittee. When asked about the importance of DFX in design, Murilo Levy Casotti of Embraer, simply replied that “[DFX is] primordial. All ‘design for’ are very important.” DESIGN007

Murilo Levy Casotti

Patrick Crawford is the manager of design programs and related industry programs at IPC. To read past columns or contact him, click here or email PatrickCrawford@ipc.org.

IPC-2581 Revision C: Complete Build Intent for Rigid-Flex

Article by Ed Acheson
CADENCE DESIGN SYSTEMS

With the current design transfer formats, rigid-flex designers face a hand-off conundrum. You know the situation: My rigid-flex design is done so now it is time to get this built and into the product. Reviewing the documentation reveals that there are tables to define the different stackup definitions used in the design. The cross-references for the different zones to areas of the design are all there, I think. The last time a zone definition was missed, we caused a costly mistake.

Continuing to review the design documents, I verify that the bend locations are defined with information about the radius of the bends with a detail about how the final product looks when all bending is complete, ensuring that the folds are made in the correct order. I hope all information is contained in the documentation, and there will be no calls from the fabricator delaying the product. With all these documents and details left open to interpretation, there must be a way to send this data more intelligently.

Enter IPC-2581 Revision C

There is a way to transfer this data digitally, reducing the need for various forms of drawing details in a document. The new IPC-2581 Revision C format eliminates the need to manually—and painstakingly—create these details

in a fabrication drawing. It uses the design data to explicitly define the multiple aspects of a rigid-flex design. How? Let's look at how some of the details are sent digitally.

First, let's look at stackup and general board structure. In the design tool, the different stackup details are created, with one or more rigid stackup definitions (8-layer vs. 4-layer, etc.) as well as several flex stackup structures (1 or 2 copper layers, etc.). In my design database a boundary is defined and the stackup data is assigned to those boundaries. This data is then placed into the IPC-2581C format containing the links of each stackup to each boundary association. These are known in IPC-2581 terms as stackup groups assigned to stackup zones. A by-product of these connections is the ability to define the outline profile for each copper and dielectric layer, a key tool for the fabricator.

These links within the IPC-2581C data take the guesswork out of calculating the different boundaries for each stackup definition to stackup zone connection and the edges for each layer. This reduces time spent on calculation and errors in interpretations for the fabricator.

Other factors related to flex information include details of each bend. The details attached to bend lines and bend areas are also preserved in the IPC-2581C data. The bend line, drawn across the apex of the bend in the CAD tool, area of the flex affected by the bend extents, or bending area, and bend definition (direction, bend type, radius, bend angle) are passed into IPC-2581C. These details can be leveraged though the manufacturing process into the final assembly.

Conclusion

Simply put, we now have a better way to get design intent for rigid-flex designs with less reliance on complex drawings and manual documentation. Passing the rigid-flex build

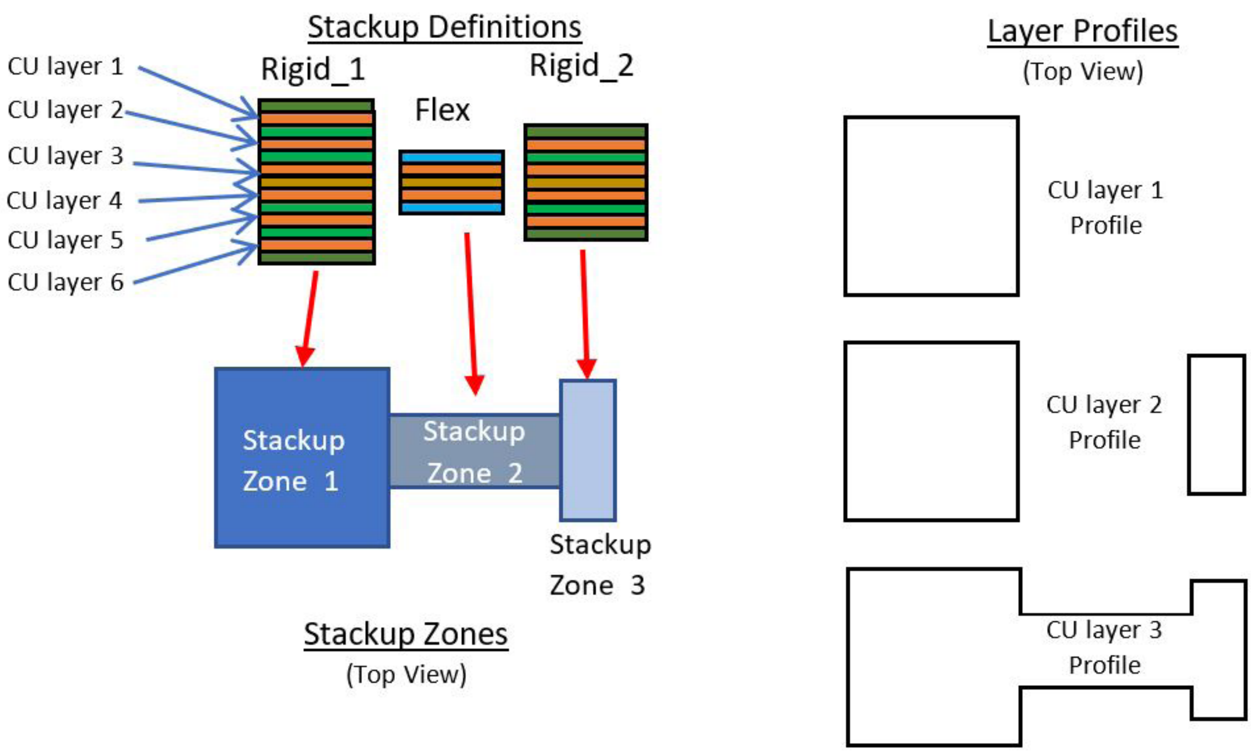


Figure 1: Stackup groups assigned to stackup zones and layer profiles are provided in IPC-2581 Revision C.

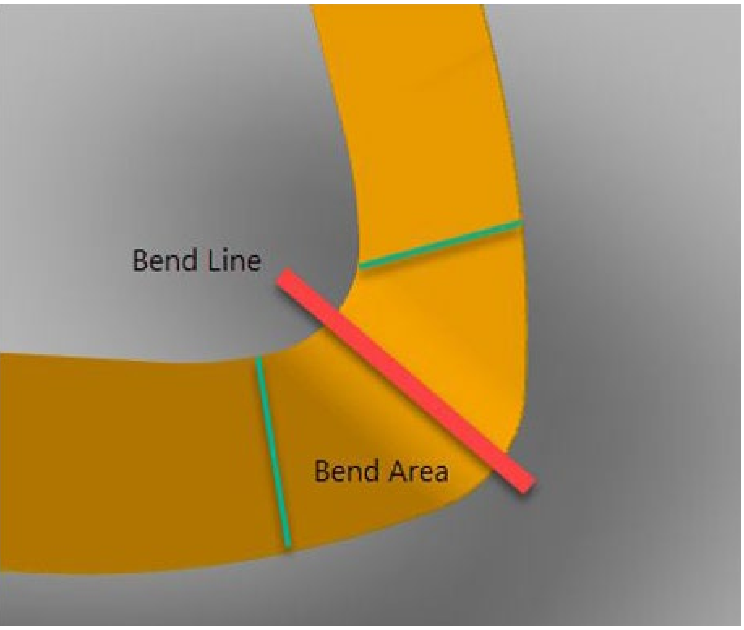


Figure 2: IPC-2581 Revision C also provides detailed information about the bend (type, direction, radius, and bend angle).

intent in the IPC-2581C format is the better way to provide the detail needed to express design intent, resulting in a lower risk of misinterpretation, and faster time to product.

IPC-2581 Revision C has been enhanced to address the latest needs of the industry for easy, intelligent design data hand-off to manufacturing. It was worked on by the industry over a period, with many detailed discussions on how to address these needs. It builds on the well-established successes many companies have already had after switching to IPC-2581.

Visit www.ipc2581.com for more information about companies that support it, test cases for revision A, B and C, as well as links to download free IPC-2581 readers from several companies. **FLEX007**

Ed Acheson is a senior principal product engineer for Allegro PCB products at Cadence Design Systems, focusing on ECAD-MCAD (EDMD Schema), rigid-flex technology and Design True DFM.

Helping Soft Robots Turn Rigid on Demand

by Daniel Ackerman, MIT News Office

Working with computer simulations, MIT researchers have developed a concept for a soft-bodied robot that can turn rigid on demand. The approach could enable a new generation of robots that combine the strength and precision of rigid robots with the fluidity and safety of soft ones.

“This is the first step in trying to see if we can get the best of both worlds,” says James Bern, the paper’s lead author and a postdoc in MIT’s Computer Science and Artificial Intelligence Laboratory (CSAIL).

Roboticists have experimented with myriad mechanisms to operate soft robots, including inflating balloon-like chambers in a robot’s arm or grabbing objects with vacuum-sealed coffee grounds. However, a key unsolved challenge for soft robotics is control—how to drive the robot’s actuators to achieve a given goal.

A simulated soft robot controlled to reach the same target (red dot) while acting either soft (left) or stiff (right).

Until recently, most soft robots were controlled manually, but in 2017 Bern and his colleagues proposed that an algorithm could take the reigns. Using a simulation to help control a cable-driven soft robot, they picked a target position for the robot and had a computer figure out how much to pull on each of the cables in order to get there. A similar sequence happens in our bodies each time we reach for something: A target position for our hand is translated into contractions of the muscles in our arm.

The researchers’ paper lays out a way to simultaneously control the position and stiffness of a cable-driven soft robot. The method takes advantage of the robots’ multiple cables—using some to twist and turn the body, while using others to counterbalance each other to tweak the robot’s rigidity. Bern emphasizes that the advance isn’t a revolution in mechanical engineering, but rather a new twist on controlling cable-driven soft robots.

(Source: MIT News Office)

With Flex, Sometimes You Gotta Break the Rules

Article by Tony Plemel
FLEXIBLE CIRCUIT TECHNOLOGIES

Sometimes in life, we need to break the rules. For example, in junior high I had a curfew but to have my first kiss, I had to break curfew. I got grounded, but it was worth it!

My last article was about reasons to follow IPC design and inspection rules. This time, we are discussing instances where, due to complex requirements, customers are not always able to follow the rules. I will also discuss some design options that will hopefully keep you from "getting grounded."

Here are a few examples of when a customer would need to stretch the rules and supersede IPC-2223 and IPC-6013.

1. Insufficient real estate for IPC-suggested blind or buried via hole sizes.
2. Limited space requires bending a flexible circuit tighter than the IPC-2223 bend rules.
3. Insufficient real estate requires placing vias closer to the transition rigid-to-flex areas on a rigid-flex circuit.
4. Requiring tighter than normal tolerances.

Now, let's go through these in greater detail.

1. Smaller than IPC-suggested blind or buried vias.

As packages get smaller and real estate is at a premium, customers sometimes go smaller than the recommended ratio of 1:1 for blind or buried vias. Due to reliability concerns, the smallest recommended microvia is 100 µm (0.004"). If the design will not support 100 µm vias, it may be acceptable to drop to 75 µm if the 1:1 aspect ratio is strictly followed. So, the 75 µm via hole can only go 75 µm deep, or less. Be sure your flex supplier runs thermal shock testing on all plated vias and through-holes (especially blind vias under 100 µm) to ensure that they are reliable.

2. Bending a flexible circuit tighter than the IPC-2223 bend rules.

A recent customer had a situation where a flex arm on a 2-layer flex circuit did not have room for a radius of 10:1 as recommended by IPC-2223. The circuit had to bend back 180 degrees upon itself. We recommended a forming fixture to create a repeatable bend. Within

this fixture, a clamp was placed to ensure that once a droplet of super glue was added, the circuit bend was stabilized. This allowed a repeatable and reliable 2:1 bend ratio. It is critical on a reduced bend ratio application that once the circuit is formed, it is immediately constrained and not allowed to relax. Once all devices were in place and a few bends were made and glued, cross-sections were taken to ensure there was no cracking or elongation of the traces, or delamination in the coverlay. Since then, hundreds of thousands of circuits have been deployed in the field with zero quality issues. When the available room and the install scenario were considered, the solution we came up with was a success.

3. Placing vias closer to the transition rigid-to-flex areas on a rigid-flex circuit.

Often, the reason customers go to a rigid-flex solution is because space is limited. In some cases, the space is so limited that the rules need to be broken.

IPC-6013 guidelines state that minimum coverlay penetration into the rigid area is to be at least 0.635 mm (0.025") but shall be sufficient to provide coverage of exposed conductors, and not to extend to the nearest via. My suggestion when a via needs to be even closer to this transition area is to use both drilling and punching to define the coverlay as shown in Figure 1. This allows the coverlay to stay out of the drilled via, but allows the via to be closer than recommended, ABUSS (as agreed upon between user and supplier).

4. Requiring tighter than normal tolerances.

Again, IPC sets up standards for tolerances, but customers will sometimes need very tight tolerances that exceed what is outlined in IPC-6013. Please be sure your supplier is able to perform and meet these requirements.

I had a customer who required openings in the substrate with a ±10 µm (0.000393") hole location tolerance in relation to other features, and a ±12.5 µm (0.0005") tolerance on the hole size. We were able to perform this by lasering fiducials while we created the substrate openings. We also adjusted the hole sizes by compensating half the laser beam diameter. While we were able to meet the requirements, yields were reduced, which resulted in increased costs.

When tolerancing your drawings, is it important to remember that these are flexible circuits, not machined metal parts. Flexible circuits can grow and shrink by 0.001" due to just temperature and humidity. You should have reasonable expectations of just how tightly dimensions can be controlled without affecting the manufacturer's yields. Also, any dimension that you put on your drawing will have some cost impact since the manufacturer will have to verify them. If there are features on your flex-

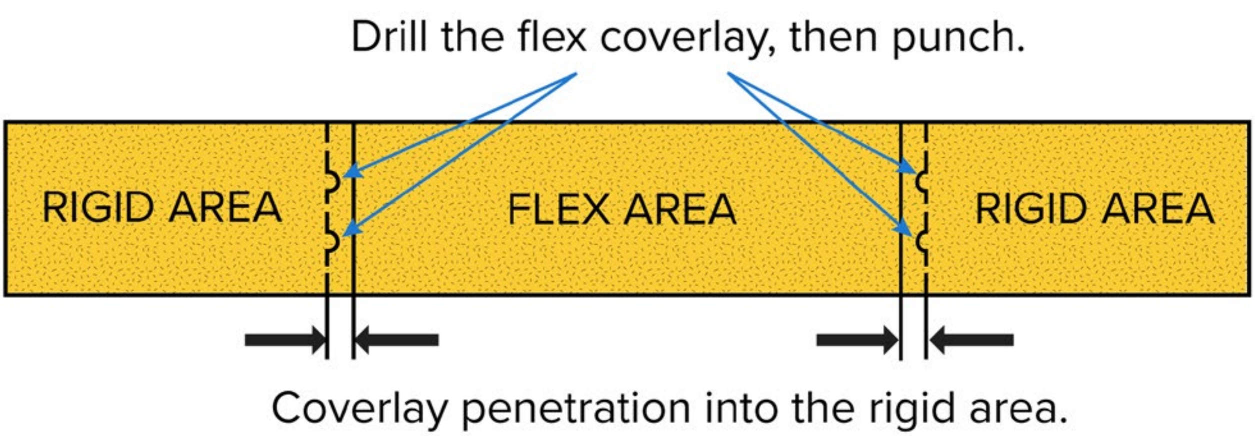


Figure 1: Define the coverlay by using drilling and punching.

ible circuit that are truly critical, include them on the drawing. If the features are not critical, either leave them off or make them a reference.

Work with your customer to ensure that drawing dimensions with tight tolerances are indeed critical, and not unnecessarily over-dimensioned. Again, dimensions with extremely tight tolerances are cost drivers!

One last thought: Always keep in mind the environment in which the circuit is used. My job as a flex engineer is to assist you by asking questions about how the circuit needs to be designed to ensure it will be able to survive the environment in which it must exist. Always inform your flex engineer about any environmental requirements such as temperature extremes, chemicals, and/or moisture exposure, vibration, static or dynamic flexing, and voltage and current requirements. This will ensure that all factors are considered in your design. FLEX007

Tony Plemel is a flexible circuit applications engineer with Flexible Circuit Technologies.

Detecting COVID-19 With a Sticker on Your Skin

One day, a wearable, bioelectronic device could wirelessly transmit a person's vital signs—potentially providing critical information for early detection of health issues such as COVID-19 or heart disease—to a healthcare provider, eliminating the need for an in-person visit while also saving lives.

University of Missouri engineers are advancing the commercial market for wearable bioelectronics by developing a large-scale manufacturing plan for a customizable device capable of simultaneously tracking multiple vital signs such as blood pressure, heart activity and skin hydration.

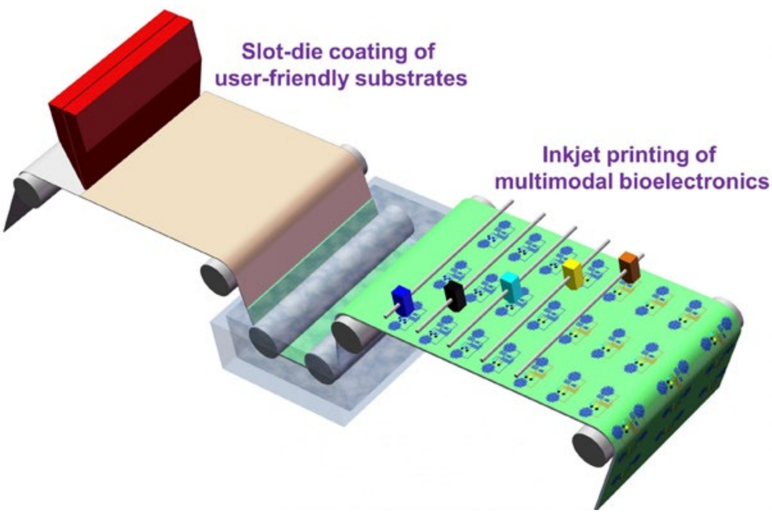
"While the biosensors for these devices have already been developed, we now want to combine them to mass produce a porous patch with multiple bioelectronic components," said Zheng Yan, an assistant professor in the College of Engineering. "The components can also be customized to fit the individual health needs of the user."

Yan recently received a more than $500,000 grant from the National Science Foundation's Faculty Early Career Development Program, or CAREER, to begin a plan for mass production of the low-cost device.

An NSF grant allows Yan to build on his previous work demonstrating a proof of concept of a small patch that works as a breathable and waterproof on-skin electronic device with passive cooling capabilities.

Yan said existing wearable devices usually consist of bioelectronics supported by a flexible, solid material—typically plastic or silicone—called a substrate. He wants to optimize the material to be soft, breathable, comfortable, lightweight, and waterproof. Also, to mass produce the bioelectronic sensors, Yan is researching how to print them directly onto the supportive material using a method called mask-free inkjet printing.

(Source: University of Missouri)



University of Missouri engineers are advancing the commercial market for wearable bioelectronics by developing a large-scale manufacturing plan for a customizable device capable of simultaneously tracking multiple vital signs such as blood pressure, heart activity and skin hydration.

Flex007 Highlights

Orbotech Flex Equipment Introduction ▶

Barry Matties speaks with Meny Gantz about the challenges for flex manufacturing, what's driving the complexity around it, and how Orbotech's new products will help their customers produce flex in much more effective way.

EPTE Newsletter: Taiwan Releases 2020 PCB Production Numbers ▶

The Taiwan Printed Circuit Association (TPCA) released December's shipment data. Full year data is also posted for 2020, so it's time to review the industry's performance.

Lenthor Engineering Receives ISO 9001:2015 Certification Renewal ▶

Lenthor Engineering, Inc., a California-based designer, manufacturer and assembler of flex and rigid-flex printed circuit boards, has successfully completed a full surveillance re-certification to ISO 9001:2015. The certification period runs through February 20, 2024. Copies of Lenthor's certification can be made available upon request.

Flex Talk: Demystify Flexible Stack-ups ▶

The sheer number of flexible laminate materials and constructions can be a bit daunting for those new to flex and rigid flex design. Tara Dunn sits down with Jeff Martin from Omni PCB to hear his insight into flexible laminates and his advice when working on a flex stack up.

Consider This: PCB Technologies We Need Now and Later ▶

RF high frequency circuits are in almost every vehicle being produced today, with side, front, and rear radar to warn of potential collisions. Specialized RF technology high frequency PCBs are in high demand.

New DuPont Production Line in Circleville Slated for Completion in 2H 2021 ▶

DuPont Interconnect Solutions, a unit of DuPont Electronics & Industrial, has announced its $220 million expansion project at the Circleville site is expected to be completed in the second half of 2021.

IPC APEX EXPO 2021 Offers More than 100 Future-focused Educational Opportunities ▶

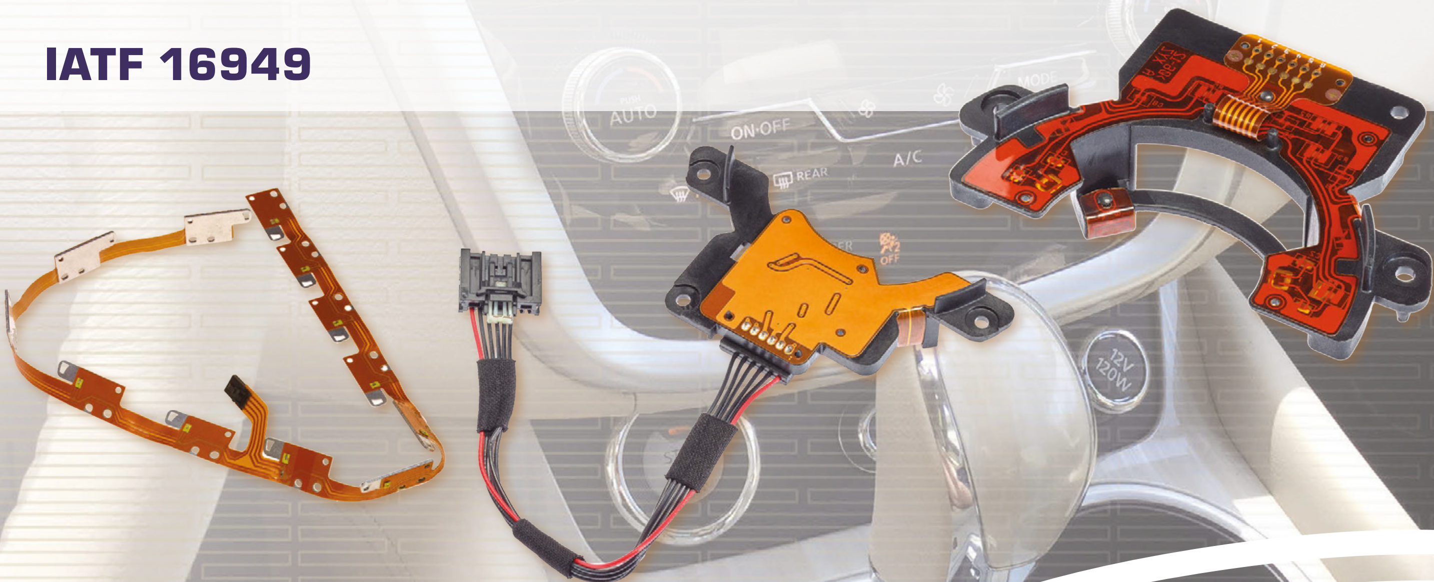
Changing technologies, advanced materials and new processes that are driving the electronics manufacturing industry will take center stage throughout the IPC APEX EXPO 2021 technical conference and professional development sessions, which will take place virtually March 8-12.

Flexible Thinking: IC Package Footprints—Why So Many and How Many Is Enough? ▶

Joe Fjelstad takes a historical look at the formation of integrated circuits and what that means for today's PCB designs.

Insulectro to Distribute Indubond Lamination Presses from Chemplate Materials SL ▶

Insulectro, the largest distributor of materials for use in manufacture of printed circuit board and printed electronics, has announced it will distribute InduBond® lamination presses and supplies manufactured by Chemplate Materials SL of Barcelona, Spain.



Simplified Assembly of Aluminum Flexible Circuits

Flex Talk

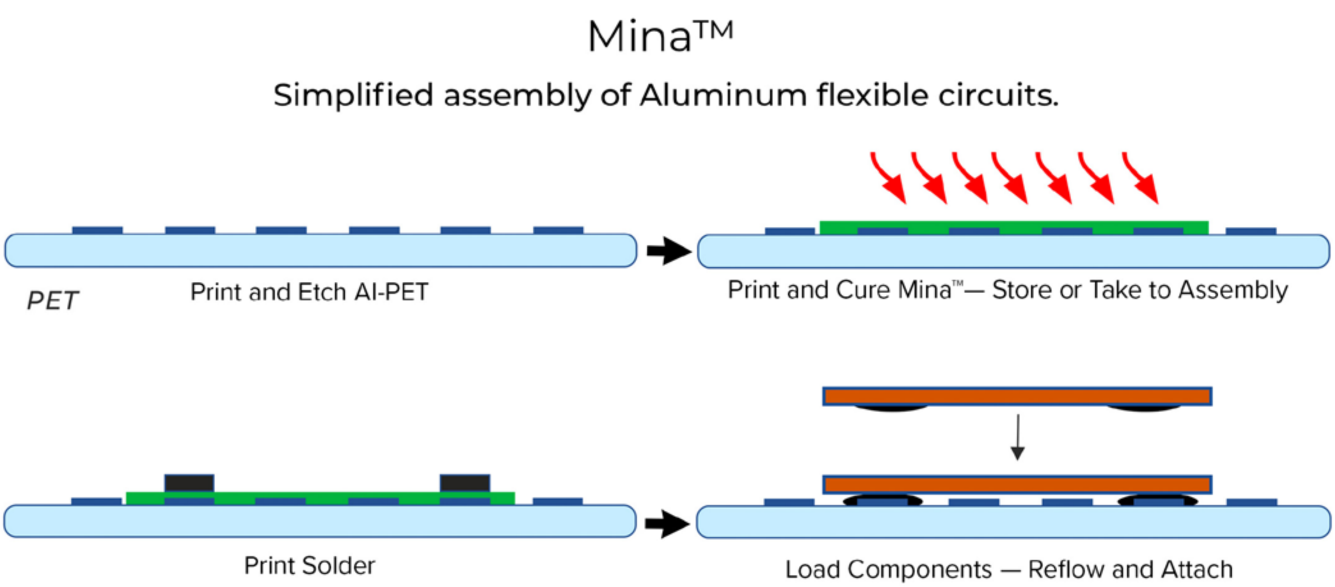
by Tara Dunn, AVERATEK

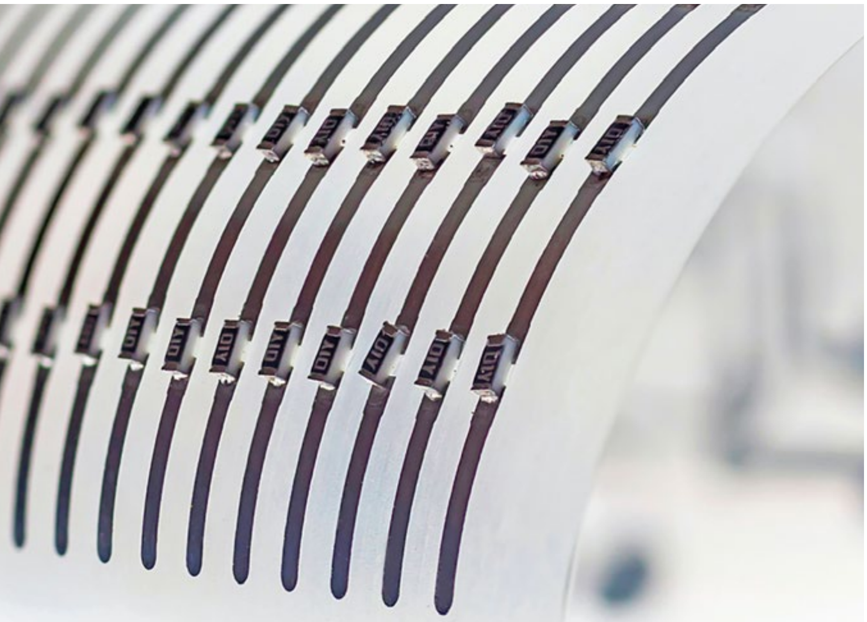
Flexible circuit designs that come across my desk are predominately constructed with copper and polyimide laminates. As I learn more about automotive applications, I am intrigued by the possibilities of using aluminum in place of copper and the potential to use polyester in place of polyimide. Both aluminum and polyester have traditionally been difficult to solder to. One very interesting development has been the Mina™ chemistry. This coating not only simplifies soldering to aluminum, but it also enables the ability to automate low temperature soldering to polyester. Having many questions about this process, I sat down to discuss the Mina process with Divyakant Kadiwala, vice president of manufacturing for Averatek. He has been instrumental in the development of this assembly process.

Tara Dunn: Divyakant, before we jump into the conversation about Mina, could you share a brief introduction to both Averatek and your background?

Divyakant Kadiwala: Averatek is a high-tech company based in Silicon Valley. It was founded by SRI International and private investors. It has two primary products: LMI™, a catalytic ink that enables the fabrication of very high-density circuits with the patented A-SAP™ process; and Mina, a surface treatment that enables soldering to aluminum. I am VP of manufacturing and my role includes overseeing process

Divyakant Kadiwala





An example of aluminum over copper.

engineering, quality control, facilities management, and business development. I am leading our efforts on productization of Mina.

Dunn: Thank you. Let's start the conversation with aluminum. What are the benefits of aluminum over other metals?

Kadiwala: Aluminum is the most abundant metal in the earth's crust. This makes it more easily available and less expensive compared to other metals. It has different benefits when compared to other metals based on the field of use. For example, in automobiles, its superior strength at lower weight is a significant benefit against iron and its common alloy—steel. But in the field of printed electronics, its main competitor would be copper.

Aluminum is more than three times lighter than copper. It has 68% of the conductivity of copper but has only 30% of the weight of copper. This means that a bare wire of aluminum weighs half as much as a bare wire of copper that has the same electrical resistance. This makes it the metal of choice for high voltage transmission cables. Also, it is three times less expensive than copper on an equal weight basis and six times less expensive on an actual usage basis. This is the biggest advantage that aluminum has over copper.

Dunn: Interesting. Why do you think aluminum is attractive specifically for automotive applications?

Kadiwala: Aluminum has been a boon for the automobile industry. Like any industry, the automobile industry has evolved due to various geo and political reasons over its 100-plus-year history. Whether it's the oil embargo of the 1970s or global warming due to climate change, it has been constantly under pressure to improve fuel efficiency standards.

The easiest way to improve fuel efficiency is to reduce the dead weight of the automobile and improve the efficiency of the internal combustion engine. Aluminum helps reduce an automobile's body weight by providing superior strength at lower weight when compared to steel and other alloys of iron. It also helps improve engine efficiency by providing better performance at lower weight.

Since most automobiles have their body and chassis made of aluminum, the integration of onboard electronics to aluminum is critical. Averatek's Mina can provide a soldered connection to aluminum without any plating or surface finish. It can help ease manufacturing and reduce costs.

Dunn: Aluminum has historically been difficult to assemble to. What are the challenges of soldering to aluminum?

Kadiwala: Aluminum PC boards, whether rigid or flex, are limited in use due to challenges associated with soldering components to aluminum pads. This is because all aluminum surface is covered with a layer of aluminum oxide. Although self-limiting, this oxide layer cannot be overcome by flux in existing solder systems during reflow. It thus prevents the formation of a metal-to-metal bond. Even if this oxide layer is removed using etchants and fluxes, a new layer forms in situ upon exposure of clean aluminum to the atmosphere. This prevents the use of conventional SMT methods for attaching SMDs to assemble PC boards.

Dunn: Can you expand on a couple of ways that Mina could be utilized in automotive applications?

Kadiwala: Integrating electronics with the aluminum body and chassis of the car is an integral part of its manufacturing and design. Since aluminum is not easy to solder to, mechanical "crimp" and "pigtail" connections are common options.

Mina™ can provide a soldered joint instead of these mechanical connectors. It can help attach aluminum wire or PCB to aluminum chassis for grounding or other such connections. It can also help with attaching copper to aluminum PCB where applicable.

Dunn: This is exciting. With any new process, reliability is always a concern. What type of reliability data has been gathered?

Kadiwala: Flexible PCBs were made using an Al 9-mm/PET 38-mm substrate and components were soldered using low temperature Sn/Bi/Ag solder paste. The fully assembled PCBs were then subjected to these tests:

- K-09 — "Damp Heat, Cyclic (with Frost)"
- M-04 — "Vibration"

Both these come under the LV-124 European automobile standards.

1. Characteristics of the Test K09 — "Damp Heat, Cyclic (with Frost)"
    - Chamber temperature: without cold phase: 23°C to 65°C; with cold phase: –10°C to 65°C
    - Chamber humidity: 95% RH
    - Duration: 10 d = 10 cycles at 24 h
    - Cycling: 5 cycles with cold phase, then 5 cycles without cold phase
    - Operation of samples: here not operated (usually intermittent operation = 30 s on, then 30 s off)
2. Characteristics of the Test M-04 — "Vibration"
    - Chamber temperature = 8 h, profile between -40 and 80°C
    - Broadband random vibration according to vibration profile D in LV-124 → 5-2000Hz → 30.8 m/s^2 → X-axis, Y-axis, Z-axis → 8 h each axis

Dunn: What materials does Mina work well with? Rigid, flex materials, others?

Kadiwala: Mina works well with both—flex and rigid materials. The alloys of aluminum that we have proven for our customers include Al1235, Al6061 and Al5052.

Dunn: Understanding the benefits of aluminum over copper and understanding the benefits of flexible materials such as polyester, the combination seems like a natural fit for automotive applications and a wide range of other applications. Which markets do you see as the early adopters of Mina taking advantage of the simplification of the soldering process for these materials?

Kadiwala: In addition to automotive, we see significant interest from the LED, SmartTag and heat-sink and high-power industries.

The LED industry would not only benefit from lower costs due to cheaper Al-PET substrate but also it would be a better product. A soldered LED operates much cooler compared to one attached using conductive epoxy. Based on information from a large LED manufacturer, the lifetime of an LED doubles with every 10°C reduction in operating temperature. Thus, soldered LED panels will be more reliable.

SmartTags have more components and are larger in size compared to generic RFID tags.

Mina is a simple pretreatment for aluminum that makes it as easy as soldering to copper.

They use a lot more silver epoxy and thus are more expensive to make than RFID tags. Also, SmartTags are designed to be used multiple times while RFID tags are usually meant for single use. Hence, longevity is important for SmartTags. A tag made with its components soldered will be more reliable and cheaper compared to one made using silver epoxy.

Heat management is important for high-power devices. Aluminum is commonly used to build heat sinks. Soldering a high-power device to aluminum would require a plated surface finish like ENIG and ENEPIG. These add costs and so other means like thermal tape, etc., are used at the cost of performance and life of product. Mina addresses all these issues by enabling direct soldering to aluminum heat sinks.

Dunn: Divyakant, thank you so much for talking with me about the Mina process. Working with flexible circuits for many years, it is always exciting to learn about new applications that change the way we look at flex assembly and integration.

Kadiwala: Thank you, Tara. FLEX007

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

TNO at Holst Centre Develops Cuddle Vest

A year onwards, COVID-19 is still among us, and restrictions to battle the disease are in place worldwide. As a result, people are expected to keep a safe distance from each other. But when people are not allowed to get close physically, feelings of loneliness and insecurity can arise.

Lotte Willems, business development manager at TNO at Holst Centre, says, "Our connected cuddle vest with integrated haptic technology gives people the opportunity to experience a friend's touch, even in times of social distancing. Fourteen haptic motors positioned around the upper body and arms create vibrotactile patterns that feel like a caress or even a firm hug. The vest is controlled at a safe distance by a phone or tablet, creating a new form of intimacy."

Lotte demonstrating the cuddle vest at Editie NL, a Dutch news show.

The cuddle-vest is designed to showcase the possibilities of haptic feedback over a distance. Lotte says, "At the moment, the most attractive opportunity this vest creates is COVID-safe cuddling. This can be of great value in, for example, elderly homes where visits are now limited and even staff has to restrict proximity. However, this technology can also be applied in many other fields, such as supporting physical therapy, enhancing the gaming experience, helping people with autism to deal with an overload of stimuli; the possibilities are endless."

(Source: Holst Centre)

Reaching the Limit on Stiffeners

Consider This

by John Talbot, TRAMONTO CIRCUITS

Flexible printed circuit boards, by their very nature, are designed to be flexible. This presents problems in securely and reliably attaching the ends of the flex circuits to a solid, stiff, main PCB or other electronic devices. A combination of hard, as well as semi-flexible stiffeners, is used for this purpose. Hard stiffeners are FR-4, aluminum, and heat conductive aluminum-backed metals. A hard stiffener has a limit as to how close the flex circuit bends to the stiffener without stressing the joint or causing the flex circuit to crease (Figure 1).

Thick flex cover-coat material is used as a semi-stiffener where the flex circuit needs to retain some flexibility close to the attachment point. The slightly less flexible stiffener allows for attachment of the flex to the FR-4 PCB, while providing an area of reduced flexibility, allowing for a more reliable connection without creasing the flex near the attachment point (Figure 2).

Double layers of thicker, cover coat stiffeners create a slightly bendable flex attachment point such as a printer head to a fixed PCB. The cover coat material allows for some stress transition from highly bendable to rigid, reducing the chance of a crease crack in the copper, similar to a short, tapered cord protector on a power tool. Flex circuitry can rip or create openings in the copper traces if the bend radius

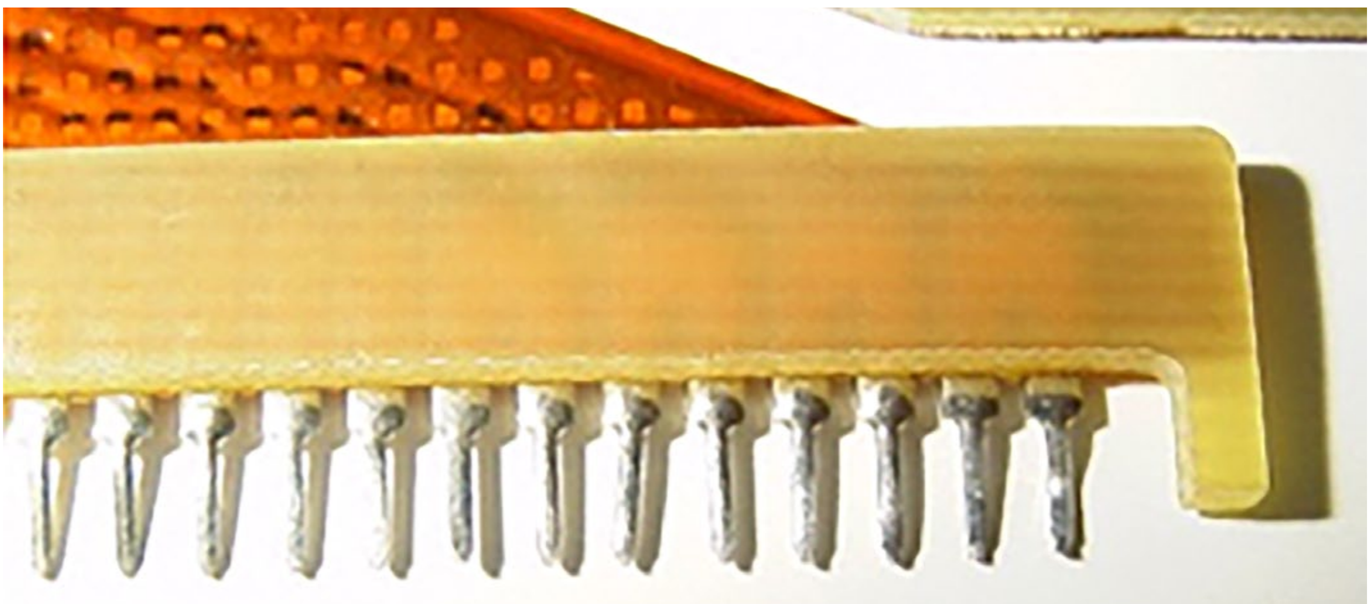


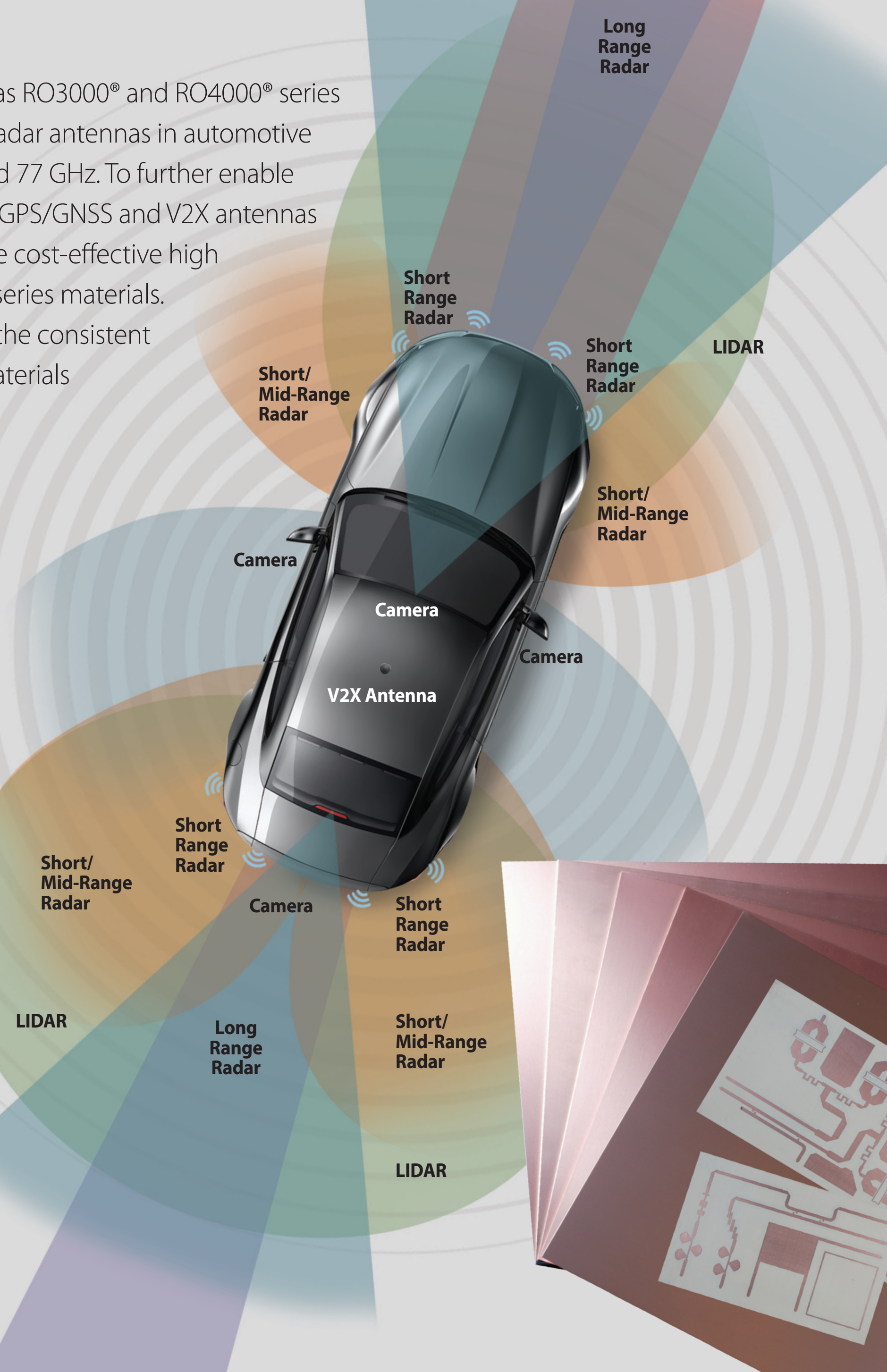
Figure 1: Solid FR-4 stiffener with locating tabs.

Rogers' Laminates: Paving the way for tomorrow's Autonomous Vehicles

Autonomous "self-driving" vehicles are heading our way guided by a variety of sensors, such as short and long range radar, LIDAR, ultrasound and camera. Vehicles will be connected by vehicle-to-everything (V2X) technology. The electronic systems in autonomous vehicles will have high-performance RF antennas. Both radar and RF communication antennas will depend on performance possible with circuit materials from Rogers Corporation.

High-performance circuit laminates, such as RO3000® and RO4000® series materials, are already well established for radar antennas in automotive collision-avoidance radar systems at 24 and 77 GHz. To further enable autonomous driving, higher performance GPS/GNSS and V2X antennas will be needed, which can benefit from the cost-effective high performance of Kappa™ 438 and RO4000 series materials. These antennas and circuits will count on the consistent quality and high performance of circuit materials from Rogers.

Material	Features
RADAR	
RO3003G2™ Laminates	Lowest insertion loss and most stable electrical properties for 77 GHz antennas
RO4830™ Laminates	Cost-effective performance for 77 GHz antennas
RO4835™ Laminates	Stable RF performance for multi-layer 24 GHz antennas
ANTENNA	
RO4000 Series Circuit Materials	Low loss, FR-4 processable and UL 94 V-0 rated materials
Kappa™ 438 Laminates	Higher performance alternative to FR-4



To learn more visit:
www.rogerscorp.com/autonomousdriving

ROGERS CORPORATION
Advanced Connectivity Solutions

USA - AZ, tel. +1 480-961-1382 • EUROPE - BELGIUM, tel. +32 9 235 3611 • **www.rogerscorp.com/acs**

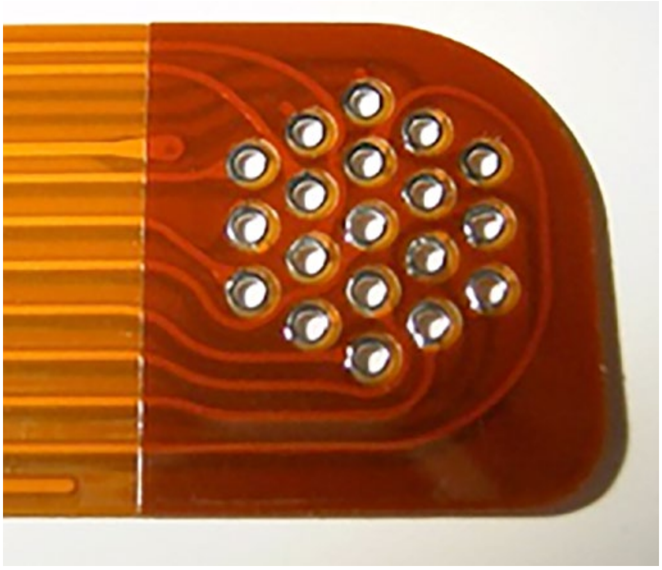


Figure 2: A 5-mil double-sided, thick cover coat used as a stiffener on a 1-mil very flexible circuit.

is too small or the number of flexing cycles is too large. A transition area from hard mount, no-flex stiffener to fully flexible will extend the life of a flex circuit. The viscoelasticity of many 3M adhesives and tapes improves resistance to vibration fatigue by imparting flexibility to a joint or bonded area.

Many modern flex ZIP FFC connectors use a self-adhesive tape wrap or a solid mount with screws, which then slip into a connector soldered to the rigid PCB.

Typically, flex stiffeners are added after the panel is laser routed or die cut, using 3M double-sided tape, epoxy glue or a lamination process using a glue sheet prepreg. The FR-4 stiffeners will be routed out of the desired thickness of FR-4. The adhesive glue sheet is applied to the flex and the two parts aligned, using etch marks or silkscreen marks, and pressed together. The double-sided acrylic adhesive tape will securely attach the two parts together. This method also allows for the stiffeners to overhang the flex area and provide mounting holes or points, as in Figure 1.

In addition to pressure sensitive tape, the parts can be thermally bonded with heat and pressure in a lamination press as a large panel. The typical method is to thermally bond the stiffener to the circuit with the same flexible adhesive that is used to attach the cover lays. Using high heat and pressure, then gluing the parts together, this method will result in a very strong, permanent stiffener bonding. However, using lamination as a method to glue stiffeners is limited to larger panels with a large number of stiffeners, which do not extend past the flex circuit. The flex final circuit and stiffener will have to be routed or laser cut out of the main panel.

Bonding with pressure sensitive, adhesive tape is the most used attachment method available. The attachment is done with PSA (pressure sensitive adhesive) material such as 3M/Tesa Tape, 3M 467MP or 3M 9077. There are over a hundred different types of PSA available from 3M alone. The specific PSA will depend on whether the flexible PCB will be subjected to a high temperature reflow cycle. The type of adhesive used will also depend upon the configuration and/or location of the stiffeners. If a stiffener extends past the flex circuit side, in most cases, it will require a PSA attachment with a locational stiffener outline added to the copper etching data or silkscreen to allow for accurate placement. The flex and stiffener can both be accurately drilled with alignment holes, fitted over a small jig with alignment pins and pressed together with PSA very accurately. Simply press the adhesive side down onto the flex and then peel off the poly-coated kraft paper lining, aligning the stiffener into the pins and applying pressure to secure together.

The PSA creates a very thin, aesthetic bond line caused by glue squish-out between the flex and the stiffener. The stiffeners can be laminated by hand or in an automation setup. The adhesive sheets are easy to die-cut, and CNC rout, as well as laser-cut. The PSA is a simple, easy-to-use, long life, industry leading solution for attaching stiffeners. 3M 467MP is the visually clearest of the 3M transfer tapes, and offers a neat, precise application with no mess or waste when applied to flat surfaces. It has excellent shear strength, as well as high temperature and chemical resistance, making it an ideal choice for stiffener attachment. The PSA tape features 3M high-performance, acrylic adhesive 200MP, an adhesive thickness of 2.0 mils (0.05 mm) and a poly-coated kraft paper liner of 4.2-mil (0.11 mm) thickness.

The 3M 9077 double-sided, pressure sensitive, adhesive tape is a high-temperature adhesive and release liner system that is easy to use. This is the recommended tape to be used for reflow soldering, as it survives short-term exposure to 500°F/260°C for lead-free solder reflow. The 9077 tape maintains high adhesive strength in high temperatures, and is excellent for heat intensive processing, as well as in high temperature attachments to heat sinks. It is ideal for flexible printed circuit attachments to clean metal surfaces with excellent holding power and low outgassing. 3M9077 PST features a 2-mil 3M ultra-high-temperature acrylic adhesive 100HT, a 3.6 mil (.05 mm) clear, heat-resistant, non-woven liner adds stability for die-cutting and converting. The long-term temperature rating is 300°F (150°C) and it is suitable for automotive under-hood applications.

Aluminum is also used as a stiffener with an insulator cover coat strip. This allows for hard mounting to a flat surface other than a PCB. In some applications, thermally conductive material, aluminum, or copper heat conductive laminate can be used to manage heat while providing a hard mounting point for the flex circuit. FLEX007

John Talbot is president of Tramonto Circuits. To read past columns or contact Talbot, click here.

Researchers Develop More Efficient Two-sided Solar Cells

To increase the performance of solar panels, a team of researchers based in Saudi Arabia, Italy, Germany and Canada has created a bifacial, or two-sided, tandem solar cell. The prototypes bring together the best of two separate technologies: silicon and perovskites.

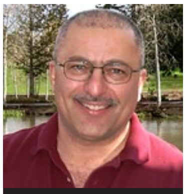
Out in the field, light primarily comes directly from the sun. Conventional tandem solar cells already convert this light into electricity more efficiently compared to traditional silicon-only solar cells by absorbing additional wavelengths of light.

Now, researchers at the University of Toronto's Faculty of Applied Science & Engineering—with colleagues at King Abdullah University of Science and Technology (KAUST), the University of Bologna and the Karlsruhe Institute of Technology—have realized that even more energy can be gathered using a two-sided tandem configuration. Light reflected and scattered from the ground—known as "albedo"—can also be collected to significantly increase the current of a tandem solar cell.

"By exploiting the albedo, we can now generate currents higher than in conventional tandems, without increasing the manufacturing costs at all," said Michele De Bastiani of KAUST, co-lead author of this study. (Source: University of Toronto)

A bifacial perovskite/silicon solar cell prototype is field-tested at King Abdullah University of Science and Technology in Saudi Arabia. (photo courtesy of Michele De Bastiani)

1 Wild River ISI-56 Platform Accelerates SerDes Testing ▸


Al Neves

I recently spoke with Al Neves, founder and CTO of Wild River Technology, about the release of their new ISI-56 loss modeling platform. Al explains why it was so critical that this tool meets the stringent requirements of the IEEE P370 specification (which he helped develop), and why he believes this is currently the best tool for SerDes testing and characterization.

2 Connect the Dots: The Case for Expansive Parts Libraries ▸

PCBs are the foundation of every electronic device, the home for the components that make up your assembly. Those integrated circuits, connectors, headers and passives are what makes it function. How it needs to function determines whether standard components alone can make it work.

3 A Library Management Cautionary Tale ▸

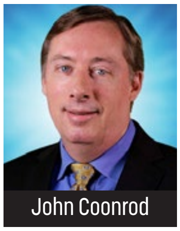

Stephen Chavez

The library management of footprints, land patterns, or cells is one of the most critical items in the foundation of any PCB or CCA design. When I was asked to write an article on this topic, so many thoughts and experiences instantly flooded my mind. After 30+ years of designing PCBs throughout the industry, I have my share of experiences and stories about footprints. One particular experience stands out.

4 Gerber File Option Added to Sunstone's PCB123 Order Process ▸

Sunstone Circuits, printed circuit board (PCB) solutions provider for prototypes, medium volume and production quantities has announced a better way to get free Gerber files when placing a PCB order through the PCB123 Free CAD Tool.

5 Lightning Speed Laminates: Things to Consider When Creating a Circuit Material Library ▸


John Coonrod

A circuit material library for the fabricator can be advantageous for multiple reasons. Sometimes these libraries are intended to be used for electrical predictions, such as impedance, insertion loss or other issues. Other times the information found in the circuit material libraries are used to assist with thermal issues, potential reliability concerns, circuit construction stackups and some processing issues.

6 Footprints: A Distributor's Perspective ▸


Geof Lipman

No issue on footprints and library management would be complete without input from a component distributor. I recently interviewed Geof Lipman of Octopart; as director of operations for part data, he's one of the brains behind the entire site. Geof explains how Octopart functions and manages millions of component data points, and he also discusses the current landscape of electronic components.

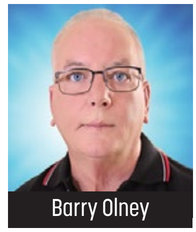
7 Elementary, Mr. Watson: The Printed Circuit Board Design ▸

John Watson addresses continuous improvement by examining the PCB design process.

8 Customers Experience Speed Increases of up to 80% on PCB Designs Using Pulsonix 11.0 ▸

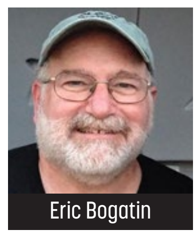
Pulsonix has announced the release of Pulsonix version 11.0, which offers significant speed improvements by using the latest technologies available, as well as the introduction of new functionality for high-speed designs.

9 Beyond Design: Stackup Planning—Three Decades of Innovation ▸


Barry Olney

Materials used for the fabrication of multilayer PCBs, absorb high frequencies and reduce edge rates thus putting the materials selection process under tighter scrutiny. Ensuring that your board's stackup and impedances are correctly configured is a good basis for stable product performance.

10 Seven Tips for Your Next Stackup Design ▸


Eric Bogatin

Rarely do we have the luxury of designing a board just for connectivity. When interconnects are not transparent, we must engineer them to reduce the noise they can generate. This is where design for signal integrity, power integrity and EMC—collectively high-speed digital engineering—are so important. Eric Bogatin offers seven tips for stackup design.

Career Opportunities

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

CAD/CAM Engineer

Summary of Functions

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

Essential Duties and Responsibilities

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

Organizational Relationship

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

Qualifications

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

Physical Demands

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

apply now

Circuit Engineering

Planning Engineer

Experience

- Minimum of 5 years' working within printed circuit board manufacturing industry

Responsibilities

- Review Gerber data and talk with the customer when necessary
- Create production traveler based on Gerber data to release the order
- Improve process capability, yields and cost while maintaining safety and improving quality standards
- Work with customers in developing cost-effective production processes

Quality Engineer/Manager

Experience

- Minimum of 2 years' working within printed circuit board industry
- Possess working knowledge of the IPC requirements and submitting PPAP reports
- Should have knowledge of working with the A16949 certification

Responsibilities

- Perform defect reduction analysis and activities
- Participate in the evaluation of processes, new equipment, facility improvements and procedures

Sales Associate/Customer Service

- Should have a minimum of 2 years' experience
- Salary plus commission

All positions will be on location at Circuit Engineering, 1390 Lunt Ave., Elk Grove Village, Illinois, not remote!

Contact: Felix Simon: (847) 867-7942

apply now

Career Opportunities

Advanced Circuitry International

We're Hiring!

Atlanta Georgia Facility

ADVANCED CIRCUITRY INTERNATIONAL is a world class supplier of RF/microwave and antenna PCBs. We have four state-of-the-art facilities on three continents to serve our customers. From rapid prototype development to large scale production ramp-ups, we supply many notable OEMs and EMS companies around the world.

As we are anticipating rapid growth for 2021 and beyond, we are recruiting for the following positions:

- Manufacturing manager
- Process engineering
- Sales and business development
- Maintenance management

Qualifications:

- 5-10 years' experience working in the PCB industry
- The ability and drive to learn about our unique product offering
- Excellent written and oral communication skills
- Strong, honest work ethic
- Degree in engineering, operations management, or related field preferred but not required

What We Offer:

- Excellent salary and benefits commensurate with experience

If you want to be part of the upcoming ***5G*** revolution and the growth in RF/microwave and antenna PCB manufacturing, consider a career at **ACI**. Were located in the Northern Atlanta suburbs, where you will enjoy a moderate climate, affordable housing, low taxes, quality school systems and numerous recreational opportunities. Please send your resume in confidence to: Career@aciatlanta.com

apply now

SUMMIT INTERCONNECT

Our Summit Anaheim, CA, division currently has multiple open positions for planning engineers.

The planner is responsible for creating and verifying manufacturing documentation, including work instructions and shop floor travelers. Review lay-ups, details, and designs according to engineering and customer specifications through the use of computer and applications software. May specify required manufacturing machinery and test equipment based on manufacturing and/or customer requirements. Guides manufacturing process development for all products.

Responsibilities:

1. Accurately plan jobs and create shop floor travelers.
2. Create documentation packages.
3. Use company software for planning and issuing jobs.
4. Contact customers to resolve open issues.
5. Create TDR calculations.
6. Assist in the training of new planning engineers.
7. Review prints and purchase orders.
8. Create stackups and order materials per print/spec.
9. Plan jobs manufacturing process.
10. Institute new manufacturing processes and or changes.

Education/Experience:

1. High school diploma or equivalent
2. Minimum five (5) years' experience in the printed circuit board industry with three (3) years as a planning engineer.
3. Must be able to cooperate and communicate effectively with customers, management, and supervisory staff.
4. Must be proficient in rigid, flex, rigid/flex, and sequential lam designs.

apply now

Career Opportunities

Career Opportunities

Career Opportunities

Career Opportunities

Career Opportunities

Career Opportunities

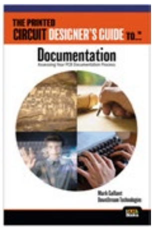
I-007e EDUCATIONAL RESOURCE CENTER

Learn from the Experts in Our On-demand Video Series

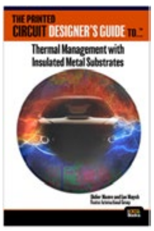
NOW AVAILABLE:
***Implementing "Digital Twin" Best Practices From Design Through Manufacturing with Expert Jay Gorajia*, a 12-part micro webinar series.**

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

Thermal Management: A Fabricator's Perspective
by Anaya Vardya, American Standard Circuits
Beat the heat in your designs through thermal management design processes. This book serves as a desk reference on the most current techniques and methods from a PCB fabricator's perspective.

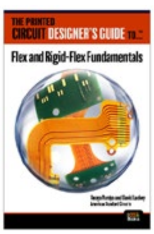


Documentation
by Mark Gallant, Downstream Technologies
When the PCB layout is finished, the designer is still not quite done. The designer's intent must still be communicated to the fabricator through accurate PCB documentation.



Thermal Management with Insulated Metal Substrates
by Didier Mauve and Ian Mayoh, Ventec International Group
Considering thermal issues in the earliest stages of the design process is critical. This book highlights the need to dissipate heat from electronic devices.

Fundamentals of RF/Microwave PCBs
by John Bushie and Anaya Vardya, American Standard Circuits
Today's designers are challenged more than ever with the task of finding the optimal balance between cost and performance when designing radio frequency/microwave PCBs. This micro eBook provides information needed to understand the unique challenges of RF PCBs.



Flex and Rigid-Flex Fundamentals
by Anaya Vardya and David Lackey, American Standard Circuits
Flexible circuits are rapidly becoming a preferred interconnection technology for electronic products. By their intrinsic nature, FPCBs require a good deal more understanding and planning than their rigid PCB counterparts to be assured of first-pass success.

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

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

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

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

ASSOCIATE EDITOR: **MICHELLE TE**
michelle@iconnect007.com

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

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

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

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

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

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

TECHNOLOGY SPECIALIST: **BRYSON MATTIES**

COVER: **SHELLY STEIN**

COVER IMAGE: **ADOBE STOCK © IURIIMOTOV**

DESIGN007 MAGAZINE

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

March 2021, Volume 10, Number 3
DESIGN007 MAGAZINE is published monthly,
by BR Publishing, Inc.

ADVERTISER INDEX