SEPTEMBER 2023

I-Connect007

DESIGN007 MAGAZINE

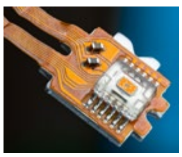
RIGID-FLEX DESIGN

DESIGN007 MAGAZINE

Rigid-flex: Designing in 3D

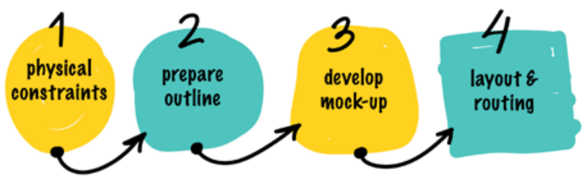
In this month's issue, our expert contributors share their best tips, tricks and techniques for designing rigid-flex circuits. If you're a rigid board designer considering moving into the 3D world of rigid-flex, this issue is just what the doctor ordered! And, as Kris Moyer says in his feature, "Don't fear rigid-flex."

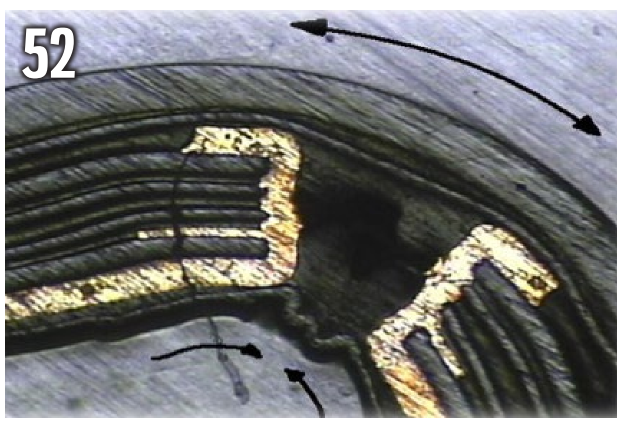
FEATURE INTERVIEWS



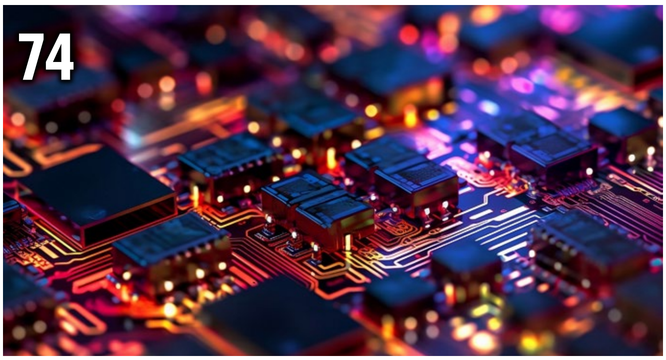
FEATURE ARTICLES

FEATURE COLUMNS





DESIGN007 MAGAZINE



SHORTS

COLUMNS

ARTICLES

HIGHLIGHTS

DEPARTMENTS

Rigid-flex Design: No Longer a Niche

The Shaughnessy Report

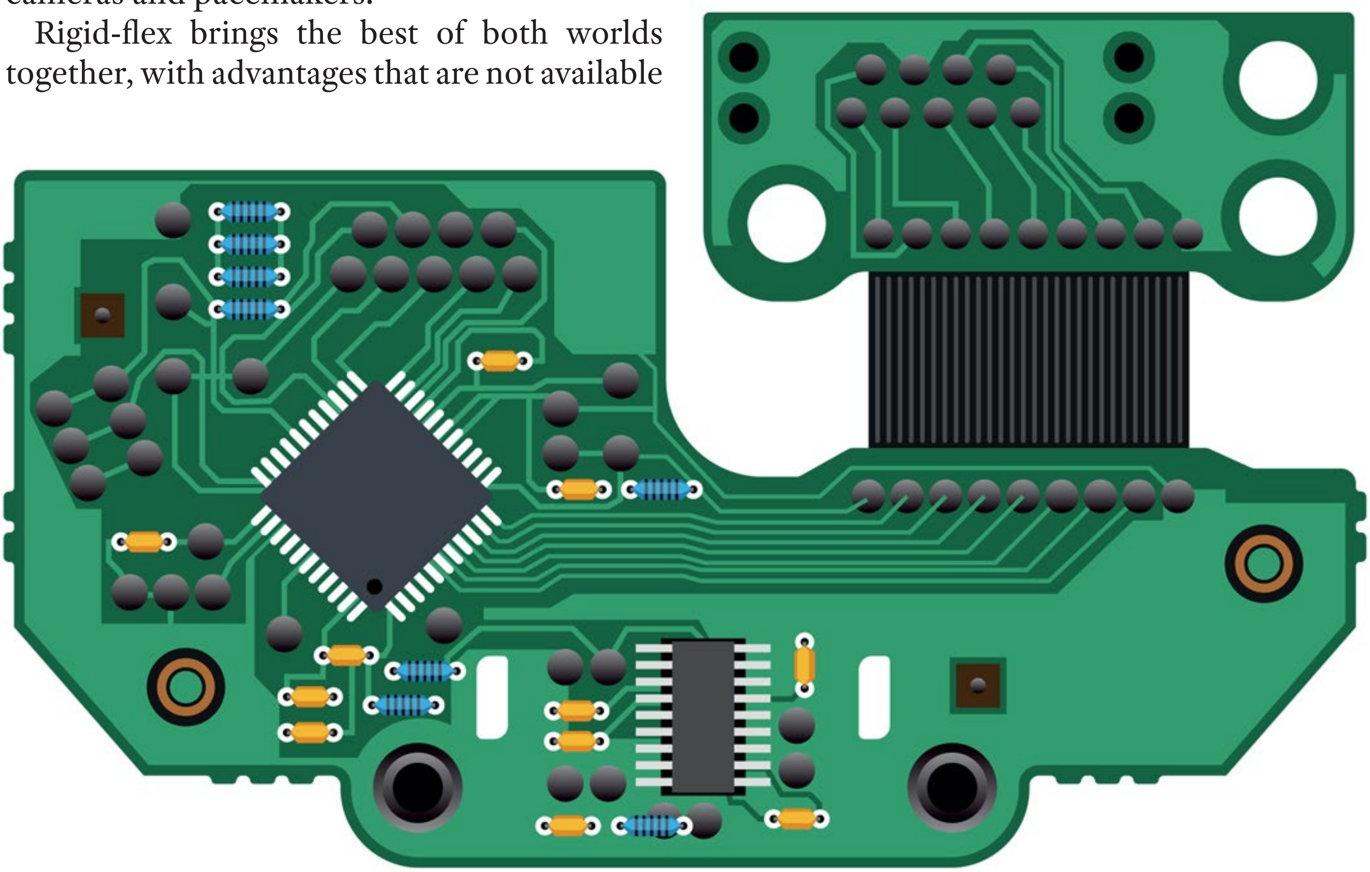
by Andy Shaughnessy, I-CONNECT007

When I first started covering this industry in 1999, rigid-flex circuits were considered a niche market. In fact, Compaq was deemed a trendsetter for using rigid-flex in its laptops.

Now, analysts at Credence Research predict that the rigid-flex market will top $5 billion by 2026, led in part by the adoption of IoT and the need to connect a variety of smart devices. That's a pretty big niche. Rigid-flex circuits are everywhere, from servers and smartphones to cameras and pacemakers.

Rigid-flex brings the best of both worlds together, with advantages that are not available with rigid or flex circuits alone. It cuts down on the number of connectors and wiring harnesses, and can fit into spaces where multiple rigid boards could never fit. Fewer connectors and solder joints means much higher reliability, as well as greater resistance to shock and vibration. Testing is simpler with rigid-flex than with a multi-board construction.

As a result, more traditional PCB designers are looking into rigid-flex design. But rigid-



flex also brings challenges for designers of traditional rigid boards. Stackup design is a whole different animal with rigid-flex, especially with multiple rigid boards. Signal integrity analysis can be a real challenge here, and proper DFM practices are critical. Your data package is more important than ever. Are you conveying your design intent to your fabricator? Are you communicating with your fabricator during the design cycle?

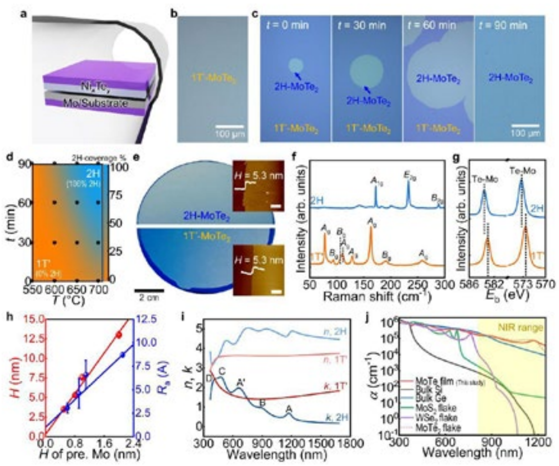
This month, we asked our expert contributors to share their best tips, tricks, and techniques for designing rigid-flex circuits. We start with a conversation with IPC instructor Kris Moyer, who explains the high points of his rigid-flex design class, while throwing in some DFA tips as well. Vern Solberg provides DFM techniques for flex and rigid-flex circuits. Bill Hargin details the finer points of designing stackups for rigid-flex. Mike Morando discusses what OEMs should consider when selecting a rigid-flex fabricator.

Tim Haag examines the often-overlooked correlation between human ingenuity and rigid-flex technology, and Cherie Litson offers a great set of rigid-flex design guidelines. Joe Fjelstad explains how to "unlock" some of the perhaps unseen benefits related to rigid-flex circuits. We also have a column by Matt Stevenson, and articles by Anaya Vardya (who begins a new series on the fundamentals of UHDI), Brad Griffin, and Léa Maurel of ICAPE, talking about sourcing diversification.

We're coming up on trade show season, and I-Connect007 will be covering PCB West and SMTA International in the next few months. Hope to see you there. **DESIGN007**

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

Fabrication of P-Type 2D Transistor Arrays with Van Der Waals Semi-metal Electrodes



Professor Soon-Yong Kwon in the Department of Materials Science and Engineering and the Graduate School of Semiconductor Materials and Devices Engineering at UNIST, in collaboration with Professor Zonghoon Lee, has embarked on a pioneering research endeavor focusing on the development of high-performance p-type semiconductor devices, utilizing molybdenum ditelluride (MoTe2)—a compound renowned for its unique properties. This pioneering technology holds great promise for application in the next-generation complementary metal oxide semiconductor (CMOS) industry, where ultrafine technology is crucial.

In this research endeavor, spearheaded by Professor Kwon's team together with Professor Lee's team, they focused on developing high-performance p-type semiconductor devices utilizing MoTe2—a compound known to exhibit unique properties. By employing chemical vapor deposition (CVD) techniques that facilitate thin film formation through chemical reactions, the researchers have successfully synthesized large-area 4-inch MoTe2 wafers with remarkable purity.

This study has been jointly carried out by Professor Soon-Yong Kwon (co-corresponding author), Professor Zonghoon Lee (co-corresponding author), Dr. Seunguk Song (co-first author) from the University of Pennsylvania, Dr. Aram Yoon (co-first author), and Sora Jang (co-first author). The results have been published ahead of their official publication in the online version of Nature Communications on August 7, 2023.

(Source: UNIST)

An Overview of Rigid-flex Design

Feature Interview by the I-Connect007 Editorial Team

For this month's issue on rigid-flex design, we spoke with instructor Kris Moyer, who teaches the IPC class "PCB Design for Flex and Rigid-flex Boards." In this wide-ranging interview, Kris breaks down the hurdles facing rigid-flex designers and offers a variety of solutions for rigid board designers taking on their first rigid-flex circuits. He also provides a few horror stories to illustrate what happens if you don't follow sound design practices, rules, and standards. And, as Kris points out, "Your fabricator is your friend."

Andy Shaughnessy: What are you trying to get across to the students when you're teaching rigid-flex design? What are the biggest challenges?

One of the biggest concepts I want to get across to students is that there are a lot of untapped, unforeseen, intangible benefits to rigid-flex, both from a design point of view and a total cost of ownership point of view. Rigid boards are usually connected with wire harnesses or cables, and we know that wire harnesses and cables are the largest point of failure in the system. Wires and cables tend to break, especially at the solder joints or pins. If we can remove all those harnesses and integrate all of that into the structure of the board, we can vastly reduce the failure rate of our designs.

If you choose to go rigid-flex, there is an initial cost expenditure: added processes, time, complexity, and all of that. But let's say I have a design with three circuit boards that would

need two cable harnesses, one from board one to board two, and one from board two to board three. That's a grand total of five different item numbers I would need for each board. I would have three drawing numbers: a number for the schematic, a number for the board, and a number for the assembly. For three boards, that's nine drawings I must maintain, and then another two drawings per cable, and let's assume two cables. That's a total of 13 separate drawings and part numbers I have to maintain in my system.

Now, if I take all of that and integrate it into a single rigid-flex design, it's much simpler to maintain. Before, when you had three different circuit boards and had to do a revision to one of the boards, you had to go through the process to make sure that the new revision of one board works 100% correctly with the old revision. I've seen that in my career multiple times.

But with rigid-flex, it's all one design. It helps with packaging because you want to get smaller and smaller packaging. Again, no more harnesses, cables, or connectors, and it's much easier to deal with.

Of course, material selection is much more complex, and there are a lot more manufacturing challenges, such as specifying rolled annealed copper for flexibility. We have to understand the stresses on copper. If I bend it, what are the bend radii and ratios that I have to deal with? I don't want to over-bend the copper and fracture it.

There's no woven reinforcement in the flex materials and you have a much higher amount of layer-to-layer misregistration, shift, and float during the lamination and bonding processes. I can't just take my part from a library that's good for a rigid board, throw it on flex, and expect that I'll get the same manufacturing producibility levels out of these boards. Again, these are all challenges you have to talk to your fabricator about as well. If we have a bunch of traces cross the flex section, we must be careful. Let's say I put one trace on top of the flex, and one on the bottom of the same flex core, so they're directly on top of each other. Now when I bend, one trace is put under compressive load, and I can have stress fractures and so on. We need to offset our traces from each other. Basically, I'm trying to get the students to understand the very high benefit you get from using rigid-flex, but also the gotchas, challenges, and pitfalls. It's not as simple as throwing a bunch of parts on a board and routing it.

I'm trying to get the students to understand the very high benefit you get from using rigid-flex, but also the gotchas, challenges, and pitfalls.

Shaughnessy: Your classes spend some time on rigid-flex tradeoffs. There seem to be quite a few.

There are multiple different factors that go into the break point of whether you should go rigid-flex, or discrete boards with flat cables, or traditional discrete boards. But I think the days of doing everything on discrete boards with traditional cable harnesses is fast going away. The fact is that cables are the single greatest point of failure in any modern system.

Happy Holden: One thing that seemed clear to me is that rigid-flex, as a design alternative, takes somebody with a lot of knowledge of all these different trade-offs. We need a complex tradeoff tool, where you can talk about each of your boards, each of your connecting harnesses and boundary

conditions, such as size, to tell you whether a rigid-flex is practical. But it may be too complicated yet for that kind of a tradeoff costing program.

Exactly. That's why I created a class to teach it. For example, whenever I bend a material of some thickness around a radius, the inside radius of the band is put under compressive load and the outside radius of the band is put under tension or tensile load. The compressive load is now hardening and forcing my atoms closer together, and if I exceed the breaking strength, the tensile loads can actually tear the copper like taffy.

So, we want to minimize the stresses on those bending points. Whether we're doing single-sided or double-sided flex cores, each flex core has its own coverlay and protection and I don't bond them together. The fabricator can extend the length of each subsequent layer of flex. We need to know how the bend is going to be, such that when each layer bends around that corner, let's say 90 degrees, each one is under the minimal amount of stress and we don't have added incurred stress. Yes, there are multiple different stackup issues we have to deal with there.

Shaughnessy: It sounds like every rigid-flex is almost a custom project every time, and communication with the fabricator is really important.

You're right. One of the mantras that I say in all of my classes is, "Your fabricator is your friend." I say with all designs, even if it's a rigid design you've done 100 times, always double-check with your fabricator, because there might be something subtle in this design that he can help you with. After you build your stackup, you build your controlled impedance, then get your fabricator to sign off on it to make sure that all of their processes, their process controls, and process variations are taken into consideration. That way you improve your success rate, your yield, and first-pass quality of your board design.

Kris Moyer

Shaughnessy: I see that you spend a lot of time talking about manufacturing in your class. What would a rigid board designer need to know about manufacturing processes for rigid-flex?

For rigid-flex, there are quite a number of touch labor steps that you don't normally have in pure rigid. Think about manufacturing a rigid panel, 18 x 24 inches. They do all the imaging, and then they come back after the fact and cut the individual boards. With rigid-flex, in the area where the flex ribbon will be, they have to pre-machine that out of the rigid panels and have an opening there, before the flex is integrated into the stackup. You have a technician who's literally going by touch labor, applying a piece of coverlay material that will be either machined or laser-cut off after it's laminated together. There are so many more touch labor steps in rigid-flex than in pure rigid fab.

Nolan Johnson: This has been very educational for me. I spent a lot of time at Sunstone Circuits working with rigid boards, but not

much rigid-flex at all. What are some of the hurdles in rigid-flex assembly?

When it comes to assembling flex only, you must have some kind of stiffener backer behind the material. If it's just a pure flex material going through the assembly equipment, when the pick-and-place machine is trying to put those ICs on the board, the flexible things move around with rigid-flex. Typically, we'll design around the board some throwaway rigid board material with little tabs that will connect to the individual little rigid boards to hold them in place and restrain them mechanically in the X, Y, and Z axes. Now, if we want to try to do assembly without the designed array, then the assembler has to have a fabricated mechanical pallet of some kind with some method of restraining and aligning all the boards relative to each other again. Assembly can definitely be more challenging.

Shaughnessy: Is the flex material itself still primarily polyimide?

Yes, polyimide is still the most common with acrylic-based adhesives. Acrylic is becoming the more popular adhesive for non-metallic layers, such as the adhesive on the coverlay backing material. Then the coverlay strip that will be on the outer surface replaces solder mask, which tends to be more acrylic-based resins.

Speaking of material, there is actually another technique out there: flexible rigid. Let's say you had your traces on top of a rigid board. They would go in and mill out the material, basically making their original material so thin that you could do a one-time bend to install it in the product. If you need low-cost bending with flexible rigid, you're bending the board into shape and relying on the resin and the copper to act as your material to do a one-time bend like that.

When it comes to assembling flex only, you must have some kind of stiffener backer behind the material.

Shaughnessy: Your class focuses on rigid-flex stackup design for signal integrity. With the whole "mixed media" structure, signal integrity could get pretty interesting in rigid-flex.

Yes, absolutely. There are EE tradeoffs and EE techniques, as well as manufacturing, mechanical, and design tradeoffs involved in planning your rigid-flex stackup. One of the biggest challenges with signal integrity in rigid-flex is the fact that 99% of the time, my transmission line structure changes when I go from the rigid to the flex section. On the rigid board, I'm usually on an internal layer, a stripline structure. As soon as I transition out onto the flex structure, it's typically a single layer or one plane below it. Now we're microstrip, so now all my geometries need to change, and I have to set up the rules accordingly. Now I microstrip across the flex and then turn back into stripline when I get to the other side.

Shaughnessy: What advice would you give a rigid board designer who's just getting into rigid-flex design?

First, take my class. Don't fear rigid-flex. It has many benefits and value-adds to your design and your system that you might not see initially. Keep your options open. Consider all of your options in order to get the optimal solution for your design.

Shaughnessy: Thanks, Kris. This was really good.

Thank you. DESIGN007

Rigid-flex DFA

Feature Q&A With Kris Moyer

Rigid-flex assembly brings its own set of issues, but designers can do quite a bit to make things easier on their downstream brethren. We asked IPC instructor Kris Moyer to give us the lowdown on DFA for rigid-flex circuitry.

Kris, what are some of the unique challenges related to assembling rigid-flex circuits?

There are several challenges to assembly of rigid-flex boards. First there is the alignment issue of the different sections of the boards. With a rigid board, the individual boards can be removed from the fabrication panel and run through the SMT assembly line without much problem, as long as the board is of sufficient size. In this case, the relative position of all the individual parts on the board remains consistent. But with rigid-flex, after the board is removed from the fabrication panel, the individual boards can move and rotate relative to each other within the limits of the flex material. This makes the placement of parts difficult to control. To overcome this, the board must be held in a restrained condition throughout the assembly process. This will require either the fabrication of a custom fixture to hold the board restrained during assembly or the design of temporary removable restraining mechanisms (rails) into the design of the board.

How does rigid-flex assembly differ from rigid or flex assembly?

The rigid-flex assembly requires the board be held in a restrained state during assembly. Additionally, if the intent is to place parts on both the rigid sections and the flex sections there is a need to provide mechanical support under the flex sections during the placement to prevent deflection of the flex material during placement. Furthermore, if placement is to occur on the flex sections the Z-axis height difference between the rigid section surface and the flex section surface needs to be considered during the pick-and-place programing to ensure proper parts placement.

What do designers need to know about placing components on rigid-flex, especially near bend areas?

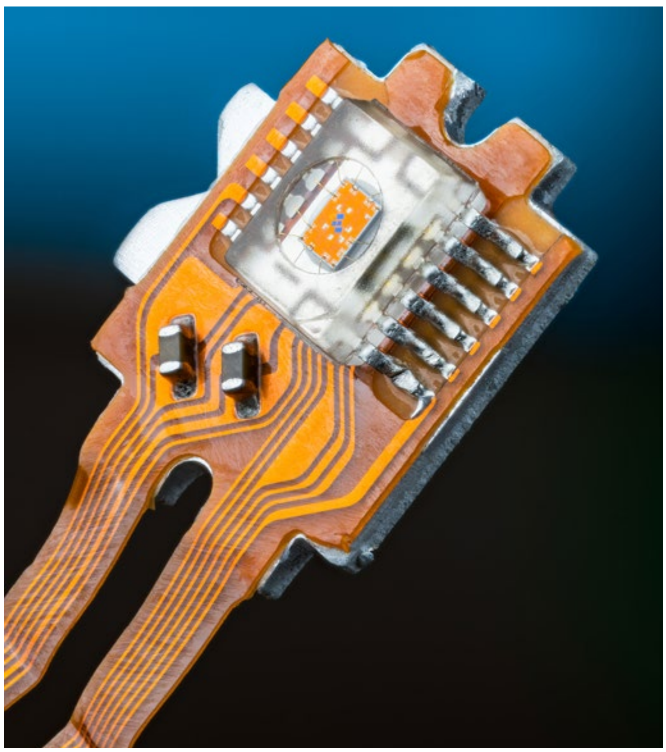
When placing parts on rigid-flex, the designer should use the same placement rules on the rigid sections as they would on a rigid-only board, with the exception of the rigid-to-flex transition areas. The placement guidelines should follow what is defined in IPC-2221, IPC-2223, IPC-6013, and IPC-J-STD-001. No parts should be placed in the flex transition area and I recommend adding additional margin to the clearance around the flex transition area.

What DFA tips would you give designers who are new to working with rigid-flex?

My main tip for good DFA for rigid-flex is to follow the IPC design, fabrication, and assembly standards for rigid-flex designs—IPC-2221, IPC-2223, IPC-6013, and IPC-J-STD-001.

Is there anything else you'd like to add?

While there are several unique challenges to rigid-flex design and assembly, the benefits of successful rigid-flex designs to the customer will outweigh the technical challenges and hurdles facing the designer in attempting to implement a rigid-flex design. **DESIGN007**



Rigid-flex Stackup: It's a 3D World

Feature Interview by Andy Shaughnessy
I-CONNECT007

Z-zero founder Bill Hargin has been studying stackup design techniques for years. He developed the company's PCB stackup planning software, and he wrote *The Printed Circuit Designer's Guide to… Stackups: The Design within the Design.*

In this interview, Bill shares his thoughts on designing rigid-flex stackups, the challenges they bring, and what rigid board designers need to know about designing stackups in 3D. "Flexperts" Mark Finstad of Flexible Circuit Technologies and Nick Koop of TTM Technologies also offer insight into the many tradeoffs that rigid-flex designers face.

Andy Shaughnessy: You literally "wrote the book" on stackup design techniques. What are some of the unique challenges designers face with rigid-flex stackups?

As I often say, the mechanical world and the electrical world are at odds with each other. We wouldn't need to concern ourselves with signal integrity if the physical/mechanical worlds weren't involved.

The rigid-flex structure carries a lot of advantages—reliability, dynamic flexure, and the ability to get things done in tight spaces. But there's the additional burden of needing to manage a mechanical world that has additional dimen-

sions or "degrees of freedom." A rigid stackup, for example, can easily be viewed in 2D, and that world is relatively easy for most electrical hardware engineers to understand and manage. You have impedance, frequency, and loss and then physical and electrical parameters that drive them. When flex substacks come into the picture, the fact that the flex portion needs to bend turns it into a 3D mechanical engineering problem that takes some time to learn. This is where you really need to lean on a flex fabricator's expertise. Toward that end, I decided to poll a few "flexperts," including Mark Finstad from Flexible Circuit Technologies and Nick Koop from TTM. Now I'll weave their thoughts in with some of my own.

Mark emphasized that rigid-flex stackups have to blend both rigid and flexible base materials into the final product, and that these materials all stretch and shrink at different rates due to processes like etching and lamination. Additionally, flex materials can stretch and shrink just due to temperature and humidity changes. To further complicate things, these materials typically don't move at the same rate in the X and Y axes. Most manufacturers have a good history with many materials, so they know how much to scale their artwork and programs to account for these dimensional instabilities. Also, laser direct imaging and smart laser and mechanical drills help with a lot of these issues, but the need to use those processing tools does come at a cost.

What do rigid board PCB designers need to understand about rigid-flex design and stackups?

I don't know if this will catch on, but I've started referring to rigid-flex design as "designing in 3D" or "3D design." I doubt that someone who's been designing rigid-flex boards (in 3D) for 20 years would learn much from my brief commentary here, so I'll throw out a challenge to the rest of the designers who may benefit from this.

Along with HDI, rigid-flex PCBs have been trending up in recent years. Younger engineers would do well to start adding rigid-flex design skills to their repertoire in order to open up career opportunities. Maybe start with the rigid-flex chapter in my book on stackups and branch out from there. For signal integrity and EMI control, we like to have nearby reference planes for the current return path. But that

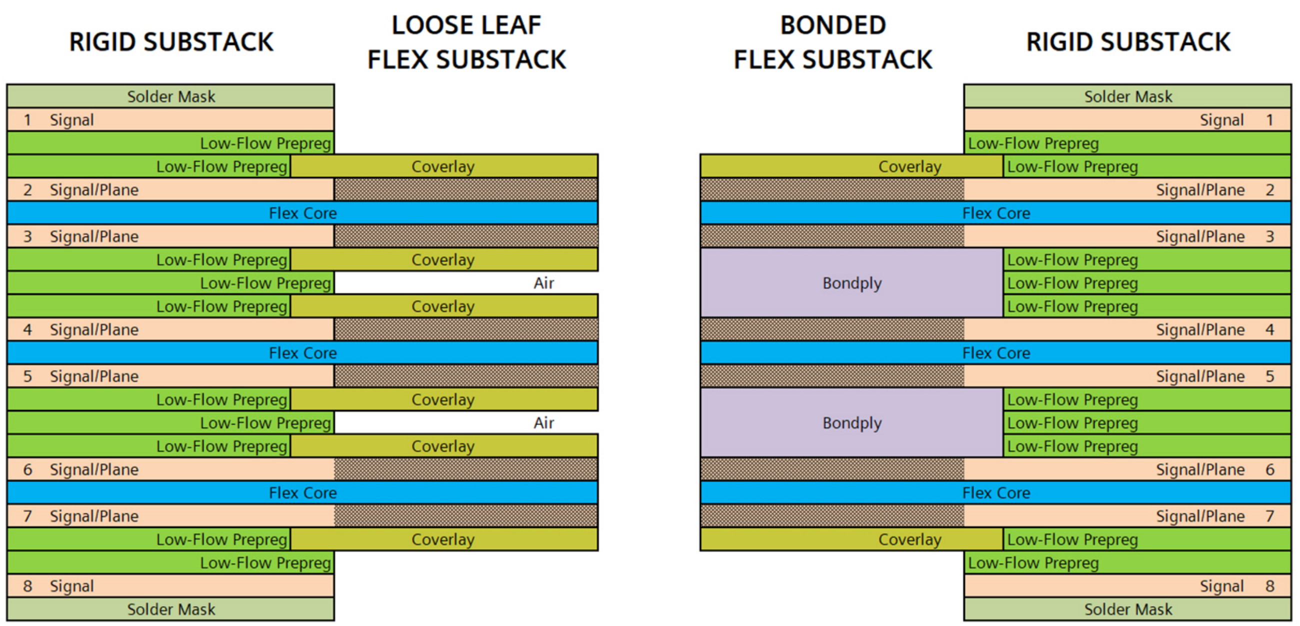


Figure 1: Loose-leaf designs are most flexible for 90-degree and 180-degree bends. They are the simplest to manufacture but not recommended for impedance control. Bonded is less flexible, thus requiring a more generous bend radius.

additional plane layer reduces the bend radius, so there are lots of new tradeoffs to make when you're designing for flexibility.

There are some important differences relative to rigid PCBs that you should consider when designing a rigid-flex project. You can connect flex cores using a loose-leaf or bonded approach (Figure 1).


Bill Hargin

With a loose-leaf design, you have air gaps between individual flex cores, and you get a less expensive and more flexible design. Bonded designs require an additional sublamination with bondply and prepreg, resulting in a stiffer design. Although at least 20x thicker than the loose-leaf design, it improves stripline impedance control because you have tightly controlled current return paths.

Nick mentioned that while impedance requirements may drive thicker dielectrics, thicker materials will result in less flexibility. Also, while any line width can be etched on flex layers, wider traces are mechanically more robust and withstand more bending and flexing. Mark added that if a rigid-flex circuit is designed with only the electrical requirements in mind, you could end up with an electrically perfect rigid-flex with traces that crack when you bend it. So, all those great electrical properties are meaningless if the circuit cannot then be formed to its final configuration without causing damage. Most successful rigid-flex designs are a balancing act to ensure that you make tradeoffs between both the electrical and the mechanical performance.

From a cost perspective, Nick offered that designers should be aware of how rigid-flex boards are fabricated. For example, often not all the layers are needed in all the rigid sections. However, if you remove the copper layers and associated dielectrics from some rigid sections and not others, you will be driving up manufacturing costs. It is cheaper to have all rigid sections using the same stack of materials. You can etch away copper you don't need but keep all the dielectrics.

How does material selection impact stackup planning with rigid-flex?

Before you can select materials, you need to know what the options are. With rigid PCBs, we have cores, prepregs, and solder mask for dielectrics. In the rigid-flex world, cores are replaced by flex cores, prepreg offerings include both "no-flow" and "low-flow" varieties, and there are bondplies, coverlays, and stiffeners.

Copper in rigid-flex designs is adhered to flex dielectrics in various ways. Adhesive may be added where copper is bonded directly onto the base material. Stiffeners are sometimes added to reinforce a flex area for component placement or routing holes. Copper can either be electrodeposited (ED, less flexible, lower cost) or rolled annealed (RA, more flexible, higher cost). Flex planes typically have a hatched pattern etched into them because the reduction in copper makes them more flexible.

Coverlays are flexible materials, typically on the outside of a flex substack. They protect and insulate the flex circuitry on the surfaces, preventing it from lifting. Coverlays are typically made from acrylic, polyimide, or polyester. A typical coverlay construction (Figure 2) has a polyimide dielectric on top and an adhesive on the side facing the copper.

Bondplies are similar to coverlays, but they are used on inner stripline layers, with adhesive on both sides and the polyimide dielectric in the middle. If you were gluing two flex cores together, this is where a bondply would come in.

Flex cores typically span all substacks, carrying the copper from one end to the other. Common brands for flex cores are DuPont Pyralux or Panasonic Felios, with lots of sub-flavors within these two product families. Flex cores without adhesive are used for high-performance rigid-flex applications. Adhesive-based

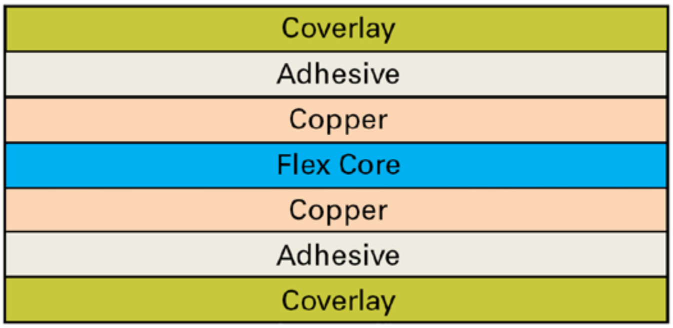


Figure 2: Two conductive layers with a flex core between them, as well as adhesive and coverlay on the outside.

flex cores are less expensive and generally used for single- or double-sided or low-layer-count flex applications.

Low-flow and no-flow prepregs are used on the rigid substacks in lieu of traditional prepregs to keep the resin from flowing onto the flex substack, causing a portion of the flex region to become brittle and creating a potential failure mode. No/low-flow prepregs are typically available in 106 and 1080 glass styles. As the names suggest, low-flow prepregs flow a bit more than no-flow prepregs. Based on my research, low/no-flow prepregs are available from the following manufacturers: AGC-Nelco, Arlon, Isola, TUC, Showa Denko (formerly Hitachi), and Ventec.

Low-flow prepregs are typically stacked one above the other, with the outer ply going to the rigid edge, while the inner layer is recessed by 50 mils so that the coverlay from the flex substack can nest into the rigid board and be held onto by the outer low-flow prepreg (Figure 3).

Flex cores and bond plies, polyimide films, and coverlays don't have glass weaves, but as noted, no-flow prepregs are glass-reinforced. Once you understand the basics, you may be able to have a more meaningful conversation with your fabricator. Nick points out that successful rigid-flex designers need to understand what materials in the stackup exist and where in the part. For example, flex cores extend everywhere throughout the stackup. Conversely, the flexible bondply and coverlay are in the flex regions only, and just extend a short distance into the rigid zones, avoiding any plated through-hole areas. It is important to keep any of the flexible adhesive out of the plated through-holes in the rigid sections as it has a low Tg and high CTE, meaning it creates extra stress on the plated through-holes if present.

Mark Finstad added the following advice on material lead times:

"There are common materials, less common materials, and some downright exotic materials available to build rigid-flex circuits.

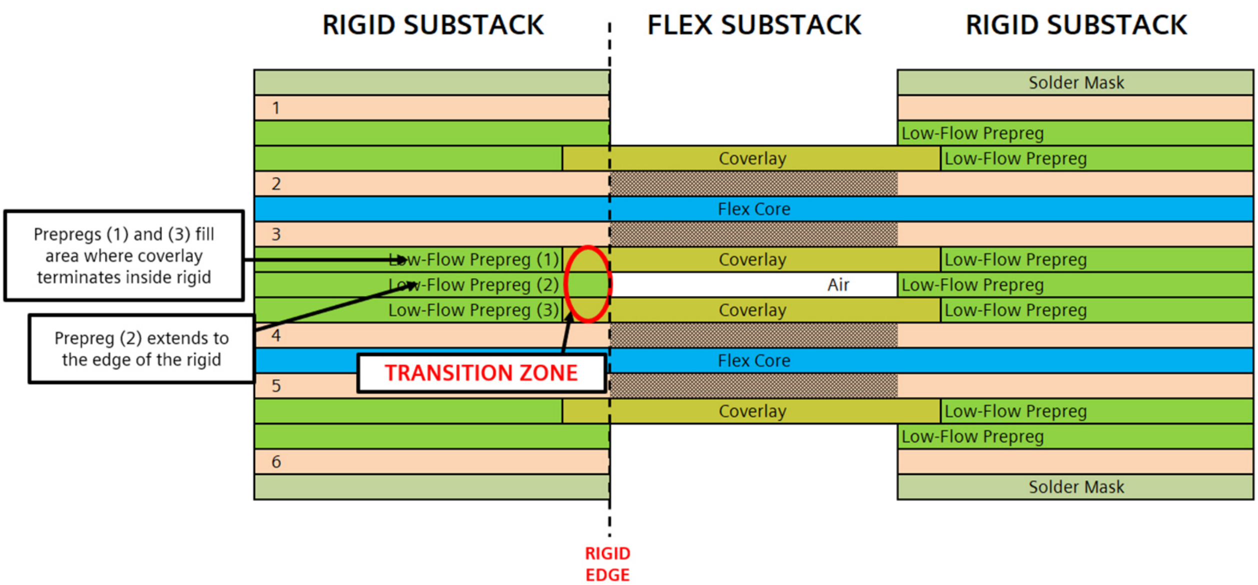


Figure 3: Coverlays from flex substacks are nested within a transition zone in the rigid substack.

Designing with common, mainstream materials will usually ensure that the materials are either on the shelf or can be procured with minimal lead time. If a manufacturer has to order exotic materials that require a two- to three-month lead time (which happens more than you would think), that extra time is going to have to be added to the processing lead time. Most fabricators like to have the materials physically in their facility before they release a job to production, so the material lead time and processing lead time run consecutively, not concurrently. This can lead to a long wait for finished product."

What can designers do to optimize their rigid-flex stackups for easier fabrication?

Joe Fjelstad, author of *Flexible Circuit Technology,* says, "Don't design *for* manufacturing. Design *with* manufacturing." In other words, he's telling us not to go it alone. Where there's a will and enough time and money, a lot of things can be built that maybe shouldn't have been built, or that you could have built with a less expensive alternative. An experienced rigid-flex fabricator is a great resource for this, and if you have enough lead time, you can even get a second opinion from another fabricator.

Mark added that a reputable fabricator will have learned from thousands of successful (and maybe a few not-so-successful) designs over many years. Fabricators can coach you on what to do, and just as importantly, what not to do to get to a successful design.

As I've noted, an entire book was written on the subject, so the list of dos and don'ts when optimizing rigid-flex designs could be quite exhaustive. Mark listed a few of his dos and don'ts for rigid-flex design:

- Use standard materials whenever possible.
- Optimize trace/space sizes where you can. Don't default the entire board to minimum feature sizes if they are only needed in a few small areas. This will increase etch yields.
- Optimize pad sizes. Increasing pad size—even an extra mil or two—can make the fabrication process a lot easier.
- Whenever possible, try to make blind vias 100 µm or larger. This will give you the most reliable plated via connections.
- Use a symmetrical material stackup to minimize warping during reflow.
- Always make each rigid area with the same construction and number of layers, even if you don't need them. It will not save you money to eliminate layers in a single area or even a few rigid areas. On the contrary, it will add significantly to the cost.

Nick contributed his own short list:

- The best rigid-flex stackups are symmetrical to minimize warpage.
- The most successful designs don't have flex layers as the outer-most layers (top or bottom) in the rigid stackup.
- Rigid-flex designs require larger annular rings to account for more variation in material movement.
- Stacking-up dissimilar materials will result in more misregistration than you might be used to seeing in a rigid PCB.

Very good. Is there anything else you'd like to add?

Take a PCB fabricator with experience in rigid-flex manufacturing out to lunch and visit their facility. Better yet, offer to buy them lunch and see if they make a counteroffer to buy *you* lunch. I tried it with Nick and Mark after their class at IPC APEX EXPO and got a pretzel out of the deal. It seemed fitting since we were talking about rigid-flex.

You're making me hungry! Thanks, Bill.

Any time. Please download my free eBook on stackup design; there's a whole chapter on rigid-flex. **DESIGN007**

DFM Principles for Flexible Circuits

Designers Notebook

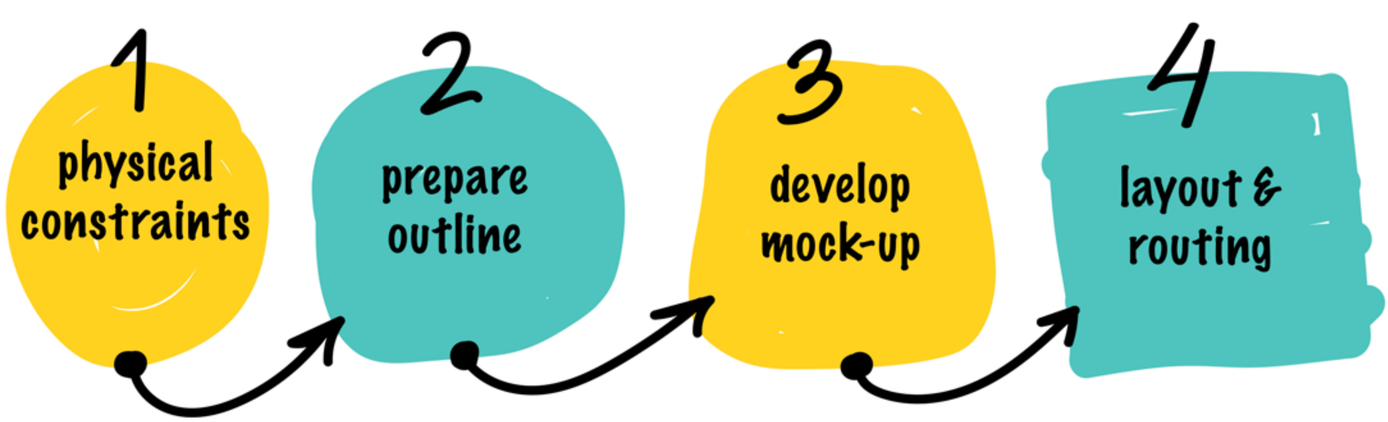
Feature Column by Vern Solberg, CONSULTANT

Flexible circuit applications can be as basic as furnishing electrical interconnect between two conventional circuit board assemblies, or to provide a platform for placing and interconnecting electronic components. During the planning and pre-design phase of the flexible circuit, there will be several material and process related questions that need to be addressed.

Most flexible circuit fabricators welcome the opportunity to discuss their customers' flexible circuit objectives prior to beginning the actual design process. The flexible circuit manufacturer will often furnish guidance that can avoid potential delays in beginning the fabrication process and ensuring satisfaction in the end product. The flexible and rigid-flex circuit development will follow a sequence of events that include timely review cycles, both internally and with the fabrication company selected:

- Determine physical constraints
- Prepare a 2D flat outline
- Develop a model (mechanical mock-up)
- Component layout and circuit routing

The design guidelines for flexible circuits, although similar to rigid circuits, have distinctive differences that are influenced by specific applications, material selection, and the intended operating use environments. The fundamental issues to be defined before begin-



ning the flexible circuit design include several primary elements:

- Quantity of signal traces required across the flexible area
- Conductor widths required for current-carrying capacity
- Spacing required for voltage isolation
- EMI shielding and impedance control requirement
- Determine the surface area required for routing the circuit

Flexible circuit complexity can have a significant impact on manufacturing lead-time and unit cost. Designers commonly create rigid boards with a more uniform rectangle outline, but flexible circuits or the flexible portions of the rigid-flex circuit outline are more likely to be profiled to meet specific application. But before finalizing the outline for the circuit, the designer is advised to prepare a physical model to confirm the dimensions initially established meets expectation. The example shown in Figure 1 represents a flexible circuit outline with holes, slots and openings, and circuit extensions for connectivity with other assemblies.

If the circuit will include surface mount components and/or interface connectors, these areas must be defined on the model as well. To ensure the finished part will hold up to the various stages of assembly and use, the designer must include attributes that will minimize physical damage: inside corners, narrow slots and slits, or in areas accommodating connectors or areas reserved for component mounting.

Figure 1: Flexible circuit outline example.

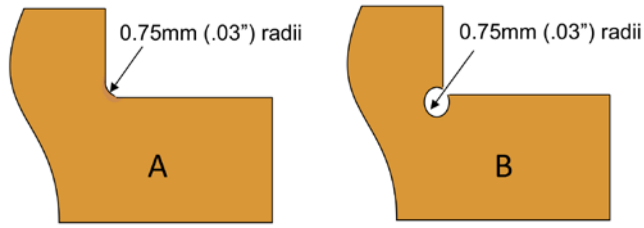


Figure 2: Inside corner tear restraint.

Corner and Slot Feature Development

Avoid sharp corners in all recesses of the flexible circuit. The designer must specify a radius at these locations in order to lessen the potential for cracking or tearing of the flexible material. Providing a radius on the inside corner (Figure 2a) will resist the propagation of tearing. When the inside corner must provide an unobstructed corner profile, the designer can position the radius beyond the corner (Figure 2b).

Manufacturers recommend providing a large radius on all inside corners. This can contribute to increasing part reliability, one that will be more resistant to tearing when twisted.

Regarding "slit" and "slot" features, flexible circuit fabricators recommend that these features be furnished with a hole or full radii where they terminate. For the "slit" features, a 1.5 mm (~0.06") diameter hole at the end of the "slit" typical of the example furnished in Figure 3a, will provide a barrier to minimize tearing when the flexible circuit is physically stressed. When a wider slot feature resembling that shown in Figure 3b is included in the flexible circuit, fabricators recommend a full radius at its end.

To further minimize tear and cracking of the material at the inside corner areas, a narrow pattern of copper foil can be retained that follows the profile of the radii. If the corner or slot is subjected to excessive strain during use, include an additional thickness of the base dielectric in the area to further reinforce the site.

Holes and cutouts in the flexible circuit should maintain a minimum distance of 1.3 mm between the exterior edge of the flex cir-

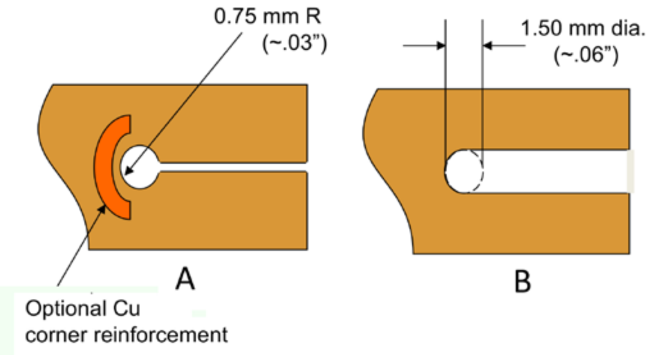


Figure 3: Slot and slit fortification.

cuit outline and edges of interior located holes and cutouts. In addition, cutouts should always maintain a minimum radius of 0.75 mm in each corner, typical of that illustrated in Figure 4.

Circuit Conductor Density

Determining conductor density and the routing area requirement on the flexible portion of the circuit will depend on the number and width established for signal conductors. For both single and two metal flex, the overall width of the base material must also provide adequate surface area for cover-layer film bonding. The general rule for a conductor to the dielectric material's outer edge is a minimum of 0.70 mm (~0.030”) but, where possible, the copper-to-edge distance should be maximized. Additionally, the minimum distance between the exterior edge of the flex circuit and the edges of interior located holes and cutouts should not be less than 1.30 mm (~0.050”). But, typical of the conductor-to-edge spacing noted above, this provides a greater distance where possible. The examples in Figure 5 furnish general guidance for providing clearance between key features of the flexible circuit.

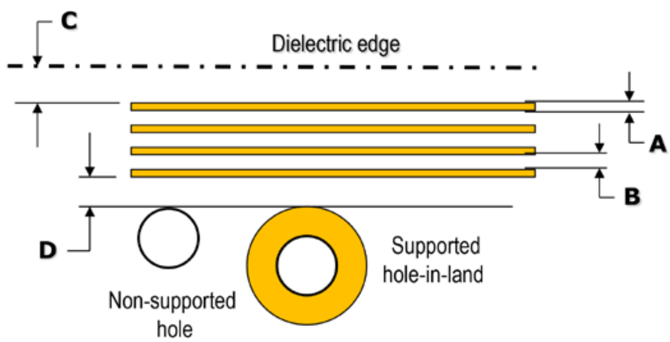
When the conductor path on the flexible circuit must change direction, the designer must avoid sharp, 90-degree turns. Ideally, these conductor paths should be routed with a nested chamfer or generous radii. Multiple conductor paths routed in parallel should also maintain uniform spacing with progressively larger radii from the center axis.

Figure 4: Maintaining distance between circuit edge to holes and cutouts.

Bend Area Consideration

The flexible circuit should not be bent or formed in areas where there is a discontinuity in the cover-layer, termination of plating, or any other stress-concentrating feature. The acceptability of this condition is determined by the thickness of materials, the radius of the bend, and the severity of the operating environment. Flexible circuits designed for dynamic applications may be more susceptible to conductor failure than the bend-to-install or mechanically formed application.

Circuit fabricators recommend that designers avoid unnecessary circuit thickness in the bend zone because it can hinder flexible capabilities. If a portion of the flex circuit needs



Dim.	Copper Foil Thickness 17µm (1/2 oz.)	26µm (3/4 oz.)	34µm (1 oz.)
A	0.13mm (~.005”)	0.19mm (~.008”)	0.25mm (~.010”)
B	0.13mm (~.005”)	0.19mm (~.008”)	0.25mm (~.010”)
C	1.30mm (~.050”)	1.30mm (~.050”)	1.30mm (~.050”)
D	0.50mm (~.020”)	0.50mm (~.020”)	0.50mm (~.020”)

Figure 5: General physical feature clearance guidelines for flexible circuits.

to be thicker for supporting components or connectors, add a stiffener. Suppliers further note that calculating bend-cycles over the projected life of the product is crucial to your design. When the circuit is flexed beyond the limit ratios established, the copper foil may elongate, become brittle, and generate micro-cracks in the bend zone. Guidelines for minimum bending radius for 90-degree "bend-to-install" for fully bonded circuits:

- Single sided, 10:1
- Double sided, 10:1
- Multilayer, 20:1

A rule of thumb for estimating minimum bend radii is 10x material thickness for single-metal layer circuits and 12x for double-metal layer circuits.

For maximum dynamic flex life and reliability, conductors in the bend area should be routed perpendicular to and evenly spaced across the bend area. Furthermore, the designer should not specify additional plating thickness in areas requiring bending or flexibility. Likewise, conductors should maintain a uniform width in the bend area and, when possible, the neutral bend axis should be at the center of the conductor pattern.

For dynamic flex applications that require repeated bending up to 90-degree from flat, the recommended minimum bending radius (R) is:

- For single-sided flex, 100:1
- For double-sided flex, 150:1
- Multilayer dynamic flexing is not recommended

A rule of thumb for estimating minimum bend radii is 10x material thickness for single-metal layer circuits and 12x for two-metal layer circuits.

In summary, for maximum dynamic flex life and reliability, conductors in the bend area should adhere to the following considerations:

- Conductors should be routed perpendicular to the bend area
- Conductors should be uniformly spaced across the bend area
- Do not apply or plate additional metals in the bend area
- Conductors should maintain a uniform width in the bend area
- Do not place SMT components or plated-through via holes in the bend area

Supplier Capability Assessment

In preparation for developing the flexible circuit, the designer must gain a general understanding of the flexible circuit fabrication process variations and review design guidelines furnished by the potential supplier(s). You will find that the capabilities and process methods can vary significantly from one supplier to another. Flexible circuit suppliers, for example, may publish their capability in two classifications: standard and advanced. While their standard level of product will typically use an image, print, and etch process, fabricators with more advanced imaging and pattern-plating capability are able to reduce conductor geometries. For controlling product cost and maximizing process yield, however, the designer may consider the more conservative approach for conductor routing.

Appearance

Vern Solberg will conduct a half-day tutorial course on "Flexible Circuit Design for Manufacturing Principles," Oct. 10, at the SMTA International Conference 2023 in Minneapolis, Minnesota. This course addresses the design, fabrication, and assembly criteria for developing and implementing both flexible and rigid-flex circuits. To register and participate in this timely tutorial event, visit smta.org. **DESIGN007**

Vern Solberg is an independent technical consultant specializing in SMT and microelectronics design and manufacturing technology. To read past columns, click here.


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

Best Practices for Prototyping

Connect the Dots

by Matt Stevenson, SUNSTONE CIRCUITS

PCB prototyping is a critical juncture during an electronic device's journey from concept to reality. Regardless of a project's complexity, the process of transforming a design into a working board is often enlightening in terms of how a design can be improved before a PCB is ready for full production.

But the PCB design needs to be carefully created and thoroughly tested before moving to prototype. Initial board testing usually happens with CAD tools and circuit simulator software. These tools tend to focus on high speed, high frequency, mixed signals, and even RF designs. Many of them represent a sizable investment for a startup or smaller electronics manufacturer hoping to take advantage of their testing features. There are numerous cost-effective testing tools, so it makes sense to do your research and find one that meets your needs and means.

There are a few different approaches to prototyping. They range from simple DIY options to partnering with a commercial prototype fabricator.

Criteria for Choosing Your Prototyping Approach

Since no two projects are the same, your approach to prototyping should be flexible. Your needs will be different every time depending on project complexity and timeline. Do you need a quick turnaround time? Will it be faster

for you to use a DIY prototyping technique? Or is your board design complex enough to require a full menu of prototyping services?

Iterative Prototyping

Prototyping often takes multiple iterations. Your first design and your first PCB may fail or not operate to specifications. With iterative prototyping, you lay out your physical components to make sure your design actually does what you want. If it doesn't work as desired, adjust the design and repeat the process. You will use actual ICs and components and try various designs until a prototype finally works. When you have a working prototype, you can copy the modified design back into your CAD software.

Today, most small projects start off with something like an Arduino, a handful of components and jumper wires, and a solderless breadboard. The resulting rat's nest can be difficult to turn into a PCB design, however, which is why you will want to quickly move to something closer to a finished PCB.

Solderable prototyping boards are perfect for your early DIY prototype.

DIY: Doing It Yourself

Solderable prototyping boards are perfect for your early DIY prototype. They come in both set form factors and perforated sheets that allow you to pick a custom size for your prototype board. You can find good walkthroughs for planning IC and component layout for your first prototype using a blank board and Kynar wire.

If you're in the mood for really taking your DIY prototype to the next level, you can always try making your own PCB at home. This requires getting a blank PCB, special transfer paper, etching chemicals, a power drill, and a lot of patience and careful work. You can also create your own PCB at home with a CNC mill, which lets you skip the chemical etching step.

Pro Tip

If you do decide to make your PCB at home, just be careful with the chemicals. They require special handling and disposal, meaning you can't just pour them down the drain and hope they don't eat a hole in your pipes. You'll end up subject to massive fines and find yourself in desperate need of a plumber.

Fabricated PCB Prototypes

When your design has progressed far enough, it's time to turn your design into a professionally fabricated PCB. When preparing your design for printing, you will need to understand that every manufacturer has different capabilities, requirements, and turnaround time. You'll need to take all these into account to manage your project efficiently.

If you already have your manufacturer identified, keep all this in mind from the very beginning of the design process. You don't want to get to the fabrication stage of prototyping and realize at the last minute you've overstepped your manufacturer's limitations.

Picking the Right Partner or Service

If your project is complex, especially if it is a brand-new design, you will want to consider expert help. Consult with your manufacturing partner for help with layout review.

Rapid prototyping fabrication services can help shorten the turnaround on complicated designs, which saves money during production while helping you stay on schedule. However, just because you can get boards quickly from a manufacturer doesn't mean you always should.

Look for manufacturing partners that can provide detailed, comprehensive feedback on your PCB design before your prototypes are fabricated. Not all do. This will reveal yield or reliability issues with your design, improve the integrity of your production schedule, and ultimately improve the quality of the boards. The manufacturer will also be able to tell you if your design fits within their capabilities and limitations, saving endless prototyping headaches later.

Some PCB manufacturers offer multiple prototyping services that range from basic service all the way up to custom PCB manufacturing including DFM checks and access to experts who can advise on design improvements.

Conclusion

Prototyping is a necessary step on the road to production, so it's important to pick the right strategy for your project. An approach that works for one board may not be right for the next, and it will take time and experience to know which strategy is the best fit for each case.

Be flexible in your approach to prototyping options. Whether you choose to etch your own prototypes in your garage or engage professional experts from your PCB manufacturer, remember to plan ahead and don't get discouraged if one prototype is not enough. **DESIGN007**

Matt Stevenson is vice president at Sunstone Circuits, a division of American Standard Circuits. To read past columns, click here.

New Ways to Suppress Lithium Plating in Auto Batteries for Faster Charging EVs

Lithium plating is a phenomenon that can occur in lithium-ion batteries during fast charging. It occurs when lithium ions build up on the surface of the battery's negative electrode instead of intercalating into it, forming a layer of metallic lithium that continues growing. This can damage the battery, shorten its lifespan, and cause short-circuits that can lead to fire and explosion.

Dr. Xuekun Lu explains that lithium plating can be significantly mitigated by optimizing the microstructure of the graphite negative electrode. The graphite negative electrode is made up of randomly distributed tiny particles, and fine-tuning the particle and electrode morphology for a homogeneous reaction activity and reduced local lithium saturation is the key to suppress lithium plating and improve the battery's performance.

"Our research has revealed that the lithiation mechanisms of graphite particles vary under distinct conditions, depending on their surface morphology, size, shape and orientation. It largely affects the lithium distribution and the propensity of lithium plating," said Dr. Lu.

The study provides new insights into developing advanced fast-charging protocols by improving the understanding of the physical processes of lithium redistribution within the graphite particles during fast charging.

In addition to faster charging times, the study also found that refining the microstructure of the graphite electrode can improve the battery's energy density. This means that electric cars could travel further on a single charge.

(Source: Queen Mary University of London)

Three Ways to Improve High-Speed PCB Signoff, Part 2

Article by Brad Griffin
CADENCE DESIGN SYSTEMS

Editor's note: Part 1 of this article is available here.

Another challenge for SerDes is losses within the channel design. At high speeds, dielectric material can be very lossy, making the appropriate selection of the right material, length, etc., critical for the channel. Many questions about stackup, trace widths, and height from the ground plane need to be defined up front. Simulating a signal with a topology explorer tool extracted from the design can be used to set up and run sweep parameters and push min/max length/spacing values into the Allegro schematic constraint manager (system capture). The preliminary constraints and schematics flow is illustrated in Figure 1. As the design progresses with final decisions on stackup and material selections, these constraints can be adjusted.

With the schematics phase finished and the layout phase in progress, the next challenge is compliance with specifications. Specs are dependent on the technology—PCI Express (PCIe), USB, etc.—and, because each one has its own requirements, this can be a complicated process. During this analysis, it is important to make sure the correct transmitter and receiver IBIS-AMI models are being used.

For the channel, Cadence tools can be used to accurately model the channels and address specifications. This is done by using the board file created by the layout designer, selecting

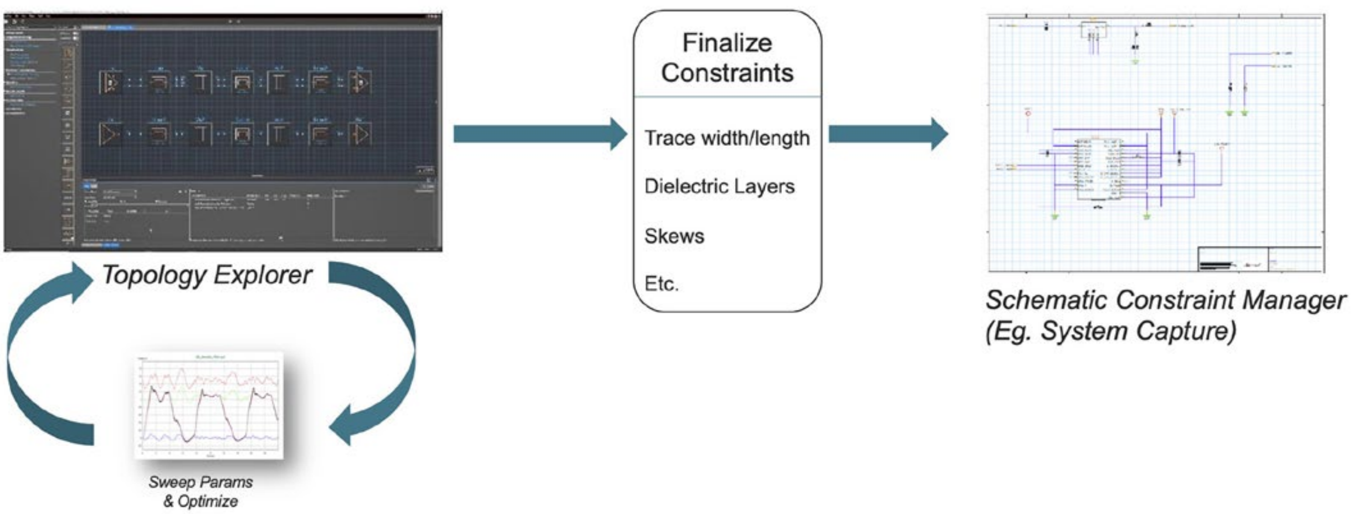


Figure 1: SerDes constraints and schematics workflow using the sweep manager utility in Sigrity Topology Explorer (TopXp).

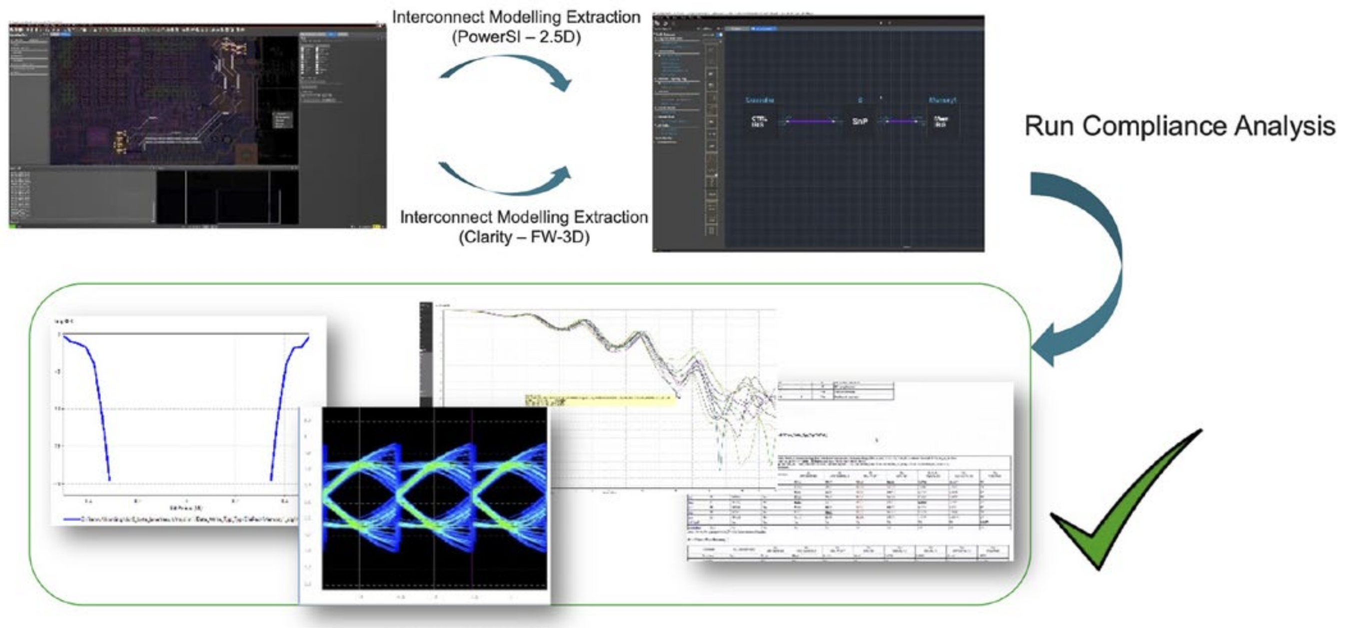


Figure 2: Compliance workflow for accurately modeling the channels and addressing compliance specifications.

several or all the lanes (depending on how much time is available), and running either a 2.5D or full 3D analysis on the entire channel (Figure 2).

Using the results of the channel extraction, a compliance analysis can be run based on the desired protocol. Most likely this will not be a one-time event, as often some obscure requirement not identified in the preliminary phase will surface, requiring additional iterations.

DDR Compliance Analysis Flow

DDR, while not as fast as SerDes, and in some ways not as sophisticated, can present more challenges for signal integrity (SI). Some DDR speeds can overlap with the lower end of the SerDes spectrum and there are many nets, both single-ended and differential, that have complex SI requirements.

DDR Design Challenges

Early-stage DDR SI issues typically involve general problems like impedance matching and discontinuities, stubs like dual in-line memory modules (DIMMs), termination optimization, and trace crosstalk. While these are basic requirements, it is important to consider them as well as the more sophisticated problems, and the earlier the better. These issues can quickly be examined by running sweeps with the Topology Explorer, such as what has been discussed with SerDes. The constraints can be determined through simulation sweeps, and those constraints can be saved with the schematics (Figure 3).

Once the preliminary schematics with constraints are generated, the layout phase begins. This is where IDA empowers PCB designers with early layout verification to provide confidence that the layout is correct and won't fail due to SI issues. This can be done by quickly running a host of general SI analyses (impedance, coupling, crosstalk, reflection, return path, etc.) in the Allegro layout environment with Sigrity Aurora workflows. These analyses provide a visual representation on the trace of where the issues are, so rather than the typical back-and-forth between the SI experts and layout designers to identify and correct the problems, the design can remain with the layout designer who can visually see the problems and correct them right then and there. This shortens the design process.

If the layout designer is unable to fix the problem, the topology explorer can be used

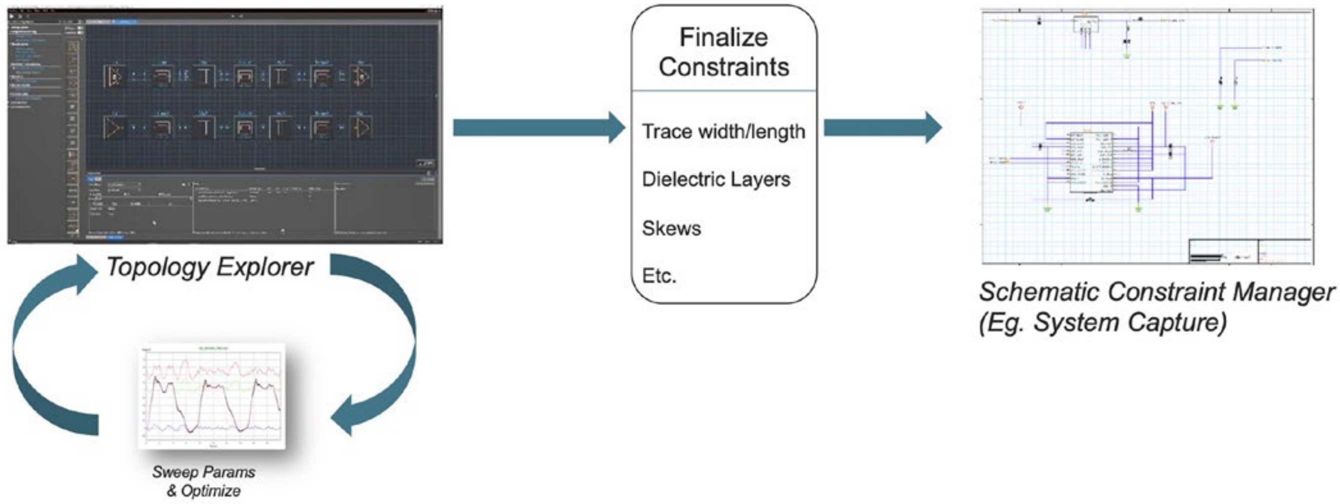


Figure 3: TopXp topology sweeps are optimized in the schematic constraint manager.

to run the iterations again. This also enables designers to fine-tune the preliminary constraints during the layout phase and update them to values that can be used on other signals.

In addition to traditional SI issues usually addressed during the schematics and layout phases, the nature of DDR makes it susceptible to three additional challenges that must be addressed: simultaneous switching noise (SSN), via-via crosstalk, and JEDEC standard compliance. Cadence software empowers the PCB design team to address these issues when SI expertise is not available.

SSN Challenge

In SSN, the transistor drives the output of a single bit, drawing power from the voltage rail as needed and dumping it into the net. The behavior of this transistor is traditionally captured in an IBIS file or analysis. With DDR, it is possible for multiple nets to toggle simultaneously and the transition between states will have a high current rate-of-change (di/dt) requirement that affects the voltage level, which in turn affects the transition (Figure 4). Two models are critical for accurately modeling SSN: a power-aware IBIS model and an accurate model of the PDN.

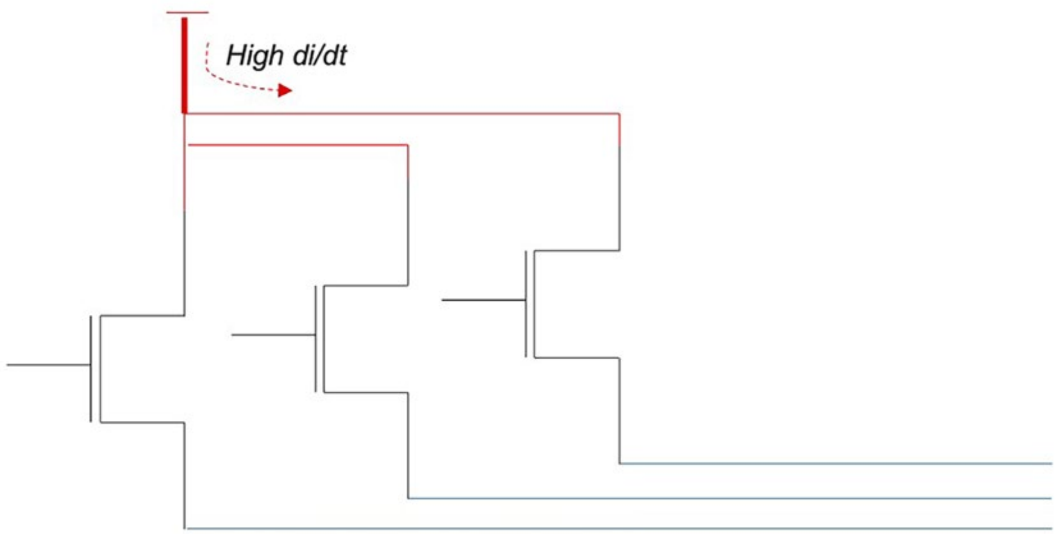


Figure 4: Multiple nets can toggle simultaneously, having a high di/dt requirement that creates interdependency between the voltage level and transition.

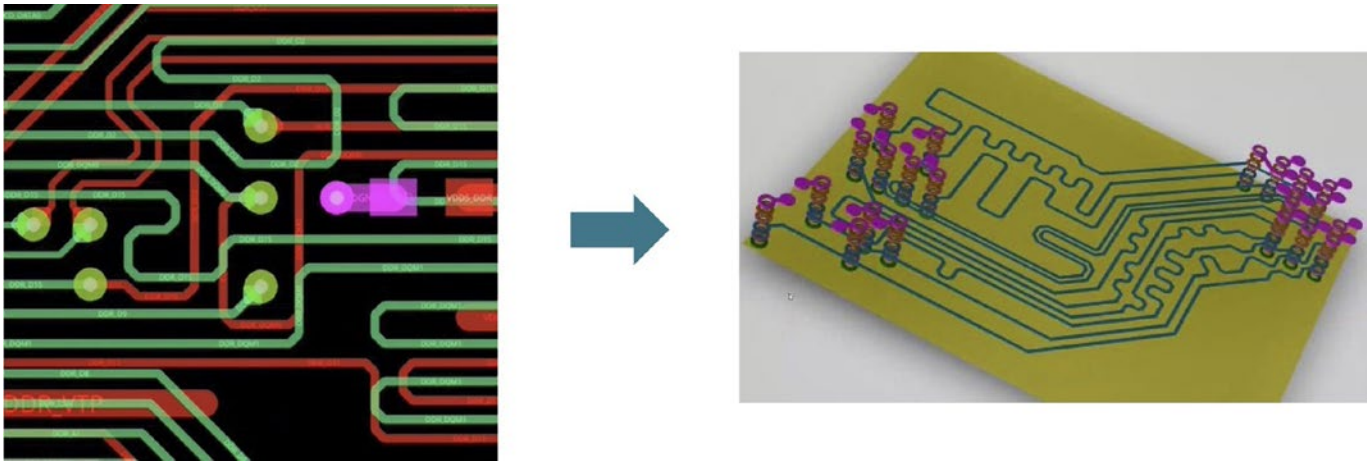


Figure 5: Return path for signal vias.

Return Path Vias Challenge

On the left side of Figure 5, the green lines are data line (DQ) signal vias and the pink area is the ground (GND) via, which is stitching two ground planes together. On the right, one of these signals has a trace on the green top layer and the second line is going to be the reference with an ideal return path.

However, the return current must traverse to the second-to-last layer, and it does so by finding the closest path possible. While the signal goes directly to the signal via, the return path is right underneath the signal and then must find its way to the stitching via and come back so it is referencing on the bottom layer again. The same thing is happening for all the signal vias—they are all using the same stitching via as the return path, which causes via-via crosstalk. This is a 3D phenomenon, not a simple 2D side-by-side problem. A 2.5D or 3D analysis is needed to accurately model this, and while it is traditionally the SI expert's wheelhouse, Sigrity Aurora analysis workflows enable the non-SI expert to do this.

JEDEC Specification Complexity

The JEDEC specification is incredibly complex and confusing with an "alphabet soup" of requirements. (V_{ix}, t_{DQSS}, t_{DSS}, t_{DS}, t_{DSH} t_{DH}, t_{VAC}, etc.) and the requirements vary by technology such as DDR, low-power DDR (LPDDR), graphics DDR (GDDR), by version (DDR3 versus DDR4), and by bus, such as address vs. data vs. clock signal (CLK) vs. DQ strobe (DQS). For example, DDR4 data has a rectangular eye mask requirement, which needs bit error rate (BER) measurements, but for DDR5 that measurement is now a diamond shape. For the PCB designer, this can be a daunting challenge, but analysis workflows from the Cadence PCB layout environment allow access to the Sigrity PowerSI and Clarity 3D Solver extraction engines to support detailed and accurate modeling of interconnect as part of a simple, easy-to-use solution for SSN, return path via, and JEDEC specification challenges.

Interconnect Model Extraction

At the later stage of the DDR workflow, the nets have been laid out and preliminary DDR checks have been completed. Now 2.5D or 3D interconnect model extraction is required for layout verification. This can be done on a section, a channel, or a handful of signals, depending on how much time is available. Based on what has been extracted, a quick waveform validation can be performed to make sure that the waveforms are correct (Figure 6), or a quick DDR analysis can be run using the PowerSI 2.5D or Clarity 3D Solver to ensure the design meets all the requirements.

This enables the designer to also observe via-via crosstalk effects, power-aware effects, and other advanced effects, and slowly and steadily

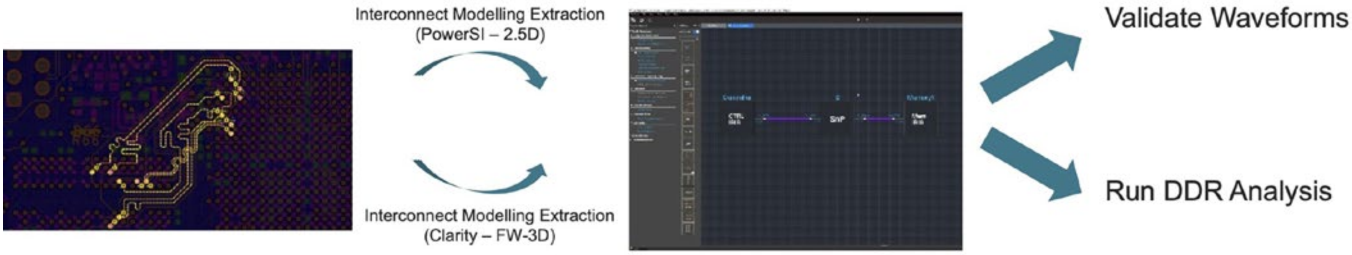


Figure 6: Interconnect model extraction is used to validate waveforms and run DDR analysis.

ensure the channel is meeting DDR specifications. Again, this flow enables the PCB designer to be less dependent on the SI expert by doing much of the validation independently.

Then, for the final check, the entire block is analyzed in a full-wave 3D simulation using the Clarity solver to make sure every detail is captured, and the DDR compliance analysis check is run (Figure 7).

For the compliance analysis, the designer doesn't need to know all the details discussed earlier, as the simulation workflow provides all the results for the specific protocol that has been selected. If the design passes this flow, the designer can be confident that the board works properly.

Board Design Example

This example describes a real-life PCB example that demonstrates how Cadence's Allegro/Sigrity/Clarity design flow streamlines the design process. Figure 8 illustrates the constraint phase of the design.

The design is in the schematic phase and the PowerTree file is used to show the power rails and validate that they have been set up correctly. During this phase, some iterative DDR simulations are also run to ensure that the first-pass constraints are correct. The same process is used to run SerDes simulations and ensure the right dielectric has been chosen, the stackup is right, etc. In parallel, either with or without the help of an SI expert, the designer

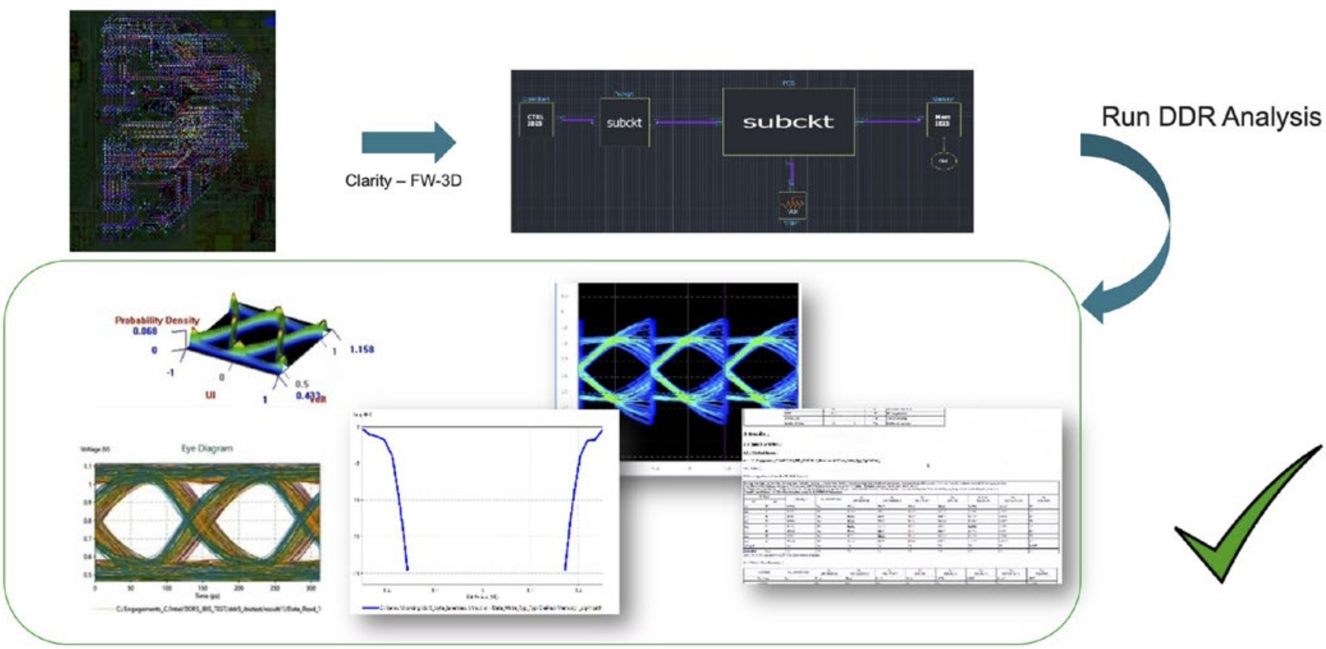


Figure 7: In the final design check, a DDR analysis is performed on the entire block using the Clarity full-wave 3D electromagnetic solver.

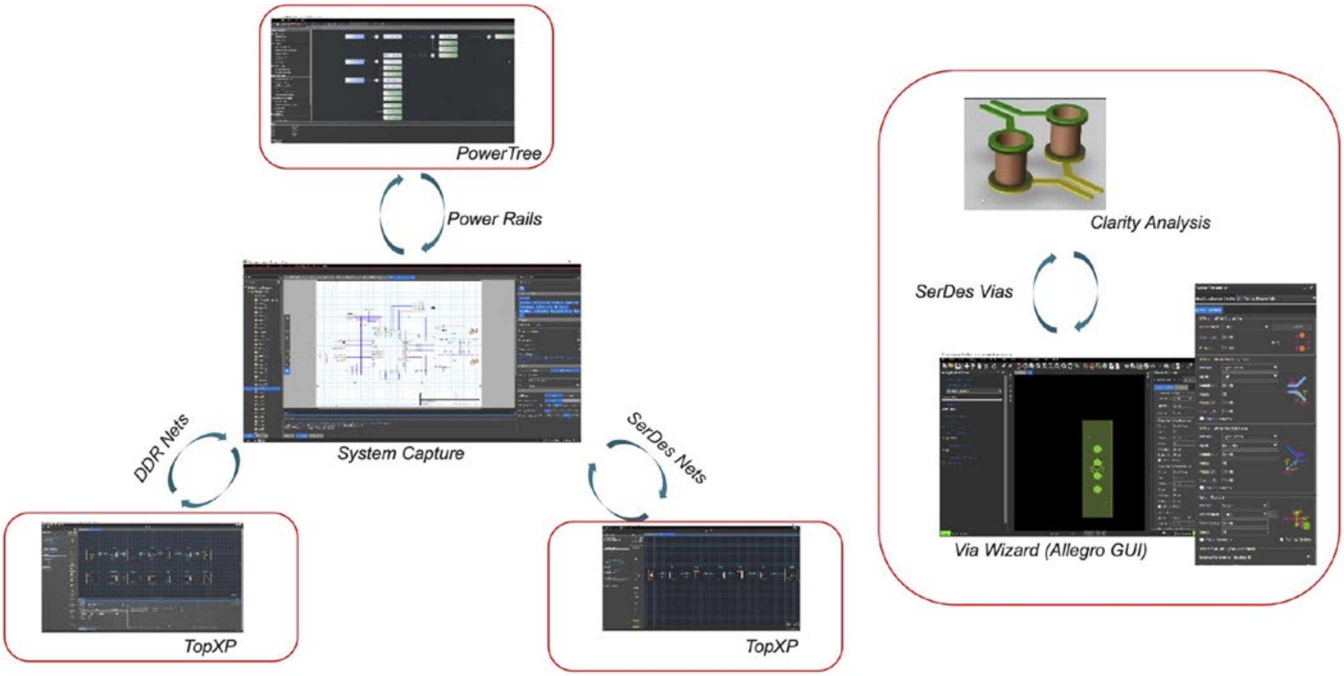


Figure 8: PCB design example constraint phase.

can use the Clarity solver to set up and analyze the right via structure for the SerDes design, which can then be used in the layout. The next phase, power analysis, is shown in Figure 9.

The preliminary layout and the previously set up PowerTree file can now be used to perform a preliminary AC and DC analysis to ensure there is enough copper for the DC drop and that the inductance is low enough for the capacitors to be effective at high AC frequencies. This phase is much faster and easier because the layout and PowerTree files were

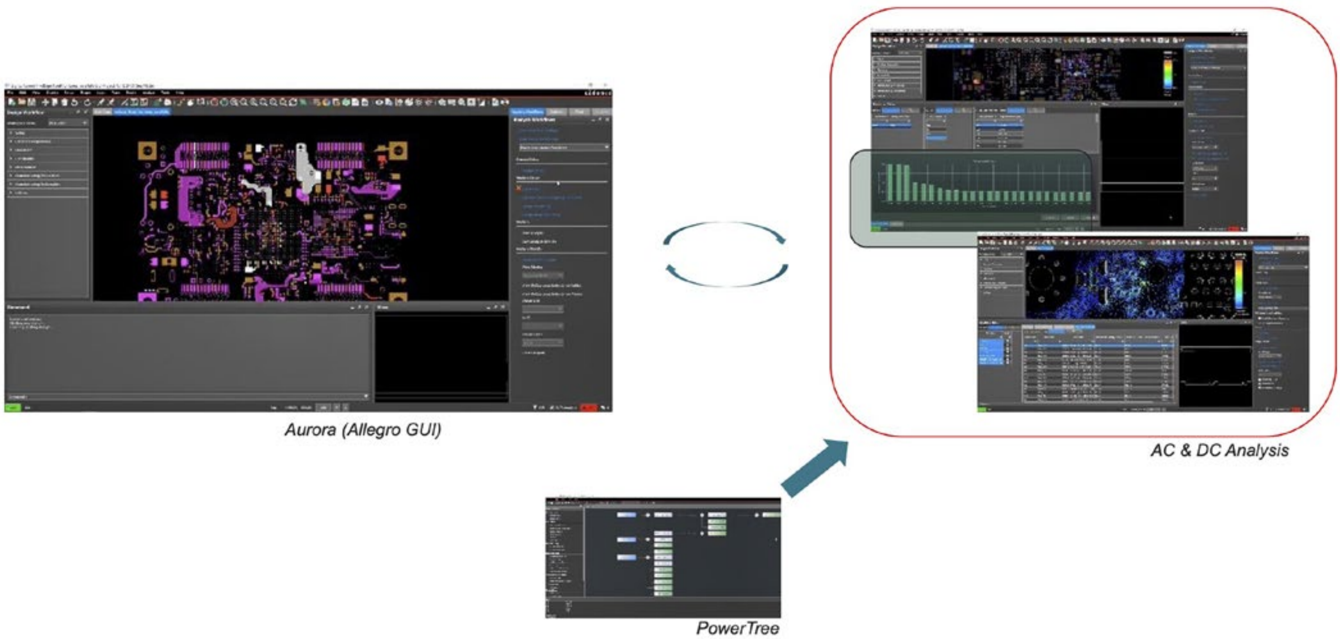


Figure 9: PCB design example power analysis phase.

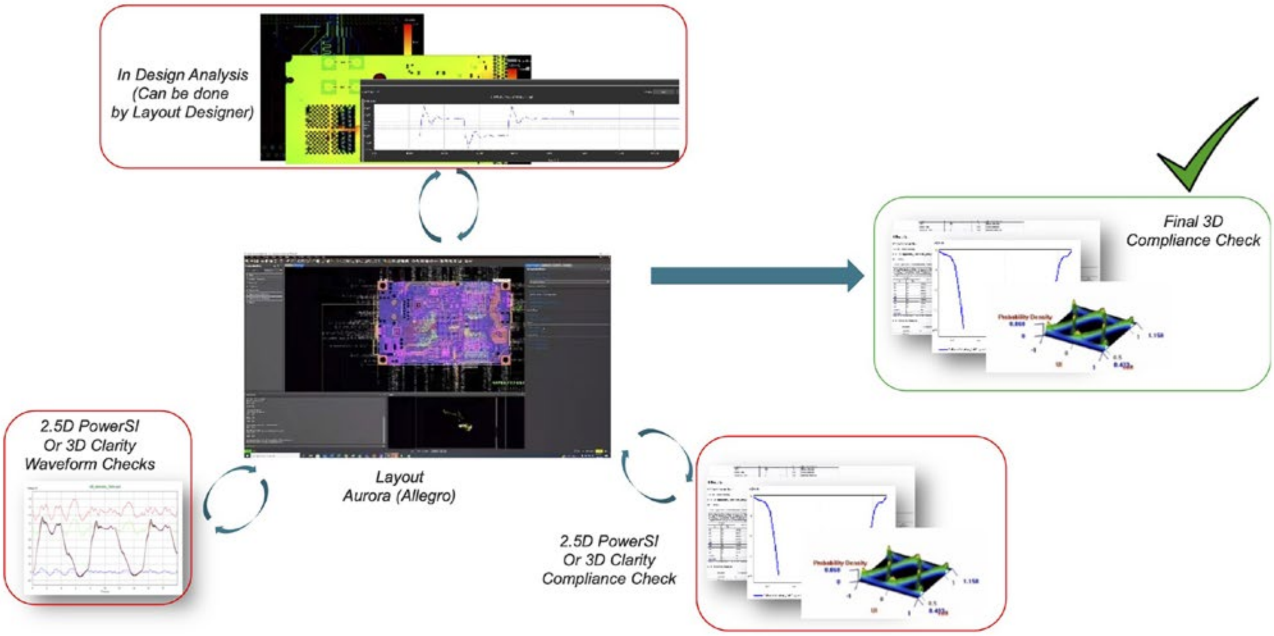


Figure 10: PCB example high-speed design phase.

already set up in the previous phase. The final high-speed SerDes and DDR phase is shown in Figure 10.

In this phase, the PCB designer is working with the layout expert on the general SI analysis to find any discontinuities. Using the Cadence flow, the layout designer can visualize and correct any problems without involving the PCB designer in an iterative cycle. When the design does come back to the PCB designer, it is much cleaner, and less time is wasted because details have already been corrected. The PCB design can move straight to extracting 2.5D or 3D channels to perform the waveform checks and even some compliance checks. It is close to the end of the design, so possibly only some layers need to be examined for compliance because many issues have already been caught and fixed by the layout designer.

The final step is to run a complete full-wave 3D compliance check on the SerDes and DDR using the Clarity 3D Solver to validate the complete design.

Conclusion

Verification as well as final signoff of today's complex high-speed and/or high-density circuit boards can be achieved more quickly by leveraging in-design analysis that uncovers SI/PI issues early in the design process. While it may take PCB designers a bit longer to complete a design iteration and correct basic SI/PI issues for themselves, the result is fewer respins on a design, which brings significant time and cost savings to the overall program.

This article has highlighted three key areas PCB design teams need to address to successfully deliver their products to market: power analysis, SerDes link compliance, and DDR memory interface compliance. Deploying the above methodology as described empowers PCB designers to create successful products on time, on budget, and on their own, without the need to wait for SI and PI specialists to find time to assist and/or work with difficult-to-use analysis point tools. DESIGN007

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

Unlocking the Key to Rigid-flex Design Success

Flexible Thinking

Feature Column by Joe Fjelstad, VERDANT ELECTRONICS

First, do the right things; then, do those things right.

Despite what some seem to believe, rigid-flex circuits are not a new technology. In fact, they are more than a half-century old. At the time of the invention, my friend Thomas Sterns was working at Sanders Associates, the pioneering flex circuit manufacturer. Like many products in the first decades of printed circuit technology, they were working on a military application. The objective was to provide a reliable method for reducing the size, bulk, weight, and questionable reliability of wire harness assemblies while simultaneously reducing cost and assuring that human error might be minimized. These were all vitally important concerns for military and aerospace products.

I'm reminded of a statement from U.S. Navy Capt. Frank Akers:

"Today with the modern high-performance airplane, electronics is a must if we are to get the benefit of the tremendous performance available. We have made electronics equipment smaller, have made it lighter, and we have made it work. Printed circuits, (have been) the greatest weight and space saver of them all."

While this sounds like a recent comment, Capt. Akers actually spoke these words in 1947, more than 75 years ago. It tells me that while the key design objectives of rigid-flex are largely unchanged, the technology—and diversity of applications—has advanced steadily. Knowing that rigid-flex circuits are an option, designers have been (with ever greater frequency) looking to rigid-flex technology to help them solve vexing interconnection problems. Most notably, and perhaps with some surprise, rigid-flex circuits have enabled a substantial percentage

of the advances we have seen in smartphones and other portable electronic products. Even though most smartphones don't have foldable screens, there are myriad interconnections required between the various features of the modern smartphone: camera, screen, speaker, microphone, GPS, gyroscope, memory storage, and yes, the telephone.

While original rigid-flex circuit designs sought to minimize the number of connectors, the number of low-profile flex connectors found in modern cellphones is impressive. Assembly appears to involve a lot of delicate handwork, which doubtless contributes to cost. Watching a few YouTube video teardowns helps me appreciate just how important it is to build in product reliability for these amazing products because rework is clearly not an inexpensive activity.

The adage, "Knowledge is king," is imperative when it comes to rigid-flex design. The importance cannot be understated that we must fully understand the intricacies of rigid-flex manufacturing in advance of setting out to design such a product.

Many useful articles and publications on rigid-flex circuits exist online, including on manufacturers' websites, all aiming to educate their customers and for the greater benefit. Most rigid-flex manufacturers have instructive websites and self-published books in some cases. I highly encourage you to visit the eBook library at I-Connect007, where you'll find a wide range of topics published by subject matter experts. Tap into this resource and greatly expand your knowledge.

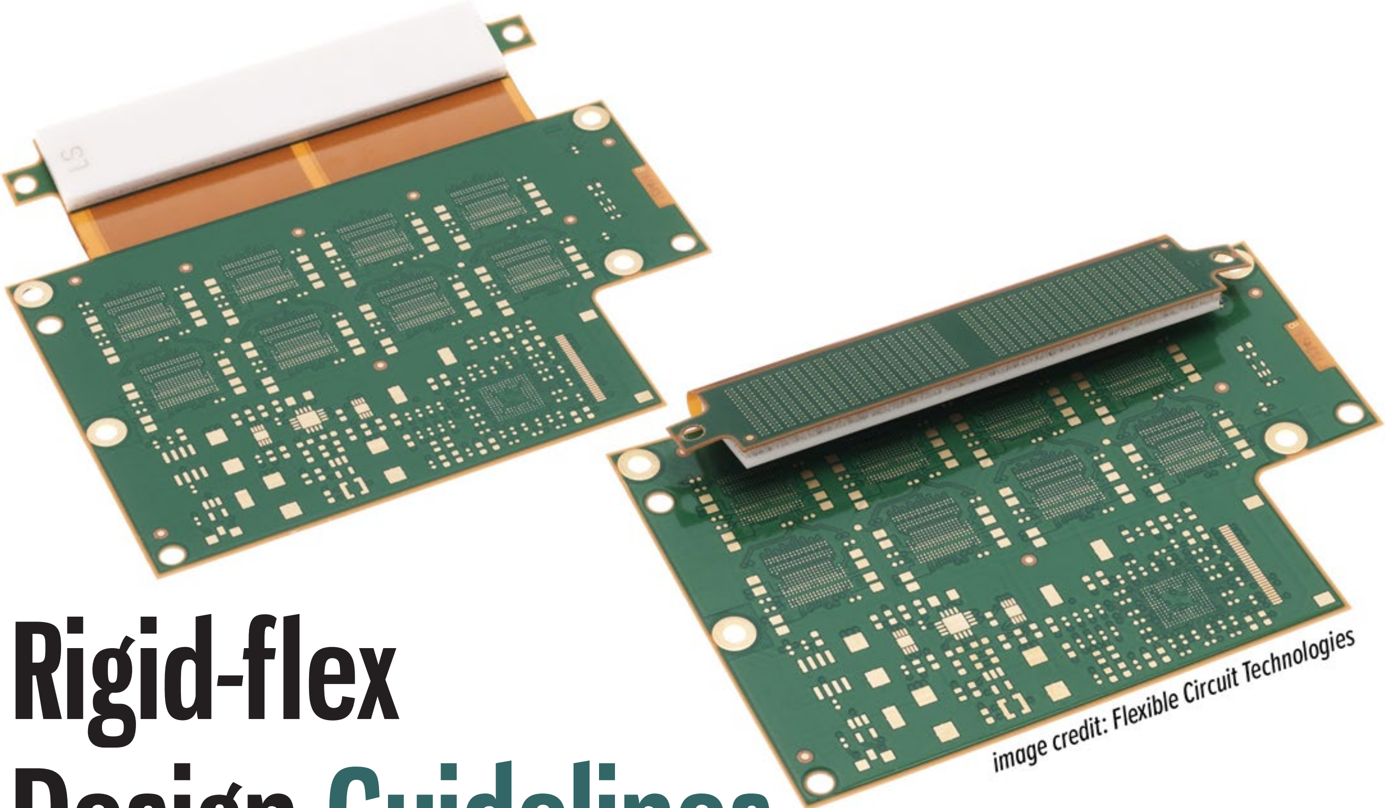
One of the best ways to avoid potential pitfalls is to engage with the manufacturer early in the design process. In a previous column, I wrote about the increasing importance of making the manufacturer part of the design team. There are many "design for" recommendations out there: design for environment (or excellence), design for testing, design for reliability, and as mentioned, design for manufacturing. But designing *with* manufacturing may be the most important design advice of all. Often, the manufacturer has "seen it all" and, if consulted early in the design process, can greatly increase your chances of first-pass design success. That "seen it all" idea reminds me of a comment from John Wooden, the Hall of Fame basketball coach from UCLA, who said, "It's what you learn after you know it all that counts." That was an important reminder as this technology has never stood still; it has continued to change and evolve. Because they are so close to the flow, the manufacturer will typically be a great source of guidance.

In conclusion, while often seen as an expensive interconnection technology alternative due to the higher degree of engineering required to produce it, rigid-flex circuits in the long run can be a much more cost-effective solution. Reducing the size, weight, and power (SWaP) requirements of products will always be overarching objectives in product development, and are intrinsic benefits of rigid-flex circuits. However, many buyers don't often appreciate that it's not just cost in, but cost out the door that must be considered when making purchase decisions.

Repackaging and condensing the words and thoughts of some historically great quality gurus of the industry, in general, I now conclude (at least to my own satisfaction) that to succeed in product design, rigid-flex, or any other product, one must first do the right things and then do those things right. Enjoy your rigid-flex design journey, and stay flexible. **DESIGN007**

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


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



Rigid-flex Design Guidelines

Feature Article by Cherie Litson
LITSON1 CONSULTING

Rigid-flex circuits are unique structures; part rigid board and part flex, they're increasingly working their way into many of the electronic devices we use every day.

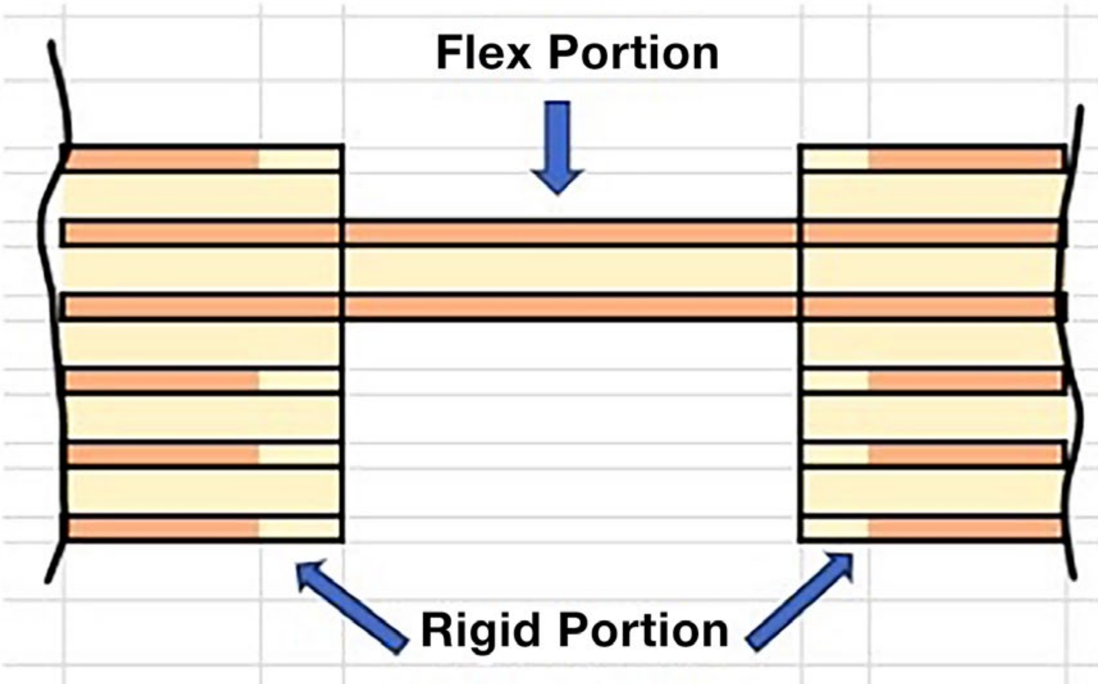
There have been many advances in rigid-flex lately as more companies find themselves exploring this technology. So, before I started writing this article about rigid-flex design, I double-checked a few things. Fortunately, there is plenty of rigid-flex information available on a variety of great websites that we all know and trust. So, here's a quick look at some of the principles of rigid-flex design.

Rigid board designers who consider entering the rigid-flex arena typically have a handful of questions. One of the most obvious is, "Why do we need rigid-flex anyway?" If we just need a way to fit circuitry into a case and it will stay that way, rigid-flex will work fine. But if the board will be moving on a regular basis during operation, then opting for just flexible circuits or attaching flex to a traditional rigid board is a better way to go.

Another common question is, "How does my PCB layout software work with curves and hatched planes?" There are a lot of curved traces and hatched planes in flex circuits. If it's a hassle to create a curved trace, you may need a different software package that is optimized for flex and rigid-flex design. Many of today's EDA tools are set up for rigid-flex design.

Rigid-flex boards use the same materials as any rigid board. One way to do this is by creating the flex portion along with the layers of the rigid board, and use a delamination process to remove any unwanted layers. No components are mounted on the flex portion of


Allegro X
NEXT GENERATION PCB DESIGN
OFFERING ENGINEERS THE INDUSTRY'S FIRST SYSTEM DESIGN PLATFORM THAT INTEGRATES LOGIC/PHYSICAL DESIGN, SYSTEM ANALYSIS, AND DESIGN DATA MANAGEMENT FOR PCB AND SYSTEM DESIGN.
GET STARTED TODAY >>
EMA Design Automation®



the board. One layer is used for the return and the other layer is used for your signals. The return is commonly a GND plane and hatched in the flex area of the board. Vias are not usually placed in this area either but may be used if needed.

Another rigid-flex approach is to first create a flex board and then laminate or adhere a rigid board (or just a stiffener) to the flex. This is done when you need more signal layers in the flex portion, the flex needs to be longer, and/or it needs to withstand multiple flexing without breaking. The rigid portions are processed separately and have the components assembled to them. In general, components should not be mounted on the flex portion. Flexing solder joints is just a bad thing.

Now you've got the basics, what's next? Here are a few guidelines, in no particular order:

Layer stackup: Keep the flex layers to a minimum of two for increased mechanical flexibility and reduced costs.

Electrical design constraints: What needs to be on the flex and what doesn't? Trace widths may need to vary. They may need to be very long. Make sure that the signals can handle these conditions. Also, designers should have a good understanding of the heat sinking capabilities of the flex portion.

Mechanical design constraints: You will need more clearance to the edges of the board, cutouts, slots, other traces, etc., as well as more space around bends and corners. Any slots or holes that must be cut will need "relief" features so that the material doesn't tear.

Component placement and finishes: As I mentioned, components in general, should not be mounted on flex, but there are exceptions. Some components can be mounted on the flex portion with a stiffener material attached beneath them to prevent solder joint breakage. The finish used to protect the open copper traces is usually an overlay material or a thicker solder mask—not LPI. Both of these have larger clearance areas around the components. Conformal coatings can be used to protect the bare copper after soldering.

Testing and verification: Because of the unique structure of rigid-flex circuits, testing and verification are critical parts of the process. Test points must be strategically placed, not too close to edges or components, and not covered by solder mask. Ask your fabricator for more information.

Rigid-flex Resources

IPC has a variety of great resources and specific industry standards on its website. IPC-6013 *Qualification and Performance Specification for Flexible Printed Boards,* features a treasure trove of information. IPC-4201 through IPC-4204 cover many of the base materials used for flex circuits. IPC-4562 covers copper foils and IPC-SM-840 defines the masks that can be used.

Once you've defined what you need in your rigid-flex design, find a flex fabricator you can work with to get the job done. Most of the guidelines listed by fabricators online focus on the company's limitations. They all have different capabilities, so be sure to match your needs with their limits for the best cost return. Pay close attention to their "standard" feature sizes, not the "highest-end capability." If your electrical and mechanical constraints are critical to your design, be sure to have a conversation with the production engineer at the facility. This will help you and the fabricator.

Your EDA software company should also have an online section titled "How to Layout a Flex Design." These are great resources and worth looking at. If you don't like your software's approach to rigid-flex, check out some of the videos from the other software companies. You might find something you didn't think of before. DESIGN007

Cheri Litson is a PCB design and engineering consultant.

Bigger and Better Quantum Computers Possible With New Ion Trap, the Enchilada

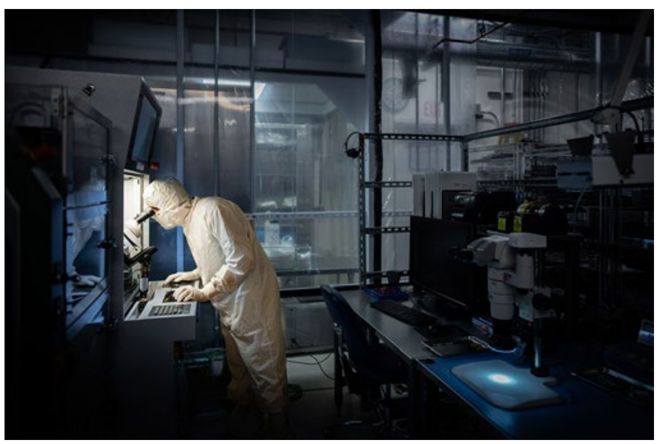
Sandia has produced its first lot of a new world-class ion trap, a central component for certain quantum computers. The new device, dubbed the Enchilada Trap, enables scientists to build more powerful machines to advance the experimental field of quantum computing.

In addition to traps operated at Sandia, several traps will be used at Duke University for performing quantum algorithms. An ion trap is a type of microchip that holds electrically charged atoms, or ions. With more trapped ions, or qubits, a quantum computer can run more complex algorithms.

With sufficient control hardware, the Enchilada Trap could store and transport up to 200 qubits using a network of five trapping zones inspired by its predecessor, the Roadrunner Trap. According to Daniel Stick, a Sandia scientist and leading researcher with the Quantum Systems Accelerator, a quantum computer with up to 200 qubits and current error rates will not outperform a conventional computer for solving useful problems. However, it will enable researchers to test an architecture with many qubits that in the future will support more sophisticated quantum algorithms for physics, chemistry, data science, materials science and other areas.

Sandia had experimented with similar junctions in previous traps. Daniel believes the branching architecture is currently the best solution for rearranging trapped ion qubits and anticipates that future, even larger versions of the trap will feature a similar design.

(Source: Sandia)



Selecting a Rigid-flex Manufacturer

Feature Article by Mike Morando
PFC FLEXIBLE CIRCUITS

Editor Andy Shaughnessy tapped me to write about rigid-flex because so many PCB designers and *Design007* readers are latching onto the subject, and there is a growing interest—and market—in rigid-flex, especially among traditional board designers.

Is this because the market for rigid PCBs has decreased a little bit, thus driving PCB manufacturers to find solutions that fill fabrication facilities? Perhaps. Could it be that OEMs are looking for ways to make their new products simpler to manufacture, be more reliable, and with fewer part numbers? Probably.

If you're coming from the rigid PCB world, you're probably wondering how to get started and how to select a rigid-flex vendor. If you're designing PCBs for an OEM that is betting everything on a next-generation rigid-flex design, you would want to use the absolute best vendor for your rigid-flex design. Your new rigid-flex design needs to be put into the right shop to leverage the best manufacturing solution.

So, let's look at the expertise, processes, and materials used to manufacture rigid-flex PCBs. Rigid-flex is a hybrid solution comprised of rigid and flex circuits and I would argue there are only two types of rigid-flex fabrication shops in North America:

1. Flexible circuit shops that specialize in the manufacturing of flex and rigid-flex.
2. Rigid PCB shops that build rigid-flex or are migrating in that direction.

Both categories are vying for your rigid-flex business. From a 20,000-foot level, I believe that some rigid-flex boards should be built in a rigid PCB shop, and some rigid-flex boards

should be built in specialized flex shops. For a PCB designer, it may not seem like a big deal either way, but understanding the difference will help in choosing the correct supplier and result in better quality and lead times. When it comes to rigid-flex, both flexible and rigid PCB manufacturers bring their own strengths and weaknesses, so I will analyze the different aspects of the processes, how the two types are differentiated, and how designers can adapt to these shops.

Rigid PCB Shops

Printed circuit board shops typically provide a multilayer FR-4 solution. About 95% of their circuits are rectangular-shaped substrates that are not required to bend. The product is relatively simple to panelize and, in many cases, the material stacks are off the shelf. PCB shops are very good at complex stacks, high-density interconnect (HDI) for blind and buried vias, and multiple lamination cycles. They have a stable, repeatable process, and PCB designers have continued using the same design rules for years. Still, as rigid boards get more complex, the available real estate shrinks while the number of components has increased dramatically.

Flex Shops

Flex shops differ from rigid board shops because they provide a largely custom mechanical solution that bends, but does not break. Once you add in circuit electrical characteristics, component placement and support, obscure outlines, copper grain direction, and coverlays, you realize that rigid-flex is something completely new. The design and manufacturing rules for a flex board are different than a rigid PCB.

Flex manufacturing is more manually intensive than rigid board manufacturing, and materials are differentiated for every job.

Materials

Suppliers for rigid PCBs and flex are very different. That includes base materials and adhesives, and processing and handling the material. A PCB shop conventionally uses FR-4 and prepregs along with electrodeposited (ED) copper. If they are supplying HDI boards, the material choices can include photosensitive dielectrics, resins, and dry films. Flex uses primarily polyimide dielectric materials, adhesives, and rolled annealed (RA) copper foils. Very thin materials are used for flexibility.

Copper Plating

A PCB shop uses panel or pattern plating, a process that has always existed with PCBs. Copper is plated on all aspects of the panels and vias, then etched away, leaving a circuit design and plated holes. A flex shop uses pad or button plating in a process that puts copper only on the pads and in the plated through-holes. If you plate up—add copper to—the traces on a section of flex that is designed to bend, there's a high potential for cracked traces. If your rigid-flex has a tight bend radius, make sure your vendor is using button plating in the flex portion of your rigid-flex.

Design Considerations

PCB and flex circuits require their own design considerations. For example, there are many rules regarding vias. Figure 1 shows what happens when a via is placed in a bend area of the flex.

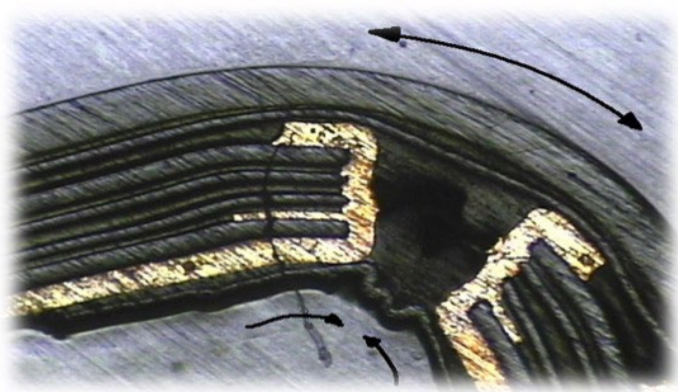


Figure 1: This is what happens when vias are placed in a bend area of flex.

Bend Radius

In most cases, PCB vendors never have to worry about bending a rigid board. With flex, bending is the most urgent issue. At the design and layout stage it's all about the bend, including material choices, bend location and degree of severity, and component placement.

IPC-2223 has a good section on bend radius calculations. It is usually recommended to calculate bend radius as 12x the thickness of the flex. The stiffness of a flex board is primarily driven by the copper weight, but polyimide thickness is critical.

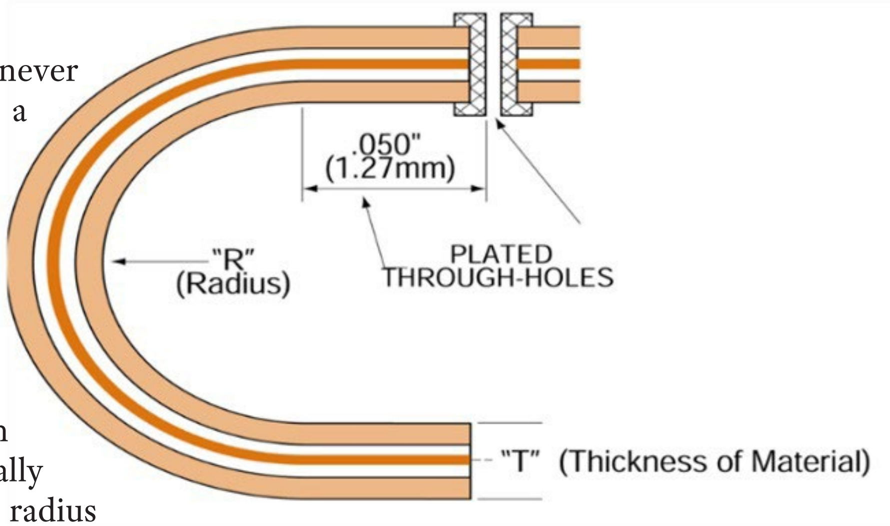


Figure 2: Bend radius is usually determined by using a formula that is 12x the thickness of the flex.

Layout Design Considerations

There are a variety of design considerations for flex that are much different than PCBs.

Filleting: It is better to add filleting (tear drop) for plated through-holes in the flex area as it helps to distribute stress over a broader area and it improves reliability.

Via location in rigid-flex: If the vias are placed in the flex area of rigid-flex, they should be at least 0.050" away from the rigid/flex junction and 0.050" away from the start of the bend area. On the rigid side of the rigid-flex, they should follow the same 0.050" keep away rule.

Routing traces: The preferred practice is to route conductors perpendicular, if possible, to bend and fold areas in a single metal layer.

Routing traces in multi flex layers: Traces should not be on top of each other layer-to-layer so as to avoid creating an I-beam effect.

HDI: PCB vendors are actively supplying blind, buried, staggered, and stacked vias. The PCB industry shines in high layer counts and HDI. While the features and vias on a flex circuit are very small, the ability to do blind, buried, and staggered vias is more difficult. However, the flex market is moving forward with technology to better support flex.

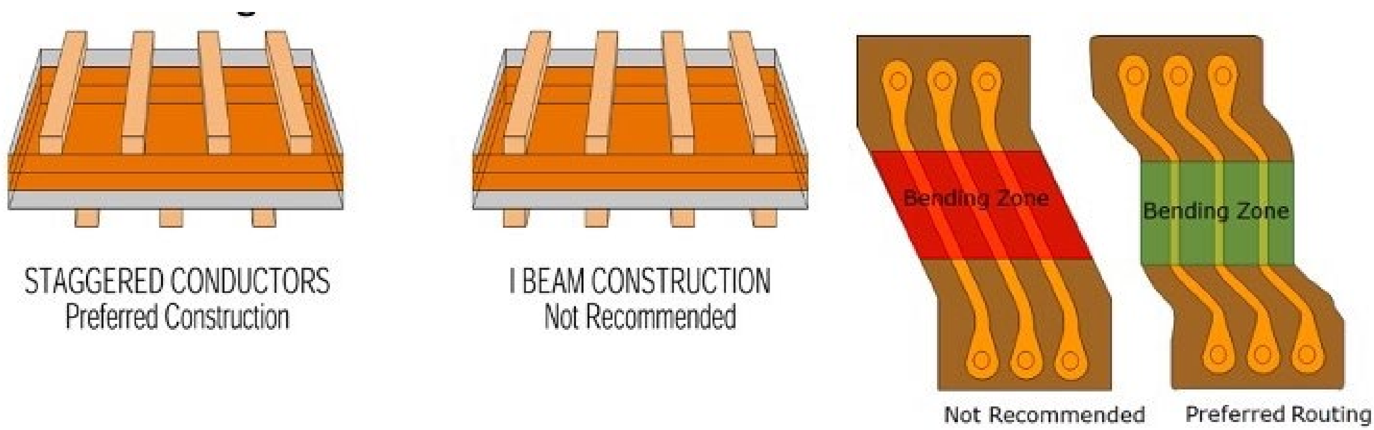


Figure 3: Examples of recommended and non-recommended constructions.

Conclusion

Rigid-flex is a hybrid solution combining rigid PCBs and flex circuits. Both rigid and flex companies can provide high-quality solutions for your requirements. As an OEM circuit designer, design engineer, or commodity/purchasing manager, pay attention to your requirements. The right answer for your solution depends on the design and using the correct vendor.

If the rigid-flex circuit you are designing has more of the features of a traditional PCB—high layer count; dense assembly requirements; HDI requirements; and blind, buried, or stacked vias—with a flex interconnect between two or more rigid sections, then a PCB vendor is probably your best bet to get your design requirements met.

If your rigid-flex has more flex circuit features—tight bend radius, small feature requirements, and thin stack requirements—and is the prominent portion of the circuit, then I would talk to a flex circuit vendor. **DESIGN007**

Mike Morando is director of sales and marketing for PFC Flexible Circuits, a subsidiary of OSI Electronics. Ata Syed, a field application engineer, assisted with this article.

New Type of Quantum Bit in Semiconductor Nanostructures

Researchers have created a quantum superposition state in a semiconductor nanostructure that might serve as a basis for quantum computing. The trick: two optical laser pulses that act as a single terahertz laser pulse.

A German-Chinese research team has successfully created a quantum bit in a semiconductor nanostructure. Using a special energy transition, the researchers created a superposition state in a quantum dot in which an electron hole simultaneously possessed two different energy levels. Such superposition states are fundamental for quantum computing. However, excitation of the state would require a large-scale free-electron laser that can emit light in the terahertz range. Additionally, this wavelength is too long to focus the beam on the tiny quantum dot. The German-Chinese team has now achieved the excitation with two finely tuned short-wavelength optical laser pulses.

The team headed by Feng Liu from Zhejiang University in Hangzhou, together with a group led by Dr. Arne Ludwig from Ruhr University Bochum and other researchers from China and the UK, report their findings in the journal Nature Nanotechnology.

The team made use of the so-called radiative Auger transition. In this process, an electron recombines with a hole, releasing its energy partly in the form of a single photon and partly by transferring the energy to another electron. In 2021, a research team succeeded for the first time in specifically stimulating the radiative Auger transition in a semiconductor.

In the current project, the researchers showed that the radiative Auger process can be coherently driven: they used two different laser beams with intensities in a specific ratio to each other. Hans-Georg Babin produced the high-purity semiconductor samples for the experiment at Ruhr University Bochum under the supervision of Dr. Ludwig at the Chair for Applied Solid State Physics headed by Professor Andreas Wieck.

(Source: Ruhr-Universität Bochum)

Flex007 Highlights

Flexible Thinking: Growth of Flex and Flex-hybrid Electronics in Mil-Aero Applications ▶

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

Stiffeners for Flex Circuits ▶

One of the most recent topics in the flex world has been the evaluation of materials required to build a rigid-flex or flex circuit. Discussions around flex and materials include variations of polyimide and copper. Here I will discuss another material that plays a very important role in the world of flex circuits: stiffeners.

Ibiden Reports Lower Revenues in 1Q FY 2023 ▶

Japanese printed circuit board (PCB) maker Ibiden Co. Ltd has reported net sales of 94,601 million yen for the first quarter ended June 30, 2023, down by 3.8% compared with the same period in the previous fiscal year. Operating profit reached 8,181 million yen, down by nearly 55% year-on-year (YoY), while ordinary profit fell by almost 50% YoY to 10,197 million yen.

Nan Ya PCB Posts 8% Growth in July Revenues ▶

Taiwan-based Nan Ya Printed Circuit Board Corp. (Nan Ya PCB), a manufacturer of single-sided PCBs, HDI PCBs, and rigid-flex PCBs, has posted unaudited sales of NT$3.33 billion ($104.86 million at $1=NT$31.79) for July 2023, up by 8% from the previous month, but down by nearly 40% year-on-year.

Trackwise Announces Appointment of Administrators, Resignation of Nominated Adviser ▶

Since September 2022, the board of directors of Trackwise has been exploring longer term strategic investment partnerships in order to support development and conversion of the pipeline of identified IHT sales opportunities, notably for EV battery cell connection systems (CCS) and also for other medical and aerospace sales opportunities.

Wearable Technology Market Size to Grow by $52.4 Billion from 2021 to 2026 ▶

The wearable technology market size is to grow by $52.4 billion from 2021 to 2026 and register a CAGR of 14.1%, according to Technavio's latest market research report estimates. With a focus on identifying dominant industry influencers, Technavio's reports present a detailed study by the way of synthesis, and summation of data from multiple sources.

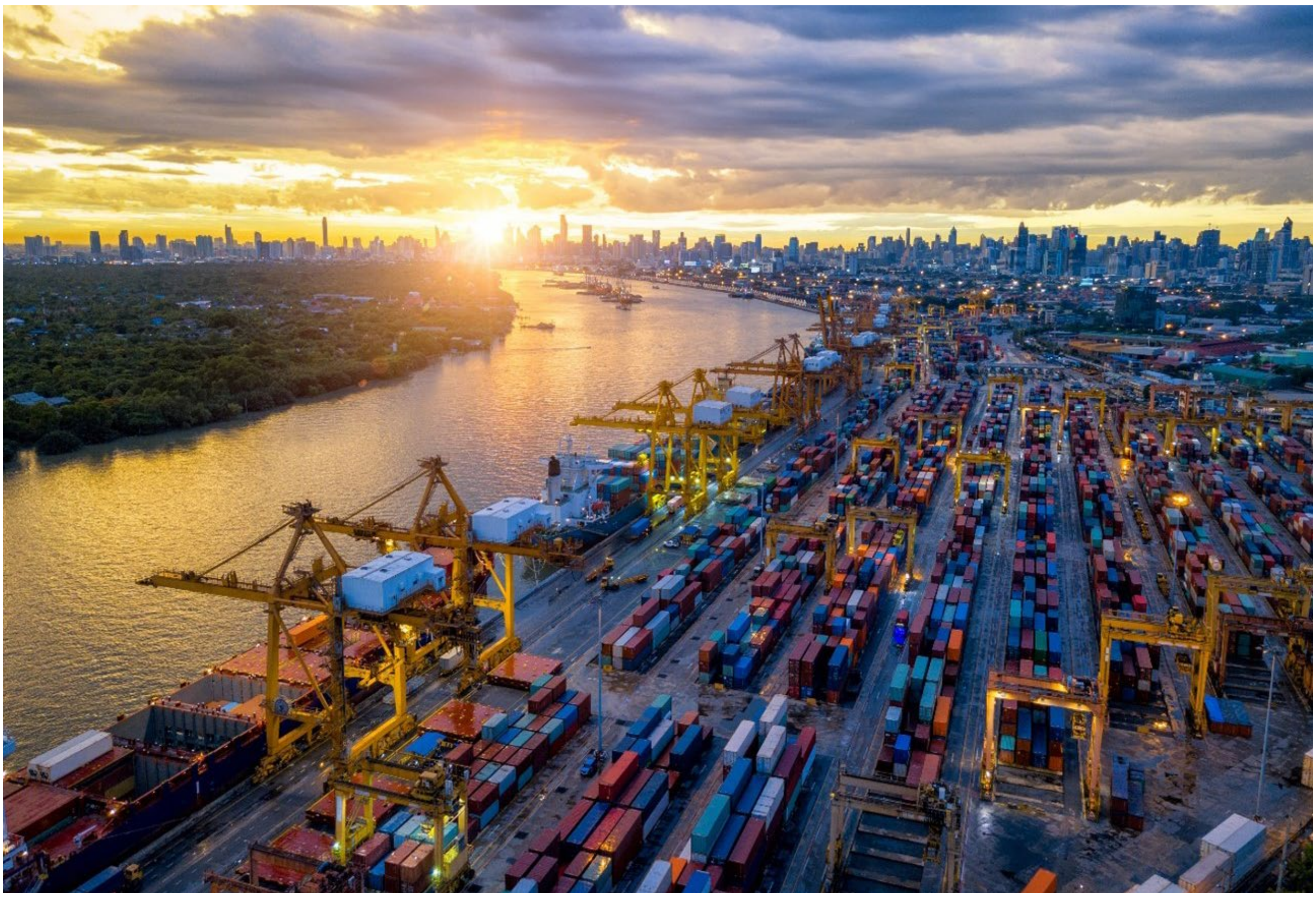
Sourcing Diversification

Empowering Your Supply Chain for Success

Article by Léa Maurel
ICAPE GROUP

In today's interconnected and rapidly evolving business landscape, sourcing diversification has emerged as a crucial strategy for organizations aiming to stay competitive and thrive in a dynamic marketplace. The traditional approach of relying on a single source for procurement is being replaced by a more sophisticated and strategic mindset that emphasizes the advantages of exploring multiple procurement channels. This approach not only opens doors to a whole different network of suppliers, but also offers numerous benefits, such as navigating sales increases, ensuring smoother deliveries, and promoting innovation within the organization.

By embracing sourcing diversification, businesses can proactively adapt to changing market conditions, mitigate risks associated with over-reliance on a single source, and tap into a vast array of resources that can fuel their growth and success. In this article, we will delve into the key aspects of sourcing diversification, highlighting its potential to revolutionize procurement practices and elevate business outcomes.



Providing Multiple Procurement Channels

Sourcing diversification encompasses the practice of establishing and nurturing relationships with multiple suppliers and vendors across various geographic locations, industries, and specialties. By expanding procurement channels, organizations gain access to a wider pool of resources and expertise, fostering healthy competition, and enabling negotiation power for better pricing and contract terms.

Navigate Sales Increases

Sudden surges in demand can be a double-edged sword for businesses. While increased sales present growth opportunities, they can strain a single supplier's capacity to meet the heightened demand promptly. By diversifying sourcing channels, companies can distribute their procurement needs across multiple suppliers, ensuring smoother sales growth and preventing potential disruptions or delays.

Smoother Deliveries

Overreliance on a single supplier exposes businesses to various risks, including production bottlenecks, logistical constraints, and unexpected disruptions. By sourcing from multiple suppliers, companies can reduce the impact of any supply chain disruptions and maintain smoother deliveries, enabling them to fulfill customer orders reliably and maintain a positive brand reputation.

Domestic vs. Global Producers

Sourcing diversification provides organizations with the flexibility to choose between domestic and global producers. While domestic sourcing ensures shorter lead times, reduced transportation costs, and supports local economies, global sourcing can offer access to lower-cost materials, specialized expertise, and a broader range of products. Striking the right balance between domestic and global sourcing can optimize cost efficiency and enhance supply chain resilience.

Nearshoring

Nearshoring, the practice of sourcing suppliers from neighboring or nearby countries, offers several compelling benefits that can significantly improve a company's supply strategy and operations.

- **De-risking the supply strategy:** Nearshoring allows businesses to reduce their dependence on a single supplier or region. By diversifying their supplier base across different regions, companies can minimize the impact of potential disruptions caused by political instability, natural disasters, or other unforeseen events, thereby ensuring a more resilient supply chain.
- **Same time-zone advantage:** One of the key advantages of nearshoring is the shared or overlapping time zones between the company and its suppliers. This synchronicity facilitates real-time communication, quick decision-making, and smoother collaboration, leading to increased efficiency and reduced delays in the supply chain.
- **Reduced lead times:** With suppliers located nearby, nearshoring enables shorter transportation distances and faster delivery times. The reduced lead times enhance the company's ability to respond promptly to changing market demands and minimize inventory holding costs,

providing a competitive edge in the market.

- **Reduced order liability:** Nearshoring helps in lowering the risk of order liabilities and financial exposure. By having multiple suppliers in different regions, a company can distribute its production volume, thereby avoiding over-reliance on a single supplier and mitigating the impact of potential disruptions on its business operations.
- **Increasing supply flexibility:** Nearshoring provides companies with greater supply chain flexibility. In times of increased demand or unexpected changes in the market, having suppliers nearby allows for more agile and responsive adjustments to production levels, helping to meet customer needs without delays.

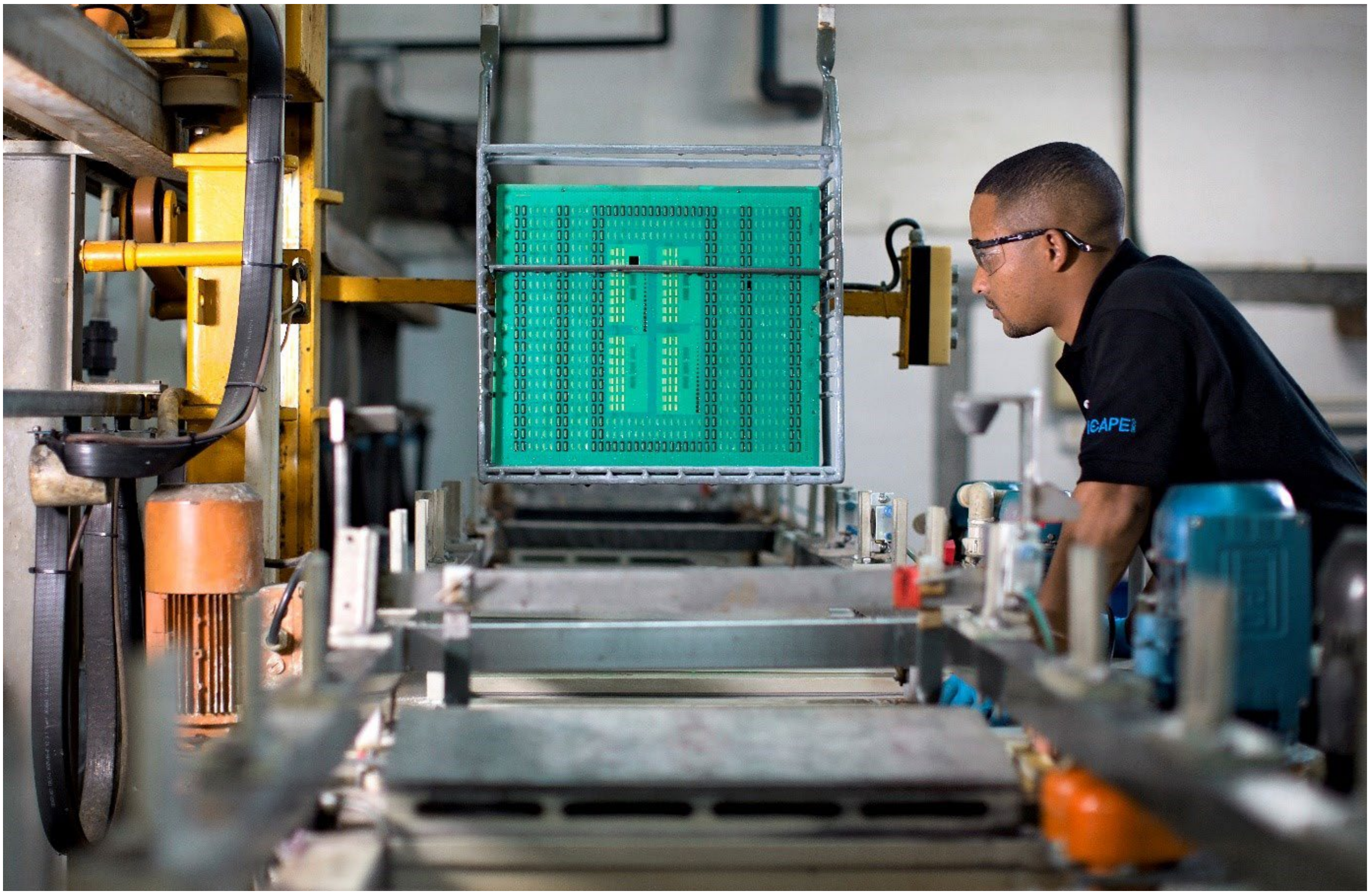
Promoting Innovation

A diverse supplier base can spark innovation within an organization. By engaging with suppliers from different backgrounds and industries, businesses can benefit from fresh perspectives, novel ideas, and unique approaches to problem-solving. Collaborative partnerships with diverse suppliers foster a culture of innovation and drive the development of groundbreaking products and services.

Diversifying Sourcing Beyond China: Benefits and Risks

The pandemic has highlighted the weaknesses of global supply chains and the lack of manufacturing autonomy in individual countries, particularly when it comes to sourcing essential medical supplies like personal protective equipment, ventilators, and medicines. The production of such items is largely concentrated in China. In addition, the U.S.-China tariffs have impacted the supply chain.

However, even before the pandemic, some companies in Europe and North America were already considering diversifying their sources of supply and reshoring production to their original sites. This was driven by factors such as increasing protectionism through tariffs, rising labor costs, and calls for environmental sustainability. As a response to COVID-19, many multinationals have been prompted to think about diversifying their supplier base, looking beyond China. For example, Foxconn, Apple's



contract manufacturer with factories in China, announced its intention to diversify its supply sources in late 2020, joining other companies like Samsung and Mazda in their efforts to enhance supply chain resilience. Moreover, Taiwan Semiconductor Manufacturing Co., another Apple supplier, decided to construct a manufacturing plant in Arizona to avoid heavy reliance on China and to serve U.S. customers directly.

Although China rebounded from production stoppages earlier than expected, the disruption had cascading effects on trading partners who faced their production problems. Consequently, a Gartner, Inc. survey in 2020 indicated that 33% of businesses have moved some sourcing and manufacturing activities out of China or plan to do so in the next two to three years.

The PCB production in Asia (except China and Japan) grew by 5.9% in 2022—especially in Taiwan and South Korea which together posted over 9% growth in PCB production. Southeast Asia seeks to become an increasingly active player in the PCB manufacturing sector in the coming years as companies seek to diversify their risk profiles in the face of increasing geopolitical uncertainty. Thailand, Vietnam, and Malaysia are particularly well-positioned to benefit from this trend.

Our Commitment to Streamlining Your Procurement Journey

Partner factories around the world

At ICAPE Group, we take pride in offering our clients a diverse and extensive network of partner factories spanning different continents. With 27 PCB factories and 61 technical part factories, we have strategically established partnerships in Europe, the Americas, Africa, and Asia, ensuring a global reach for our clients' needs.

In Europe, our partner factories are in Portugal, Slovenia, and Macedonia, providing proximity and accessibility to cater to clients in the

region efficiently. Meanwhile, in the Americas, we have partner factories in the U.S. and Mexico enabling us to serve clients in North, Central, and South America effectively.

In Asia, we have established strong relationships with partner factories in Taiwan, Thailand, Vietnam, China, and South Korea. This allows us to leverage the capabilities and expertise of manufacturers in this dynamic region, providing clients with a competitive edge and access to cutting-edge technologies.

More than a supplier, we are a manufacturer

While our extensive network of partner factories sets us apart, we go beyond traditional PCB suppliers. We are also a PCB manufacturer with two company-owned factories dedicated to quick-turn services in Sweden and South Africa. Additionally, we possess a company-owned keyboard factory in France, showcasing our commitment to innovation and offering comprehensive solutions to our clients.

Proactive purchasing team to combat price and volume disruption

To ensure competitive pricing and stability in the supply chain, our dedicated team of buyers closely monitors raw material prices. Through direct contact with manufacturers of copper-clad laminates in China and Taiwan, we stay informed about price trends, enabling us to take proactive measures in minimizing potential price disruptions.

Local warehousing that spans the world

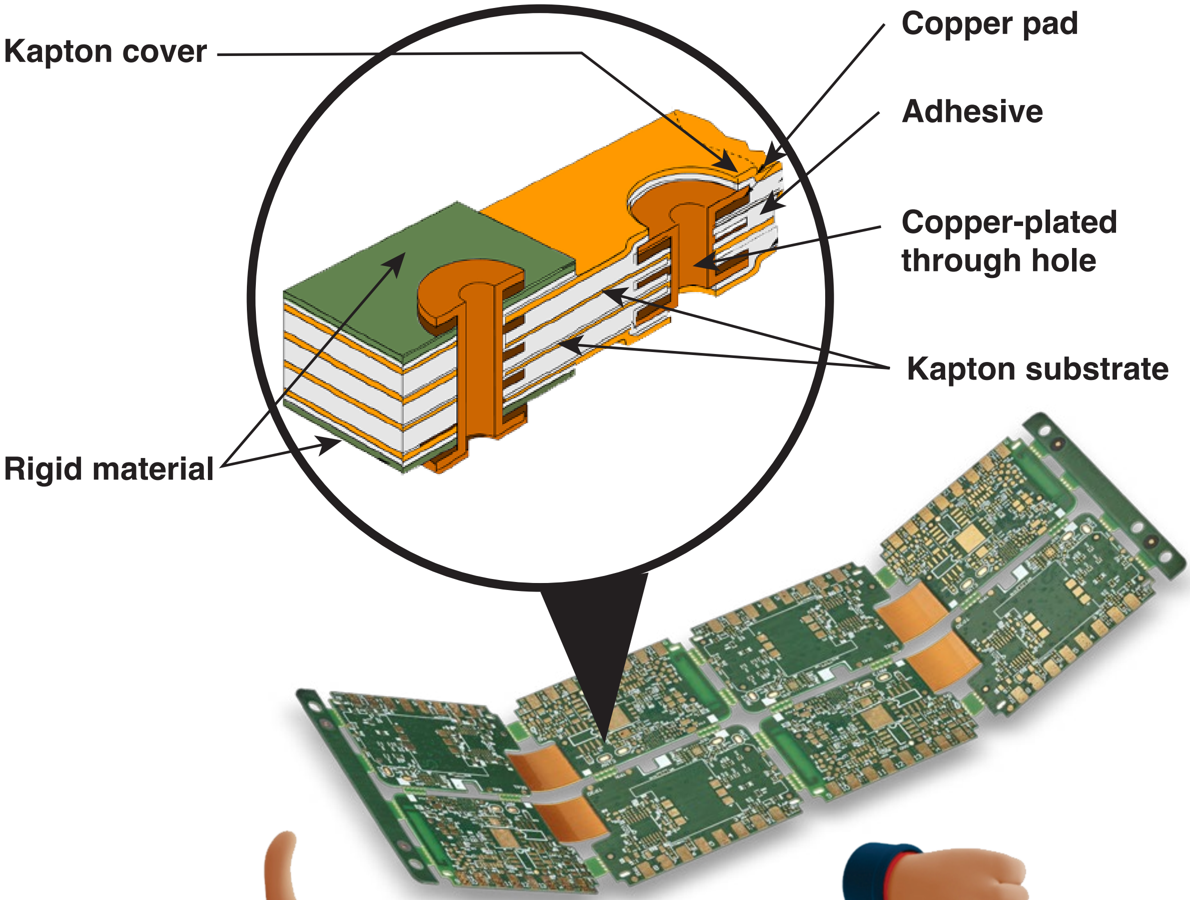
We prioritize client satisfaction and seamless service delivery. To achieve this, we maintain local warehouses near our customers, facilitating quick and efficient distribution. This approach allows us to respond swiftly to clients' demands and provide timely support whenever needed.

Sourcing Diversification: Your Path to Resilience and Success

As the business landscape continues to evolve in complexity and uncertainty, organizations must recognize sourcing diversification as an indispensable tool for sustainable growth. By weaving a tapestry of procurement channels, companies not only widen their network but also fortify themselves against the volatility of the market. This approach enhances supply chain resilience, ensuring continuity even in the face of disruptions. Moreover, the practice fosters an environment of innovation, as exposure to various sources of input sparks creativity and adaptability. In the pursuit of long-term success, embracing sourcing diversification isn't just an option—it's a strategic imperative that empowers businesses to thrive amidst change. DESIGN007

Léa Maurel is Americas marketing manager at Icape Group.

Anatomy of Rigid-flex



Advantages

Denser PCB packaging gives designer more options

Denser system packaging

Simplified assembly reduces manufacturing costs

Disadvantages

20–50% higher cost than a PCB and 20–30% higher than flex

Lower run volumes means higher cost per unit

Can be difficult to manufacture depending on materials/layers

Human Ingenuity and the Rigid-flex PCB

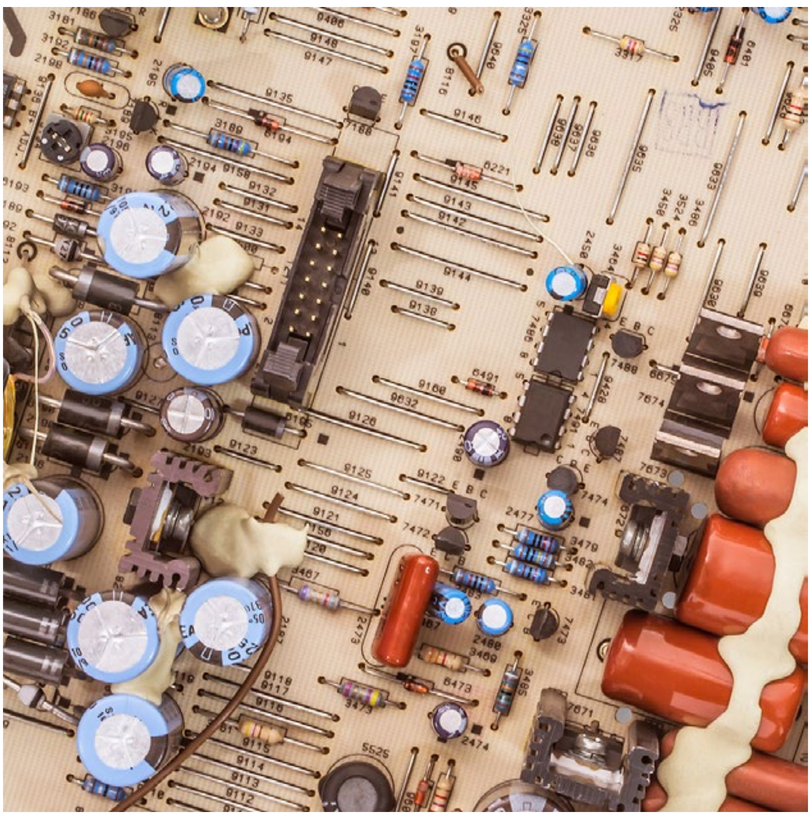
Tim's Takeaways

Feature Column by Tim Haag, FIRST PAGE SAGE

I'm always amazed by human ingenuity. For example, when Frank Walker in the 2015 movie "Tomorrowland" was asked why he made a jetpack, he responded, "I guess I got tired of waiting around for someone else to do it for me."

I'm inspired by that kind of drive and determination in life. I suspect it's why most of us entered the engineering field in the first place. But even though we can trace many advances in technology to various personal motivations, ultimately it has been necessity that serves as the mother of invention. Just look at our own printed circuit board design and manufacturing as an example.

Although the development of the printed circuit board is a story rich in historic and scientific significance, in general it can be said that the creation of PCB technology was fueled by necessity. The roots of our industry go back a hundred years to when electronic products were hand-built with point-to-point wiring connections. As you can imagine, this manual process was very tedious and time-consuming. However, with the introduction of the circuit board, radio and other electronics, manufacturers were able to simplify and speed up their processes. As this new technology matured, the circuit board printing process—and the materials used for substrates—evolved rapidly, making PCBs ideal for military and other advanced applications.



Example of point-to-point wiring connections.

You know the rest of the story. Electronic components evolved by increasing their pin counts and decreasing in size while PCB manufacturing technologies responded with advancements in multiple board layers and new component mounting processes. Today, the modern circuit board bears little resemblance to what was invented 100 years ago, which, if memory serves, were actual boards with holes drilled in them for component wires. But the core pur-


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

pose remains the same: to provide a stable platform for electronic circuits that simplifies their manufacturing through more efficient processes. But necessity continued to drive invention, and human ingenuity responded with the next step—the rigid-flex circuit board.

Interestingly, electronic circuits started out as flexible boards 100 years ago when you consider that they were all hand wired from point to point. However, these circuits were large, bulky power hogs that generated tons of heat, and could never be used for anything more advanced than the family radio. If you've ever had the opportunity to crack open a really old radio or TV set, you know how brittle and dirty those things could be. As electronic devices shrank in size and grew in functionality, the need for flexible circuitry continued to grow. Once again, the great minds of the day got back to work.

Starting in the 1950s and '60s, the advanced needs of military and aerospace applications demanded a method of connecting electronics together that didn't rely on traditional cabling and connectors. This eventually resulted in rigid-flex PCBs being used in aircraft to reduce weight and increase reliability. Just as with traditional printed circuit boards, flexible PCB technology changed and grew over the years in response to the increasing demands and requirements of the industry. From simple circuit patterns on film, flex circuit technology has evolved to include multiple substrate materials, ultra fine line width and spacing, and multilayer embedded components. Flex circuitry has now become an essential part of nearly every aspect of electronics manufacturing. Consider the following:

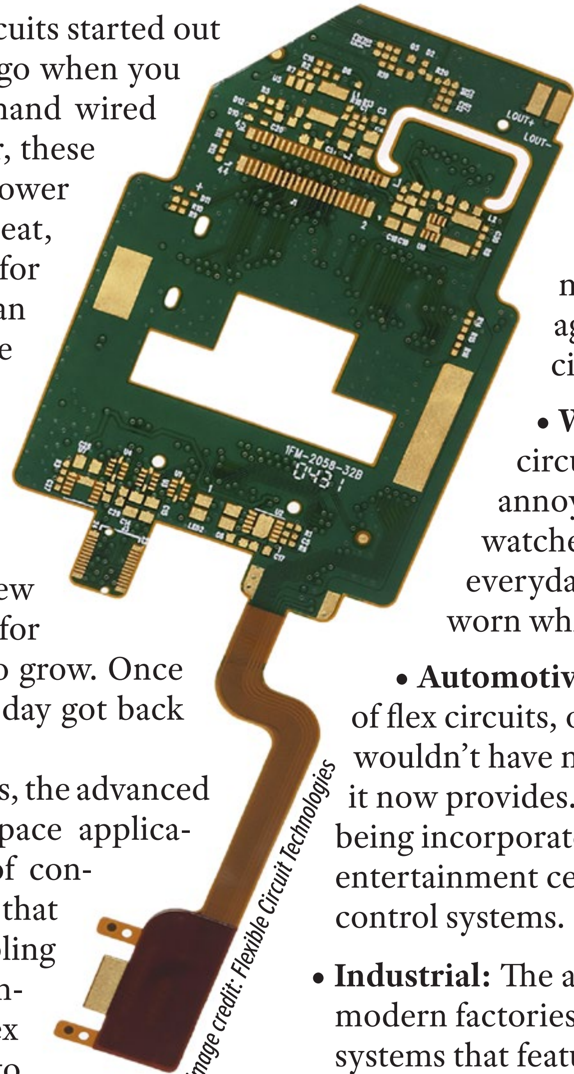


image credit: Flexible Circuit Technologies

- **Computers:** To provide direct connections between displays, keyboards, and other major components, laptops rely on flex circuits for high-performance interconnectivity that is small, robust, and reliable.
- **Mobile devices:** I'm not sure, but I think that the iconic flip-phone is back in style again, and that just won't happen unless manufacturers can leverage the latest in flexible circuitry.
- **Wearables:** Thanks to flex circuits, we can avoid the annoyance of bulky and heavy watches, earbuds, and other everyday devices designed to be worn while we use them.
- **Automotive:** Without the benefit of flex circuits, our car's electronics wouldn't have nearly the capabilities that it now provides. Flex circuitry is now being incorporated into everything from entertainment centers to engine and control systems.
- **Industrial:** The automation in today's modern factories depends on control systems that feature flex circuits. Robotics, sensors, and inventory control systems all have pieces that are constantly in motion that must be connected reliably with flexible circuitry.

Everywhere you look, flex technology is being incorporated into the electronics that are part of our daily lives. The industries listed above rely on rigid-flex PCBs to not only save space but increase their durability thanks to the new materials in use today. This makes the modern rigid-flex PCBs more reliable than traditional cable and connectors. Additionally, rigid-flex PCBs are designed for their specific application. They are crafted to precisely fit the device they are intended for instead of requiring the device be altered to fit the circuitry. At the same time, they are designed with the same constraints as a regular PCB for signal performance and EMI control.

Rigid-flex boards provide precision fit, durability, space and weight reduction, and better performance and reliability. It is tempting to use the phrase, “It really is a dream come true.” However, the importance of rigid-flex circuitry in today’s electronics is very real and has some very important shoes to fill. One of the most mission-critical uses of rigid-flex PCBs can be found in the medical field. Consider the importance of these medical devices in use today that incorporate flex technology within their designs:

> **Rigid-flex boards provide precision fit, durability, space and weight reduction, and better performance and reliability.**

- **Scanners:** Rigid-flex PCBs give ultrasound, MRI, and CT scanners the ability to be positioned correctly for diagnosing a patient. Not only is the PCB able to move as required, but the flexible portion of the design allows for placement of the device within tight areas, while the rigid sections contain the components necessary for processing.
- **Diagnostic equipment:** Depending on the application, diagnostic equipment can connect multiple sensors to their data processors. For large pieces of equipment, flex circuitry saves space over traditional cabling, while the flexible nature of the circuits allows for compact design of smaller hand-held units.
- **Surgical instruments:** Going beyond the traditional scalpel, modern surgical procedures utilize high-technology equipment such as cameras and robotics. Rigid-flex circuitry is essential for motor control and video communication and processing.
- **Defibrillators and pacemakers:** Implantable medical devices need to be able to conform to the shape and contours of a patient’s body. Rigid-flex circuitry allows for this while delivering the performance and durability that is essential for the environment that these life-saving devices are operating in.

No matter what, necessity will keep driving invention. But who knows what directions this invention will take once it is coupled with human ingenuity? I’m still waiting for neural interfaces on my CAD applications so I can route traces with my mind without ever touching the mouse. In the meantime, there are so many new and wonderful inventions happening within our industry. If you haven’t designed a rigid-flex board yet, chances are you will soon. I’m looking forward to reading more from the experts in this issue about the pros, cons, challenges, and benefits of working with rigid-flex PCBs. If you’ve got questions, this is a great place to start, and I’m looking forward to seeing what they have to say.

Until next time, 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, click here.

MilAero007 Highlights

And Now, a Word About Sustainability With Stanley Merritt, Northrup Grumman ▶

IPC Community spoke with several members of IPC's Sustainability for Electronics Leadership Council on their company's sustainability mission, reasons for joining IPC's Council, and future Council leadership projects.

Mil-Aero Design: Not Just Another High-rel Board ▶

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

Lockheed Martin: Flying High With Digital Twin ▶

While at a conference in Dallas, Barry Matties listened to a presentation on digital twin from Don Kinard, a senior fellow at Lockheed Martin. Later, we reached out to Don, who was happy to provide a deeper understanding of the role of the digital twin in the manufacturing space. What does digital twin mean when the product your manufacture is an eight- or nine-figure combat aircraft packed full of electronics?

Good News for U.S. National Security ▶

In case you missed it, President Joe Biden recently issued a presidential determination that prioritizes the domestic development of printed circuit boards (PCBs) under the Defense Production Act (DPA). Translation: It is now a tenet of U.S. policy that manufacturing more of the building blocks of modern electronics in the United States is essential to America's economic and national security.

American Made Advocacy: Taking the Fight to Capitol Hill ▶

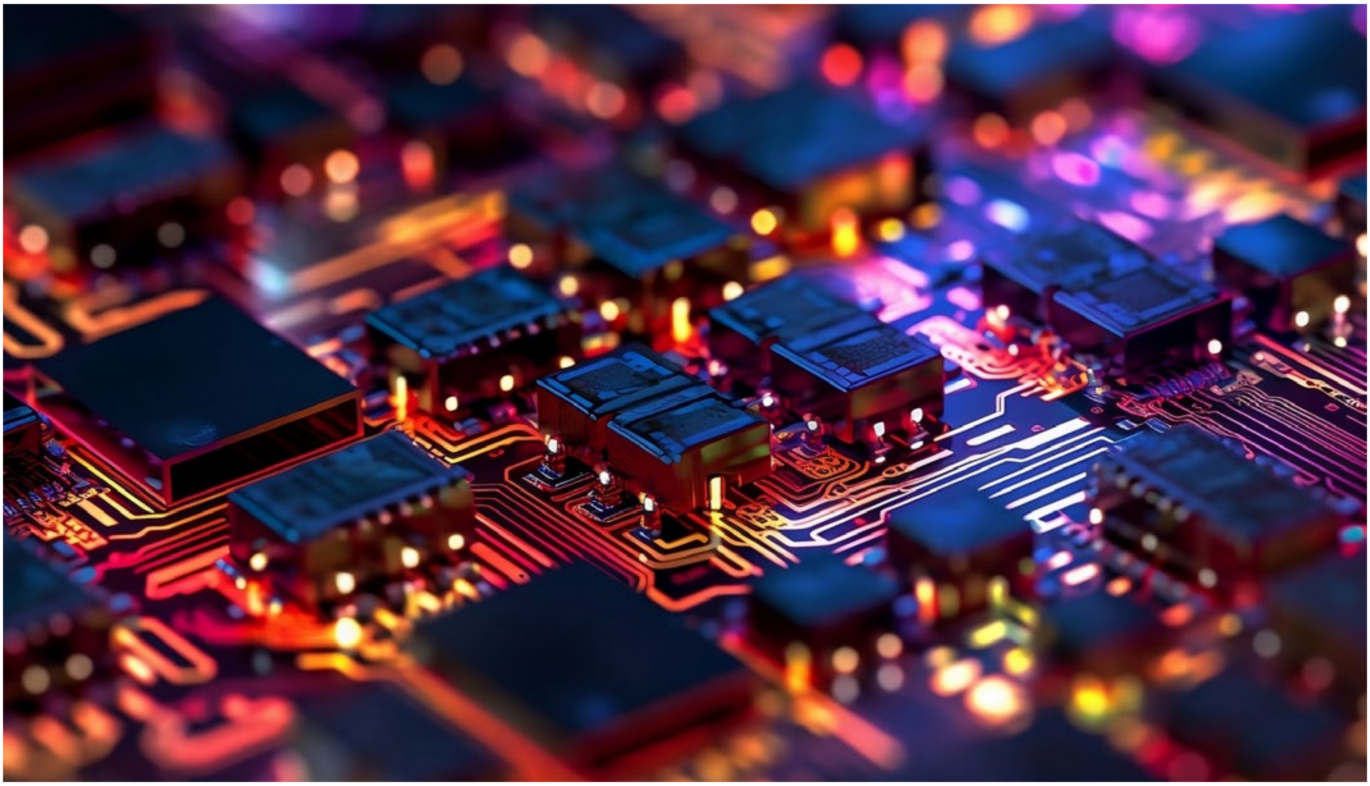
PCBAA hosted its second annual meeting, June 13–14, in Washington, D.C. It was great to see our founding members as well as many new corporate and individual members. On the first day, we heard from senior officials at the Departments of Commerce and Defense, as well as several members of the House and Senate. We spent the second day on Capitol Hill lobbying for the Protecting Circuit Boards and Substrates Act (PCBS).

Northrop Grumman Opens Taipei Office, Names Country Executive ▶

Global aerospace and defense leader Northrop Grumman has opened an office in Taiwan to accelerate access to the company's technologies and strengthen partnerships with customers and local industry.

From AI to Nuclear: The Technologies Driving UK Space Exploration ▶

The Space Exploration Technology Roadmap will guide research and development activity and future funding decisions over the next decade, putting the UK's growing space sector in a stronger position to collaborate with international partners including NASA, the European Space Agency and JAXA (Japan's space agency).



A Primer on UHDI

Article by Anaya Vardya
AMERICAN STANDARD CIRCUITS

There has always been pressure to reduce line and space as we have seen the bleeding edge technology go from 8 to 5 mils and then to 3 mils. The difference between then and now is that the prior advancements, for the most part, used the same processes, chemistry, and equipment going from 8 mils to 3 mils. But going from 3-mil to sub 1-mil trace and space is a quantum leap in printed circuit board (PCB) technology that requires a whole new set of processes and materials.

High density interconnect (HDI, the predecessor of UHDI) deals with line width and space, but primarily employs via structures to increase density. In broad terms, HDI printed circuit boards are defined as PCBs with one or more of the following via structures: microvias, stacked and/or staggered microvias, buried and blind vias, and all with sequential lamination cycles. Printed circuit board technology has been evolving with changing technology that demands smaller and faster features. HDI boards allow smaller vias, pads, lines and spaces—in other words, higher density. This increased density also allows a reduction in the number of layers and a smaller footprint. For example, one HDI board can house the functionality of multiple standard technology PCBs. Conventional state-of-the-art technology has been stuck at the 3-mil line and space capability for the longest time, but that is just not good enough to meet the increasingly tighter real estate constraints of today's products. That is where Ultra HDI comes in.

What is UHDI?

As soon as we cross below the 1-mil (0.001") line-width threshold, we need to stop talking in terms of mils and ounces and begin talking in terms of microns. For reference, a 3-mil trace is 75 microns, so a 1-mil trace is 25 microns. In general terms, ultra high-density interconnect (UHDI) refers to lines and spaces on a printed circuit board that are sub-25 micron. As electronics continue to shrink, so does the printed circuit board, not only in the X- and Y-axes, but also the Z-axis. Designers are challenged with reducing the form factor as well as the thickness of printed circuit boards to meet these demands.

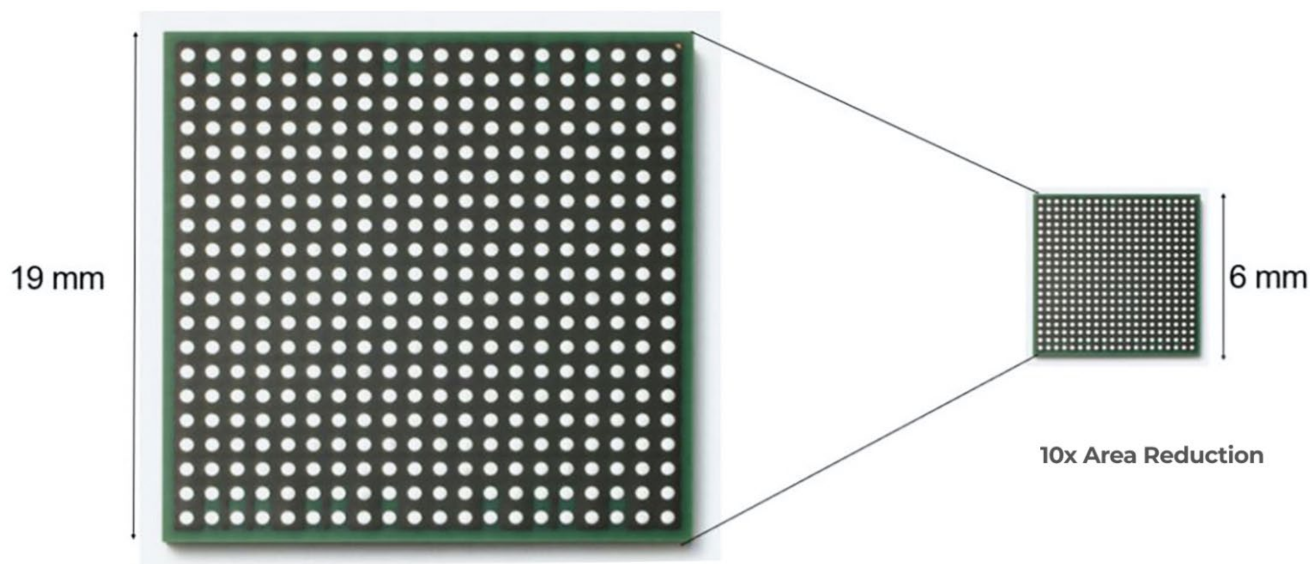
Subtractive vs. Additive

Subtractive technology is how we have primarily produced printed circuit boards since their inception. The subtractive process refers to the selective building up of traces and features on a copper-clad substrate through plating processes, and then removing, or subtracting, the base copper to leave the circuit pattern. The limiting factor of the subtractive process is the thickness of the base copper, typically 5-micron, or 2-micron ultra-thin foil. This base copper thickness defines the minimum trace that can be achieved through the etching process. That is the technology that got us as an industry to the 3-mil line and space. The etched sidewall of the trace is tapered, not straight, with the taper getting larger the closer you get to the base substrate. The amount of taper is defined by the copper weight; the higher the weight, the more the taper. This limitation is what drove the development of UHDI technology.

UHDI additive technology starts with unclad substrate and adds an ultra-thin 0.2-micron layer of liquid ink to the substrate. The selective trace pattern is applied and the circuit pattern is then built up through the plating processes. The game-changing difference here is only having to etch 2 microns of base material, which results in perfectly straight trace sidewalls.

SAP and mSAP

The semi-additive process (SAP) and modified semi-additive processes (mSAP) have



0.8mm pitch 20 x 20 grid
3 routing layers using standard PCB with L/S 3mil/3mil

0.3mm pitch 20 x 20 grid
3 routing layers using A-SAP™ PCB with L/S 1mil/1mil

Source: Averatek Corporation

been around for a while and have brought the capability of line width and space down to the 1–3 mil level. mSAP began in the IC substrate industry but has proliferated throughout PCB manufacturing shops for use with HDI products. These processes utilize a layer of base copper on the substrate between 5 and 2 microns to achieve line width reductions. However, production of sub-1-micron traces requires the ultra-thin 0.2-micron layer of liquid ink used in the A-SAP process.

A-SAP

The pioneer of UHDI processing is Averatek, which has brought its A-SAP™ process to market. A-SAP stands for "Averatek Semi-Additive Process" and is the industry leader in this technology. American Standard Circuits has partnered with Averatek on the technology which allows for the manufacture of PCBs with line width and space down to 15 microns. Some of the benefits of using A-SAP are:

- Significant size and weight reduction
- Improved reliability and signal integrity
- Improved RF performance
- Reduced costs
- Biocompatibility

These processes have allowed devices like cellphones to continually shrink in size while increasing functionality. As technology continues to push everything smaller and faster while integrating with other devices, UDHI is here for the foreseeable future.

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

Fast Electrical Signals Mapped in Plants With New Tech

Most people know that the nervous system in humans and other animals sends electric impulses. But do plants also have electrical signals even though they lack a nervous system? Yes, plants have electrical signals that are generated in response to touch and stress factors, such as wounds caused by herbivores and attacks on their roots.

It turns out that in some plants electrical signals are correlated with rapid movements. The carnivorous plant Venus Flytrap (Dionaea muscipula) is used by researchers as a model system for fast electrical signalling in plants.

Electrical signaling in living organisms is based on a difference in voltage between the inside of cells and the outside environment. This difference in voltage is created when ions, i.e., electrically charged atoms, are moved between the inside and the outside of the cell.

There is ample knowledge about how nerve impulses function in humans and other animals. But when it comes to plants, which do not have a nervous system, a lot remains to be discovered.

This new technology was developed by researchers at Linköping University in collaboration with researchers from Columbia University, who use this technology for neuroscience studies in animals.

(Source: Linköping University)

TOP TEN EDITOR'S PICKS

Beyond Design: Integrating AI Into PCB Design Flow

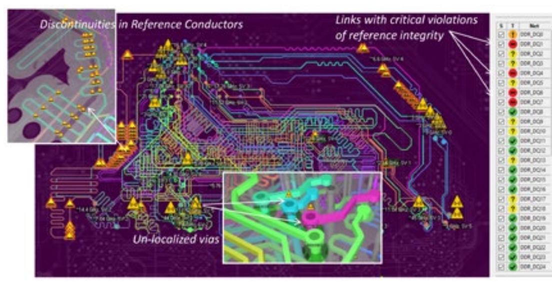
Today's PCB design tools are a far cry from the rudimentary tools we used 30 years ago, but even though the algorithms are now quite clever, they are still fairly basic as far as intelligence goes. Comprehensive design rules can be established to accommodate myriad constraints to limit placement and routing which advise us when we have overstepped the boundaries.

Book Excerpt: *'The Printed Circuit Designer's Guide to... Manufacturing Driven Design,'* Chapter 2

In a recent survey on the increasing complexity of electronics design, it was reported that 68% of companies were facing increasing pressure to miniaturize their electronics form factors. This report mirrors global trends seen within the industry. It represents an increased burden placed upon the printed circuit board designer to put more functionality into a smaller space, and on the manufacturer to produce with a smaller margin of error.

3D Electromagnetic Analysis

Data rates in PCB interconnects are increasing in all signaling protocols (PCIe, DDR, GDDR, Ethernet, USB, SAS, InfiniBand, CEI, OIF, 5G). Most of those high-speed signaling standards have one-lane data rates over 6 Gbps (GT/s) and some up to 112 Gbps with signal spectrum in microwave and even millimeter wave bandwidths.



Dana on Data: Filling the Gap When Tribal Knowledge Runs Out


Dana Korf

There are probably only a few hundred front-end engineering experts in North America and probably even fewer in the EU. So, where will the new facilities get that expertise? There aren't a sufficient number of training classes or courses in available to teach new potential employees in colleges and universities. How can we mitigate this crisis? The first step is for companies to quit sending manufacturing documentation that requires human intervention.

American Standard Circuits Celebrates 35 Years in Business

Anaya Vardya, president and CEO of American Standard Circuits, has announced that on Aug. 25, 2023, American Standard Circuits celebrates 35 years of doing business. Starting with a small facility in the Libertyville, Illinois, area, ASC has grown to be one of the major leaders in the North American printed circuit board market. In 1994, the company moved to a larger facility in Franklin Park and then to its current facility in West Chicago in 2004.

Elementary, Mr. Watson: Circuit Simulation, SPICE, and AI

Many PCB designers feel that circuit simulation is something they will never need to learn, and it is best handled by that infamous "someone else." But in our present business environment, with the lack of PCB design and engineering resources, many designers are pulling multiple duties throughout the design process.

How Far Can We Trust AI?

Sheldon Fernandez, CEO of DarwinAI, discusses the difference between true artificial intelligence and machine learning (ML), and whether we can trust what AI gives us. Is AI only as good as the training it's given by a human?

Latest Advancements in Safety and Reliability Requirements at IPC's High Reliability Forum

The critical reliability of electronic prosthetics in the brain to help restore movement to a paralyzed hand, enhancing the durability of EV automotive infrastructure, testing criteria and challenges of working with lead-free materials, and evaluation of reliability and maintainability for safe airborne structures are just a few of the agenda items offered at IPC's High Reliability Forum October 17–19 in Linthicum (Baltimore), Maryland.

APCT Now Offering Full Turnkey Solutions

With the recent acquisition of Advanced Circuits and the subsequent acquisition of San Diego PCB Design, APCT has become the second largest printed circuit manufacturer in North America, offering design, fabrication, and assembly.

The Shaughnessy Report: Simulation, Analysis, and AI

Getting today's designs "right the first time" is critical, especially with costly advanced PCBs. Companies are slowly realizing that building two, three, or seven respins into the process—and budget—is like flushing cash down the toilet. I can hear some of you thinking, "But that's how we've always done it. Why don't you knock it off with those negative waves?"

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

Career Opportunities

Find Industry-experienced Candidates at jobConnect007

For just $975, your 200-word, full-column ad will appear in the Career Opportunities section of all three of our monthly magazines, reaching circuit board designers, fabricators, assemblers, OEMs, suppliers and the academic community.

In addition, your ad will:

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

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

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

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

+1 916.365.1727

I-Connect007
GOOD FOR THE INDUSTRY

Career Opportunities

ventec INTERNATIONAL GROUP 騰輝電子

Senior Sales Representative

Ventec Central Europe

Location: Kirchheimbolanden, Germany/Remote

We are looking for a self-motivated Senior Sales Representative—Ventec Central Europe, ideally with experience in the PCB industry. This position requires significant selling experience (15+ years) in the electronics and PCB industries. Candidates must possess a proven & consistent history of proactive sales growth with OEM customers. Most notably, they must be able to connect with OEM contacts that have decision-making capabilities.

Key Responsibilities

- Promote, sell, and close business for all Ventec product lines with focus on key OEM and PCB manufacturing customers.
- Track projects and submit monthly updates to management.
- Coordinate cross-functional resources when applicable.
- Assist in coordination and set-up of relevant trade show events.
- Assist in strategic planning initiatives.
- Assist in market and customer intelligence gathering.
- Recommend pricing strategies.

Job Requirements

- Entrepreneurial spirit, positive, high energy, and desire to win.
- Proactive and self-motivated work strategy to develop and win business for all business units.
- Excellent written and oral communication skills in German and English
- Excellent computer skills (Microsoft Office, especially Excel).
- Proven track record securing new business at OEM accounts.

Please apply in the strictest confidence, enclosing your CV, to: accountingde@ventec-europe.com

apply now

ViTrox®

Technical Support Engineer

USA Region

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

Key Responsibilities:

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

Qualifications & Requirements:

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

apply now

Career Opportunities

The Test Connection, Inc.

Rewarding Careers

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

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

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

Associate Electronics Technician/ Engineer (ATE-MD)

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

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

apply now

TAIYO AMERICA

Europe Technical Sales Engineer

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

PRIMARY FUNCTION:

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

ESSENTIAL DUTIES:

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

QUALIFICATIONS / SKILLS:

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

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

apply now

Career Opportunities

Career Opportunities

KYZEN
Where Science and Care Converge.

Regional Manager

Southwest Region

General Summary: Manages sales of the company's products and services, Electronics and Industrial, within the Southwest Region. Reports directly to Americas Manager. Collaborates with the Americas Manager to ensure consistent, profitable growth in sales revenues through positive planning, deploy-ment and management of sales reps. Identifies objectives, strategies and action plans to improve short- and long-term sales and earnings for all product lines.

DETAILS OF FUNCTION:

- Develops and maintains strategic partner relationships
- Manages and develops sales reps:
  - Reviews progress of sales performance
  - Provides quarterly results assessments of sales reps' performance
  - Works with sales reps to identify and contact decision-makers
  - Setting growth targets for sales reps
  - Educates sales reps by conducting programs/ seminars in the needed areas of knowledge
- Collects customer feedback and market research (products and competitors)
- Coordinates with other company departments to provide superior customer service

QUALIFICATIONS:

- 5-7+ years of related experience in the manufacturing sector or equivalent combination of formal education and experience
- Excellent oral and written communication skills
- Business-to-business sales experience a plus
- Good working knowledge of Microsoft Office Suite and common smart phone apps
- Valid driver's license
- 75-80% regional travel required

To apply, please submit a COVER LETTER and RESUME to: Fernando Rueda, Americas Manager

fernando_rueda@kyzen.com

apply now

EMA Design Automation®

Technical Marketing Engineer

EMA Design Automation, a leader in product development solutions, is in search of a detail-oriented individual who can apply their knowledge of electrical design and CAD software to assist marketing in the creation of videos, training materials, blog posts, and more. This Technical Marketing Engineer role is ideal for analytical problem-solvers who enjoy educating and teaching others.

Requirements:

- Bachelor's degree in electrical engineering or related field with a basic understanding of engineering theories and terminology required
- Basic knowledge of schematic design, PCB design, and simulation with experience in OrCAD or Allegro preferred
- Candidates must possess excellent writing skills with an understanding of sentence structure and grammar
- Basic knowledge of video editing and experience using Camtasia or Adobe Premiere Pro is preferred but not required
- Must be able to collaborate well with others and have excellent written and verbal communication skills for this remote position

EMA Design Automation is a small, family-owned company that fosters a flexible, collaborative environment and promotes professional growth.

Send Resumes to: resumes@ema-eda.com

apply now

Career Opportunities

pluritec®
MACHINES FOR PRINTED CIRCUIT BOARDS

Field Service Engineer

Location: West Coast, Midwest

Pluritec North America, ltd., an innovative leader in drilling, routing, and automated inspection in the printed circuit board industry, is seeking a full-time field service engineer.

This individual will support service for North America in printed circuit board drill/routing and X-ray inspection equipment.

Duties included: Installation, training, maintenance, and repair. Must be able to troubleshoot electrical and mechanical issues in the field as well as calibrate products, perform modifications and retrofits. Diagnose effectively with customer via telephone support. Assist in optimization of machine operations.

A technical degree is preferred, along with strong verbal and written communication skills. Read and interpret schematics, collect data, write technical reports.

Valid driver's license is required, as well as a passport for travel.

Must be able to travel extensively.

apply now

Koh Young America

Technical Service & Applications Engineer

Full-Time — Flexible Location

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

Responsibilities

- Provide support, preventive and corrective maintenance, process audits, and related services
- Train users on proper operation, maintenance, programming, and best practices
- Recommend and oversee operational, process, or other performance improvements
- Effectively troubleshoot and resolve machine, system, and process issues

Skills and Qualifications

- Bachelor's in a technical discipline, relevant Associate's, or equivalent vocational or military training
- Knowledge of electronics manufacturing, robotics, PCB assembly, and/or AI; 2-4 years of experience
- SPI/AOI programming, operation, and maintenance experience preferred
- 75% domestic and international travel (valid U.S. or Canadian passport, required)
- Able to work effectively and independently with minimal supervision
- Able to readily understand and interpret detailed documents, drawings, and specifications

Benefits

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

apply now

Career Opportunities

ARLON ELECTRONIC MATERIALS

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

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

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

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

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

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

apply now

INSULECTRO

Are You Our Next Superstar?!

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

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

apply now

Career Opportunities

MivaTek Global

Field Service Technician

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

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

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

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

More About Us

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

apply now

eptac
TRAIN. WORK SMARTER. SUCCEED.

Become a Certified IPC Master Instructor

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

Qualifications and skills

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

Benefits

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

apply now

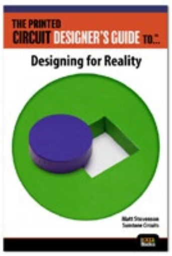
Career Opportunities

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

NEW! Manufacturing Driven Design

by Max Clark, Siemens

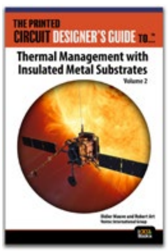
This book introduces a new process workflow for optimizing your design called Manufacturing Driven Design (MDD) and is a distinct evolution from DFM. When defining Manufacturing Driven Design, it is important to recognize that this is, foremost, an element of the design stage. Manufacturing certainly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



Designing for Reality

by Matt Stevenson, Sunstone Circuits

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



Thermal Management with Insulated Metal Substrates, Vol. 2

by Didier Mauve and Robert Art, Ventec International Group

This book covers the latest developments in the field of thermal management, particularly in insulated metal substrates, using state-of-the-art products as examples and focusing on specific solutions and enhanced properties of IMS. Add this essential book to your library.

High Performance Materials

by Michael Gay, Isola

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

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

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

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

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

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

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

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

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

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

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

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

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

ART DIRECTOR/PRODUCTION MGR. **SHELLY STEIN**
shelly@iconnect007.com

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

TECHNOLOGY SPECIALIST: **BRYSON MATTIES**

COVER DESIGN: **SHELLY STEIN**

COVER IMAGE: **ADOBE STOCK © DEAGREEZ**

DESIGN007 MAGAZINE

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

September 2023, Volume 12, Number 9
DESIGN007 MAGAZINE is published monthly
by IPC Publishing Group, Inc., dba I-Connect007

ADVERTISER INDEX


Problems solved!
PCB007 MAGAZINE JUNE 2023
Growing Your Workforce
PCB007 MAGAZINE JULY 2023
Winning With TQM
PCB007 MAGAZINE
Developing High-flying Strategies
DESIGN007 MAGAZINE JULY 2023
Emerging Design Strategies
DESIGN007 MAGAZINE AUGUST 2023
SIMULATION • ANALYSIS • AI
DESIGN007 MAGAZINE SEPTEMBER 2023
RIGID-FLEX DESIGN
SMT007 MAGAZINE AUGUST 2023
FLYING HIGH WITH DIGITAL TWIN
SMT007 MAGAZINE SEPTEMBER 2023
PAGE 68: PRESHOW SECTION SMTA INTERNATIONAL 2023 CONFERENCE & EXHIBITION
SUBSCRIBE NOW