I-Connect007

AUGUST 2022

DESIGN007 MAGAZINE

LESS IS MORE

I-007eBooks

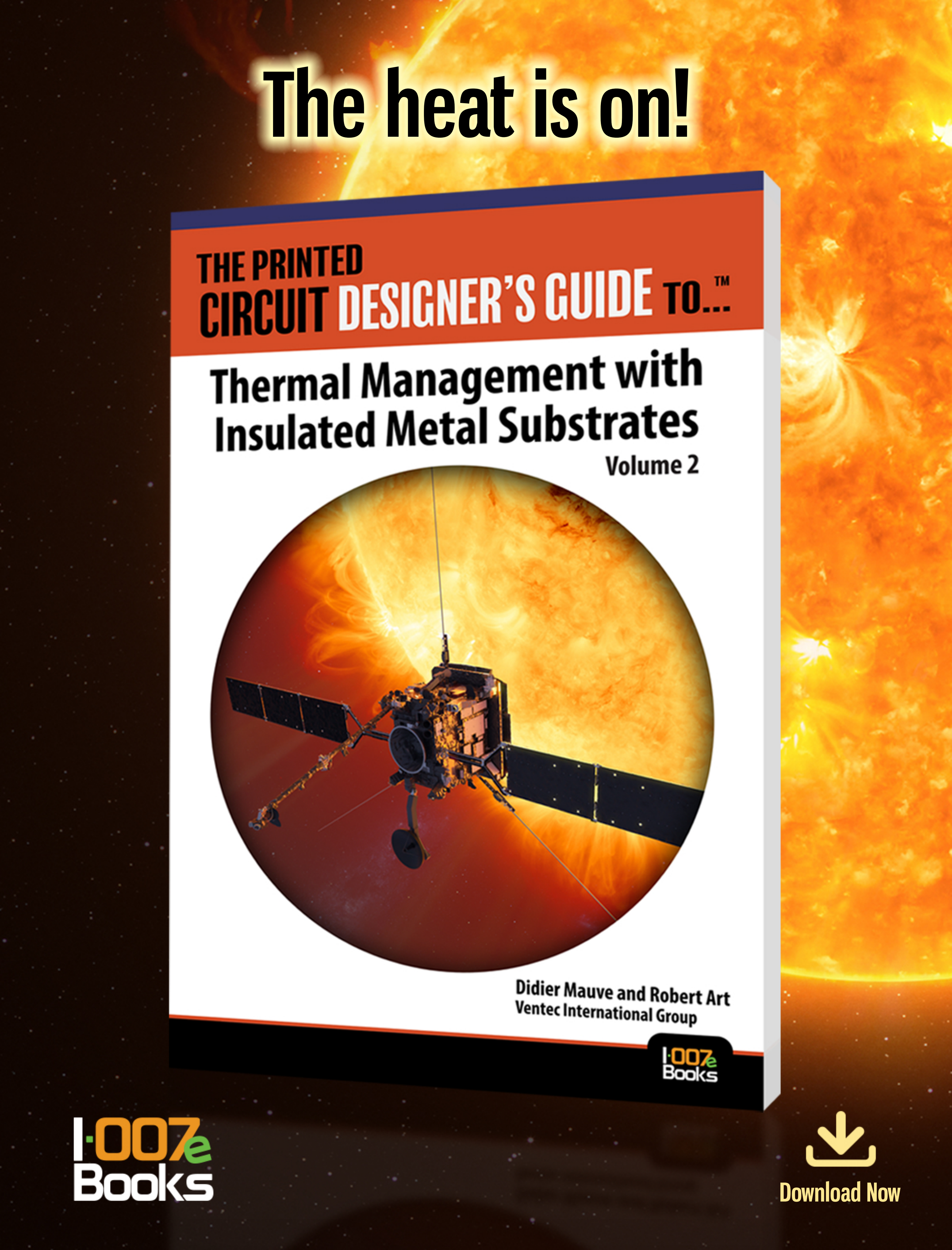
Distinctly different.

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

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

Stephen V. Chavez
PCEA Chairman, MIT, CID+

VISIT OUR LIBRARY



The heat is on!
THE PRINTED CIRCUIT DESIGNER'S GUIDE TO...™
Thermal Management with Insulated Metal Substrates
Volume 2
Didier Mauve and Robert Art
Ventec International Group
I-007e Books
I-007e Books
Download Now

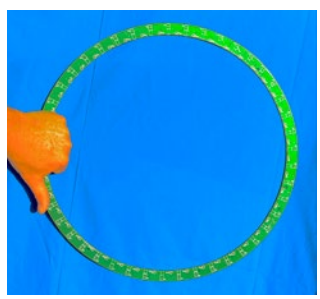
DESIGN007 MAGAZINE

Less Is More

We're seeing all sorts of interesting tactics for dealing with 50-week lead times. One of the most basic concepts that we've heard about lately is material conservation—why not just design PCBs with fewer parts? As we point out in this issue, sometimes less is more. Our experts share a variety of strategies and technologies to help reduce your overall material consumption, allowing you to lower costs and add competitive advantage.

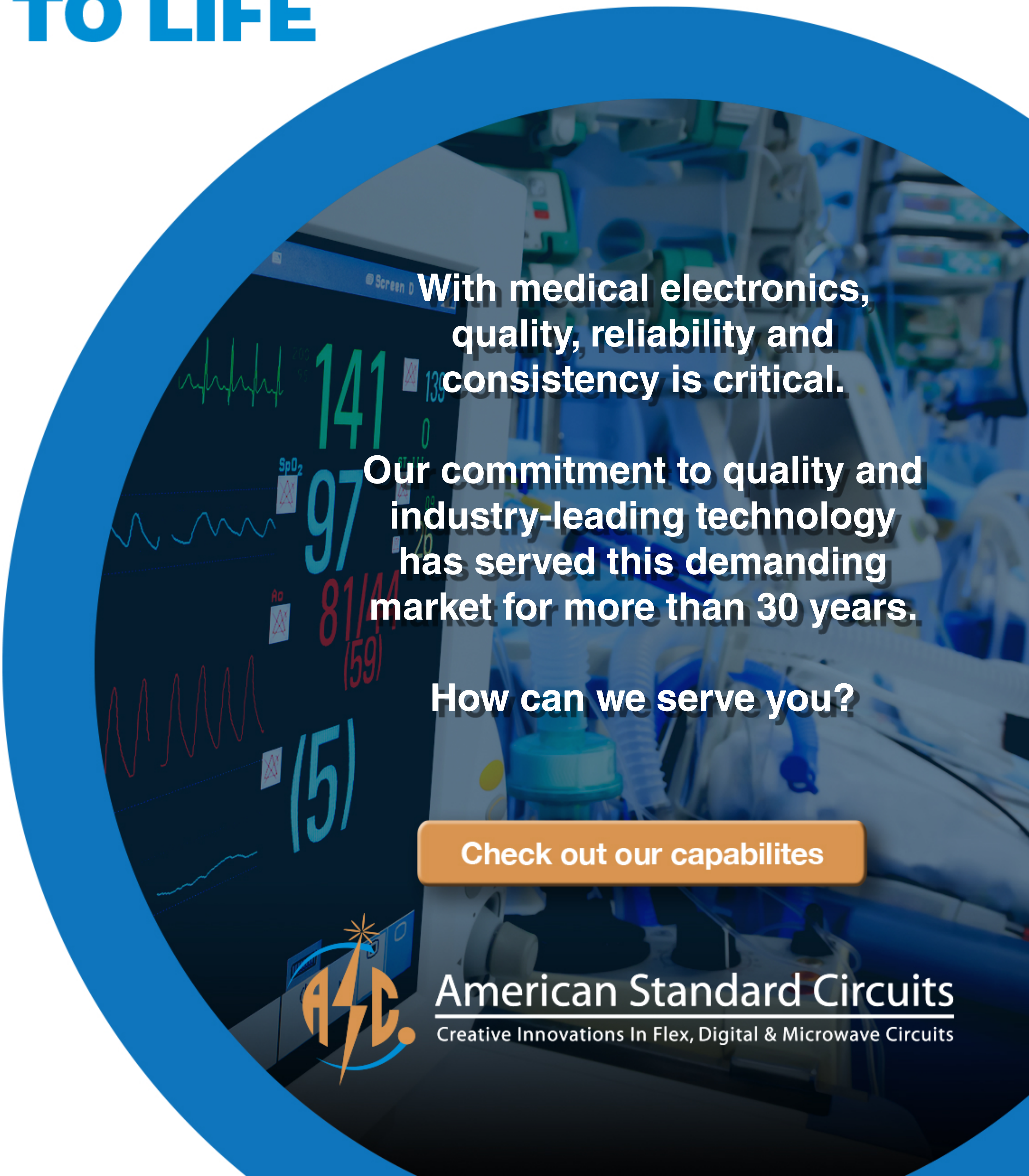
FEATURE INTERVIEWS & ARTICLES

FEATURE COLUMNS



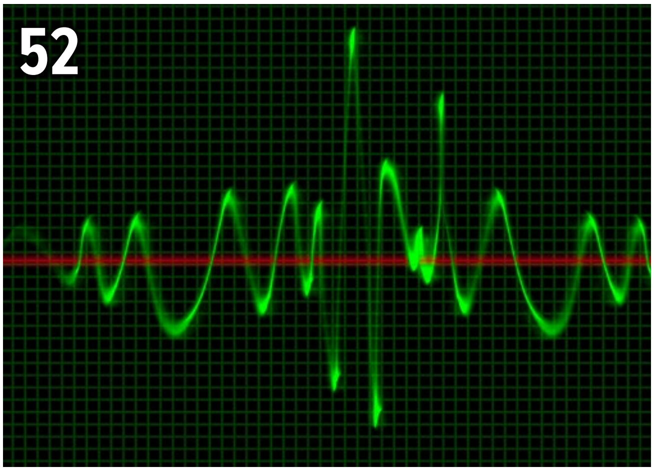
asc-i.com

WE BRING MEDICAL ELECTRONICS TO LIFE

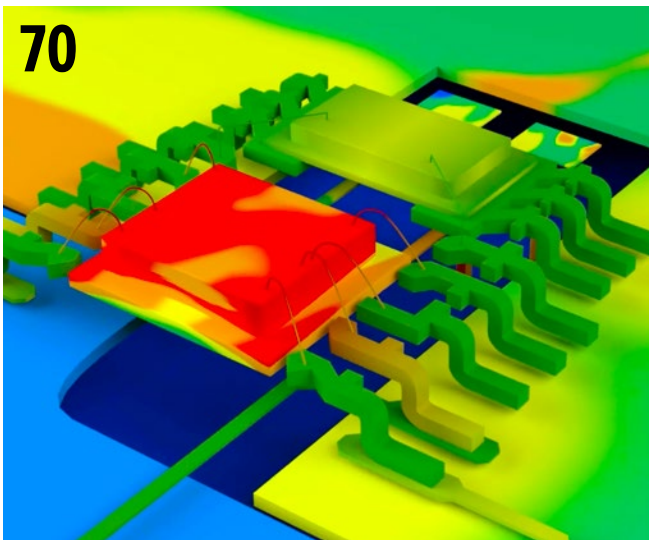


DESIGN007 MAGAZINE

COLUMNS



ARTICLES & INTERVIEWS



HIGHLIGHTS

SHORTS

DEPARTMENTS

PegaClad Series

for Sub-6 GHz Antenna (Dk 3.00, 3.88, 3.45, 3.65)

PegaClad 1&2

for mmWave

Developed for Sub-6 GHz antenna applications, TUC's PegaClad 300, PegaClad 338, PegaClad 345, and PegaClad 365 - with respective Dks of 3.00, 3.38, 3.45, and 3.65 - are advanced materials designed to minimize passive intermodulation and meet designers' needs. PegaClad 300, PegaClad 338, PegaClad 345, and PegaClad 365 can be used in multilayer and double-sided circuit board designs requiring excellent thermal reliability.

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

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

tuc

Taiwan Union Technology Corporation

www.tuc.com.tw

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

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

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

Designing for Material Conservation

The Shaughnessy Report

by Andy Shaughnessy, I-CONNECT007

The supply chain issues plaguing our industry don't seem to be going away any time soon. Like an annoying mother-in-law, they've moved into our guest room, rearranged the furniture, and generally overstayed their welcome. Why don't they take a hint?

We're seeing all sorts of interesting tactics for dealing with 50-week lead times. One of the most basic concepts I've heard lately is material conservation—when it's hard to get the parts you need, why not just design PCBs with fewer parts? Materials typically make up 20% of the cost of the board, so we're not talking nickels and dimes.

It seems like a simple idea: Just design boards with fewer components and less laminate. Do you really need all those decoupling capacitors? And, as Happy Holden points out in this issue, there's no real reason that most boards are still 0.062" thick. They were originally that thick because they had to plug into motherboards, but why are boards in our handheld devices still that same thickness?

It's because we've always done it that way. Maybe it's time to consider something different.

It's just common sense. If the industry designed boards with fewer parts and less laminate, the savings could be in the tens of millions of dollars. Thinner PCBs with fewer components could mean improved signal integrity, but there are a lot of tradeoffs to investigate.

Obviously, this would change everything. Designers are accustomed to crafting PCBs a certain way. It would require looking at design in a completely new way. Are we ready? Judging from the prognostications of the supply chain experts for 2023 and 2024, we may not have a choice.

In this issue, we share a variety of strategies and technologies to help reduce your overall material consumption during the PCB design cycle. This will help lower costs, streamline the supply chain, and add competitive advantage.

We start out with a conversation with Happy Holden, who explains why risk-averse designers have not traditionally tried to conserve materials in their designs, and why it's time for designers to embrace new technologies such as VeCS. Happy includes some tips on material conservation, as well as his relative cost index (RCI) that helps designers figure out the cost per square inch of a new design. Columnist Barry Olney discusses a raft of design strategies that can cut costs, including choosing the correct laminate and using simulation early in the design cycle. Next, Cherie Litson shares her tips for lowering the cost of manufacturing your board, and as she points out, sometimes reducing layer count is actually a bad idea.

Alun Morgan provides a great macro view of the whole supply chain situation. As he points out, even if the container ship bottleneck opened tomorrow and the components we needed were suddenly available, it would still take months for these parts to reach their final destination. And the docks are still trying to fill open positions. Columnist Dana Korf walks us through how we got into this mess as he explains why designing for conservation may be at least part of the solution. Columnist John Watson discusses some new design techniques for these inflationary times (DFI, anyone?). One of his best pieces of advice: Stop making knee-jerk decisions.

Columnist John Coonrod takes us through the benefits of hybrid multilayer construction, which can increase reliability while cutting costs. And columnist Kelly Dack pontificates about the importance of avoiding component "logjams" by maintaining practical packaging density during the design phase. We also feature columns from many of our regular contributors, and welcome new columnist Beth Massey from Electrolube. You'll also find our printed electronics roundtable and my review of Cadence's thermal integrity webinar.

Our Big News

As many of you have heard, IPC has acquired I-Connect007. It makes a lot of sense; we've been their media partner at IPC APEX EXPO for over a decade, and we enjoy working with them. Not much has changed, really. Barry Matties is still our publisher, but now he might be able to take a day off every now and then.

See you next month! DESIGN007

Andy Shaughnessy is managing editor of *Design007 Magazine.* He has been covering PCB design for 20 years. To read past columns, click here.

Material Conservation Demands Stakeholder Buy-in

Feature Interview by the I-Connect007 Editorial Team

With the supply chain problem showing no signs of letting up, the idea of designing PCBs with fewer materials has begun to take hold. So, as we mapped out this issue on conserving materials, we knew it was important to hear from Happy Holden about this topic. Here's why:

Happy has been a proponent of cost-aware PCB design for decades. At HP, Happy helped develop the relative cost index (RCI), which allows PCB designers to compare the costs of PCB structures and their alternatives (see page 18). With this in mind, we spoke with Happy about the potential benefits of designing for material conservation, and why many of the old design concepts may be ripe for updating, such as the 0.062" board.

Barry Matties: Happy, in the past you mentioned that 75% to 80% of the cost of the board is controlled by design and just 20% by the fabricator. To help designers control cost, you developed a relative cost index. Tell us about that.

Happy Holden: We came up with the relative cost index because it was almost impossible to get fabricators to provide cost predictors. First, they didn't want to have people hold their feet to the fire in case you left something out. Second, they were unsure if the information would get out to their competitors. So, we came up with the relative cost index. It allows you to compare architecture design alternatives as a percent savings or percent increase in cost from that fabricator. It doesn't give you the absolute value, although the RCIs are calculated based on the costs of a conventional eight-layer FR-4 through-hole multilayer PCB.

Matties: So, you have a benchmark board, and then you're either adding or subtracting based off that benchmark.

Holden: In terms of percentages, not in terms of dollars. It's only good for that fabricator because some other fabri-

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

Every day we get questions like those. And we love it. We have more than 500 PCB experts on 3 continents speaking 19 languages at your service. **Regardless where you are or whenever you have a question**, contact us!

What's your PCB question?
www.ncabgroup.com/pcb-design-mistakes/

Reliable answers. Reliable PCBs.

NCAB GROUP

Happy Holden

cator, depending on their eight-layer through-holes and what they charge, will have different RCIs. But at least RCIs give you some help in design when you're comparing alternatives.

Matties: Are the designers driven to reduce cost or conserve material, or are they just driven by schedule?

Holden: They're singularly driven by schedule.

Matties: Now with supply chain issues hitting, I would think that they have to now start thinking about reducing the number of materials they use in each PCB.

Holden: Ah, yes, and that's the big bugaboo. This came out of nothing. Now that people can't get to components, they have to change the component, which changes its footprint, etc. The whole supply chain and virus lockdown has changed many rules, but nothing is available. Nobody was prepared for this. I've had that RCI chart out for 30 years now. Manufacturers actually like how bad designers are because that gives them a much greater profit. If designers were designing optimally, much of their extra profits would go away, so what's their incentive to educate the designer to do it better?

Matties: That's the question that's percolating in my mind. What's the incentive for the designer to conserve material?

Holden: Right now, his biggest incentive is getting his boards out, and figuring out what he's going to do when fabricators come back with a huge increase in pricing on materials because of the scarcity.

Matties: Inflation is the driver, of course. We have to lower the cost by whatever means we have, because of supply chain issues. So, you have double incentive to create a shift in the paradigm.

Holden: Yes, but nobody teaches this. So, how does a designer learn this?

Dan Feinberg: Happy, I don't think that the design team's management rewards them for this sort of behavior.

Holden: Yes. It certainly will punish them if they don't meet the schedule, but it won't necessarily reward them. You're right.

Matties: So, you fall back on tried-and-true methods. You just need to make sure that this is functional, rather than spending a lot of time saving the company money when there's little in it for you individually.

Holden: If you sandbag and over-design a board, it still works, and only its cost goes up. But if you're trying to minimize the cost, then you've taken on risk, in terms of meeting the schedule and everything else. You know how much everybody likes to avoid risk. The solution to risk aversion is just to make things more complicated and expensive. At least it won't come back to you.

Feinberg: There's one factor that we're not talking about: There's the cost of materials and of design, but let's not forget profit. You must put it into the equation, and you don't know how much the board or the owners or the senior management of a particular company have included for profit, what percentage it is. That's a factor. It's one of the reasons why companies previously did their own circuit board fab: you didn't need to have two segments of the industry that you had to provide profit for. Now, the cost came down and there were all kinds of other reasons why you didn't continue to do that. But I just wonder if that will come back, to some degree.

Holden: I think it motivated Schweitzer Engineering Laboratories (SEL) to vertically integrate. And Whelen said it was building boards at half the price of China in two days instead of 12 weeks.

Matties: The soft advantage was that their design team had an ability to improve product easily. They could design it in the afternoon and build it the next day.

Holden: That's one of the things that SEL contacted me about; they wanted to know about the dynamics of making your own printed circuit boards. I told them that at Hewlett Packard, the printed circuit fabrication guys were there with assembly from the very beginning of the project, to look at ideas and ways that we get higher performance, lower cost, or quicker delivery. In almost every one of HP's famous and hugely profitable products, there was a printed circuit board contribution that they couldn't have gotten from the outside market.

Matties: If we circle back to the material conservation side of things, is this the time for the OEMs to come in and say, "We're going to respin or redesign our electronics?"

Holden: Well, that's who most of the designers work for.

Matties: Right. But won't they just keep designing PCBs the same old way?

Nolan Johnson: They are being forced to do that somewhat, based on supply chain stuff. Emmalee Gagnon, who was a columnist for Manncorp, recently discussed having customers who are reverting back to through-hole components because they can't get the surface-mount components. So, there are all sorts of pressures to redesign in order to keep shipping product right now.

Matties: What I'm talking about is that it's time to redesign your products. This is where designers can consider HDI, additive, semi-additive, and VeCS. Happy, didn't you say you can often go from 14 layers to eight layers with HDI?

Holden: Yes.

Matties: That's substantial. Then you start looking at your power planes; what did you call that technology, Happy?

Holden: Power mesh. When you eliminate power planes, you drop a number of layers too, as well as improve your electrical performance, because each power rail is tightly coupled to ground.

Matties: While everybody is busy looking for parts, it seems like equal effort is made to look at alternate technologies and design methodology.

"Like the adage says, we never seem to change until we have an emergency or a crisis."

Holden: Yes. But how many OEMs employ anybody who even understands what you've just listed there, much less the power to say to the boss, "Hey, let me re-design this a different way"?

Matties: I think we need to highlight these alternatives, because the designers can be the heroes if they come in and say, "Listen, we can save you X dollars if we utilize this, and we can eliminate 30% of the component usage or whatever it happens to be."

Holden: Like the adage says, we never seem to change until we have an emergency or a crisis. If everything that we think might happen actually happens and you have to start and you can't get material, then this thing says, "All right, figure out how to do it, less material or make the board smaller." The material you have provides more boards, so you can ship a product.

Matties: We've been talking about the design side of this, but what about on the solder mask? For example, can we eliminate cost by going to inkjet solder mask? The designer has to account for that.

Holden: I think there is a savings there. Especially with an inkjet solder mask, you don't cover the whole board with solder mask anymore. You can put it just around the traces and save 50% of the cost of the solder mask. That's a material cost savings. You get twice as many boards out of the same amount of solder mask material you purchased.

Matties: A huge labor savings too, and you can ask for this when you're designing a board and you're saying, "I'm going to use inkjet solder mask. What's the labor or what's the reduction?" I think you have to ask for that cost reduction because if you don't ask for it, they may charge you the standard solder mask fee.

Johnson: Right. Because that's been the traditional pricing structure; you're actually, as a customer, forcing them to do a new kind of pricing around solder masks.

Matties: Who decides the final finish? Is it the designer?

Holden: I just finished an upcoming Tech Talk column where I discuss palladium on copper, which is especially suitable for additive and fine line. It costs significantly less than ENIG and yet it's suitable for harsh environments and things like that; OSP, immersion silver, and immersion tin don't work well in harsh environments.

Matties: It may be time to rethink final finishes, to add in some cost of material conservation.

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

Every day we get questions like those. And we love it. We have more than 500 PCB experts on 3 continents speaking 19 languages at your service. **Regardless where you are or whenever you have a question**, contact us!

What's your PCB question?
www.ncabgroup.com/pcb-design-mistakes/

Reliable answers. Reliable PCBs.

NCAB GROUP

Holden: Yes. I was specifically addressing new electric vehicles because an automobile is a harsh environment, and because it's vibration, temperature, and time, and you want the board to last at least 15 years.

Matties: When we talk about the fabricator controlling 20% of the cost, what are the variables in that, Happy?

Holden: It's their material handling options. You may specify the material, but a fabricator varies that cost depending on how he buys the material, stores it, and deals with it. The manufacturing yield, especially, will change that cost; otherwise, almost everything is spelled out by the designer in the specs. The specs tell you the material and what the trace/space widths are and how many layers you will have and the thickness of the layers, etc. There are not very many degrees of freedom a fabricator has once he gets all these specs.

Matties: Well, that's the other point you made the other day, we have a board thickness based on connector type. And that's been the standard ever since. I think that's what you mentioned, Happy.

> "Because designers control 80% of the cost of the board, the potential savings would be unbelievable."

Holden: It's the paradigm paralysis system. We've been making 0.062" boards because that was the thickness of the connector. And now we don't usually have tab fingers on boards anymore, and connectors come in all different thicknesses. Now the main board doesn't have to be 0.062" because it's not plugging into the gold. If it's half the number of layers, then it's going to be less expensive. Can we make it smaller and get more per panel? The number of boards per panel is a major cost driver. And the total number of layers is a major cost driver.

Matties: If you could get 20% more out of a panel, that's huge.

Holden: It's just all tracked in the same old way. The nice thing about the same old way, since it's not optimized, is that it gives the smart fabricator a way to increase the profitability or save costs and provide some of that back to the OEM in terms of reduced price on quoting, and they keep mark. It all stems from that fact that there is no planning tool to optimize the design before you start the design.

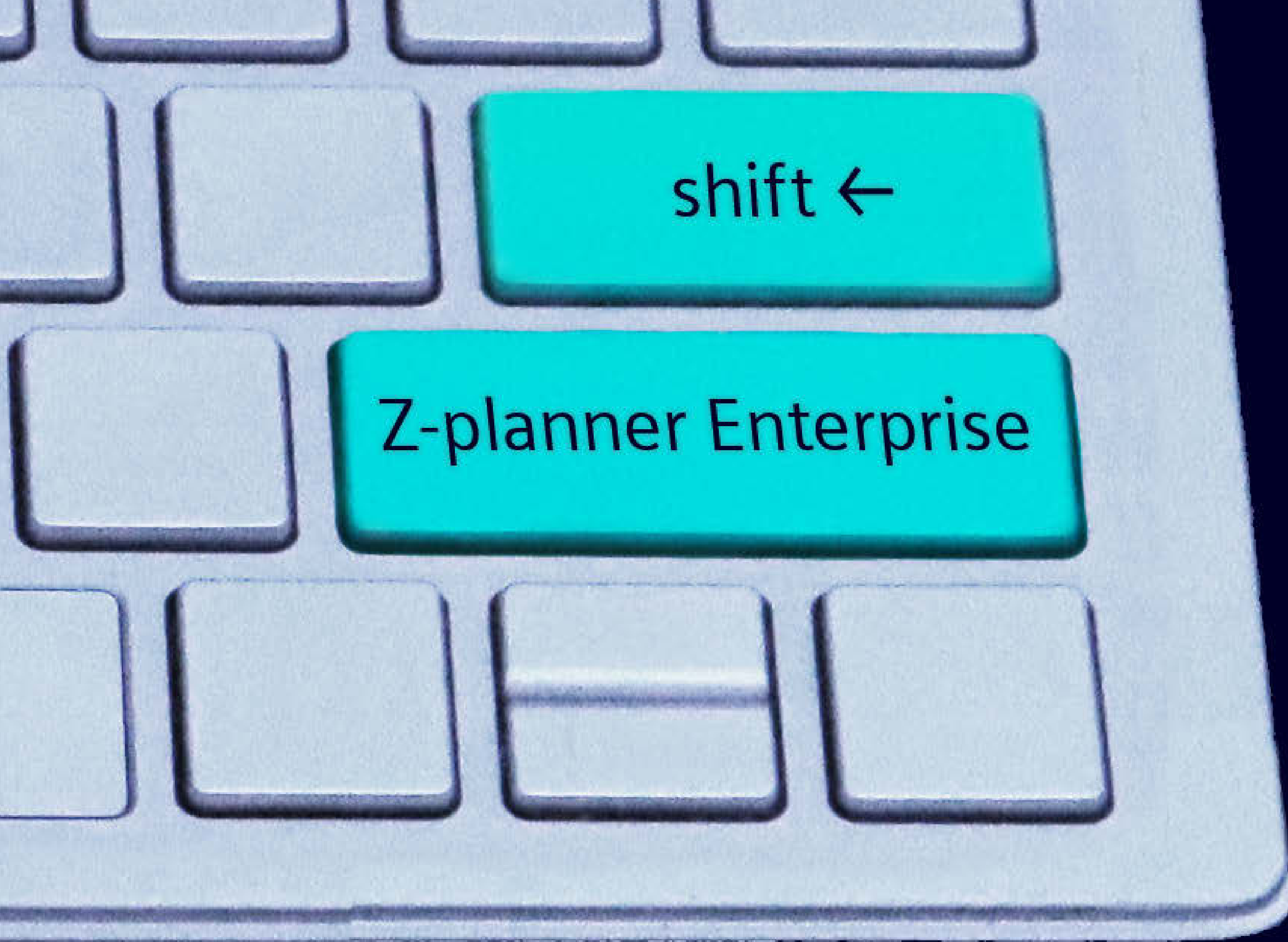
Andy Shaughnessy: I think it's going to be tough to get designers to buy into designing for conservation, because the extra costs are already baked into the cake, along with extra respins. It's going to take a concerted effort from everyone involved.

Matties: Yes. I think if you could get the material suppliers to buy in, along with the designers and the OEMs, then you can create change. But that's what it's going to take.

Holden: Because designers control 80% of the cost of the board, the potential savings would be unbelievable.

Matties: Always great to talk with you Happy. Thanks so much. DESIGN007

Happy Holden has worked in printed circuit technology since 1970 with Hewlett-Packard, NanYa Westwood, Merix, Foxconn, and Gentex. He is currently a contributing technical editor with I-Connect007, and the author of *Automation and Advanced Procedures in PCB Fabrication,* and *24 Essential Skills for Engineers.* To contact Holden or read past columns, click here.



Shift Stackup Planning into Design

Siemens' Z-planner Enterprise software provides detailed stackup planning early in the initial design process, allowing you to optimize your PCB stackup early. Conduct pre-layout signal-integrity simulations based on the actual materials your fabricators will use. By shifting this awareness 'left' in the NPI process, you will eliminate the expensive delays typically experienced after a design has been released.

Shift left with Z-planner Enterprise today.

siemens.com/Zplanner

SIEMENS

Happy's Design Tips for Material Conservation

Sidebar by Happy Holden

Note: For this issue, Happy Holden provided a range of options for designers who are seeking to conserve materials in their next design. He also offered an example of the relative cost index, or RCI, that he developed at HP exclusively for PCB design. With this RCI, designers can figure out the relative cost of a new design compared to an eight-layer through-hole board.

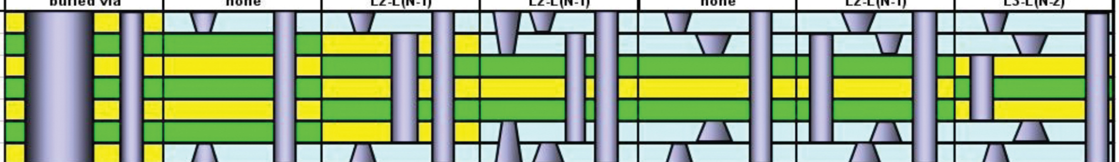
- Consider making the board thinner; eliminate prepregs
- Design to HDI for smaller board or fewer layers (Figure 1 and Reference 1)
- Design inner layers with 1 to 2 mil smaller traces/spaces
- Swing vias for BGA breakout can reduce layer count
- Nesting and rotation on panels
- Using HDI "skip vias" to eliminate a build-up layer

Consider using new technologies:

- VeCS can reduce layer count (Figure 2 and Reference 2)
- Additive creates more routing per layer and smaller boards
- Switch HDI layers to RCF (RCC)
- Power mesh can yield fewer layers (Reference 3)
- Landless vias on the surface can mean higher routings (Figure 3 and Reference 4)
- Smaller panel peripheral border
- Different panel sizes for higher utilization
- Add-on 0.062" gold fingers as components
- Board replacement for X-outs and repanelization (Reference 5)
- Inkjet solder mask just on traces/pads

TH vs. HDI – Price/Density Comparison

N Layers	A		B		C		D		E		F		G	
	THRU-HOLE		HDI BLIND		HDI BL/BU		1BU BLIND		2BU BLIND		2BU BL.BU		2BU BL.BU	
	N		1+N+1		1+bN+1		1+N+1		2+N+2		2+bN+2		2+bN+2	
blind via*			L1-L2		L1-L2		skip via L1-L3		staggered L1-L2, L2-L		skip via L1-L3		staggered L1-L2, L2-L	
buried via			none		L2-L(N-1)		L2-L(N-1)		none		L2-L(N-1)		L3-L(N-2)	
	RCI	DEN	RCI	DEN	RCI	DEN	RCI	DEN	RCI	DEN	RCI	DEN	RCI	DEN
4L	0.62	20	0.83	40	1.05	80	1.26	135	1.38	135	- -	- -	- -	- -
6L	0.78	20	0.99	60	1.24	160	1.46	200	1.60	200	1.74	260	1.91	280
8L	1.00	30	1.21	120	1.49	180	1.74	240	1.90	240	2.06	300	2.25	320
10L	1.30	40	1.51	200	1.83	210	2.11	260	2.30	260	2.50	400	2.73	440
12L	1.70	60	1.92	210	2.31	230	2.62	300	2.85	300	3.10	600	3.37	650
14L	2.24	70	2.48	220	2.95	250	3.32	360	3.61	360	3.91	800	4.25	860
16L	2.97	80	3.22	260	3.81	300	4.25	420	4.61	420	5.00	1000	5.43	1100
18L	3.92	100	4.21	300	4.95	400	5.47	480	5.93	480	6.42	1250	6.96	1350
20L	5.14	105	5.48	360	6.41	500	7.04		7.62		8.23		8.90	
22L	6.67	110	7.08	400	8.23	600	8.99		9.70		10.45		11.27	
24L	8.53	125	9.03	460	10.41	700	11.32		12.17		13.07		14.04	
26L	10.68	130	11.30	500	12.92		13.96		14.96		16.00		17.11	
28L	13.09	135	13.83	540	15.65		16.82		17.93		19.09		20.32	
30L	15.63	140	16.50	580	18.47		19.73		20.94		22.18		23.48	
32L	18.17	145	19.17	620	21.21		22.53		23.79					
34L	20.59	150	21.69	660	23.73		25.09							
36L	22.79	160	23.96	700	25.94									
38L	24.68	180	25.91	740										
40L	26.26	200												



© COPYRIGHT HHolden 2006

Figure 1: Replacement chart for through-hole to HDI with Relative Cost Index.
Note: RCI = the relative cost compared to an eight-layer through-hole board (RCI=1.0).
DEN = total wiring in inches per square inch (diagonals of equivalent density).

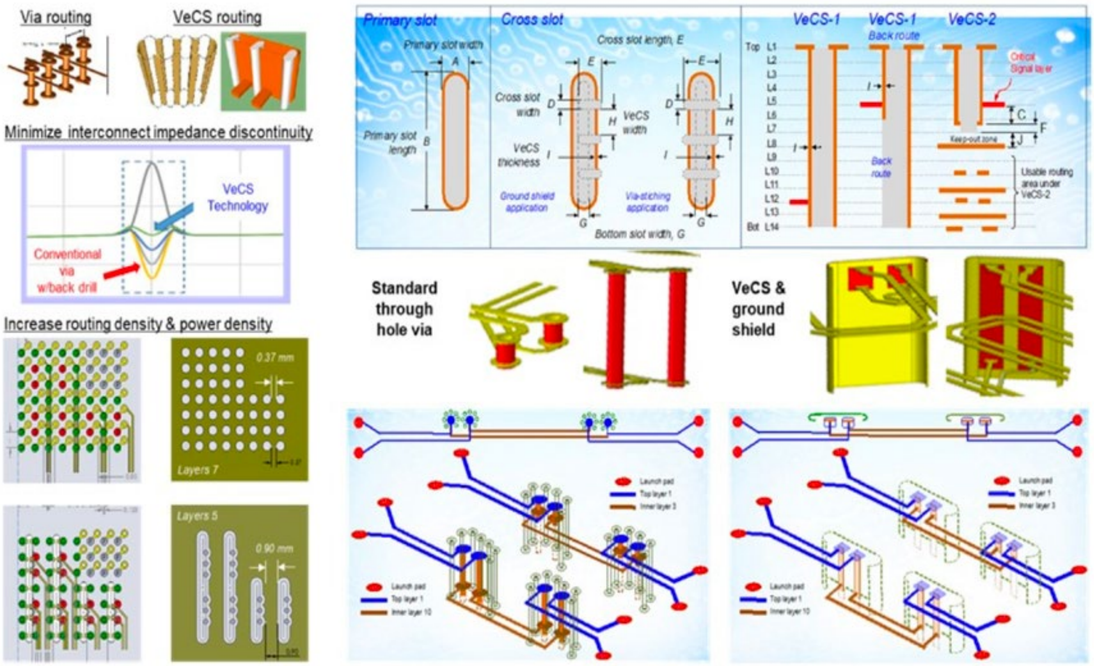


Figure 2: VeCS (Vertical Conductive Structures) is a licensed technology using low-footprint vertical controlled impedance interconnect with low-leakage ground shielding capable of matching interconnect impedance with SE and differential microstrip and striplines. VeCS also provides high-density differential routing solutions in =/< 1.0 mm BGA area. Unique "anti-pad" layers provide a higher cross-section area for power and heat dissipation.

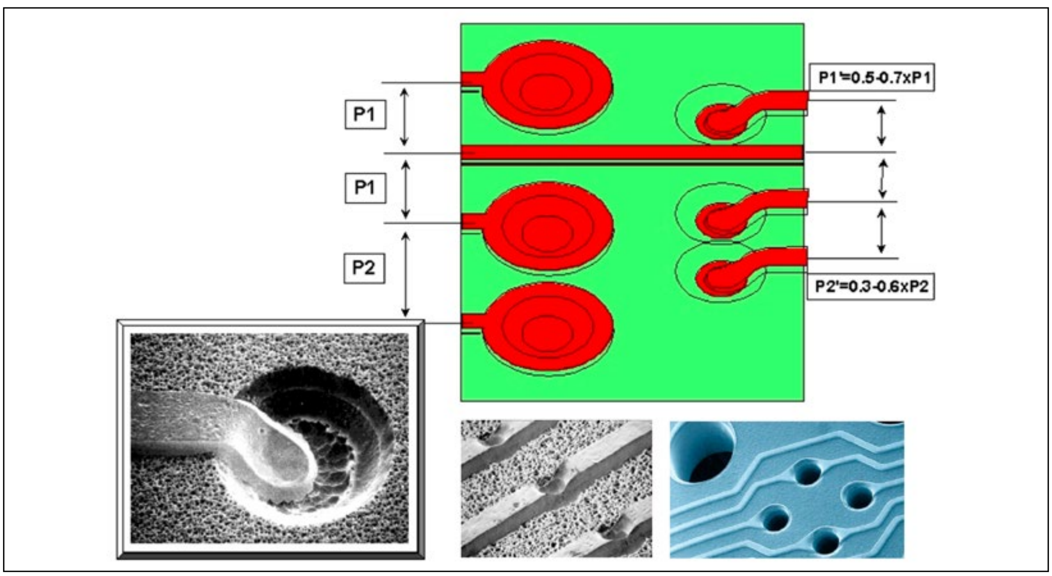


Figure 3: A variety of examples of landless vias on the outer layers of a PCB.

References

1. *The HDI Handbook,* by Happy Holden, et al, BR Publishing, Salem, Oregon, 2009.

2. Happy's TechTalk #1: Vertical Conductive Structures (VeCS), by Happy Holden, *PCB007 Magazine,* October 2021.

3. "Innovative Use of Vias for Density Improvements," by Happy Holden, *The PCB Magazine,* November 2016.

4. "Against the Density Wall: Landless Vias Might be the Answer," by Happy Holden, *The PCB Magazine,* June 2016.

5. Happy's TechTalk #6: Looking at the Process of Repanelization, by Happy Holden, *PCB007 Magazine,* March 2022.

PCB Design Strategies to Reduce Costs

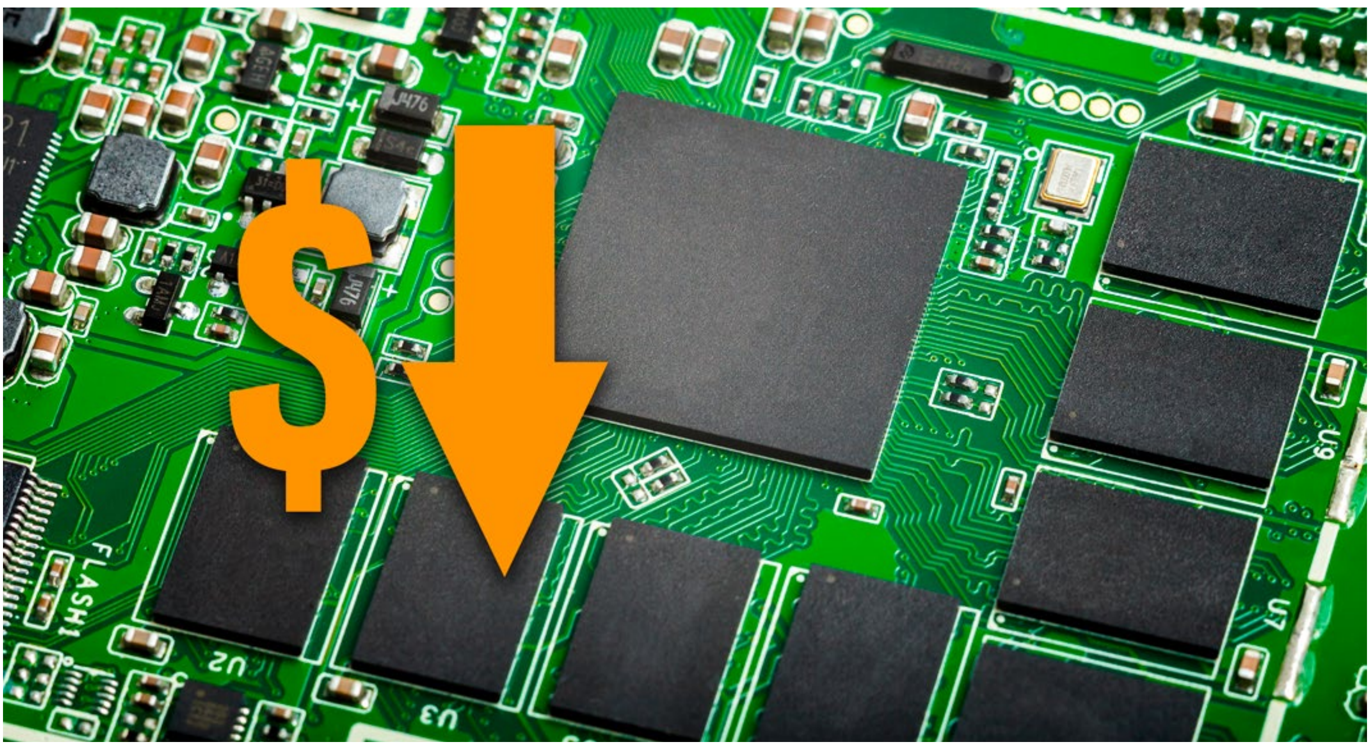
Beyond Design

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

There are numerous ways to improve the PCB design and production processes and thereby reduce costs, from fundamental improvements involving a standard form factor and reducing the board size and complexity, to technology choices and simulation to reduce iterations. A good starting point would be the IPC standards developed by the electronics industry to enhance manufacturability, testability, and assembly. Anyone new to PCB development should initially begin with these standards and then fine-tune them to capture the essence of their design style.

The adoption of surface mount technology (SMT) and the increased use of high-density interconnects (HDIs) has enabled more functionality per unit area than conventional plated through-hole (PTH) PCBs. The semi-additive process (SAP) takes this one step further by reducing the size of wearable devices. These technologies are also ideal for high-speed design as they reduce transmission line length, hence, reflections and lower inductance of the power/ground system. With no component leads the assembly can be automated, dramatically reducing production time and cost.

Via technology is another consideration. Plated through-hole vias are the most affordable and should be used whenever possible. Blind and buried vias will increase your costs.




ENSURING
EXCELLENCE FOR AMERICAN MANUFACTURING
Stuart
Fills complex internal via structures with conductive materials for high-density PCB designs.
Zach
Routs mechanical features and depanelizes into final customer parts and arrays.
Heather
Ensuring PCBs are conforming and have zero defects on surface finishes.
Mikal
Routs mechanical features and depanelizes into final customer parts and arrays.
Elizabeth
Ensures the smooth operation of soldermask and other factory workgroups.
Megan
Expert in Automatic Optical Inspection (AOI), inspecting PCBs to ensure conformance to customer specs.
Calumet
ELECTRONICS
DOMESTIC MANUFACTURING READY
100% MADE IN AMERICA
ONSHORE TRUSTED FOR DOD AND COMMERCIAL OEM ELECTRONICS MANUFACTURING
AMERICA'S ONLY HUBZONE PCB MANUFACTURER
calumetelectronics.com

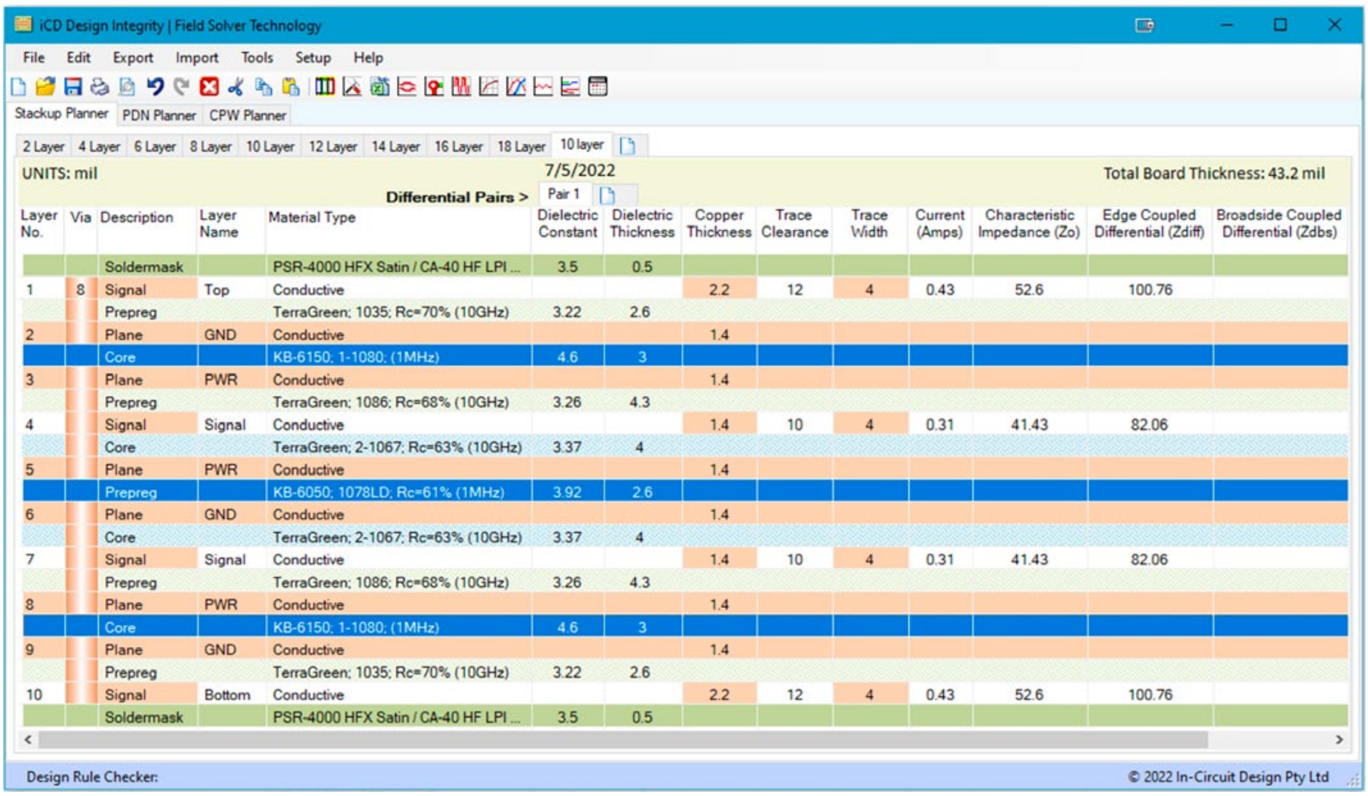
They're only necessary on high-density and high-frequency complex boards, so you should typically not need them. Microvias are relatively costly for simpler designs, but for more sophisticated PCBs and prototypes, they are the prime solution. They also reduce layer count which is a cost-saving. Blind and buried vias are also used to reduce the board aspect ratio. The minimum via hole is determined by drill size as well as the aspect ratio, which is the thickness of the PCB divided by the diameter size of the drilled hole. There is usually an additional manufacturing cost for aspect ratios higher than 8:1 for PTH and 0.8:1 for microvias.

The efficiency and design freedom provided by blind and buried microvias are some of the reasons why PTHs are rapidly becoming a thing of the past for complex designs, especially since most of the components that require PTHs to be mounted on a PCB are bulky and space-consuming, whereas dense BGA designs are more suited to the use of microvias.

Cost is also relative to the trace/clearance requirements of the technology employed. The larger the trace/clearance, the less the cost. Going below 4/4 mil technology will incur a cost premium. The use of mixed signal/power planes can reduce the need for additional plane layers to keep the layer count down. Look for different design options to make sure your board is as simple as possible. Not only do you want to optimize the elements of the board, but you'll also want it on a small form factor that still provides appropriate clearance for every element.

Material selection is another cost consideration. When each material is used for the right target application, the resultant PCB will have the lowest possible cost while still satisfying the design and performance goals of the project.

In Figure 1, I have selected a 10 GHz Isola TerraGreen material with a Dk of 3.3 and a dissipation factor (Df) of 0.003 for its low loss capabilities. Now, this may be necessary for our high-speed DDR4 signals but is not a



Layer No.	Via	Description	Layer Name	Material Type	Dielectric Constant	Dielectric Thickness	Copper Thickness	Trace Clearance	Trace Width	Current (Amps)	Characteristic Impedance (Zo)	Edge Coupled Differential (Zdiff)	Broadside Coupled Differential (Zdbs)
		Soldermask		PSR-4000 HFX Satin / CA-40 HF LPI ...	3.5	0.5							
1	8	Signal	Top	Conductive			2.2	12	4	0.43	52.6	100.76	
		Prepreg		TerraGreen; 1035; Rc=70% (10GHz)	3.22	2.6							
2		Plane	GND	Conductive			1.4						
		Core		KB-6150; 1-1080; (1MHz)	4.6	3							
3		Plane	PWR	Conductive			1.4						
		Prepreg		TerraGreen; 1086; Rc=68% (10GHz)	3.26	4.3							
4		Signal	Signal	Conductive			1.4	10	4	0.31	41.43	82.06	
		Core		TerraGreen; 2-1067; Rc=63% (10GHz)	3.37	4							
5		Plane	PWR	Conductive			1.4						
		Prepreg		KB-6050; 1078LD; Rc=61% (1MHz)	3.92	2.6							
6		Plane	GND	Conductive			1.4						
		Core		TerraGreen; 2-1067; Rc=63% (10GHz)	3.37	4							
7		Signal	Signal	Conductive			1.4	10	4	0.31	41.43	82.06	
		Prepreg		TerraGreen; 1086; Rc=68% (10GHz)	3.26	4.3							
8		Plane	PWR	Conductive			1.4						
		Core		KB-6150; 1-1080; (1MHz)	4.6	3							
9		Plane	GND	Conductive			1.4						
		Prepreg		TerraGreen; 1035; Rc=70% (10GHz)	3.22	2.6							
10		Signal	Bottom	Conductive			2.2	12	4	0.43	52.6	100.76	
		Soldermask		PSR-4000 HFX Satin / CA-40 HF LPI ...	3.5	0.5							

Figure 1: Ten-layer high-speed PCB stackup. (Source: iCD Stackup Planner)

good pick for planar capacitance. For fast rise time signals, we need a low dissipation factor with low loss. However, for planar capacitance (between power/ground planes), we need a very thin dielectric with a high dielectric constant. The capacitance (C) of a parallel plate capacitor (plane) is relative to plane area (A), the dielectric thickness (d), and the dielectric constant (Dk).

$$C = Dk \cdot A/d$$

For a high distributed capacitance, we need a high Dk, so I have chosen a low-cost Kingboard material (highlighted) with a high Dk of 4.6. Not only will this help reduce the cost but will also help reduce the AC impedance of the power distribution network. You do not have to use the same expensive material throughout the stackup; in this case, the low-cost Kingboard material performs better for its application.

Utilizing efficient EDA tools can also streamline the PCB development cycle. Many improvements in EDA software have helped to structure the design flow effectively and have reduced expenses. The availability of an elaborate component library reduces the rebuilding time and effort. Team sharing software can also drive comprehensive design reviews to improve the quality of the product.

Following the best design constraints won't necessarily produce a premium product. Across multiple sectors, fields, and disciplines, simulation is having a profoundly positive effect. One of the primary advantages of using simulation software is the fact that it enables one to obtain valuable feedback when designing real-world circuits. Furthermore, this feedback does not come with the time or expenditure typically required to procure this much-needed design insight. This, in turn, allows the designer to determine the efficiency and integrity of their designs without the need to build the product first. Lifecycle Insights (September 2018) found that the average number of respins per project was 2.9, and the average cost of a respin was $28,482. That's an incredible $82,598 for each new product.

Simulation also allows the designer to explore the worthiness of alternate designs without ever building the actual product. Moreover, examining the effects of your design decisions during the design phase rather than the construction phase saves iterations, time, and money, as well as increases design quality.

The cost of development is dramatically reduced if simulation is employed early in the design cycle. If changes are made late in the design process, then it takes more time, people, material, and therefore money, to complete the project. The advantage of simulation is that it identifies issues early in the design process and rectifies them before they become a major problem.

Design changes that occur:

- In the conceptual stage it costs nothing
- During the design stage it requires just a little extra time
- During the test stage means that you must regress one stage
- During production, or worse still, in the field, can cost millions to fix and possibly damage the company's reputation

Having the project completed on time and within budget, means that costs are cut by reducing the design cycle and generating higher profits due to shorter time-to-market and an extended product life cycle. Having validated the design by simulation, you can be assured of reliable performance.

Key Points

- Anyone new to PCB development should initially begin with the IPC standards and then fine-tune them to capture the essence of their design style.
- The adoption of SMT and the increased use of HDIs have enabled more functionality per unit area than conventional PTH PCBs.

- Plated through-hole vias are the most affordable and should be used whenever possible. Blind and buried vias will increase your costs.
- Dense BGA designs are more suited to the use of microvias.
- The larger the trace/clearance, the less the cost. Going below 4/4 mil technology will incur a cost premium.
- When each material is used for the right target application, the resultant PCB will have the lowest possible cost while still satisfying the design and performance goals of the project.
- Utilizing efficient EDA tools can also streamline the PCB development cycle.
- Simulation enables one to obtain valuable feedback when designing real-world circuits.
- The cost of development is dramatically reduced if simulation is employed early in the design cycle.
- The advantage of simulation is that it identifies issues early in the design process and rectifies them before they become a major problem. DESIGN007

Reference

1. Beyond Design columns by Barry Olney: The Key to Product Reliability; Designing for the SAP PCB Fabrication Process; Simulation Slashes Iterations; It's a Material World.

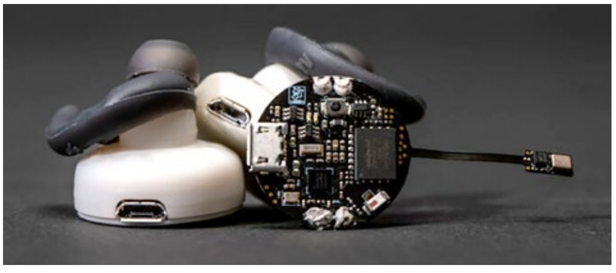
Barry Olney is managing director of In-Circuit Design Pty Ltd (iCD), Australia, a PCB design service bureau that specializes in board-level simulation. The company developed the iCD Design Integrity software incorporating the iCD Stackup, PDN, and CPW Planner. The software can be downloaded at www.icd.com.au.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

ClearBuds: First Wireless Earbuds That Clear Up Calls Using Deep Learning

As meetings shifted online during the COVID-19 lockdown, many people found that chattering roommates, garbage trucks and other loud sounds disrupted important conversations.

This experience inspired three University of Washington researchers, who were roommates during the pandemic, to develop better earbuds. To enhance the speaker's voice and reduce background noise, "ClearBuds" use a novel microphone system and one of the first machine-learning systems to operate in real time and run on a smartphone.

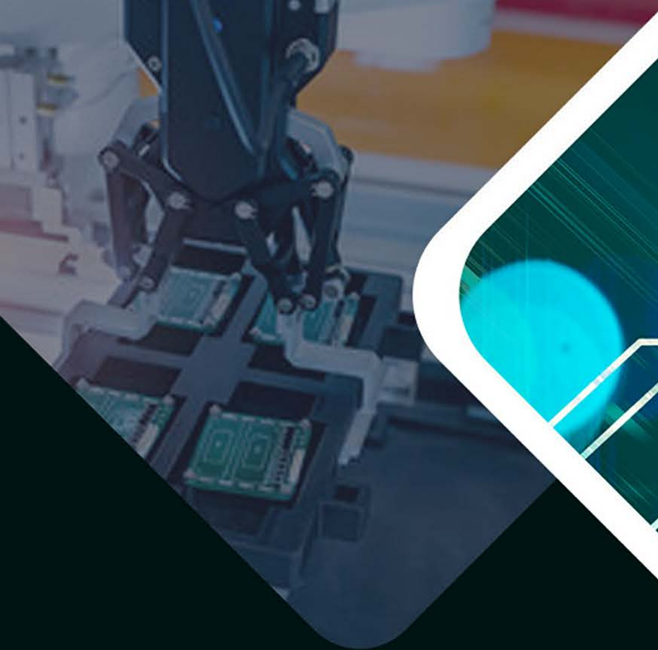
"ClearBuds differentiate themselves from other wireless earbuds in two key ways," said co-lead author Maruchi Kim, a doctoral student in the Paul G. Allen School of Computer Science & Engineering. "First, ClearBuds use a dual microphone array. Microphones in each earbud create two synchronized audio streams that provide information and allow us to spatially separate sounds coming from different directions with higher resolution. Second, the lightweight neural network further enhances the speaker's voice."

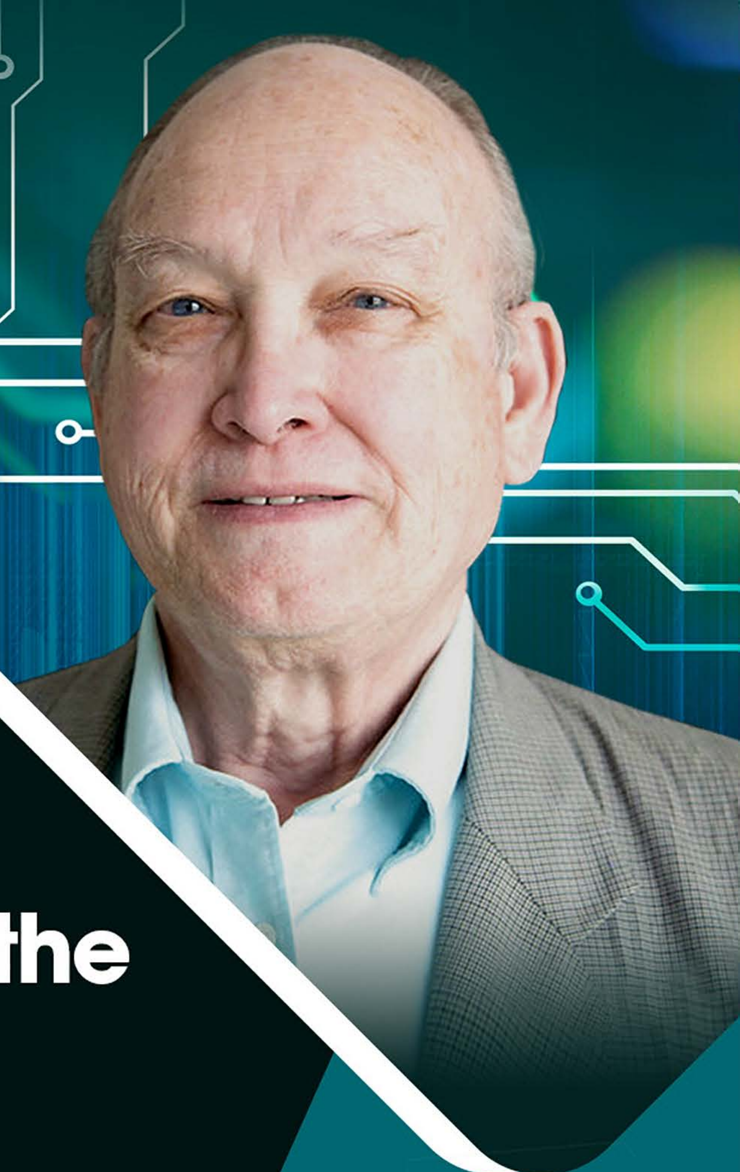


With ClearBuds, each earbud sends a stream of audio to the phone. The researchers designed Bluetooth networking protocols to allow these streams to be synchronized within 70 microseconds of each other.

The team also tested ClearBuds "in the wild," by recording eight people reading from Project Gutenberg in noisy environments, such as a coffee shop or on a busy street. The researchers then had 37 people rate 10- to 60-second clips of these recordings. Participants rated clips that were processed through ClearBuds' neural network as having the best noise suppression and the best overall listening experience.

(Source: Sarah McQuate, UW News)





ICT Webinar

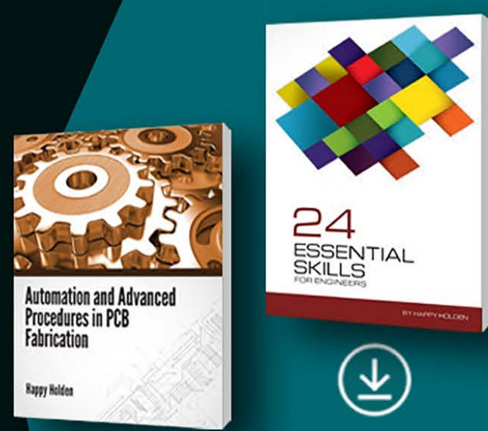
Automation & the Journey to the Smart Factory

Presented by Happy Holden

Essential Skills

Industry 4.0

Please join us beginning September 7th for eight 60-minute online sessions with industry icon Happy Holden. This webinar is based on his books ***Automation and Advanced Procedures in PCB Fabrication*** and ***24 Essential Skills for Engineers***.



AGENDA & REGISTRATION

www.instct.org info@instct.org

Design Tips for Lowering Fab and Assembly Costs

Feature Article by Cherie Litson, CID+
LITSON1 CONSULTING

This is the $1 million question of every project: How can I cut the cost of the PCB?

There are about a thousand answers to this question. I may be exaggerating a little bit, but not much, especially when you consider that there are about 42^{18} different ways a PCB could fail. That's a lot, but fortunately you really need to have a significant combination of these failures before it makes the boards unusable.

That said, there are a few simple guidelines that everyone can follow to reduce costs. I talk about them in my IPC CID and CID+ courses. Designers, fabricators, and assemblers talk about them in a variety of articles. Some professionals who have published some great articles on cost-saving strategies include Tara Dunn, Happy Holden, Chris Church, Kella Knack, Judy Warner, Julie Ellis, Lars Wallin, and many, many others.

It's not as simple as saying, "Just cut down the layer count" or "Just use smaller parts and traces." Here's another: "Just use standard FR-4 material." Then there's, "Just don't use blind and buried vias."

These will certainly work if you make them happen, but they are not always the go-to answers on how to reduce costs. I've actually reduced the cost of some boards by doing the opposite of what you would normally think you should do. Here are some examples:

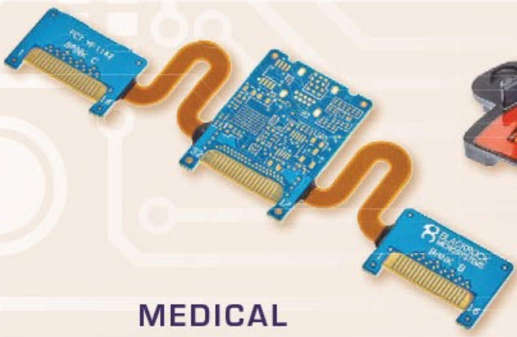
- **Adding layers:** This cut the cost of the board because I could increase the size and spacing of the traces. I was able to add an extra GND layer for shielding and better electrical performance. I had less fallout, less bow and twist, and easier manufacturing; thus, I cut the final costs.

PREMIER GLOBAL SUPPLIER of **FLEXIBLE CIRCUITS**

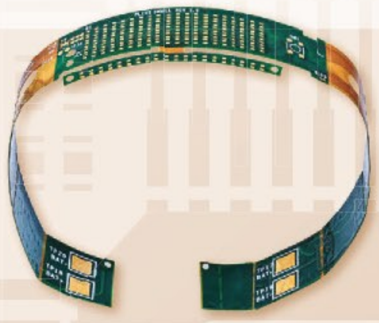
FLEXIBLE CIRCUITS | EMS/ASSEMBLY | CONTRACT MANUFACTURING

Superior design consultation & support
Flexible Circuits
Rigid Flex
Flexible Heaters
Flex/Rigid Flex Assemblies
Membrane Switches
Plastic Moldings
Specialized EMS/Assembly
Product Module to Complete Box Builds

EXPLORE the POSSIBILITIES!

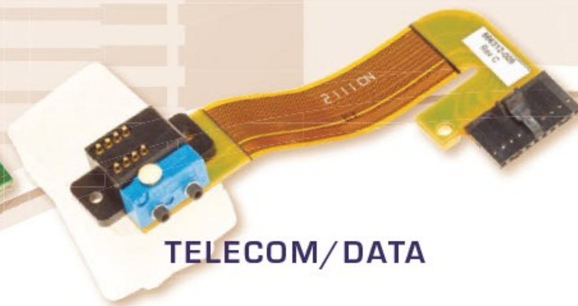

MEDICAL


AUTOMOTIVE


CONSUMER PRODUCTS


WEARABLES


INDUSTRIAL


TELECOM/DATA

FLEXIBLE CIRCUIT Technologies®

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

- **Using larger components:** When only one component on the board had pin spacing less than 0.5 mm, it didn't save any space at all. This part needed a special paste mask and we had to have extra spacing for the masking. Replacing it with a larger package saved us space on the board and cost less in manufacturing.
- **Using higher-temp materials:** This helped the board to withstand the stresses of manufacturing. It cut down on stress failures and fallout, thus cutting the overall costs.
- **Using blind and through-vias:** This improved breakout from fine-pitch parts. It wasn't as expensive as using blind, buried, and through-vias; it improved power connectivity, and saved space on the board.
- **Split one board into two boards:** I modularized high-power, larger-pitch circuits and low-power, small-pitch circuits. The electrical requirements for these are different and become cost-adders for manufacturing when combined. Creating two boards, one with thick copper and larger features, the other with thin copper and smaller features, allowed each to be easily created at less overall cost.

So, here's my take on how to reduce your costs:

Planning Ahead

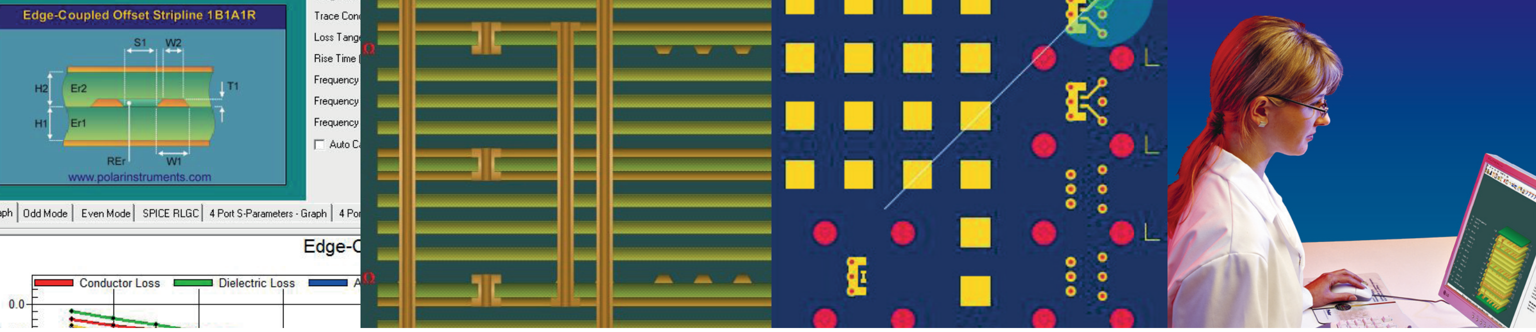
This is one of the most important jobs of the program manager (PM). You'd be surprised how many DFM issues start with the PM. So, how does the PM influence the costs?

- Setting the IPC classification of the design. This puts limits on everything else that follows. Class 2 is common and doesn't incur any cost adders. Class 3 will often triple the cost of manufacturing. It can be used in specific areas only and not have too much of a cost hit. Class 1 can reduce the costs if the features (trace/space/vias/components) are larger. Classification is all about the risk factors a product can support.
- Coordinating all the players for the design features—ME, EE, designer, fabricator, assembler, and test engineer. Having regular design reviews along the way helps to curb cost-increasing decisions.
- Setting the time frame for all processes. The PM must be aware of the domino effect. If too little time is spent on planning and design, then it can lengthen the time in fab and assembly.
- How cutting-edge do you want to be? Some new processes can save time in fab and assembly, but cost time in the design and engineering phase.
  - This is not a bad tradeoff if you are building a lot of boards (over 1,000 per year). If you're building only a few boards, then you may want to cut back on those smaller components and advanced processes. You're not going to save that much in the manufacturing processes. And they may end up costing you more.
  - If you're building very small products or mixed high power/high speed products, you will want to take more time in the design phase and maybe look closely at some new processes. This can help to save significant cost adders even for smaller volume products.

Dummy Boards

Prepare a couple of different types of dummy boards ready for quotes when qualifying a fabricator and assembler. I've heard so many comments from clients that they get vastly different prices from fabricators and assemblers. Yes, they each have different capabilities and ways that they can or can't produce a product.

- The websites will get you a lot of information about what they consider "cost adders" and their capabilities. However,

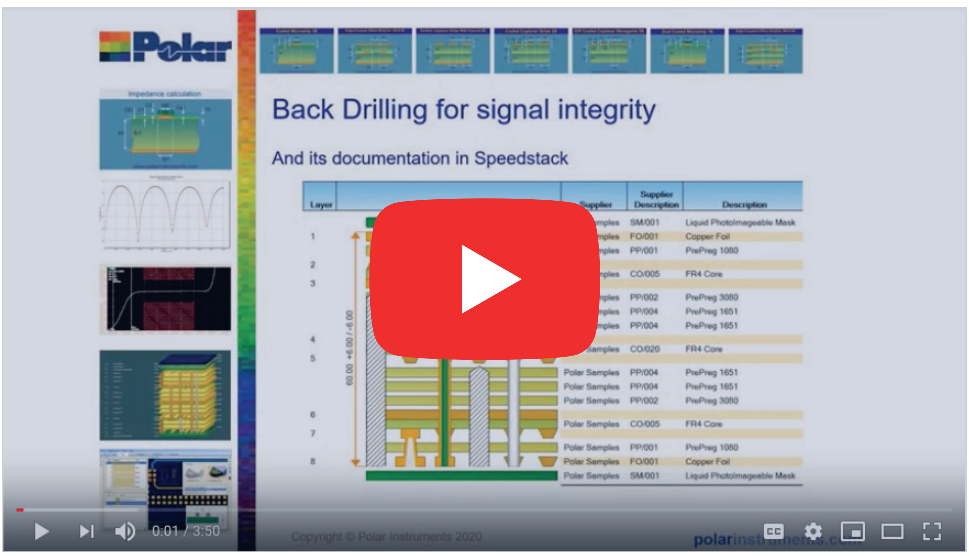


SUBSCRIBE

Subscribe to the Polarinstruments YouTube channel for helpful impedance and stackup videos

PCB Signal integrity tools for design & fabrication

- Impedance & insertion loss modeling with Si9000e
- PCB stackup design & documentation
- Test systems for controlled impedance & insertion loss
- Application notes on a wide range of Si topics



polarinstruments.com

they mostly give you the overall picture. Don't forget to verify the details with the production manager or engineer, not just the salesperson.

- Go to *your extremes* with one of the dummy boards and stay within *common parameters* with the other. Get both quoted and then do your comparisons.
- Visit the manufacturing sites if you can, either virtually or in person. If the facility is clean, organized, and the production engineer seems competent, you've got a good manufacturer.
- Price vs. lead time is important. So are shipping time and other delivery delays if you're going overseas.
- Inspection and testing capabilities will impact the design. What are their limitations?
- Qualify two or three manufacturers and stick with them.
- Find out what documentation formats they prefer, how much information is enough, and how much is too much. Sending ODB++, Gerbers, and IPC-2581 files to your fabricator and assembler actually costs more. They must look at and compare all of them.
- Good customers get discounts. Become a good customer.

Listen and Learn

Don't be afraid to engage in collaborative creativity with others. Where there's a will, there will be a way to accomplish what you need. It may not always look like what you thought it would.

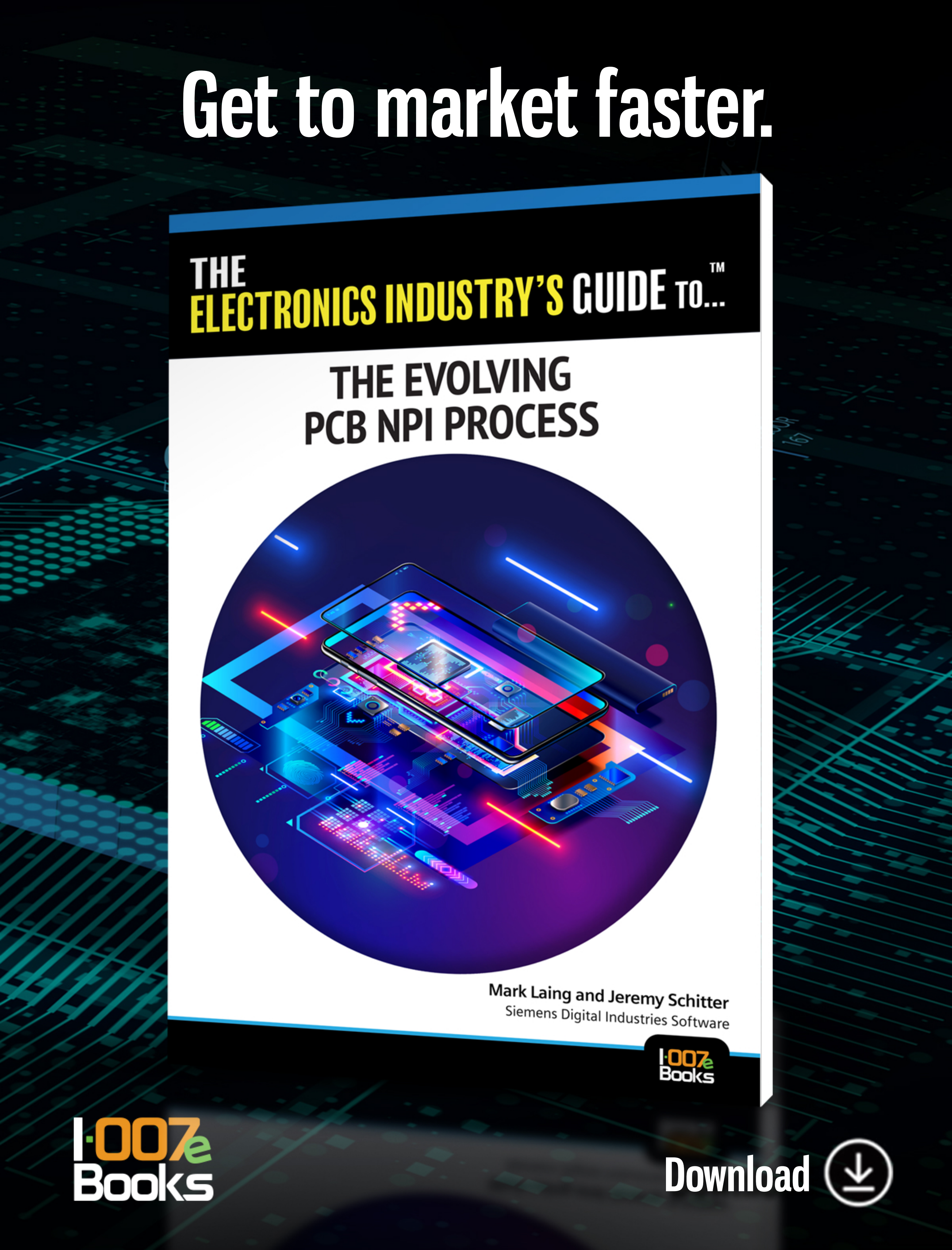
- Read articles, take classes, go to industry product shows.
- Teach/mentor others when appropriate.
- Join organizations like PCEA, IEEE, SMTA, IPC, and EDA.

One of my favorite publications that defines the feature costs for fabrication and assembly, titled "PCBAChecklist17" is free from IPC. There are so many gems of knowledge in that single publication. I give it to every designer and engineer I know. I've created presentations from data embedded in its pages. I even recreated one of the tables that I felt applied best for design teams to follow regarding DFM issues in the board.

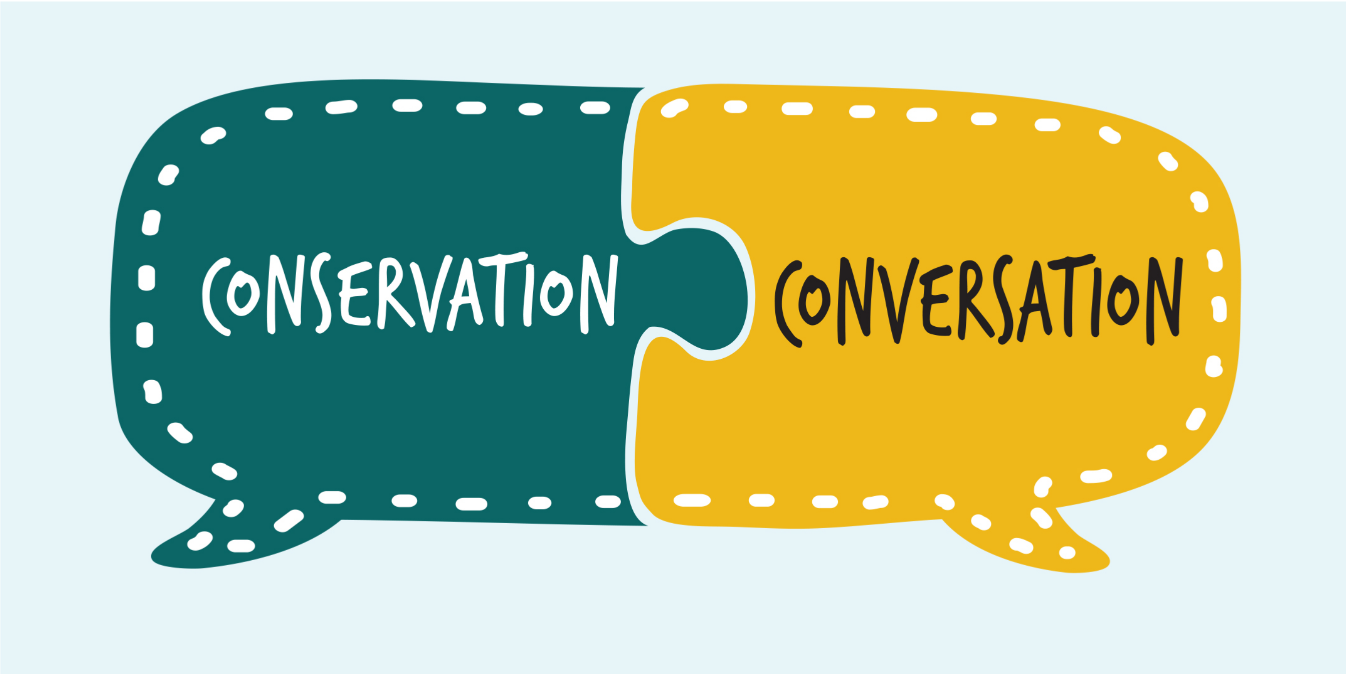
One of my favorite publications that defines the feature costs for fabrication and assembly, titled "PCBAChecklist17" is free from IPC.

My version defines who should be involved in the decisions and who needs to be notified or consulted for those decisions. The charts in the publication do have more information than what I've extrapolated and I'm sure there are more things that can be added to my list, but it's a good start. I hope you take it and make it your list. DESIGN007

Cherie Litson, CID+, is a master design instructor for EPTAC and an instructor at Everett Community College. She is also the fouder of Litson1 Consulting.



Get to market faster.
THE ELECTRONICS INDUSTRY'S GUIDE TO...™
THE EVOLVING PCB NPI PROCESS
Mark Laing and Jeremy Schitter
Siemens Digital Industries Software
I-007e Books
I-007e Books
Download



Material Conservation: The PCB Designer's Role

Feature Interview by the I-Connect007 Editorial Team

During these times of supply chain uncertainty, many product developers are considering new ways to conserve materials—from laminates to components, layer reduction, and everything in between. Barry Matties and Happy Holden recently spoke with Alun Morgan, president of EIPC and technology ambassador for Ventec, about material conservation strategies for today's PCB designers and design engineers. Alun explained why this may be the perfect time to educate PCB designers about conserving materials: When a model is broken, the people involved are much more open to new ideas.

Barry Matties: As companies try to navigate these challenging times, they're considering changing their strategies. What other options do they have?

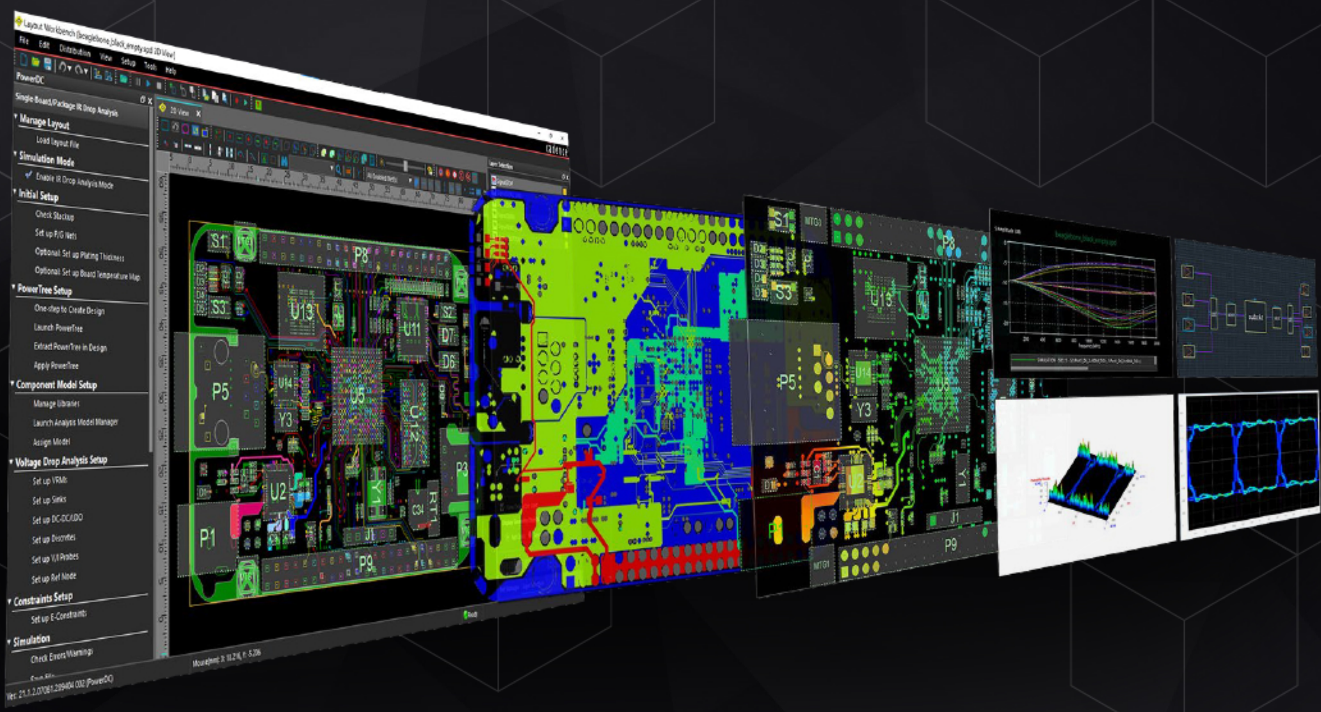
Alun Morgan: The only other option is stopping completely. Cease operations. And that really is an issue.

Matties: Material conservation must be a conversation in many facilities today.

Morgan: Yes. And actually, I hope we can come out of this and start accepting Happy's very simple suggestion. He said, "Why don't we just reduce the thickness of the boards?" That would save a huge number of resources, just a huge number, which has got to be a good thing. Using less material is a good thing in any way, shape, or form.

Happy Holden: Everybody is buying direct imaging equipment now, but the thing is they're still designing 4-mil lines and spaces.

One Environment
Endless Possibilities



Solve All Your SI, PI, & EMC Design Challenges With Sigrity

Easily determine optimal design configurations, validate performance, and sign off on critical interconnects BEFORE prototype. Confidently achieve first-pass success with unified electrical analysis in Sigrity.

Get The FREE Trial Now »

EMA Design Automation®

cādence®

Well, if you went to 2-mil lines and spaces and had half the number of signal layers, you would also need fewer power and ground layers. Unfortunately, currently in North America, if you can't route the board with 4-mil lines and spaces, you add more layers, you make the thing more complicated, and use more materials.


Alun Morgan

You don't go back and reconsider: Should I use 3/3? No, you just slap in more signal layers. Well, that's great, but what if your vendors can't buy the material? That's not a solution. North America is particularly bad at that. You've got to start looking at other solutions because doing it the same old way (SOW) that we did 40 years ago will not work. When we have a material crisis, you can't get copper foil. You can't get prepreg. Well, stop using copper foil. Stop using as much prepreg. Extend what you have.

Morgan: Or use less. The message you're giving, Happy, is one that we've heard before, which is that you must get to the designers and explain because if you don't get the designers on board, they won't design the boards that way. Many of us have spent our working lives talking to designers. Of course, there are a lot of them and every time we've talked to one set, there's a new batch of designers coming along.

They need to understand what this is all about, because they would never come to that kind of conclusion naturally—to make the board thinner. They would never come to that conclusion because they don't understand. But in the space of a short presentation, you could probably give designers a huge amount of information and help them make the change. Maybe this will be the catalyst for designing for conservation (DFC), because we haven't seen the worst of it. I'm certain of that. Maybe it will be the catalyst that will drive some proper change.

Happy, you and I know what some of the possibilities are if we can get designers to think in terms of material conservation. I'm not saying that we know everything, but we know some things that could work, and this might be the time. This may be our chance to actually do something to reduce material consumption, starting with board design.

First, you must break the status quo. Then you can make the changes you want to make and move on toward the kind of vision you have for the future. Then you set it again. One positive thing about now is that we've pretty much broken the model, and when the model is broken, then you can change things.

When things are running normally, you can't change much. "Oh, I get this stuff. Why should I bother?" But when it's broken and they can't get the materials, then they'll be forced into making changes. And now is a good time, and we should take advantage of it. If we don't make the changes now, all we'll do is have the same issue in a few years.

We have a chance now to change the way designers look at design and layout. We really need to get the message out, and quickly. I don't mean just taking out one layer of glass but taking out half the layers or two-thirds of them and getting down to what we actually need.

Stop designing boards the same old way. I know it's tough to break away from the SOW, but now we're in a perfect position. Designers, you're standing at the precipice.

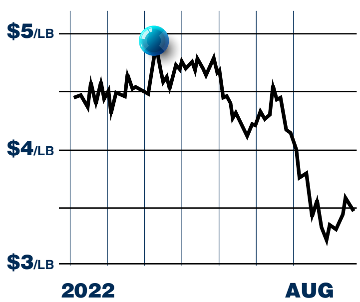
Matties: You may be right, Alun. Thank you for the inspiration. Always a pleasure.

Morgan: The pleasure is mine, Barry. DESIGN007

Designing for Material Conservation

The three factors that most impact PCB manufacturing cost breakdown are the type of material, the size of the board and the **number of layers.**

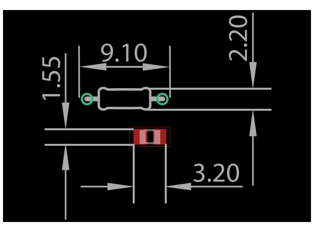
MATERIALS HAVE A BIG IMPACT ON PRICE



COPPER IS 50% OF BOARD COST

Source: Barrons

REDUCE BOARD SIZE

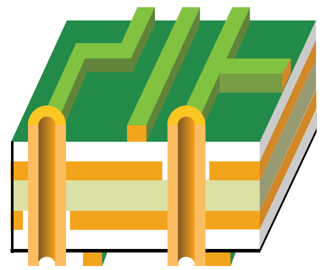


USING SMD COMPONENTS CAN REDUCE BOARD COST

30-40%

Source: Circuit Digest

REDUCE UNNECESSARY LAYERS

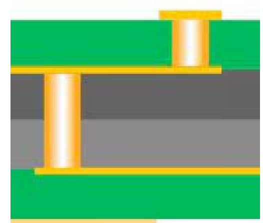


EACH ADDED LAYER INCREASES BOARD COSTS

10-20%

Source: Cadence

AVOID BLIND AND BURIED VIAS



USING THIS METHOD CAN INCREASE COSTS

50%

Source: EPEC

MilAero007 Highlights

Lean Digital Thread: The Secure Digital Thread ▶
Securing intellectual property has become a priority for manufacturers, and recent reports from the U.S. and EU governments highlight the risks and direction for securing the supply chain.

Time to Get Serious About CMMC Readiness ▶
Divyash Patel of MX2 Technology is a leading cybersecurity expert who's sounding the alarm about getting your company into a state of readiness. But he's not yelling fire in a theater. "This is a must-have compliance program," he says. "It needs to be taken seriously and maintained."

The U.S. Economy Needs the Bipartisan Innovation Act and the PCB Act ▶
In this IPC-created interview between Dale Curtis, IPC Advocacy Communications, and Chris Mitchell, IPC VP Global Government Relations, the importance of the semiconductor and PCB manufacturing legislation moving through U.S. Congress is highlighted.

Ultra Selected in Six of Seven Categories for USAF ABMS $950M IDIQ Contract ▶
Ultra has been awarded a $950 million ceiling indefinite-delivery/indefinite-quantity contract for the maturation, demonstration, and proliferation of capability across platforms and domains, leveraging open systems design, modern software, and algorithm development in order to enable Joint All Domain Command and Control.

FedDev Ontario Invests Over $7M for FTG Green Operations ▶
Aerospace is a pillar of the economy and of innovation in Canada. It is one of the most research-intensive and export-focused manufacturing industries, employing thousands of highly qualified people in every region across the country.

Zentech Appoints Rich Fitzgerald, Experienced EMS Leader, as Chairman of the Board ▶
Zentech Manufacturing is pleased to announce that Rich Fitzgerald has joined the business as Chairman of the Board. Rich has 26 years of executive management experience in electronics manufacturing and distribution.

NASA's Perseverance Scouts Mars Sample Return Campaign Landing Sites ▶
NASA's Perseverance Mars rover is conducting its science campaign, taking samples at Jezero Crater's ancient river delta, but it's also been busy scouting.

ESCATEC Taps Noted Industry Talent as COO to Execute Strong Growth ▶
The Swiss-owned and Malaysian-headquartered Group welcomes noted industry talent HL Wong as its new Chief Operating Officer.

Dymax Announces New Partnership with Quantum Systems ▶
Dymax, a leading manufacturer of rapid curing materials and equipment, proudly welcomes its newest sales partner, Quantum Systems, to their growing list of business partnerships.

www.prototron.com

Serious About Speed

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

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

Prototron Circuits

***America's* Board Source**

Designing for Material Conservation Means Changing Attitudes

Feature Interview by the I-Connect007 Editorial Team

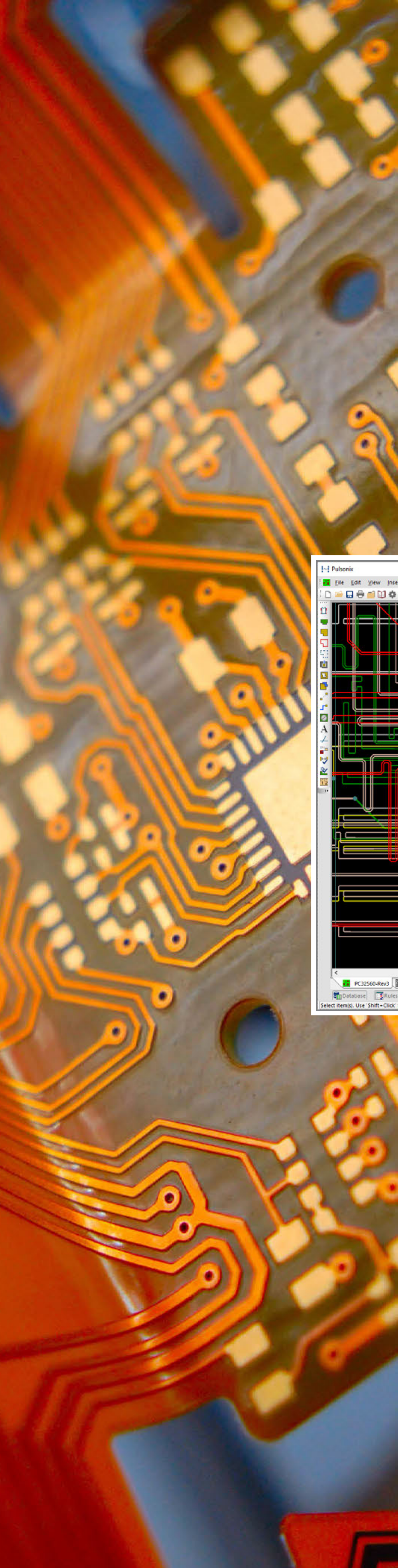
It makes a lot of sense: During times when the supply chain is stretched to the breaking point—and the last few years certainly qualify—what if PCB designers created boards that used fewer components and less laminate? Do PCBs still have to be 0.062" thick? Why not reduce layer count while they're at it?

Andy Shaughnessy and Nolan Johnson spoke with I-Connect007 columnist Dana Korf about the idea of designing a PCB with material conservation in mind. Is it a great new idea, or are we opening a whole new can of worms and a separate group of problems?

Andy Shaughnessy: Dana, our August design issue focuses on material conservation, which has become a topic of conversation during the supply chain snafu. It seems simple—let's just use fewer components and less laminate, right? What are your thoughts on that? You were working at this huge volume level in China, where saving a few ounces of copper or a few components here and there could mean millions of dollars. Was conserving materials something on your radar screen?

Dana Korf: Actually, I just saw an interesting email this morning that included Happy Holden on that subject. The one comment that struck me was someone saying that boards don't have to be 0.062" thick anymore; that was set in the 1950s. Can we make thinner boards? Use thinner materials? Consume less of everything? That's a very interesting comment. But they're right; we're stuck on a 1950s architecture, basically. And in a regular FR-4 board, traditionally the material cost is about 16-20% of the total cost.

With high-performance boards, it could be 80% of the board cost. People drive to use lower-end materials because as performance goes up, that ratio goes up. Could we use thinner materials and consume less? That's one of the big advantages of 3D printing for circuit boards—we don't waste anything. You don't rout out a panel. You don't use layers. You print your trace of any X, Y, Z fashion you want, so you don't need to drill holes and you don't consume drill bits, copper plating chemistries, etc. That very topic is one nice advantage about the 3D world; it's one of the side advantages.

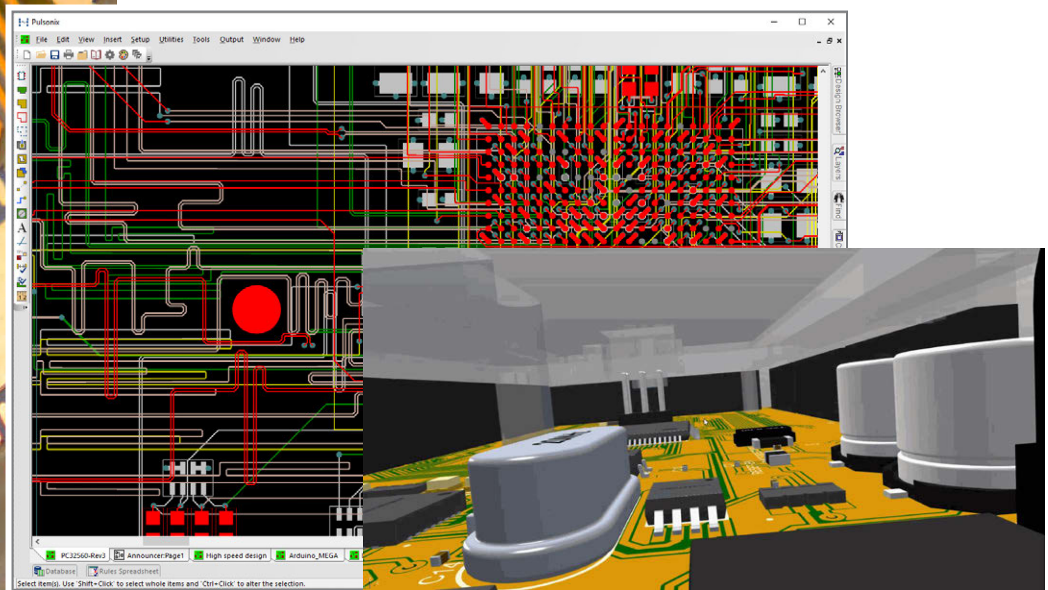


Pulsonix PCB Design Software

PCB design done right

New Version 11.0 Increases Speed up to 80%

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



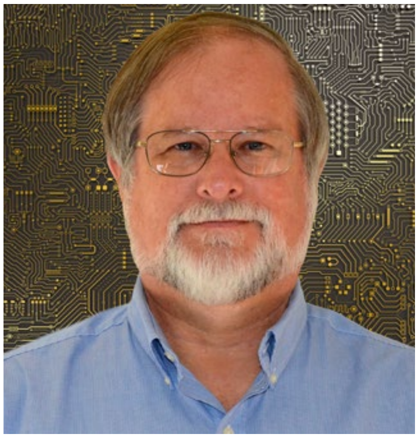
Advanced PCB Design Features

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

Try Pulsonix yourself at pulsonix.com/trial

PULSONIX

Do I see a lot of people thinking that way? No, but as someone pointed out in our email thread, on the average, our typical board is four or six layers, and 0.062" thick. Why can't we make it 0.050"? Why not? We would save material. That's very interesting. The industry needs to change the way we think.



Dana Korf

Shaughnessy: As Happy was saying, if you start making the board itself thinner, you may run up against ultra HDI problems, as well as changing things like impedance, too. You'd have pros and cons.

Korf: Yes. In a traditional board shop, you're trying to use the entire sheet size that the laminator makes, so they don't throw away or have to recycle anything. Then you get to the panel, and you try to use everything because you pay for the entire panel whether you receive it or not. That work has been going on for a long time just to help reduce costs related to throwing material away. As I like to say, "Minimizing that effect to save money." A lot of it is already recycled anyway.

I haven't been in any meetings with designers who say, "I'm trying to use fewer materials to save the environment." I've really never heard those words. Maybe in the back of their head they're thinking that, but not in the forefront.

Nolan Johnson: It makes an interesting spin.

Korf: Yes. It's a good, new way of thinking if you're trying to lead the industry a little bit. It would be great to get people thinking about it more.

Johnson: This is yet another place to shave your margins, be more efficient, make more money, and release capacity to the whole industry.

Korf: From a cost standpoint? Absolutely, it's true. You must go to the next level of expensive technology for the line width and space and/or interconnects, vias. It could be very true. One time, I was working on a design where the customer had specified just one blind via and one laser via on the whole board. I said, "We can make that a through-hole and save you 20%." He says, "No, I need it." I said, "Okay, it's your money. We'll make it for you." Conservation is an interesting topic. How can we save money by using less material, changing the way we do layers and stackups?

Realistically, material conservation isn't something that I've heard come up much at all; it's usually more talk about performance and which technology you should use.

Shaughnessy: Technologies like additive, semi-additive, and printed electronics are all examples of ways of getting boards built with fewer materials. Though some would argue that it's not really a PCB at some point, like certain printed electronic circuits.

Korf: Obviously, fundamentally, if you go additive and you're not throwing away as much material, it should save you money, assuming all loss is equal. And we might save the planet along the way.

Shaughnessy: Thank you, this gives us a lot to think about.

Korf: Glad I could help. But it's not going to happen overnight. I think designers are going to need to change the way they look at PCB design overall to really design for material conservation.

Johnson: Thanks, Dana. We appreciate it. Always a pleasure. DESIGN007

IPC ADVANCED PACKAGING SYMPOSIUM:

Building the IC-Substrate and Package Assembly Ecosystem

October 11-12, 2022

Kimpton Hotel Monaco
Washington, D.C.

EXPERIENCE AND LEARN:

- 28 speakers, eight sessions, and three keynotes focus on strengthening the IC-Substrate and Package Assembly Ecosystem in North America and Europe
- Identify key challenges to overcome to enable sustainable businesses over the long run
- Focus on ‘punching through’ into actionable research, development, design, materials, manufacturing, and business operations execution needs
- Actionable next steps and an expanded network for continued development efforts

IPC® BUILD ELECTRONICS BETTER

ipc.org/event/advanced-packaging-symposium

Design Tips for Inflationary Times

Elementary, Mr. Watson

Feature Column by John Watson, ALTIUM

It was recently reported that inflation hit a new 40-year high at 9.1%. Have you felt that kick in the gut when filling your car with gas or making a purchase at a grocery store? I'll leave it up to those more intelligent than myself to analyze the whys of the steep rise in cost.

This is also another tough blow to the PCB design industry among what seems a never-ending list of adversities. Those who got through the pandemic shutdowns now must deal with the supply chain shortages (which are still ongoing). Our industry has not gone unscathed with the newest problem on the block: inflation. Every step of the PCB process now costs more. It's much like a long line of dominoes just waiting for the first one to tip over.

Can I be honest with you? I consider you all my friends and I know you won't judge. At this point, I feel a bit like Charlie Brown trying to kick the football, only to remember that Lucy Van Pelt will just pull away the football at the last minute, resulting in me laying in a prone position, thinking about how blue the sky looks. The bottom line is that, to survive, we need to change how we conduct business.

During rough times (our present situation certainly qualifies), the standard practice for companies involves the balance sheet, and the call goes out for everyone to tighten their collective belts. The company does this balancing act between what they provide and increasing price—giving less for more.

Clara Peller perfectly illustrates this practice. She was born in Russia in 1902 and worked as a manicurist at a Chicago beauty salon for 35 years. You may not know who she is, but I'm sure you're familiar with her catchphrase,

Do You Feel Left In The Dark By Your Current PCB Supplier?

APCT Is Your Shining Partner, Not Just A Supplier

- **Operational Support** to Assure Availability & Required QTA Lead Times
- **Engineering Expertise** to Advise on New & Emerging Technologies
- **Customer Service** “That Is Best In Class”

The Solution Is APCT

APCT
Passion I Commitment I Trust

Work With A Partner, Work With APCT

APCT.com

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

which was everywhere in the mid-1980s. She was the elderly lady in Wendy's commercials who asked, "Where's the beef?" When belts get tighter, even fast-food companies try to give the customer less.

This practice is no different for the PCB industry. For some, they pinch those pennies so hard they turn into quarters. That can be done successfully but requires some fundamental principles, which is fascinating when it all unfolds. As PCB designers, we have a unique opportunity to reduce costs. First, you determine much of the cost of the board during the design process. It is true that every step of the PCB process now costs more, but that is also an opportunity to examine each step for cost reduction.

I would like to put forth a few basic principles:

1. Stop the knee-jerk decisions.

I know this is easier said than done. When you're in the heat of the battle, emotions get involved, and judgment can become clouded. A firm decision-making process is nowhere to be found. Those making the choices don't fully understand the ramifications. The idea is, "We need to do something, anything." The knee-jerk is reactive rather than proactive, and the results are never pretty. You've seen this: Instead of using scalpels, the chainsaws come out and are flying, looking like a scene from *The Texas Chainsaw Massacre.*

There is a book I think everyone should read today to succeed in buiness. It changed how I looked at difficult situations and the decisions I made. I know it would change how your company operates. The book is called *The Tortoise and the Hare.* Yes, it's the children's story. Our industry is full of hares hopping around when an event occurs; hopping here, hopping there, running around. But what was the lesson of *The Tortoise and the Hare*? It's the competitors who are slow, steady, and focused that win the race. Make your decisions count. Don't just decide to decide because you need to be doing something.

2. Know your design limitations.

If we are not making design decisions based on our "knees," something must replace them. Decisions are made on knowing the customer design specifications and expectations. Some say a glass is half full and others half empty, but there is a third option; as an engineer, I would say that the glass was not correctly designed to the customer's specifications. It is crucial to know your end product requirements.

Answer this question: What are we doing? You may bend those objectives, but don't break them. When you make changes, they will impact the design in many ways, and it doesn't matter if the finished product comes in under budget if the resulting product is unreliable or doesn't meet the customer requirements. There is the old adage that you may win the battle but will lose the war. You may get over the immediate emergency but lose customers forever.

3. Identify your design tradeoffs.

This rule is the heart and soul of these principles. A tradeoff means a balance between two desirable but incompatible features, a compromise, we'll say. Everything comes with tradeoffs. Causality rules our day, but many of us act as if there are no compromises. You must be aware and identify them. Page 5 of IPC standard IPC-2221 has common physical features in your PCB design, the type of change, and the effect on resulting performance. Learn this information. The writers of that document put this at the beginning of the go-to standard for most designers because it drives everything after it. Many go through the design process without considering their decisions' impact on final cost. We place our components, connect the dots, throw up our hands, and chant the designer's mantra: "Everything is OK. Everything is OK," as we slowly slip into a Zen state of denial.

You must consider the four tradeoff areas: electrical performance, mechanical perfor-

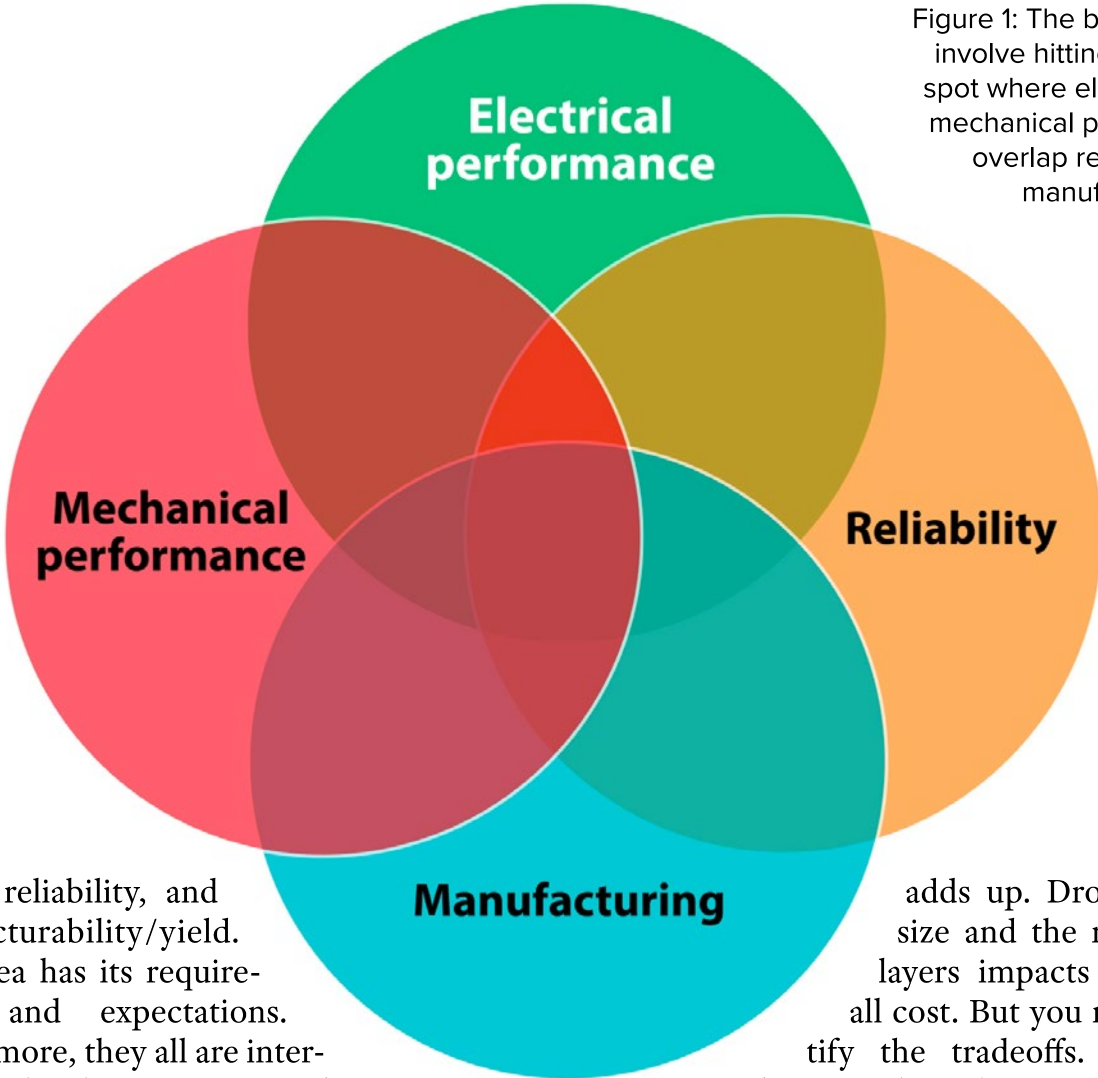


Figure 1: The best designs involve hitting the sweet spot where electrical and mechanical performance overlap reliability and manufacturability.

mance, reliability, and manufacturability/yield. Each area has its requirements and expectations. Furthermore, they all are interconnected and impact one another. The "sweet spot" is that small area in the center where everything is balanced. It is a great designer who knows that spot.

If you feel we don't have enough acronyms, such as design for manufacturing (DFM) or design for reliability (DFR), etc., let me give you another. Our current situation has brought about DFC—design for cost. As if there isn't enough for me to think about in your PCB, I know must keep in mind how much this costs? Designing for cost begins with the very first steps of your design. Know what contributes to the cost in both the fabrication and assembly process.

For example, most of the cost of a PCB design is in the raw materials, specifically the bare PCB itself. It's found in the size of the PCB, the type of FR-4, and the number of layers. It all adds up. Dropping the size and the number of layers impacts the overall cost. But you must identify the tradeoffs. Reducing the PCB design's size impacts the manufacturing or electrical performance significantly when reducing layers has a critical impact on electrical performance.

Almost every time I have reduced the number of layers in a design, it hurt the design in a way that far outweighed the financial cost savings. For instance, taking a four-layer board and reducing it to two to save money resulted in problems. First, it involves physics, with a four-layer design producing 15 dB less radiation than a two-layer board. It allows you to route signals in a microstrip (or stripline) configuration. Finally, the ground plane significantly decreases the ground impedance (and, therefore, the ground noise).

Am I saying we should never use a two-layer design? No. I am saying that you must consider the outcome of your decisions. With changes, does the PCB still meets the customer's desired quality outcome? Also, consider this: Are you introducing problems into the design that may not show up way down the road? However, that is another subject for another day.

4. Manage the risk.

Lastly, manage your risk; don't let risk manage you. We all are too young to get ulcers. I don't need that stress. Stay in complete control of your design process and know the impact of your decisions. Honestly, this may happen by trial and error, and that's okay. It is a learning process but be informed by asking questions of your team—designers, engineers, fabricators, assemblers, etc.—about how you can improve that process.

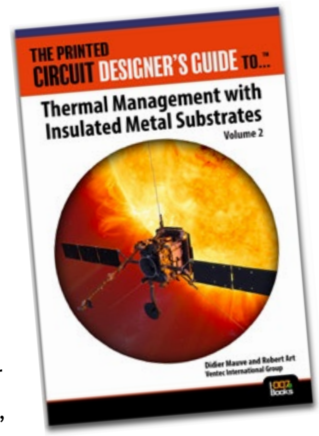
Finally, let us all take a collective deep breath; come on, everyone join in, including you in the back, and know that our present situation will pass. We will come out the other side stronger and better designers because of it. It's during the rough times that we find who we are but, more importantly, what we are made of on the inside.

We are PCB designers. **DESIGN007**

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

BOOK EXCERPT

The Printed Circuit Designer's Guide to... Thermal Management with Insulated Metal Substrates, Volume 2



Chapter 1: High Emissivity

Regarding basic principles of thermal dissipation there are three ways of dissipating energy:

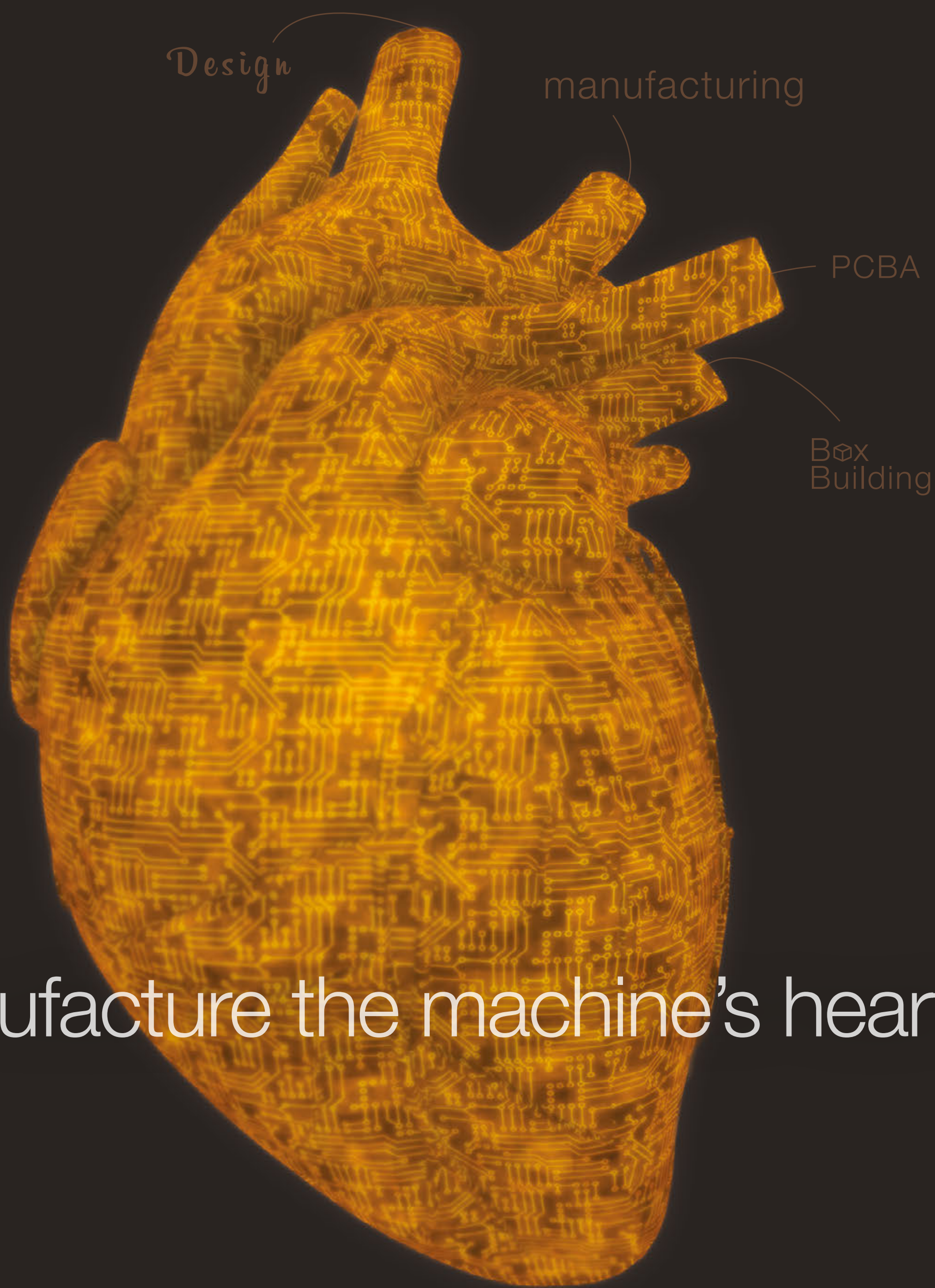
- Conduction
- Convection
- Radiation

Integrated metal substrate (IMS) printed circuit boards rely predominantly on heat conduction all the way through the different layers of the substrates from a hot point (the base of the component) to a cold point (the furthest surface of the metal base) and, usually, thereafter, through a dissipator. Ideally, the thermal engineering of the package, soldered connection(s) to the substrate, dielectric layer, IMS metal base, and dissipator should conduct heat as effectively as possible away from the die and into the ambient temperature. That is, they should have the highest possible thermal conductivity (or low thermal resistance, the one being the inverse of the other). However, various other properties are needed, and the entire system cannot be designed solely around thermal performance. The different elements of the "stackup" (noting that inside the package there is typically some combination of metallic parts such as bond wires and/or leadframe, ceramics or plastics, FR-4 or similar substrate in the case of a system-in-package device or high-performance MPU, and adhesives) can be modelled as a series of dissimilar thermal resistances.

Ultimately, thermal energy conducted into the metal base is transferred into the ambient by radiation. The effectiveness of this transfer is influenced by factors such as the emissivity of the surface and the radiating area in contact with the surrounding air. The emissivity is dependent on the material as well as the surface finish (basic physics tells us that a matte black surface radiates heat more effectively than a shiny surface–and also absorbs heat radiated from nearby sources more effectively than a polished surface).

Download this title today!

© PCB Technologies Ltd. All rights reserved.



Manufacture the machine's heart...

We offer an all-in-one solution through our experience, integrity, and clear communication. Get on board for our joint journey into the future.

www.PCB-technologies.com

ALL IN ONE

PCB TECHNOLOGIES™
leading the way

Benefits of High-Performance Hybrid Multilayer PCBs

Lightning Speed Laminates

Feature Column by John Coonrod, ROGERS CORPORATION

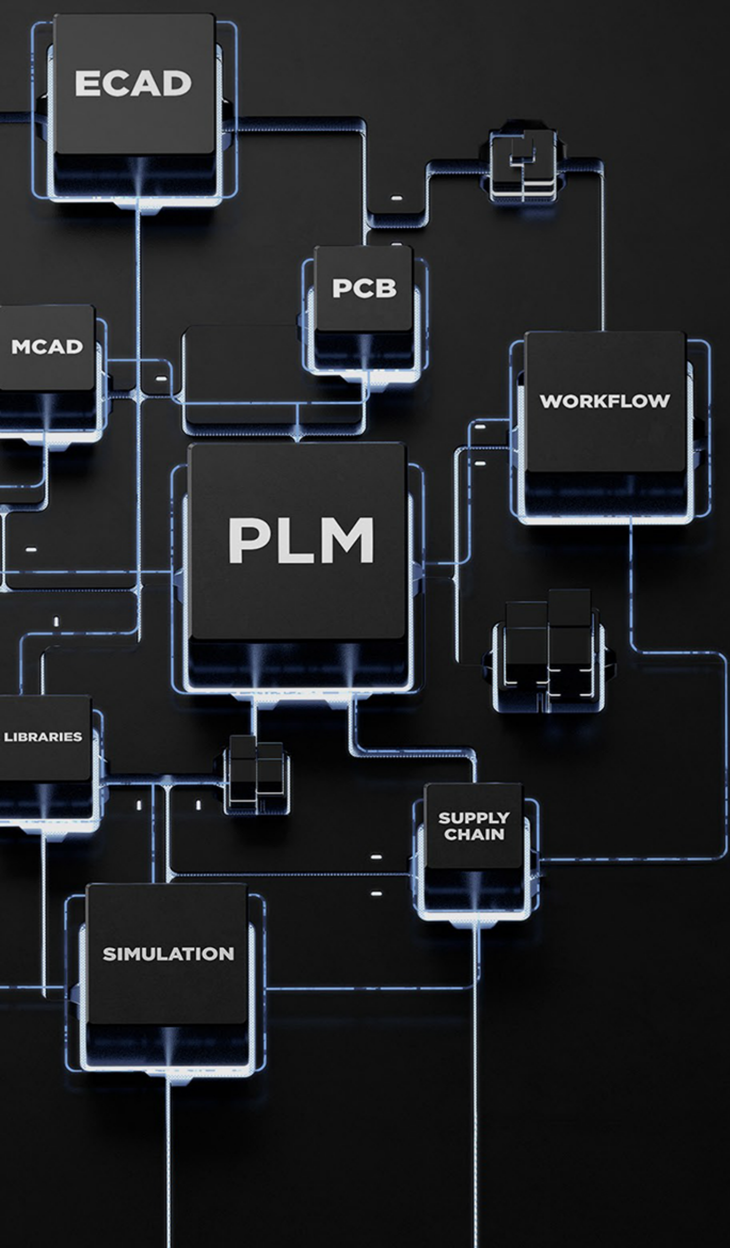
A hybrid multilayer PCB is a circuit consisting of several layers of different circuit materials. This is typically done for cost reasons, but there are other reasons as well. Sometimes using a combination of different materials in a circuit stackup can assist with adjusting the overall coefficient of thermal expansion (CTE) of the circuit and that can be advantageous for circuit assembly and reliability. Additionally, there have been different materials combined for hybrid circuits with the intent of impacting thermal management, adjusting coupling, and customizing electrical properties for different layers within the circuit. Another good reason for combining dissimilar materials for a hybrid multilayer circuit is to improve circuit fabrication.

Probably the most common reason for using hybrid multilayer PCBs is cost. Over the past many years, PCBs have been getting more complex. Today's circuits can be used to process many different types of electrical signals. The circuit may have high speed digital signals, RF signals, control signals, power planes, ground planes, etc. The high-speed digital and RF signals will typically require a higher quality circuit material that is formulated for very good and consistent electrical performance. The high frequency circuit materials, which are commonly used for high-speed digital and RF applications, are generally higher in cost as compared to more standard FR-4 circuit materials. The control signals, power planes, and ground planes do not need the higher quality circuit material and for the circuit layers with those signals, they can use the cheaper FR-4 materials.

Combining low-cost FR-4 materials with higher quality high-frequency circuit materials is often done with today's hybrid PCBs. There are circuit fabrication concerns which need to be addressed when combining dissimilar materials and most

ALTIUM
NEXUS

A comprehensive, digitally integrated electronic design solution for enterprises of all sizes.



- Digitally-connected ECAD to the rest of the enterprise
- Cross-domain and cross-discipline collaboration
- Digitally-managed workflows, processes and resources

As the most lightweight, easy-to-implement electronics design solution for enterprises, Altium NEXUS makes the last mile of digital transformation not only possible - but easy.

Learn more at altium.com/altium-nexus

PCB fabricators are aware of these issues. When a multilayer hybrid PCB is made by combining FR-4 and a PTFE-based material, through-hole drilling can be a major concern. The PTFE material is very soft, and the FR-4 material is rigid, and that interface can be a concern for drilling the through-holes. The drill tool and parameters are critical to ensure a good quality hole is drilled without overheating the PTFE material. If the PTFE material is overheated, it can cause "flap" at the FR-4 and PTFE interface. The flap is a small amount of PTFE that has been smeared over the metal interconnect and that can be a reliability concern. Unfortunately, if PTFE is smeared and a flap exists, there is nothing that can desmear PTFE and the circuit may be discarded.

Combining low-cost FR-4 materials with higher quality high-frequency circuit materials is often done with today's hybrid PCBs.

Another circuit fabrication issue for hybrids is the treatment of the through-hole prior to electroless copper plating. From the above example, the PTFE material will need a different process to make it wettable as compared to the FR-4 material. Generally, the FR-4 material can go through a plasma or permanganate process to get the material wettable, which allows it to accept the electroless copper plating in the drilled hole. However, the PTFE material will usually require a very different process and the circuit fabricator should know the correct processes, the sequence to run these processes, and any post processing that may be necessary. If the circuit fabricator is unaware of these processes, our technical service engineers can get involved and assist with the process definition.

There can be benefits to combining dissimilar materials to help reliability issues. If one of the layers in the hybrid multilayer circuit has a poor CTE, it is possible to combine dissimilar materials with good CTE to get an overall good CTE for the circuit. In many cases this can help assembly concerns and long-term reliability. However, there can be exceptions, and this will need to be well understood, because it is still possible to get poor reliability performance on the layer with the high CTE property.

Thermal management can also benefit from the use of dissimilar materials. If a hybrid PCB has one layer with poor thermal conductivity and other layers with very good thermal conductivity, that can help the overall thermal performance. Again, there are exceptions, and this will need to be well understood by the designer and circuit fabricator to get the improved thermal performance.

There are also hybrids that combine dissimilar high frequency circuit materials for customized electrical performance. One concept that has been used many times is to use a broadside coupled stripline circuit with the material between the signal planes having a high Dk, and the materials between the ground-signal planes having a lower Dk. The lower-Dk materials will allow a wider signal conductor and that will give lower conductor losses. High Dk material is very good at condensing electric fields, so having the high-Dk material between the coupled signal planes will increase the coupling. This is one way to get a tightly coupled circuit with lower losses.

Hybrid multilayer PCBs have been used for many years and the trend will continue. It is a way to get more performance out of a circuit, sometimes improving reliability and possibly reducing the cost of the circuit. DESIGN007

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

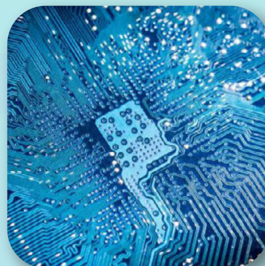
Candor
Circuit Boards

Innovative solutions for complex circuit board manufacturing

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

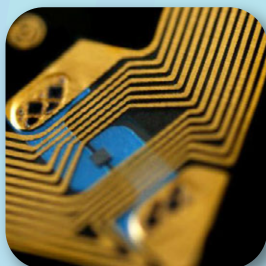
RF/Microwave PCBs

Power Electronics



High-Speed Data

Automotive Electronics



Lamination

Startups

Our technology produces what our customers dream!

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

Controlled Impedance: The Devil Is in the (Math) Details

Connect the Dots

by **Matt Stevenson,** SUNSTONE CIRCUITS

Controlling impedance is critical to signal integrity and board performance in devices powering everything from high-speed digital applications to telecom and RF communication. It is common practice for designers to include impedance-related notes with their PCB designs and rely on the manufacturer to determine the proper trace parameters. This inherently passive methodology often leads to delay, cost overrun, and even batches of useless boards.

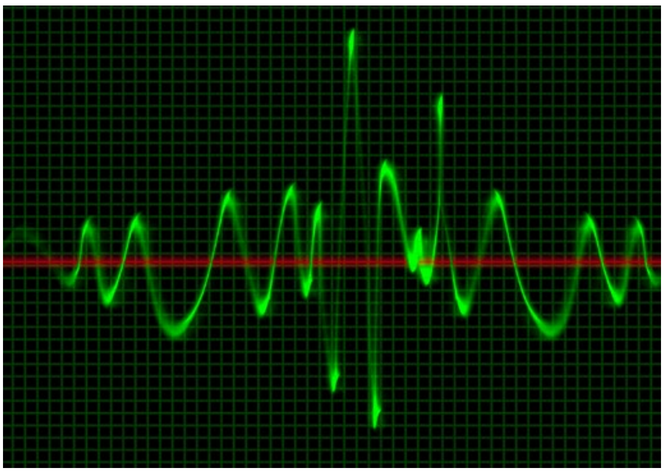


Figure 1.

Designers can save time, money, and effort if they are aware of the impedance math as they design their board. Impedance math is heavily influenced by the distance an electrical signal must travel between components.

Comprised of a signal trace and a return path, a PCB transmission line transmits an electrical signal from one component to another. To function as a transmission line, the line length must be at least one quarter of the signal wavelength. For most transmission lines, the return path is a ground plane located on a layer above or below the signal trace.

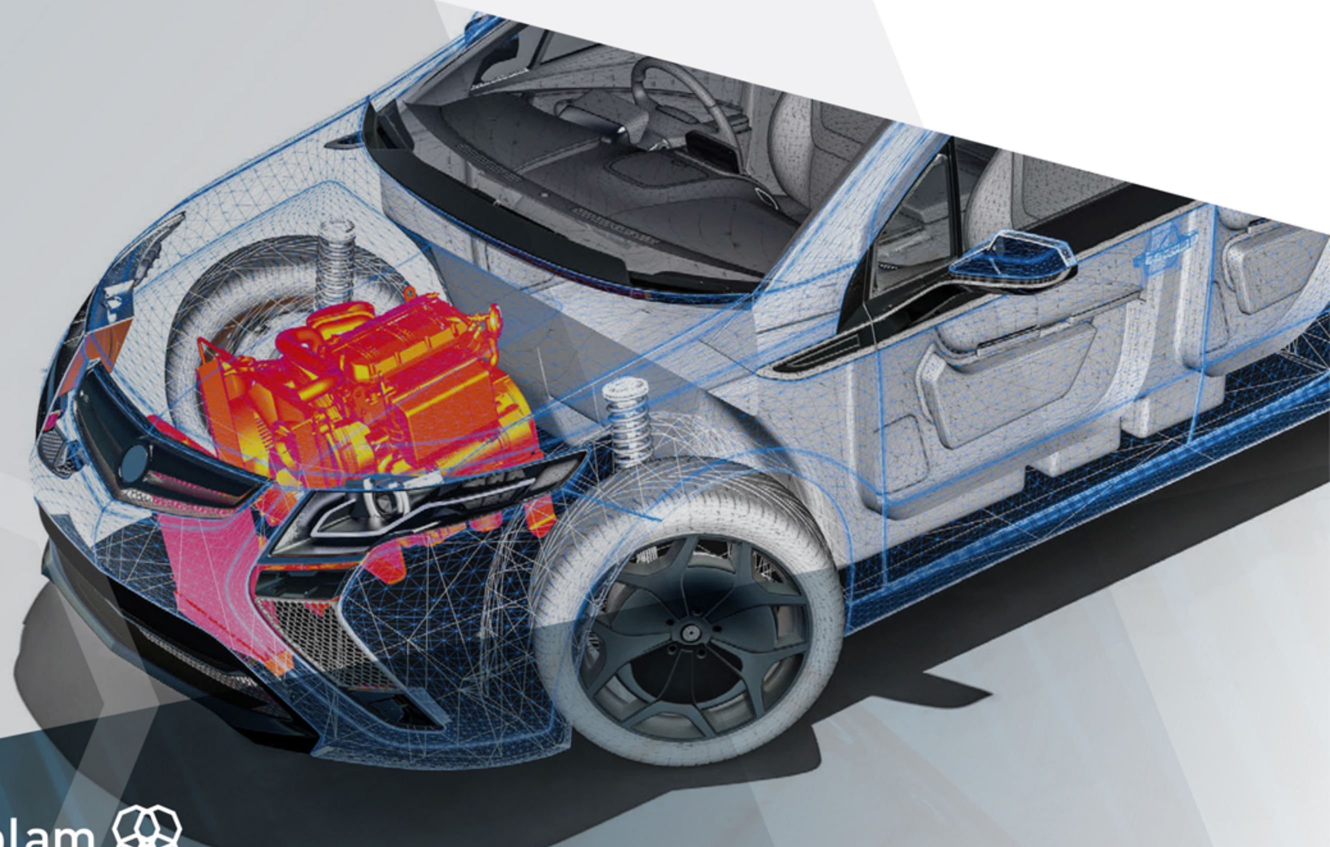
Impedance comes into play when transmission lines are part of the PCB design. It is equivalent to the measure of resistance in a DC circuit and is also measured in ohms. The higher the impedance, the higher the input voltage must be to achieve a desired current in the circuit.

For high speeds greater than 100 MHz, frequencies and noise sensitive signals are used to control the characteristic impedance and are important to maintaining signal integrity. Mismatches in the impedance of these high-speed signals can cause signal reflections—meaning a portion of the signal is traveling in the opposite of the intended direction. The greater the difference in impedance, the greater the amount of signal reflected in one portion of the circuit, which will cause the signal to be noisy and could even cause a disruption of the signal.

Figure 2.

ventec
INTERNATIONAL GROUP
騰輝電子

autolam: Base-Material Solutions for Automotive Electronics



autolam

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

IATF 16949:2016 CERTIFICATION

venteclaminates.com

Several factors that will influence the impedance of a circuit are:

- Cross-sectional area of the transmission line (height x width)
- The height of the dielectric material between the transmission line and the reference plane
- The dielectric constant of the dielectric material
- Spacing between the transmission lines (differential pairs)

The devil is in the mathematical details when it comes to controlling impedance. Documenting impedance requirements properly is more onerous than most people realize. Simply stating target impedance, trace requirements, and material tolerances often leaves knowledge gaps for the manufacturer and increases the likelihood of producing boards that do not make the cut during testing.

The math is the math when it comes to controlled impedance and it must be done at some point. Why not during the design phase? Using an impedance calculator will help designers build the right tolerances into their designs and avoid issues like signal mismatch. Impedance testing then becomes a double-check instead of the tool relied upon to determine if impedance documentation is correct.

When it comes to the math of controlling impedance, even simple equations really aren't that simple. The following equation,

$$Z = \frac{87}{\sqrt{\varepsilon_r + 1.41}} \ln\left(\frac{5.98H}{0.8W + T}\right)$$

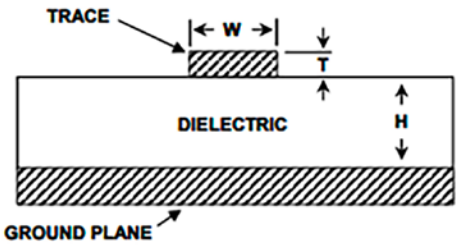


Figure 3: There are a variety of free online controlled-impedance calculators available to make this math easy for designers.

taken from IPC-2141, is used for a single ended microstrip structures. It is a good demonstration of the influences on impedance. This is the simplest mathematical representation of impedance, and it can present up to a 7% error in calculations at high speeds.

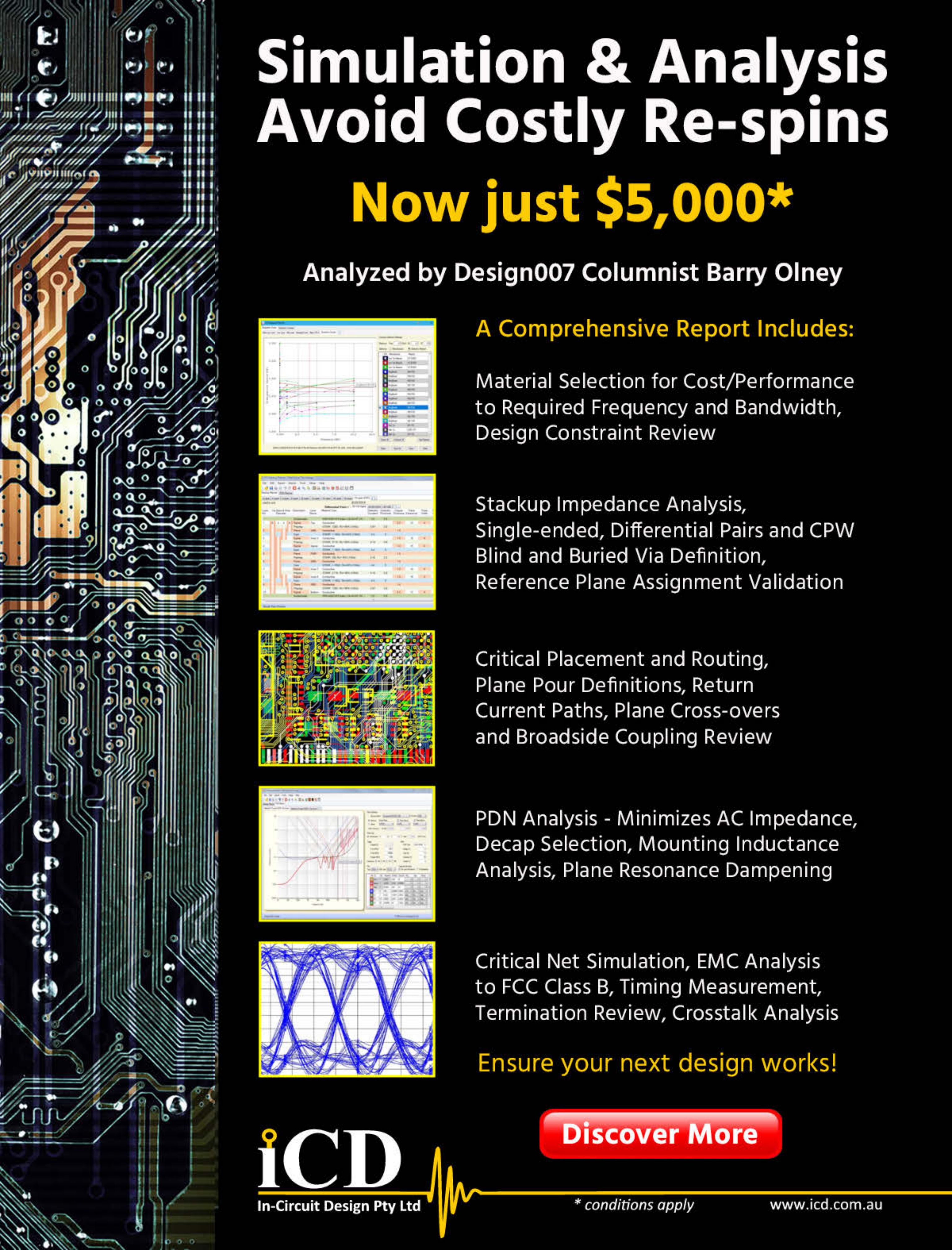
W = Width of trace
T = Copper thickness
H = Dielectric Thickness
ε_r = Dielectric Constant (D_k)

Lack of specificity in the design notes can create risk during the transition from design to manufacture. The components of this equation all come with a +/- tolerance during manufacturing and raw material creation. For a PCB to have the desired result of matching or controlled impedance for critical structures, it is imperative that the PCB manufacturer understand these factors.

Manufacturing doesn't happen in a vacuum and lessons learned from past experience will help reduce risk associated with controlling impedance. PCB manufacturers should be collecting data on every board produced, and they can leverage that in predictive models that will provide a good starting point to fill in documentation gaps and achieve the desired outcome.

A good partner for producing boards that reliably handle high-speed signals uses sound methods for process control and data collection. They fundamentally understand the influence of impedance and scrutinize signal transmission. But it is a two-way street. The designer should consider impedance factors in advance of manufacture. Matching the material characteristics (D_k), targeted layer spacing, and impedance needs during the design phase will make the process go more smoothly and help avoid costly rework. DESIGN007

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


Simulation & Analysis
Avoid Costly Re-spins
Now just $5,000*
Analyzed by Design007 Columnist Barry Olney
A Comprehensive Report Includes:
Material Selection for Cost/Performance to Required Frequency and Bandwidth, Design Constraint Review
Stackup Impedance Analysis, Single-ended, Differential Pairs and CPW Blind and Buried Via Definition, Reference Plane Assignment Validation
Critical Placement and Routing, Plane Pour Definitions, Return Current Paths, Plane Cross-overs and Broadside Coupling Review
PDN Analysis - Minimizes AC Impedance, Decap Selection, Mounting Inductance Analysis, Plane Resonance Dampening
Critical Net Simulation, EMC Analysis to FCC Class B, Timing Measurement, Termination Review, Crosstalk Analysis
Ensure your next design works!
Discover More
iCD
In-Circuit Design Pty Ltd
* conditions apply
www.icd.com.au

PCB007 Highlights

One World, One Industry: Opening New Opportunities in Mexico ▶

IPC and WHMA have long supported the electronics assembly and wire harness manufacturing industries in Mexico, but recent regional growth coupled with supply chain disruptions necessitated a closer relationship.

IPC: Over 100 Industry Executives Urge U.S. Congress to Strengthen Electronics Supply Chain ▶

More than 100 top executives representing companies in the U.S. electronics manufacturing industry are urging the U.S. Congress to address critical shortcomings in the printed circuit board industry and the entire U.S. electronics supply chain.

The New Chapter: My Interview With Happy Holden ▶

This past year, I set up several informational interviews with individuals across the industry. I saw this as an avenue to both enhance my own career and provide insight for my peers. To that end, I had the incredible honor of interviewing Happy Holden, the father of HDI PCBs. His insight into what it takes to be an excellent engineer and grow exponentially in this industry is unrivaled.

The Plating Forum: Plating in Electronic Applications ▶

Plating is the deposition of a metal layer on a substrate in order to modify its properties. It occurs when the metal ion in an aqueous solution is reduced to the metal: M+ metal ion + e- reducing electron = M0 deposited metal.

Testing Todd: ET and the DoD ▶

Building printed circuits can be a tricky business. There are many attributes that go into the production process. Initially, there is the sales interface with the customer, and the receipt of the data for the initial quotation. Then there is the procurement process for raw materials. This month, let's dive into the DoD and how this affects electrical test.

Ventec Meets Demand for Taiyo LPI Solder Mask Products in Europe ▶

Following Ventec's announcement of its exclusive distribution agreement with Taiyo, customers in mainland Europe and the UK are now guaranteed reliable, immediate, and flexible access to the full color range of Taiyo Liquid Photoimageable (LPI) Solder Mask inks.

Happy's Tech Talk #9: Radars, Missiles, and the World's Costliest Computer ▶

Let's have a little fun and walk back nearly 70 years into the history of electronics and computers. What was the world's costliest computer and why? The answer is not today's supercomputers, nor computers built during World War II. Instead, it lies in a real-time air defense radar system built during the height of the Cold War of the 1950s that had left the U.S. extremely vulnerable to a Soviet bomber attack.

Additive Reality: Drop It, and Enjoy the Greenback ▶

The columns so far have brought up the technical aspects of inkjet solder mask and made readers more familiar with this technology. This last column instead will show how the technical aspects match the commercial ones.

BENDING THE POSSIBILITIES

FLEXFINER

BY TAIYO

AVAILABE IN TWO COLORS

AMBER

BLACK

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

OUR BIGGEST FLEX YET!

Contact your local sales representative today!

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

TAIYO AMERICA

Follow us on social media

All Resins Are Not Created Equal

Sensible Design

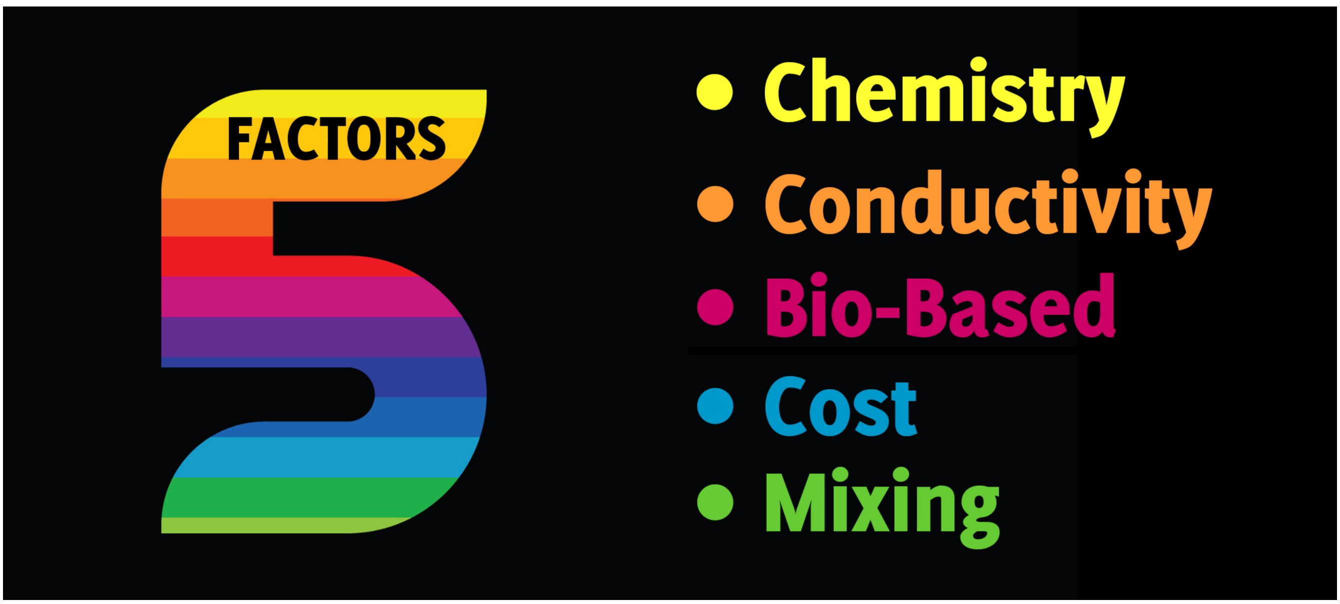
by Beth Massey, ELECTROLUBE

With so many different encapsulation resin options on the market, selecting a resin that is best suited for your application can present a real challenge. Today, there is a wide range of different resin systems available that offer a comprehensive range of different properties. It is often assumed that properties can vary moderately, however, resins can offer significantly different benefits. This month, I will explore the different factors you should consider when comparing resins for your application. Every project will differ in terms of the unit, the dispensing method and equipment that will be used, the cure time, and temperature limitations during the production process.

First, I establish the the resin's ultimate operating conditions, temperature range, and likely chemical exposures. I will look at resins that have the biggest impact on performance and reliability. Let's look at where to start and what you need to factor in using our signature five-point format.

Resin Chemistry

It's easy to assume that all polyurethane resins or epoxy resins would have broadly similar strengths and weaknesses as they belong to the same chemical family, but this isn't the case. Different materials used in the part A and part B can have a big impact on the performance of the end material. Different fillers and loading can affect thermal conductivity, and various additives can influence processability, cure time, and operating temperature range. An example is whether a polyurethane part A is polybutadiene (polyBD) or polyether/polyester based.



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
DEPARTMENT OF DEFENSE
UNITED STATES OF AMERICA
ITAR

PolyBD resins generally have exceptional resistance to water, even holding up well against saltwater immersion, but are susceptible to attack by organic solvents such as oils and fuels. Resins with a polyether/polyester part A base, however, generally have a better resistance to organic solvents. But while they offer good water resistance at lower temperatures, they typically are not as strong as the polyBD based materials when the conditions are both hot and wet. Even within those broad categories there are differences in performance, which is why it's best to bring your application and conditions to specialists in the first instance so a resin can be matched with the best combination of properties for your needs.

Thermal Conductivity

The trend for miniaturisation means high power density devices are increasingly common. Thermal dissipation away from hot areas of the device to heat sinks and metal casings is more important than ever.

The trend for miniaturisation means high power density devices are increasingly common.

The thermal conductivity of a resin is influenced by the type of conductive filler used and even particle size distribution and particle shape, as this will affect the packing of the particles, and thus heat transmission, through the material. Thermal conductivities quoted on datasheets are of limited value as different test methods can give different values for the same material. This means that two resins with the same stated TDS value can perform very differently in-application.

Some methods are most accurate when measuring lower thermal conductivity materials, such as the guarded hot plate method, while others such as the laser flash method are particularly strong for testing at high temperatures. Several methods (including both already mentioned) rely on precise test sample dimensions, which means the testing is time-consuming and can be influenced by operator technique and precision. We use a modified transient plane source (MTPS) instrument which is a non-destructive technique suitable for liquids, solids, pastes, and powders, and which gives accurate and repeatable measurements, even for smaller sample sizes.

To achieve the most accurate comparison it is important to test side by side in-application, if possible, to see the real-world heat reduction achieved. This is not just relevant for resins, it also applies when considering thermal interface materials.

Bio-based Raw Materials

Sustainability is becoming ever more of a priority globally, for manufacturers and for us here. We take our ESG targets very seriously and have been devoting a lot of R&D time to the use of bio-based raw materials in our resins, which led to the introduction of our UR5645 resin, a chemically resistant, high temperature bio-based PU. There is an element of future-proofing in using bio-based resins, as synthetic raw materials derived from crude oil will not be available forever. Results demonstrated by our resin development chemist, Beth Turner, have highlighted that there can be significant technical benefits from transitioning over to bio-based raw materials. The results also show that not all bio-based raw materials are created equally. A lot depends on their origin as some cheaper bio-based fillers did not offer the same performance benefits as other options, and users cannot assume that all resins advertised as using bio-based materials will offer a technical advantage. There are numerous opportunities for bio-based encapsulation resins for use

Integrated PCB Design, Effortless Collaboration

Collaborate From Anywhere, Innovate From Everywhere With OrCAD for Teams

Optimize every part of your design process with the only real-time PCB design and data management platform built to scale with you and your team. From automated supply chain insights and constraint-driven, high-speed design and analysis to one-click release-to-manufacturing, OrCAD for Teams is architected to help you work more efficiently and effectively than ever before.

Get OrCAD for Teams »

EMA Design Automation

OrCAD™ CADENCE PCB SOLUTIONS

in the electronics industry, given the distinct performance advantages they provide in harsh environments, underwater applications, and hot and humid operating environments.

Resin Cost

Comparing the cost of resins can potentially cause confusion because resins are often priced in kilograms but used in litres as the unit to be potted is of a fixed volume—which means density can have a big effect on the actual cost per potted unit.

If we consider two resins which have the same apparent cost per kg, but resin A is 1 kg/L (typical for an unfilled resin) and resin B is 2 kg/L (a highly filled resin), it will cost twice as much to pot a unit of fixed volume X using the denser resin vs. the lighter one. The difference in densities is not usually this extreme as other requirements which are influenced by filler content, such as thermal conductivity and flame retardance, mean it's unlikely for both a completely unfilled and a heavily filled resin to be in the running for the same application. However it can still be an important consideration when choosing a resin for a cost-critical end-use.

Mixing

Not all mixers are created equally, both in terms of manual operators when using the resin bi-packs for prototyping, and also when using a mix-and-dispense machine for larger production runs. An improperly mixed resin will not cure properly, leaving voids and a sticky or uneven finish. Even if there are no visible defects, it is highly likely the material will not have the final cured properties expected, which could lead to unit failures later on and will certainly not allow a fair comparison between materials at the approval stage.

It's important to have a good resin pack mixing technique. There are several videos available on our website and our YouTube channel that give a good grounding in the best way to use our resin packs.

If using a machine, it's important to work with your resin provider and your machine manufacturer to identify a static mixer with the correct attachment for your machine, the correct shape and mixing style, and with sufficient elements, diameter, and length to give the resin a good mix. The resin chemistry, densities, and viscosity difference of parts A and B, as well as the mix ratio, desired flow rate, and shot size, will all influence the amount of mixing required and consequently the selection of the mixer. Depending on whether the resin is filled, heating, stirring, and recirculation may also need to be considered on the machine's storage tanks as well.

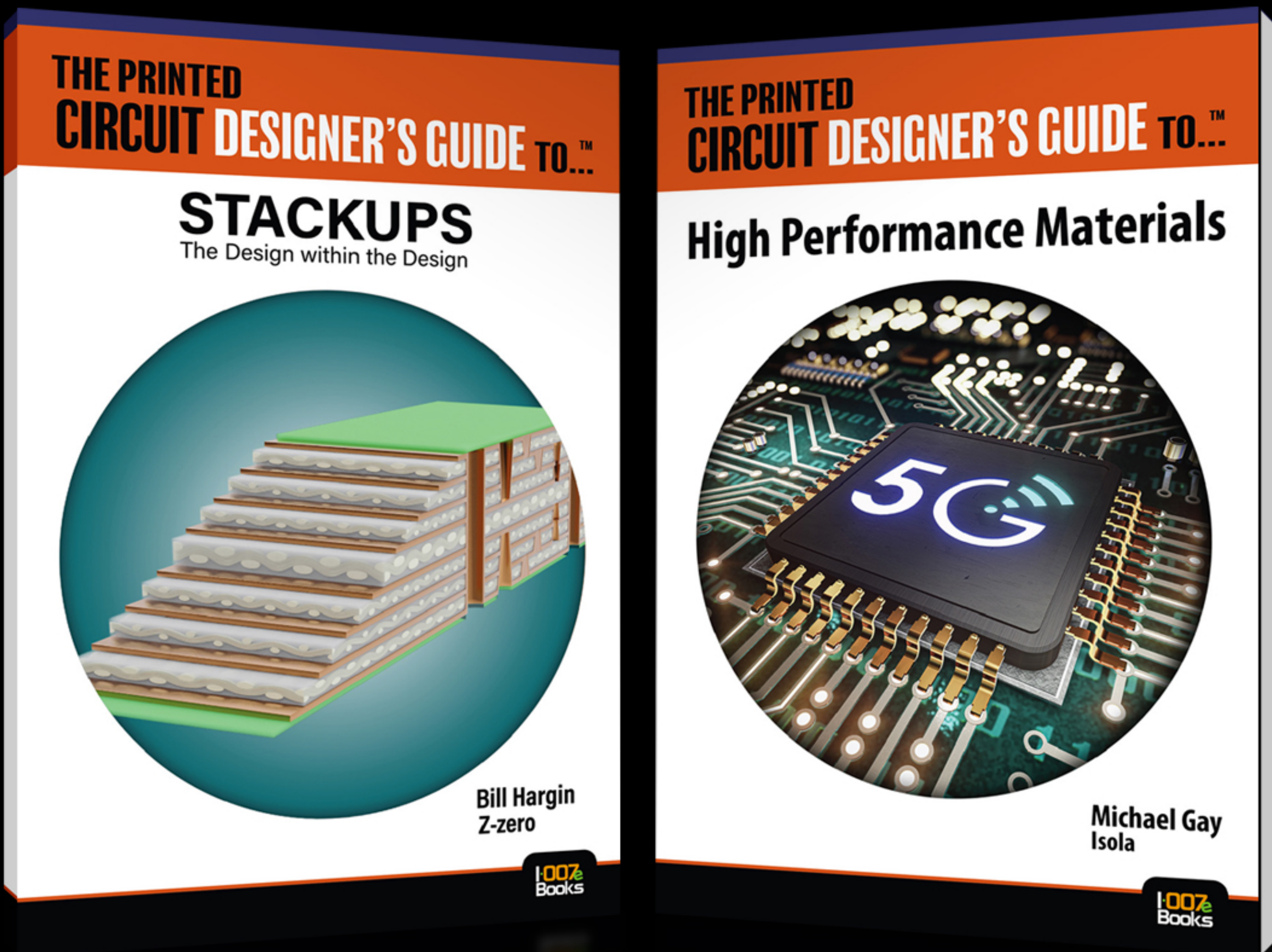
An improperly mixed resin will not cure properly, leaving voids and a sticky or uneven finish.

Not all resin manufacturers offer the same level of customer service or technical support, so do ask them what level of support you can expect to receive before embarking on your project with them. I hope you have enjoyed this month's column and picked up some top tips on comparing resin systems. I cannot stress enough how looking at bio-based resin systems could be of significant benefit when considering a resin system for your next application. **DESIGN007**

Beth Massey is head of encapsulation resins at Electrolube. To read past columns from Electrolube, click here. Download your free copy of Electrolube's book, *The Printed Circuit Assembler's Guide to... Conformal Coatings for Harsh Environments,* and watch the micro webinar series "Coatings Uncoated!"

Better Together.

These books can help you streamline your process.



See the Reviews

I-007e Books

Practical Packaging Density in PCB Design

Target Condition

Feature Column by Kelly Dack, CIT, CID+

Have you ever sat in an overcrowded meeting room? For me, this is never fun. I think of elbows knocking against others and poor ventilation. My performance degrades when I'm overcrowded. Conversely, when I sit in meetings within oversized conference hall rooms containing too much empty volume, I feel uncomfortable. I can't help being distracted by the hollow, reverberating sounds as well as thinking about the costs and waste of resources. To engage effectively, I need to be closer to my fellow attendees and the presenter.

PCB Design Agoraphobia

Just as I tend to exhibit discomfort in oversized conference halls and meeting rooms, I self-diagnose as agoraphobic when it comes to entering PCB layouts full of wasted space. Mechanical constraints which utilize the printed circuit board substrate material as a mechanical or structural "filler" within an electronic product, under-utilizing the material for its intended purpose to support conductive circuitry, grate on me like nails on a chalkboard. I've seen an eight-inch, triangular shaped support bracket designed out of 0.093" thick FR-4 laminate material. It was contrived and specified by an uninformed mechanical engineering overachiever who proposed transforming the original sheet metal bracket into a "dual purpose" PCB solution in order to support the electronic engineer's connector and associated circuitry. The circuit added to this clumsy "printed circuit bracket" monstrosity occupied just over a single square inch and seemed a ghastly waste of resources.

I've lived through other well-intentioned mechanical engineering efforts to cut assembly costs—without scratching off my skin. Once, an uninformed mechanical engineer proposed a PCB to reduce assembly costs by eliminating connector harnessing between multi-sectioned PCB assembly sets. It did not go well. It involved creating a single piece LED board which could be installed quickly into a machine using only six fasteners—a novel requirement but without consideration of all the materials and processes associated with the design for bare board manufacturability. Each single 19-inch diameter board ring—dubbed "the panel burner"—wasted an entire, oversized panel of copper-clad FR-4 material. The board could only be processed by a custom supplier at the time.

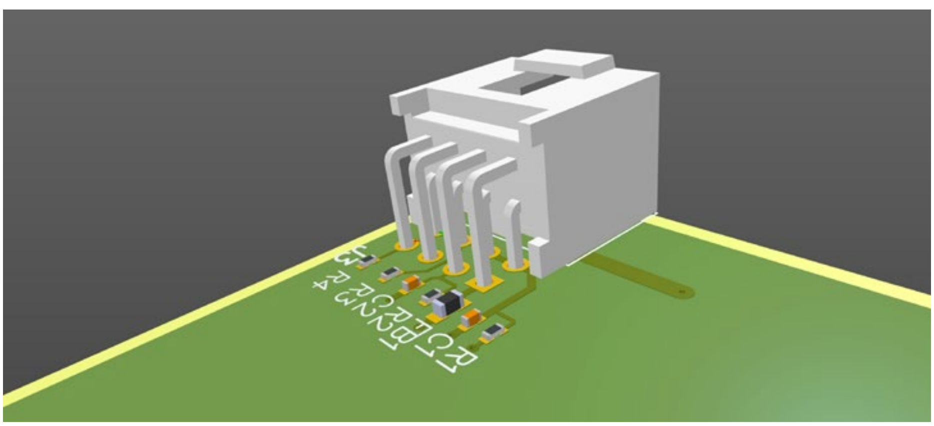


Figure 1: 3D pic of empty boardscape.

$10 MILLION IN RECENT TECHNOLOGY CAPEX INVESTMENTS
NIST 800-171 COMPLIANT CYBERSECURITY PROCESSES
250,000 SQUARE FEET SUPPORTING ALL TECHNOLOGIES
4 STATE-OF-THE-ART NORTH AMERICAN FACILITIES

VIEW OUR QUICK TURN VIDEO NOW

U.S. Partnership for Assured Electronics USPAE

SUMMIT INTERCONNECT
www.summit-pcb.com

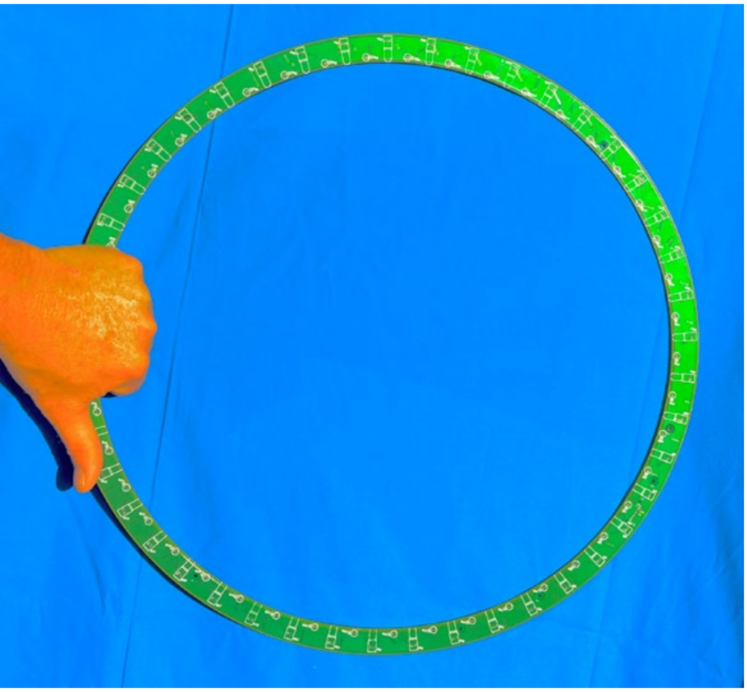


Figure 2: The "panel burner" consisted of a lot of material, very low design density, and was a very expensive-to-manufacture bare PCB.

PCB Design Claustrophobia

Sometimes, when our front-end packaging stakeholders seem to be working for "Team Crazy," their PCB mechanical constraints can send me in an opposite direction. Over many decades we have been watching as electronic components shrink in size. Each new generation of components seems to include a smaller packaging scale which is construed to be good for shrinking products and using less material. But from an actual product design standpoint, electronic products have gone through an identity crisis with regard to packaging size. For example, all that must happen for an electronic product to be rendered obsolete and discontinued is for a competitor to come out with a smaller, more tightly packaged alternative. Industrial designers are kept busy sketching out increasingly smaller electronic products from the outside-in and passing on their concepts to the mechanical engineering teams. Mechanical engineers seem to only re-design their molds and choose higher performance materials to meet the industrial designer's smaller constraints while electronics packaging engineers and PCB designers are often left scratching their heads trying to select available components and circuitry which can be manufactured to fit within a new, sleek, ergonomic, and attractively small product housing.

There is a classic, ongoing challenge for electronics design stakeholders today. It is commonly attributed to EEs residing at the end of the product design cycle. Creative, responsive, industrial design and mechanical engineering stakeholders are usually well on their way to offering their deliverables well before components can be found and the electronic assembly bill of materials is even finalized. Electronics packaging stakeholders are often faced with pressure to cram their circuitry onto less-than-ideal PCB real estate, robbing process efficiency from every other electronics manufacturing stakeholder involved.

As a PCB design stakeholder, I can't tell you how many times I've created a PCB outline per mechanical specification only to discover there is not enough space for the component footprints to fit on the board. This condition is typically resolved, however, by a quick discussion with the project team that will need to review its whole design process and either make the PCB larger or continue to find smaller parts. We usually won't see them back for an average of about two weeks, which frees us up to work on other, more practically dense PCB layouts. There is a more difficult condition, however, occurring from time-to-time when the parts within a design layout happen to just barely fit within a specified PCB outline. This PCB design condition is on the hairy edge of "doable." The parts fit, but will most certainly result in an extremely dense layout with compromised design rules. While this condition is a welcome challenge to any emerging PCB design layout engineer, designs exhibiting extreme density cause me to self-diagnose as claustrophobic. It's not because I don't like a challenge. I'll work overnight as hard as any PCB designer to push and shove design features to intelligently set design rule constraints. But when it comes to compromising on well-established manufacturing proximity constraints, I become anxious. These days, it makes my skin crawl to have to tighten up a tolerance or nudge a trace or component knowing that I'm pushing a systemic problem downstream which may cause a logjam of challenges for my fellow stakeholders.

Over the years there have been many logjams and I have developed special empathy for all the valuable stakeholders in our electronics industry struggling to play a part in a PCB assembly project's success when DFM considerations are ignored. Dense packaging solutions are not easy. I seriously consider any decision moving a layout from the IPC general design producibility level A to a reduced producibility level C without substantial justification and buy-in from all project stakeholders. Sometimes though, any of us can get caught up in the shimmer of new technology without counting the project costs from a holistic viewpoint.

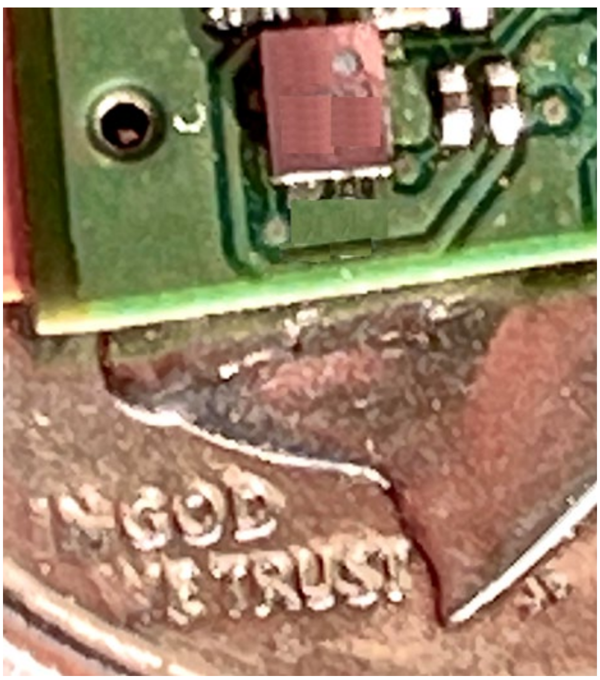


Figure 3: Extremely-dense PCB means minimal material usage, micro-scale packaging density; 0.24 mm pitch BGA shown presumed to exceed IPC producibility class C.

Feng Shui in PCB Design?

The traditional practice of feng shui claims to use energy forces to harmonize individuals with their surrounding environment. I recall hearing a production floor manager once complain about a dangerously unorganized area of a stockroom, facetiously pointing out that the room was not feng shui. Was he serious? Regardless, I've always wondered if there was a place for feng shui in PCB layout. Looking at a printed board assembly from a distance, one can imagine it looks much like a city with components appearing as neatly placed buildings and traces as pristine roads. A well laid out PCBA should provide a feeling of positive energy from the product. I've experienced this feeling many times while visiting PCB fabrication or assembly suppliers. The front lobby features an impressive glass cabinet display housing various bare printed boards or full assemblies—examples of their work—and I feel the energy of project stakeholder success. The high-tech examples in these cases radiate the case that the PCB engineering and manufacturing teams in this company have worked together to achieve the satisfaction of their customer and are proud of what they have produced.

But what happens to these designs if a part or material becomes unavailable? The design will no longer be feng shui because it has become "out of balance" and must change. Now what?

Practical Packaging Density, Plus

We see how using an abundance of material to solve a single problem can create other problems. On the other hand, we see efforts to shrink and shave every last scrap of real estate or material can cause adverse effects regarding producibility. We now consider the concept of feng shui in establishing just the right balance of PCB yin and yang to come up with an ideally balanced PCB assembly. Consider the words of Greek philosopher Heraclitus, who is credited with exclaiming: "Change is the only constant."

Over the past few years, project stakeholders have been faced weekly with rippling pandemic effects causing severely disruptive component and material shortages. The extenuating supply chain challenges are causing perfectly performing board design layouts to require making room for substitute parts which may only be available in larger form factors or less-than-ideal packaging technologies which do not match the "qi"—the vital life force—of the PCB design. Argh! Enough on the philosophies already. What can be done to fix this?

Enough on the philosophies already. What can be done to fix this?

I've actually been incorporating a PCB design concept of my own for quite a while. I call it practical packaging density, plus (or PPDP). It's rather simple.

First, packaging density for electronics has long been considered an essential measurement for determining ease or difficulty of manufacturing, reworking, or even re-designing a product. Packaging density can have a monumental effect on electronic performance. We should strive to start our electronics product design methodology by considering all materials, manufacturing, and performance constraints from the inside out for a product, to establish a practical packaging density for our PCBs. This means that the design layout will be stakeholder friendly and priced right for all project stakeholders, including the customer, throughout its design cycle. This means a project team will have to spend a lot more time on the front end of a project researching availability, manufacturing constraints and, most importantly, dialing design rule settings away from limited IPC class C producibility to achieve the definition of practical packaging density.

However, I've met too many designers who are in the habit of calling up a local PCB prototype shop to ask about pushing the shop's capabilities: How small holes can be drilled, how close copper images can be spaced and etched, and how tight solder mask patterns can be registered. They ask and are typically given minimum producibility requirements. Upon receiving the values, they go right to the design constraint settings in their layout tools and enter the (minimum) values given. This philosophy is counter-productive for all. Designing to minimum manufacturing constraint capability is like continually red lining a running motor. It's hard on the systems, which are bound to break down.

The final step in the PPDP concept is to incorporate some extra space. To move toward successful packaging design, designers need to reverse thrusters by adding more space and clearance to what is already considered practical. This is the plus in PPDP. It is the plus that will end up helping reduce the domino effect of component placement if your team cannot find drop-in replacements for those components which have gone EOL (end-of-life). It is the plus that will help out that tech who needs just a bit of extra clearance to rework that component. It is the plus that is very minor regarding increasing material costs but can save countless dollars in time and effort in manufacturing, rework, test, and re-redesign.

Regardless of our viewpoints on metaphysics, feng shui, or any unfamiliar cultural traditions involving proximity, energy, crystals, harmonic forces, and even a little "black magic," it may be surprising for us to realize that creating a holistically successful printed circuit assembly already includes most all these ancient ingredients plus a little bit more. DESIGN007

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

Integrated Tools to Process PCB Designs into Physical PCBs

Visualize

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

Verify

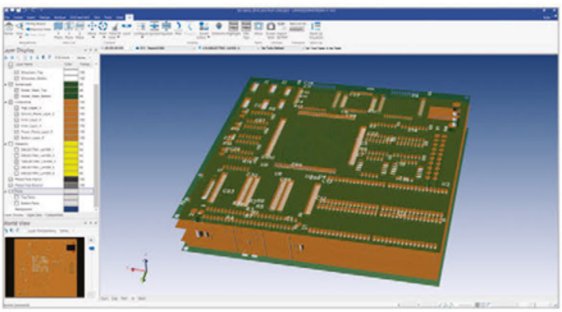
Ensure that manufacturing data is accurate for PCB construciton.

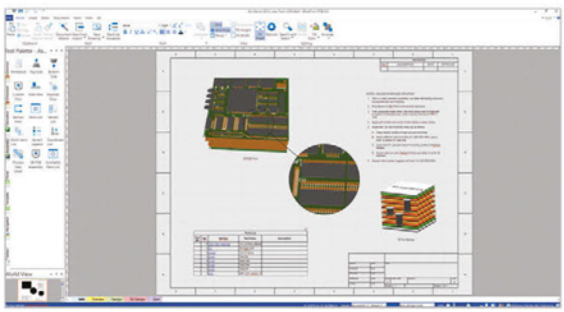
Panelize

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

Document

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





CAM350®

Verify and Optimize PCB Designs for Successful Manufacturing

BluePrint-PCB®

Create Comprehensive Documentation to Drive Fabrication, Assembly & Inspection

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

DownStream Technologies

800-535-3226 | downstreamtech.com

©2019 DownStream Technologies, Inc. All rights reserved.

NOW WITH 3D

Webinar Review:

Thermal Integrity of High-Performance PCB Design

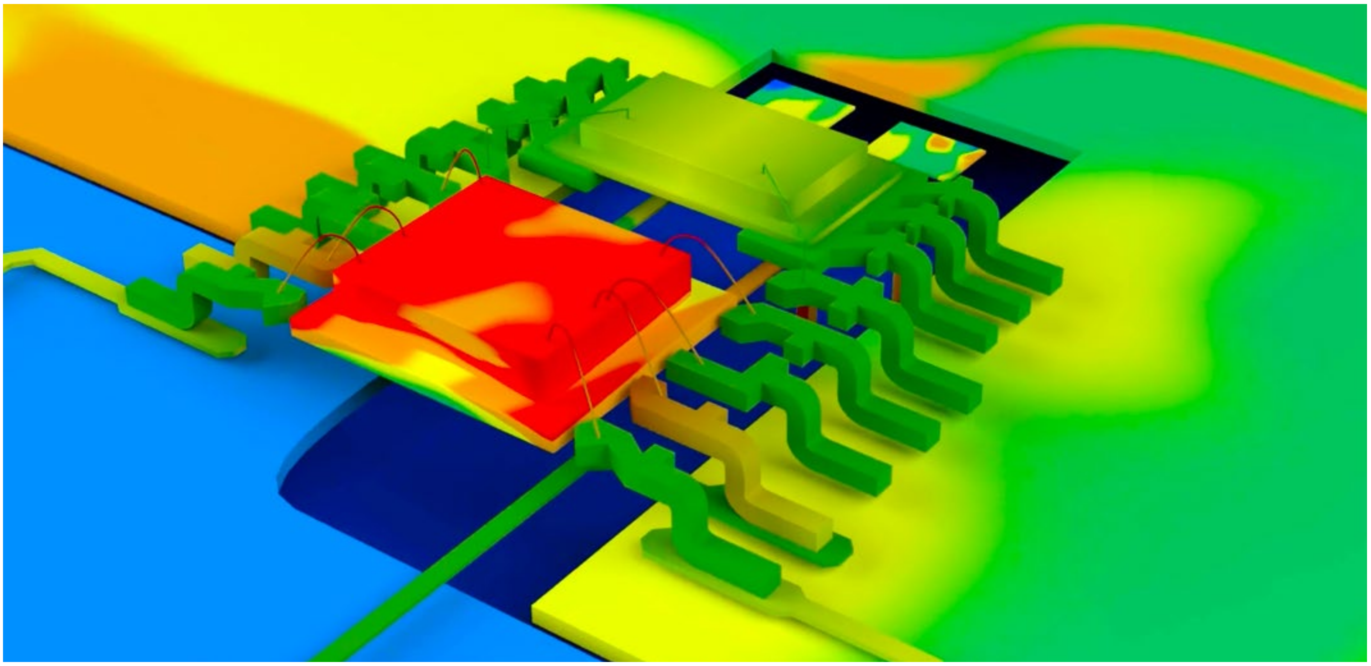
by Andy Shaughnessy
I-CONNECT007

Electrical and mechanical engineers may be working on the same product development teams, but they speak different languages, and they have completely different objectives. As a result, these folks almost never use the same software tools.

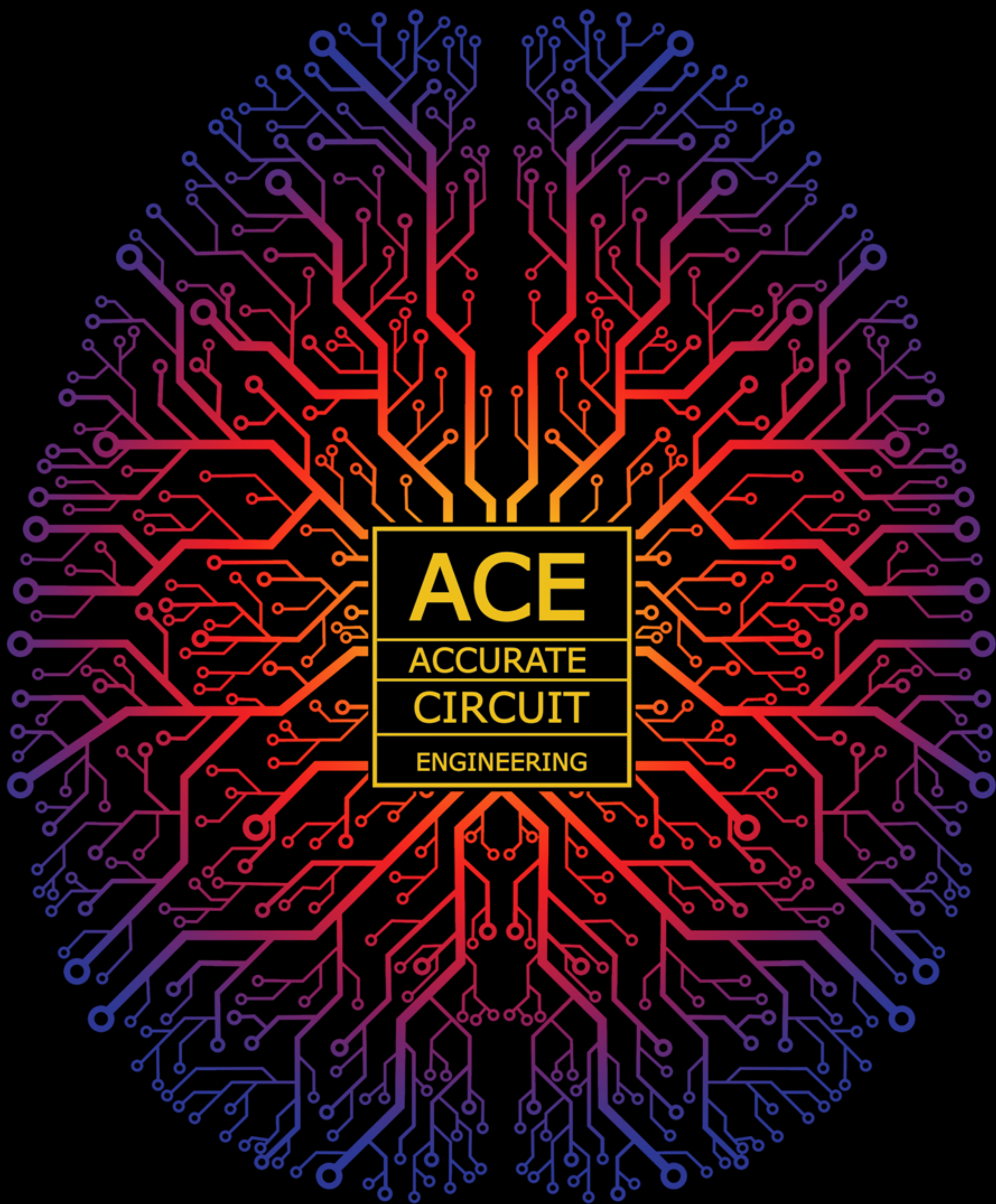
But Cadence's new Celsius Thermal Solver is an exception to the rule. In a new CadenceTECHTALK webinar, "How Static and Dynamic IR Drop Analysis Can Help PCB Designs and Challenges," product manager Melika Roshandell and SerDes SI/PI engineer Karthik Mahesh Rao explain how the EE and ME can both use the Celsius Thermal Solver to achieve their disparate objectives—at the chip and package/board levels.


Melika Roshandell

Melika discusses the solver's capabilities on the PCB and packaging side, which are accessible during pre-layout stages of the design cycle. Built on FEA and CFD engines, the solver can identify Joule heating on boards and packages, and help determine copper density, component spacing, and proper placement and size of thermal vias. She compares the results of the solver's dynamic



Don't Overthink Your Prototype Builds



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

RF • MICROWAVE • HIGH-SPEED • QUICK TURN EXPERTS

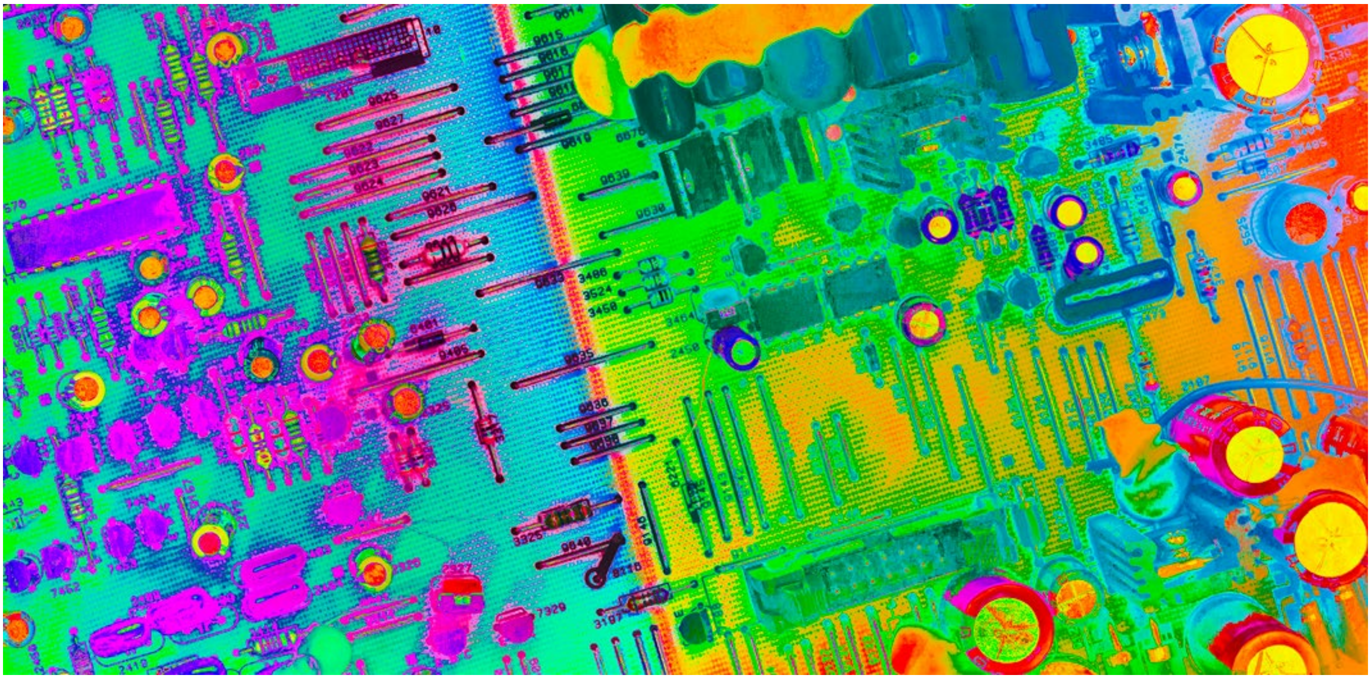
ACE ACCURATE CIRCUIT ENGINEERING

Accurate Circuit Engineering
3019 Kilson Dr
Santa Ana, CA 92707

(714) 546-2162 • sales@ace-pcb.com

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

www.ace-pcb.com



IR drop analysis with DC analysis, which only works with uniform temperature. Melika also points out the differences between typical thermal simulation and the solver's electrothermal co-simulation.

Karthik then takes over, demonstrating electrothermal co-simulation by importing an entire CAD file into the Celsius environment. He utilizes PowerTree technology to organize heat sources and heat sinks—definitions, model names, target impedance constraints, etc. He places probes at different points on the PCB and tracks the transient results of components across various points of time. He shows how to trace power loss across each layer across time, and changes in power distribution as well. The 3D modeling capability depicts any potential hot spots on the board.

To illustrate how this PCB would operate in the real world, Karthik switches to the CFD analysis environment and places the design inside an enclosure. He adds a heat sink to one of the thermally problematic packages, and a cooling fan on the side of the enclosure, then traces the cooling flow across the component and board. Karthik closed with examples of the more comprehensive results attained through transient electrothermal co-simulation vs. traditional steady state analysis.

The Celsius Thermal Solver continues the "shift left" of signal integrity horsepower into the front end of the design cycle, and it offers functionality that will appeal to electrical and mechanical engineers. Maybe EEs and MEs will eventually speak the same language after all.

Melika and Karthik pack quite a bit into this half-hour webinar. If you're dealing with thermal challenges in your high-speed PCB designs, you'll want to check out the CadenceTECHTALK, "How Static and Dynamic IR Drop Analysis Can Help PCB Designs and Challenges."

Watch now! **DESIGN007**

Related content:

- Shift Left: Moving Multiphysics into the Mainstream
- *The System Designer's Guide to... System Analysis: Electromagnetic Interference and Thermal Analysis of Electronic Systems* by Brad Griffin
- *The Cadence System Design Solutions Guide*
- "Cadence Provides 'Clarity' in Design Tool," Interview with Brad Griffin, *Design007 Magazine*, (July 2022)

IPC BUILD ELECTRONICS BETTER

IPC — PROVIDING THE STANDARDS YOU **NEED FOR OPERATIONAL EXCELLENCE**

Stay ahead of the curve with resources to enhance your work, help train your staff and support your company's strategic planning efforts.

- Industry standards on design, board fabrication, and assembly
- IPC certification programs
- Information on Solutions
  - IPC-CFX
  - Factory of the Future

Learn more: Meet Your Standards | IPC International, Inc.

Flex007 Highlights

EIPC Summer Conference 2022: Day 1 Review ▶

At last, a live EIPC conference and this time in the Swedish city of Örebro, "where history and contemporary culture converge," a pleasant and convenient location for an event that included a privileged visit to the Ericsson facility in Kumla. Around 100 delegates made the journey and the Örebro Scandic Grand Hotel was an excellent conference venue for the June 14-15 conference.

NCAB Group Releases Q2 2022 Interim Results ▶

Net sales increased by 47% to SEK 1,122.0 million (762.2). In USD, net sales increased 25%. For comparable units, net sales increased by 35%, and in USD 15%.

Flexible Patch Detects Real-time Changes in Water Temperature ▶

Researchers at Tokyo Tech invent a flexible patch containing carbon nanotubes and stretchable conductors that can fit inside a pipe to detect real-time changes in water temperature or the presence of contaminants, which may improve sanitation during industrial processes.

Compeq 1H22 Revenue Up 31.5% YoY ▶

Taiwan-based Compeq Manufacturing Co. Ltd, a manufacturer of HDI, rigid-flex PCBs, and flex PCBs, has reported unaudited net sales of NT$5.97 billion ($199.4 million at $1:NT$29.93) for June 2022, a jump of nearly 58% year-on-year, and an increase of 16% from the previous month.

FLEX Conference 2022: If You Build It, We Will Buy It ▶

Key presentations from companies like Boeing highlight FLEX Conference and Exhibition 2022, held in concert with the annual SEMICON West show this week in San Francisco's Moscone Center.

Nan Ya PCB 1H22 Revenue up 27% ▶

Taiwan-based Nan Ya Printed Circuit Board Corp. (Nan Ya PCB), a manufacturer of single-sided PCBs, HDI PCBs, and rigid-flex PCBs for motherboards, desktop and notebook PCs, home appliances, smartphones, and gaming consoles, has reported unaudited sales of NT$5.5 billion ($184.7 million at $1=NT$29.90.) in June 2022, up by 5.6% from the previous month and by 30% year-on-year.

Eltek Resumes Deliveries to Customers ▶

Eltek Ltd., a global manufacturer and supplier of technologically advanced solutions in the field of printed circuit boards, reported that it has resumed making deliveries to its customers after a fire in one of the production rooms in the company's plant in Petach-Tikva.

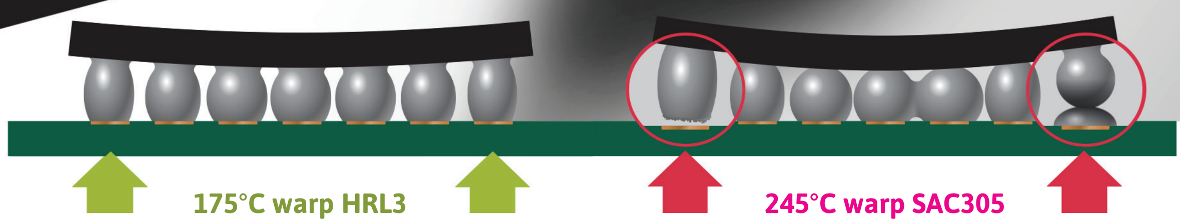
The Importance of Rigid-Flex PCB Design Guidelines ▶

I have the tendency to try to replicate the delicacies I've ordered at restaurants in my own kitchen. While I'm still struggling to figure out the trick to bringing together the different textures of a Korean pancake, I've had more success bringing together the hardboard elements and flexible PCB elements of a rigid-flex PCB.

ALPHA® OM-565 HRL3 Solder Paste

Low Temperature, High Reliability

- **Enhanced** mechanical reliability over traditional SnBi solder pastes
- **Superior** wettability minimizing warpage induced defects
- **Enabling** peak reflow temperature of 175°C
- **Increases** energy efficiency and cost savings

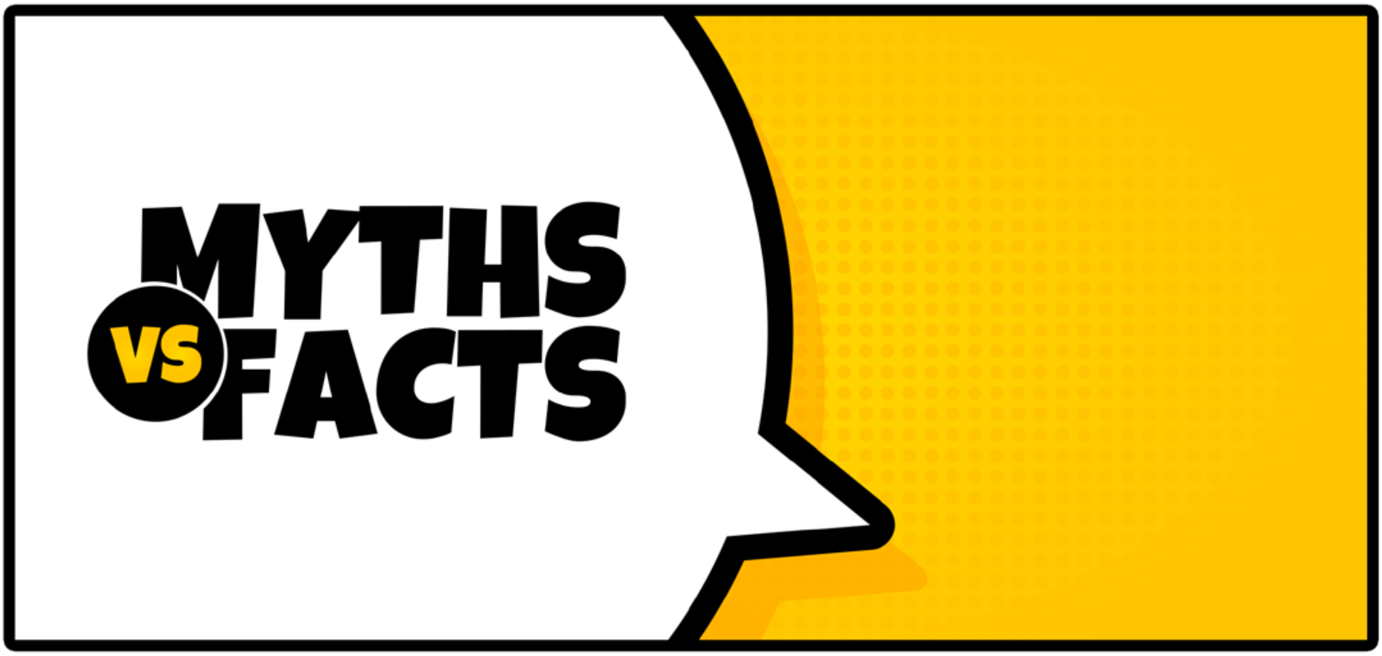


Minimize Warpage Induced Defects to Maximize Assembly Yield.

www.macdermidalpha.com

150 YEARS OF INNOVATION

alpha



The Printed Electronics Roundtable, Part 1

Interview by Andy Shaughnessy
I-CONNECT007

We recently held a roundtable with a team of printed electronic circuit experts from companies that run the gamut: John Lee and Kevin Miller of Insulectro, Mike Wagner of Butler Technologies, Tom Bianchi of Eastprint, and John Voultos of Sheldahl Flexible Technologies.

In this first part of the roundtable, the participants dispel 10 common myths that have been floating around regarding printed electronic circuits (PEC). They also discuss the progress that's been made in PEC development in just the past decade, and what the future may hold for this technology.

Andy Shaughnessy: Welcome to the printed electronics roundtable. How about a quick introduction? John, why don't you go first?

John Lee: I oversee Insulectro marketing and brand strategy for both the PCB side of our business and the printed electronics side.

Mike Wagner: I serve in two roles: chief operations officer and chief innovation officer for Butler Technologies. We're located in western Pennsylvania, near Pittsburgh. We design and manufacture graphic overlays, membrane switches, and other printed electronics, and we have a big focus on flexible wearable technology.

Tom Bianchi: I'm a vice president and managing partner of Eastprint. We have factories here in North Andover, Massachusetts and Juarez, Mexico, manufacturing human-machine interfaces, keypad assemblies, and various types of printed electronics, whether it be a wearable patch or a screen-printed electrode.

John Voultos: I'm glad to be here, and I think most of the characters on this call know me. I'm the vice president of business development for Sheldahl Flexible Technologies. I've been with the company over five years. Prior to that, I had 29 years with DuPont.

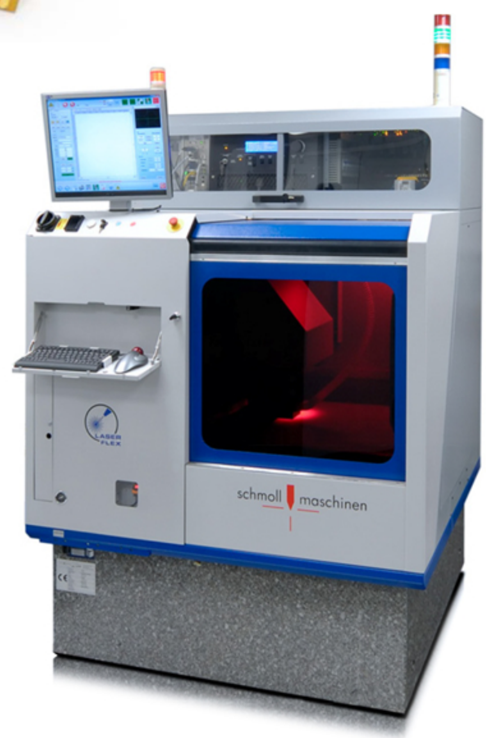
Kevin Miller: I am VP of national sales at Insulectro and have been with them for 28 years. I

Focused on Flex

schmoll optronics

LaserμFlex series

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



Typical applications:

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

1991
2021
BÜRKLE
NORTH AMERICA, INC.

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

burkleamerica.com

started off in the PCB part of the business, supporting that industry for probably 20–25 years. About five years ago, I moved into printed electronics to start up that division and I am responsible for sales, but also play a role in operations and other things within printed electronics.

Kevin Miller

Shaughnessy: So, Chris Hunrath with Insulectro sent me a list of 10 popular myths about printed electronics, so let's address those.

MYTH #1

The first myth: Printed electronics is only for single-sided, low-end applications. Is there any truth to that?

Bianchi: That's a complete myth. We produce ECG electrodes with printed through-holes on two-sided polyester circuits. I would consider that a critical component.

Wagner: Yes, I can add to that. Butler Technologies will print two-sided circuits for membrane switches, especially in some high-density applications where there's just not enough room with one-sided. So, it's pretty straightforward. We've been doing this for more than 10 years.

MYTH #2

Shaughnessy: The next myth is that printed electronics are only good for flat applications and flat parts.

Wagner: I do remember when inks couldn't stretch, but now that they stretch, you can do in-mold structural electronics, in-mold electronics, where you're actually printing circuits onto, let's say, a polycarbonate sheet and then forming that into a three-dimensional shape and shooting it with plastic resin and graphics on the film as well. So, no, it's not just for flat circuits. Many of the capacitive touch interfaces on appliances and some high-end automobiles are all in-mold structural electronics that are three dimensional in nature.

Bianchi: It's definitely an addition to the IME type of structures. There's a printing of conductive ink on stretchable TPU (thermoplastic polyurethane) material, which can be formed around curves or the body for different applications as well.

Miller: Yes. We sell to a wide variety of manufacturers, and there is nothing that's flat about printed electronics, except for the screening process. Companies use a rotary process as well in high volume applications. The end product usually requires flexibility and it's really more of a flexible circuit. In the PCB market, we have rigid boards, flex boards, and rigid-flex boards. But I think in PE, it's strictly flexible boards because it's usually on a thinner dielectric—five mils and below.

Bianchi: There was a time once when you wanted printed electronics in membrane switch applications to be flat. But with the developments by the ink manufacturers to make this stretchable and formable, that's changed.

Wagner: Wearables are our growth area. Right now, it's biometric sensors as well as printed heaters that are meant to stretch. Wearables are a huge area. You can create garments, for example, that have sensing, heating, and hopefully some other technologies in development. You can have a one-stop shop health-monitoring garment.

Shaughnessy: This has a broad reach because everyone wears clothes. People who don't know much about PCBs have thoughts on wearables.

Wagner: The nice thing about printed electronics is that it's primarily an additive process, as opposed to subtractive process, and we can do it in our shop.

Lee: Do the wearables wear out? Is there obsolescence about them eventually?


John Lee

Wagner: Sure. From what we see in garments, they'll have abrasion issues after a while. Just like with any clothes, you can get fibers that break down from wearing and washing them. But, depending on the market, if it's medical, it doesn't have to last 10 years. It's lasting maybe a diagnostic period, for a few weeks or a month, for example, or maybe 25, 30, or 50 washes. So, it's not necessarily meant to be highly durable at this point. It will come as the inks are developed further along in the process. But right now, there is some finite life to that.

Bianchi: There are more medical applications than consumer applications for wearables right now, in terms of maybe a garment or in a T-shirt. They do exist and they're out there, but from our perspective, it's more in the medical field.

Wagner: From our customer base and perspective, we're seeing from the biometric side of it, a split between sports monitoring and medical garments. But from the heater standpoint, it's almost all OEM-type clothing and some things that already have heating and they're looking for a different technology to put into those.

MYTH #3

Shaughnessy: Polymer thick film printed electronics are not good for high-temperature applications. What are your thoughts on that?

Miller: There are polymer thick films we're using in heater applications and other types of things. It is high-temperature circuit. You have to look at the substrate that you're putting it on, which is key here. When you look at PET, it does have a much lower operating temperature. But there are applications where you can print polymer thick films onto polyimide films, onto other types of substrates that are much higher temperature.

Voultos: Yes, polymer thick film (PTF) inks have their limitations, however, companies like DuPont and Nagase, as well as others, continue to develop inks by developing additives to the metals and polymers, which leads to new possibilities.

MYTH #4

Shaughnessy: Printed electronics is only suitable for very large volumes, for instance, screen printing, flexographic printing. I guess that's basically saying it's only suitable for large volumes.

Bianchi: I would suggest that's definitely a myth because the benefit of the screen-printing process is you can do low and high volumes; and low, medium, and high volumes. You don't need to set up a flex graphic press or even a rotary press and print for three days in a row. You can set up and print for two hours, and then do a changeover and print another job that's a similar volume.

Wagner: When you're screen printing, it's so easy to do prototyping on a screen press, as opposed to something rolled, right? There's a much shorter span of time to get that product out. Just think about applications, like Tom said. If you consider membrane switches to be printed electronics or capacitive touch, it's a newer version of a user interface. Those can be low volume because of the process of screen printing. And then if you do have it, you can do faster screen-printing equipment for high volumes. It's there either way you want it.

Mike Wagner

MYTH #5

Shaughnessy: The main use for printed electronics is in membrane touch switches.

Bianchi: Right. But that's not the only use; in my opinion, it's the original use and it probably is still used primarily for membrane switches. Other applications are growing faster than applications for membrane switches but that's still our core business—membrane switches and user interfaces.

Wagner: Companies like Butler Technologies, and I'm sure Eastprint, leverage their knowledge of how to do circuits or membrane switches into these other printed electronic applications. It was not that hard of a transition for some parts of those printed electronics projects.

Bianchi: Right. There are similar manufacturing processes: screen printing, die cutting, lamination, and surface mount. Then you go from a membrane switch to a wearable patch or printed heater in a garment.

Wagner: From our standpoint, we don't even consider membrane switches as printed electronics, just because they've been around so long. It's not cutting edge compared to printed electronics as we think of it.

MYTH #6

Shaughnessy: Printed electronics cannot do fine line circuits. I've heard that before.

Bianchi: I think they can do fine lines, but not as fine as some other circuit technology.

Wagner: I was at the ID TechEx conference recently and they were talking about screen printing fine lines. Most people in applications don't necessarily need to print that fine, but people are printing sub-25-micron lines to create transparent heaters or transparent capacitive touch sensors instead of printing things like PDOT or other clear conductors. We can easily print with the right substrate ink. We just did a test for one of the presentations. No problem doing 75-micron lines and spaces and 50 microns with the right combination. It's not that difficult, but it's not that common in certain applications.

Miller: There are specialty inks that the suppliers provide if you're doing fine lines. We have many different types of conductive inks, and I know some of our competition does as well. There are also different types of meshes that take that into account if you're doing fine lines.

Bianchi: The one variable is that the finer you go, the less ink you're laying down, and as the resistance changes, how long is the trace, and how big is the part? The current and voltage are all impacted.

Wagner: Exactly. That was a big thing. A discussion at the workshop was how much current these tiny traces can carry in certain applications.

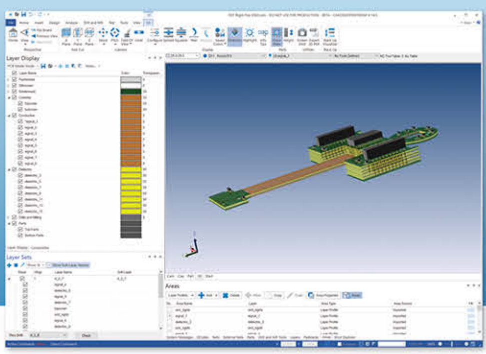
Bianchi: There are a lot of customer applications we get saying, "We can print at that fine line, but it won't meet the electrical requirements." So, they have to change the design a little bit to meet the electrical requirements.

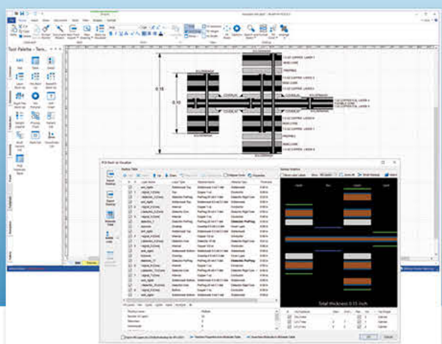
Support For Flex, Rigid Flex and Embedded Component Designs Now Available.

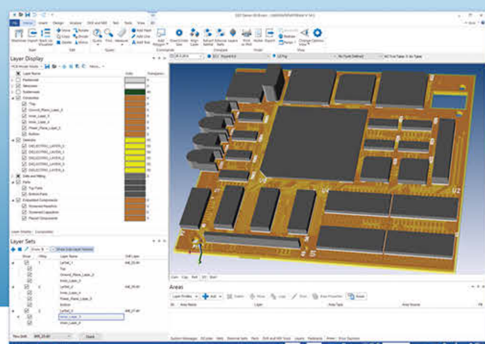
BluePrint-PCB®

CAM350®

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







DownStream Technologies

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

©2021 DownStream Technologies, LLC. All rights reserved.

MYTH #7

Shaughnessy: In printed electronics, wearable applications are limited by the number of uses, washes, and types of fabrics. What say you all?

Miller: That was a concern when wearables first came out, but I think there have been different revs and modifications to both the ink sets and the fabric sets to provide more washes to them. In the beginning, that was one of the things suppliers and manufacturers struggled with, but there have been advances over the last couple years. At this point, to me, it's not true.

Wagner: It just depends on what you're comparing it to. I mean, something's always better than the next thing, but it's improving every time, like Kevin said. It just depends on how quickly the material sets develop. More and more people are looking at the stretchable inks for wearables and the TPU are stretchable materials, as well as the hot-melt adhesive on the backside of those that actually attach to the fabric.

I've seen a lot of improvements. I've printed on material that wasn't meant to do what it was doing because inks wouldn't stretch. But we were practicing that more than 10–12 years ago. We could print stretchable, wearable circuits, but the ink wasn't ready yet; it would just fracture. Then along comes, for example, DuPont and its Intexar™ series, and that was able to meet some of the demands of those applications pretty easily. It keeps them improving from there. We had a visit with them last week and they're still working on modifications of their materials to improve it. Again, like anybody else, they're trying to make it better as they learn more in research.


John Voultos

Bianchi: It's definitely not limiting the people who are designing it and putting it into production. It's good enough right now. It's just going to get better.

MYTH #8

Shaughnessy: Printed electronics are mostly limited to polyester film substrates, PET.

Bianchi: I don't know if it's limited to that. That's probably the most common substrate that thick film polymer is printed on, but one of the reasons people are going to printed electronics is cost effectiveness. We've printed on polyimide films, but it somewhat defeats the purpose of getting a lower cost.

Voultos: Speaking of films, there are also TPUs. There is a wide variety of polycarbonates. As the industry is evolving, we're seeing its evolution. As it branches out and utilizes other substrates than polyester, I think the industry will grow.

Wagner: Yes. There are even things like synthetic papers that are used in printed electronics, RFID tags, things like that. You can get really low-cost substrates to print these things on.

Miller: We sell to customers who are printing on glass, TPUs, and polycarbonates. We also have customers that are printing on polyimide films and other types of substrates and utilizing conductive ink. We do cross over to the printed circuit board side of the business, and we have different types of polyimide-based materials that can be utilized on top of circuitry to reduce processing costs.

Wagner: Sure, a lot of common materials, polycarbonate for in-mold electronics.

MYTH #9

Shaughnessy: Printed heaters need complex control systems to work correctly. What do you think?

Wagner: There are PTC heaters that are basically self-regulating, and many are used in the automotive industry—just simple co trols. You can get into more complex controls with fixed-resistance type heaters. But it's not that complex for things like wearables where you're not cooking anybody at those temperatures.

Bianchi: The PTC printed heater is probably the simplest. You just need to put power to it, and it regulates its temperature on its own. You don't need much for a microcontroller at all if anything.

Wagner: You pick the right ink and the right design, and it's pretty self-sufficient as far as regulating itself.

MYTH #10

Shaughnessy: Finally, printed electronics cannot be used with soldered components. I've heard that it's doable but difficult.

Bianchi: If you're using a low-temp solder, you can use a solder. It's more common to use a conductive epoxy to go onto a silver ink, primarily because it's usually printed on a polyester circuit or a polyester sheet. A solder would be too hot for the polyester, so we use conductive epoxy instead.

Wagner: Some companies have perfected their own IP to do some soldering onto PET when they're putting down specific inks and things like that on the soldering pads and then putting them through reflow ovens, but it's not common per se through the industry.

Tom Bianchi

Bianchi: But more importantly, you can surface mount components onto printed electronics.

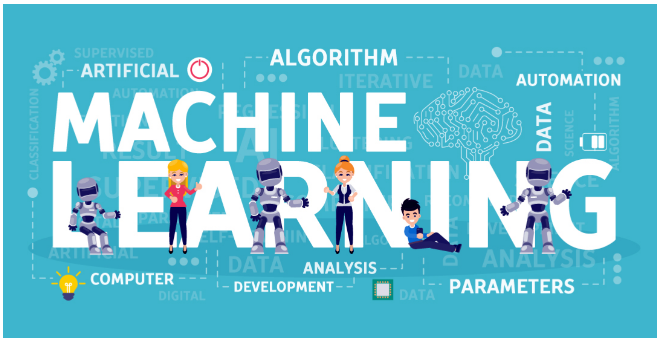
Wagner: Yes. You're just dispensing conductive epoxy and then using the same pick and place machine to put components on. You can even screen print an epoxy paste as well.

Shaughnessy: Thank you everyone for addressing these myths today. **DESIGN007**

Editor's note: Watch for the continuation of this roundtable discussion in next month's issue.

TOP TEN

EDITOR'S PICKS



All Systems Go: Accelerate Your PCB Designs with Machine Learning

Even though we hear the terms artificial intelligence (AI) and machine learning (ML) almost daily, there's still a lot of confusion about the actual meaning of these designations.

Lightning Speed Laminates: An Overview of Copper Foils

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

Rambus Driving a CXL Memory Option

Arjun Bangre

In this interview with Arjun Bangre, director of product for high-speed interface IPs for PCI Express and CXL at Rambus, the discussion revolves around new developments in CXL, PCI Express, and interoperable IP solutions that Rambus has developed.

The Shaughnessy Report: With Field Solvers, GIGO Hurts

The "left shift" concept has been under way for at least five years, as EDA tool providers offer more powerful functionality earlier in the stages of PCB design and layout. This month, we focus on one tool that's been shifting leftward for some time now: the field solver.

What Happens When You Assume?

What is design with manufacturing, and what does true DWM look like in operation? In this interview, I-Connect007 columnist Dana Korf explains what it will take to achieve total communication among all the stakeholders in the PCB development cycle.

Keysight Launches New Design and Simulation Software for Radio Frequency and Microwave Designers

Keysight Technologies, Inc. has introduced the new PathWave Advanced Design System (ADS) 2023, an integrated design and simulation software that rapidly addresses increasing design complexity and higher frequencies in the radio frequency (RF) and microwave industry.

Fresh PCB Concepts: Part 5—How to Handle Possible Moisture During Shipping, Handling, and Storage

This is the fifth part in a series titled "What Damage Does the Assembly Process do to a PCB?" In part four of this series, I discussed the effect of moisture on the printed circuit board at soldering temperatures. I explained the material properties of FR-4 laminate and how they are hygroscopic.

Electronic System Design Industry Logs 12.1% YoY Revenue Growth in Q1 2022

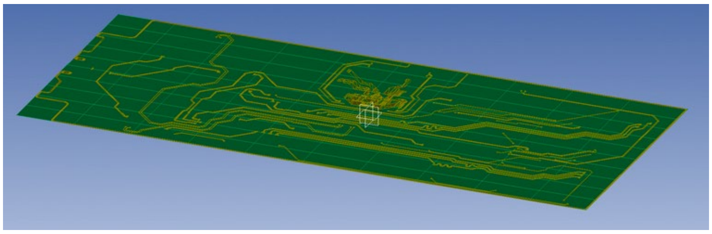
Electronic System Design (ESD) industry revenue increased 12.1% from $3,157.7 million in Q1 2021 to $3,540.5 million in Q1 2022, the ESD Alliance, a SEMI Technology Community, announced in its latest Electronic Design Market Data (EDMD) report.

Cadence to Acquire Future Facilities, A Pioneer in Data Center Digital Twins

Cadence Design Systems, Inc. announced that it has entered into an agreement to acquire Future Facilities, a provider of electronics cooling analysis and energy performance optimization solutions for data center design and operations.

Avishtech Introduces Gauss Stack Pro Software

Avishtech, a provider of PCB simulation solutions, announced the availability of its Gauss Stack Pro software. Gauss Stack Pro takes PCB manufacturability and reliability simulations further by accounting for the full board layout.



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

Is your team growing?

Find industry-experienced candidates at I-Connect007.

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

In addition, your ad will:

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

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

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

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

+1 916.365.1727

I-Connect007
GOOD FOR THE INDUSTRY

smt007 pcb007 pcb design007

Career Opportunities

Technical Support Applications Engineer

Full-Time — Duluth, GA

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

Responsibilities

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

Skills and Qualifications

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

Benefits

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

apply now

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

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

Responsibilities:

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

Qualifications:

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

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

apply now

Career Opportunities

ventec
INTERNATIONAL GROUP
騰輝電子

European Product Manager
Taiyo Inks, Germany

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

Duties include:

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

Skills and abilities required:

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

What's on offer:

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

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

apply now

CHEMCUT

Director of Operations
State College, PA

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

Objectives of the Role:

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

Qualifications:

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

Preferred Qualifications:

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

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

apply now

Career Opportunities

Prototron Circuits

Sales Representatives

Prototron Circuits, a market-leading, quick-turn PCB manufacturer located in Tucson, AZ, is looking for sales representatives for the New England and Northern California territories. With 35+ years of experience, our PCB manufacturing capabilities reach far beyond that of your typical fabricator.

Reasons you should work with Prototron:

- Solid reputation for on-time delivery (98+% on-time)
- Capacity for growth
- Excellent quality
- Production quality quick-turn services in as little as 24 hours
- 5-day standard lead time
- RF/microwave and special materials
- AS9100D
- MIL-PRF- 31032
- ITAR
- Global sourcing option (Taiwan)
- Engineering consultation, impedance modeling
- Completely customer focused team

Interested? Please contact Russ Adams at (206) 351-0281 or russa@prototron.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 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

KYZEN
Where Science and Care Converge.

Regional Manager
Midwest Region

General Summary: Manages sales of the company's products and services, Electronics and Industrial, within the States of IL, IN & MI. Reports directly to Americas Manager. Collaborates with the Americas Manager to ensure consistent, profitable growth in sales revenues through positive planning, deployment 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

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, and major credit card for travel.

Must be able to travel extensively.

apply now

Career Opportunities

Manncorp™

SMT Field Technician
Hatboro, PA

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

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

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

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

apply now

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

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

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

Laminator Technician

Nature of Duties/Responsibilities

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

Education/Experience

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

Wet Process/Plating Technician

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

Purpose

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

Nature of Duties/Responsibilities

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

Education and Experience

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

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

ARLON ELECTRONIC MATERIALS

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

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

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

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

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

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

apply now

Career Opportunities

The Test Connection, Inc.

Rewarding Careers

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

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

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

Associate Electronics Technician/ Engineer (ATE-MD)

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

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

apply now

Test Engineer (TE-MD)

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

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

apply now

Sr. Test Engineer (STE-MD)

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

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

We proudly serve customers nationwide and around the world.

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

apply now

Career Opportunities

BLACKFOX
Premier Training & Certification

IPC Instructor

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

Independent contractor, possible full-time employment

Job Description

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

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

Qualifications

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

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

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

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

apply now

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

CAD/CAM Engineer

Summary of Functions

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

Essential Duties and Responsibilities

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

Organizational Relationship

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

Qualifications

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

Physical Demands

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

apply now

Career Opportunities

U.S. CIRCUIT

Plating Supervisor

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

Competitive benefits package.
Pay will be commensurate
with experience.

Mail to:
mfariba@uscircuit.com

apply now

APCT
Passion | Commitment | Trust

APCT, Printed Circuit Board Solutions: Opportunities Await

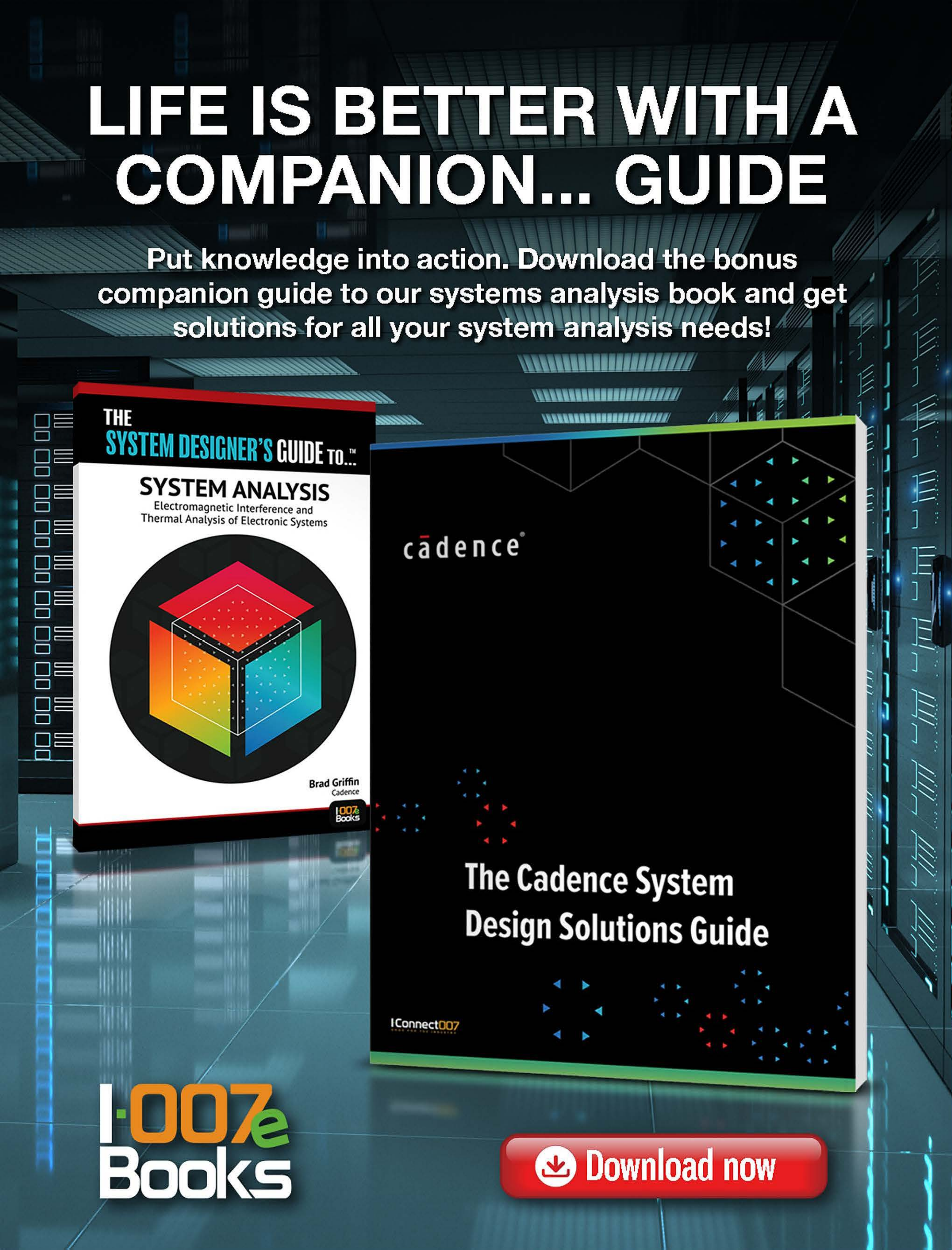
APCT, a leading manufacturer of printed circuit boards, has experienced rapid growth over the past year and has multiple opportunities for highly skilled individuals looking to join a progressive and growing company. APCT is always eager to speak with professionals who understand the value of hard work,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

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

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

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

apply now


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

Educational Resources

I-007e

The Electronics Industry's Guide to... The Evolving PCB NPI Process

by Mark Laing and Jeremy Schitter, Siemens Digital Industries Software

The authors of this book take a look at how market changes in the past 15 years, coupled with the current slowdown of production and delivery of materials and components, has affected the process for new product introduction (NPI) in the global marketplace. As a result, companies may need to adapt and take a new direction to navigate and thrive in an uncertain and rapidly evolving future. Learn how to streamline the NPI process and better manage the supply chain.

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

NEW! 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

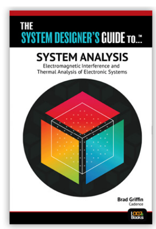
This book provides the reader with a clearer picture of what to know when selecting which material is most desirable for their upcoming products and a solid base for making material selection decisions. Get your copy now!

Stackups: The Design within the Design

by Bill Hargin, Z-zero

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

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



by Brad Griffin, Cadence

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

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

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

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

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

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

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

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

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

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

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

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

TECHNOLOGY SPECIALIST: **BRYSON MATTIES**

COVER DESIGN: **SHELLY STEIN**

DESIGN007 MAGAZINE

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

© 2022 BR Publishing, Inc. does not assume and hereby disclaims any liability to any person for loss or damage caused by errors or omissions in the material contained within this publication, regardless of whether such errors or omissions are caused accidentally, from negligence or any other cause.

August 2022, Volume 11, Number 8
DESIGN007 MAGAZINE is published monthly,
by BR Publishing, Inc.

ADVERTISER INDEX


Problems solved!
PCB007 MAGAZINE
MAY 2022
GOLDEN ROI OPPORTUNITIES
PCB007 MAGAZINE
JUNE 2022
PLATING
Cybersecurity & Counterfeiting
PCB007 MAGAZINE
DESIGN007 MAGAZINE
JUNE 2022
DESIGN007 MAGAZINE
JULY 2022
GIGO HURTS
DESIGN007 MAGAZINE
AUGUST 2022
LESS IS MORE
SMT007 MAGAZINE
JULY 2022
TRIPLE WORD SCORE
TRIPLE LETTER THREAT
SMT007 MAGAZINE
AUGUST 2022
MANAGING THE SUPPLY CHAIN PAIN
SUBSCRIBE NOW