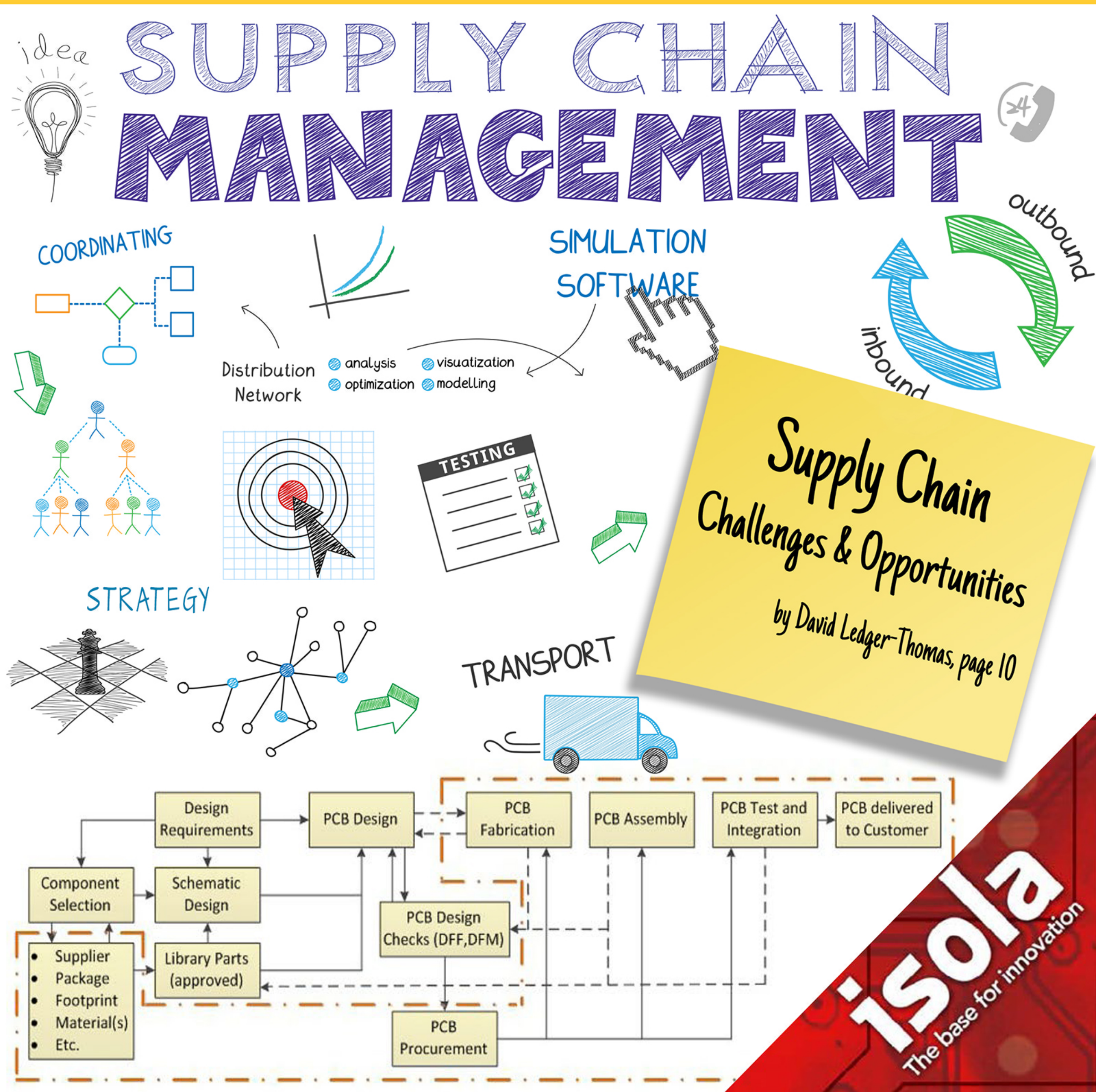
THE PCB DESIGN MAGAZINE

an I-Connect007 publication

July 2015

An Update on the Rogers Material Supply Line
p.18

Gary Ferrari Shares His Thoughts on PCB Design and More
p.28



TECHNICAL SALES MANAGER

(Midwest Territory)

Taconic, a leader in the manufacture of PTFE coated fiberglass fabric, belts, tapes and circuit board laminates is seeking a **Technical Sales Manager** to grow our Advanced Dielectric Division (ADD) market. Responsibilities will include servicing existing accounts and developing new markets and customers in assigned territory for microwave circuit laminate product line.

Key qualifications include a Bachelor's degree in Engineering or related. Three years of field sales experience desired. Good problem solving skills with the ability to communicate effectively while maintaining professional working relationships with customers and internal staff. Previous microwave design or printed circuit board fabrication experience desired. The ideal candidate will have a materials background, a background in PWB fabrication, and/or a background in RF design. A seasoned salesperson with OEM experience will be preferred. Substantial knowledge of MS Office programs with ability to develop and conduct sales presentations required. Candidate should be self-motivated and have a bias for action. Our Technical Sales Manager will work from a home office and will be expected to travel 80% of the time within the assigned territory.

We offer a competitive benefits package including medical, dental, vision and life insurance, a competitive compensation and retirement package, tuition assistance, vacation and more!

For immediate consideration, email your resume to the HR Director at: hr@4taconic.com.

To mail your resume, send to:

TACONIC
Attn: Human Resources
P.O. Box 69
Petersburgh, NY 12138-0069

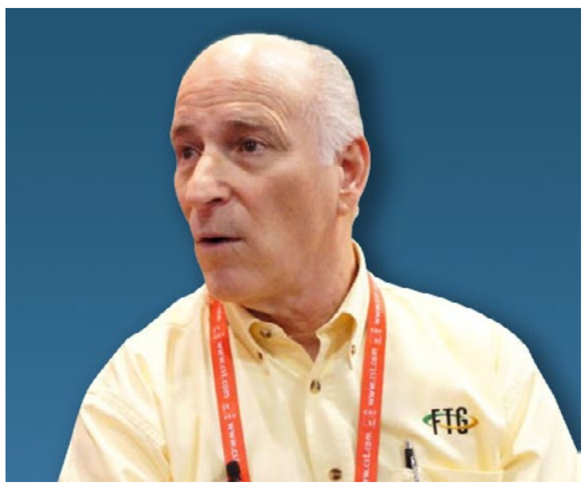
www.4taconic.com

EOE/AAP

This Issue: SUPPLY CHAIN MANAGEMENT

FEATURED CONTENT

Supply chain management is one aspect of the design world that often takes a back seat to the latest PCB design tricks and techniques. But if you don't manage your supply chain, it will wind up managing you. This month, our cover story by David Ledger-Thomas and our interviews with Gary Ferrari of FTG and John Pavlak of Rogers Corporation discuss supply chain management from the PCB designer's viewpoint.



isola RF/Microwave Materials & Resources

- **IS680** materials offers a complete laminate materials solution for single- and double-sided printed circuit designs and is a cost-effective alternative to PTFE and other commercial microwave materials.
- **I-Tera® MT RF** materials are available in 0.010", 0.020" and 0.030" in 3.38, 3.45 and 3.56 Dk.
- **I-Tera® MT** materials are suitable for both high-speed digital and RF/ microwave designs. A full compliment of cores and prepregs allowing flexibility in design is available in core thicknesses from 0.002" to 0.018".
- **TerraGreen®** halogen-free, very low-loss, thermoset materials are available in a variety of laminate and prepreg offerings. This material is inexpensive to process – improving your company's bottom line, as well as the environment.
- The revolutionary **Astra® MT** ultra low-loss thermoset laminates are a replacement for PTFE. These materials have been used for hybrid applications with Isola's 185HR and 370HR products.

	RF/MICROWAVE MATERIALS				
	IS680	I-Tera® MT RF	I-Tera® MT	TerraGreen®	Astra® MT
Tg	200°C	200°C	200°C	200°C	200°C
Td	360°C	360°C	360°C	390°C	360°C
Dk @ 10 GHz	2.80 - 3.45	3.38, 3.45 & 3.56	3.45*	3.45*	3.00
Df @ 10 GHz	0.0028 - 0.0036	0.0028, 0.0031 & 0.0034	0.0031*	0.0030*	0.0017
CTE Z-axis (50 to 260°C)	2.90%	2.80%	2.80%	2.90%	2.90%
T-260 & T-288	>60	>60	>60	>60	>60
Halogen free	No	No	No	Yes	No
VLP-2 (2 micron Rz copper)	Available	Available	Available	Standard	Standard
Stable Dk & Df over the temperature range	-55°C to +125°C	-55°C to +125°C	-55°C to +125°C	-55°C to +125°C	-40°C to +140°C
Optimized global constructions for Pb-free assembly	Yes	Yes	Yes	Yes	Yes
Compatible with other Isola products for hybrid designs	For use in double-sided applications	Yes	Yes	Yes	Yes
Low PIM < -155 dBc	Yes	Yes	Yes	Yes	Yes

* Dk & Df are dependent on resin content NOTE: Dk/Df is at one resin %. Please refer to the Isola website for a complete list of Dk/Df values. The data, while believed to be accurate & based on analytical methods considered to be reliable, is for information purposes only. Any sales of these products will be governed by the terms & conditions of the agreement under which they are sold.

RF Conversion Service

- Isola's **Design Review Service** can facilitate your conversion to Isola's RF/microwave products and get you to market faster with the newest, ultra-low-loss materials.
- As part of this new service, Isola's technical staff will provide turn-key calculations, testing, characterizations and material recommendations to assist PCB fabricators and OEMs in converting to Isola's RF-materials, which will help overcome the current material shortages of other vendors and accelerate time-to-market. The design review service will also address the perceived conversion issues when migrating from a currently used material to an Isola material.

http://www.isola-group.com/conversion-service

FREE! Impedance and Power-Handling Calculator

- Isola's free **Impedance and Power-Handling Calculator** predicts the design attributes for microstrips and striplines based on the design's target impedance and dielectric properties of the company's RF, microwave and millimeter-wave laminate materials.
- This software tool provides a design or an equivalent dielectric constant to facilitate modeling for PCB designers to predict impedance and other design attributes. The software computes changes in the effective dielectric constant due to dispersion at higher frequencies. The software then computes the total insertion loss – a measure of power lost through heat for power handling calculations, including the dielectric loss, conductor loss, and the loss due to the surface roughness. The main factors affecting the typical power-handling capability of a material are its thermal conductivity, the maximum operating temperature, and the total insertion loss.

https://isodesign.isola-group.com/phi-calculator

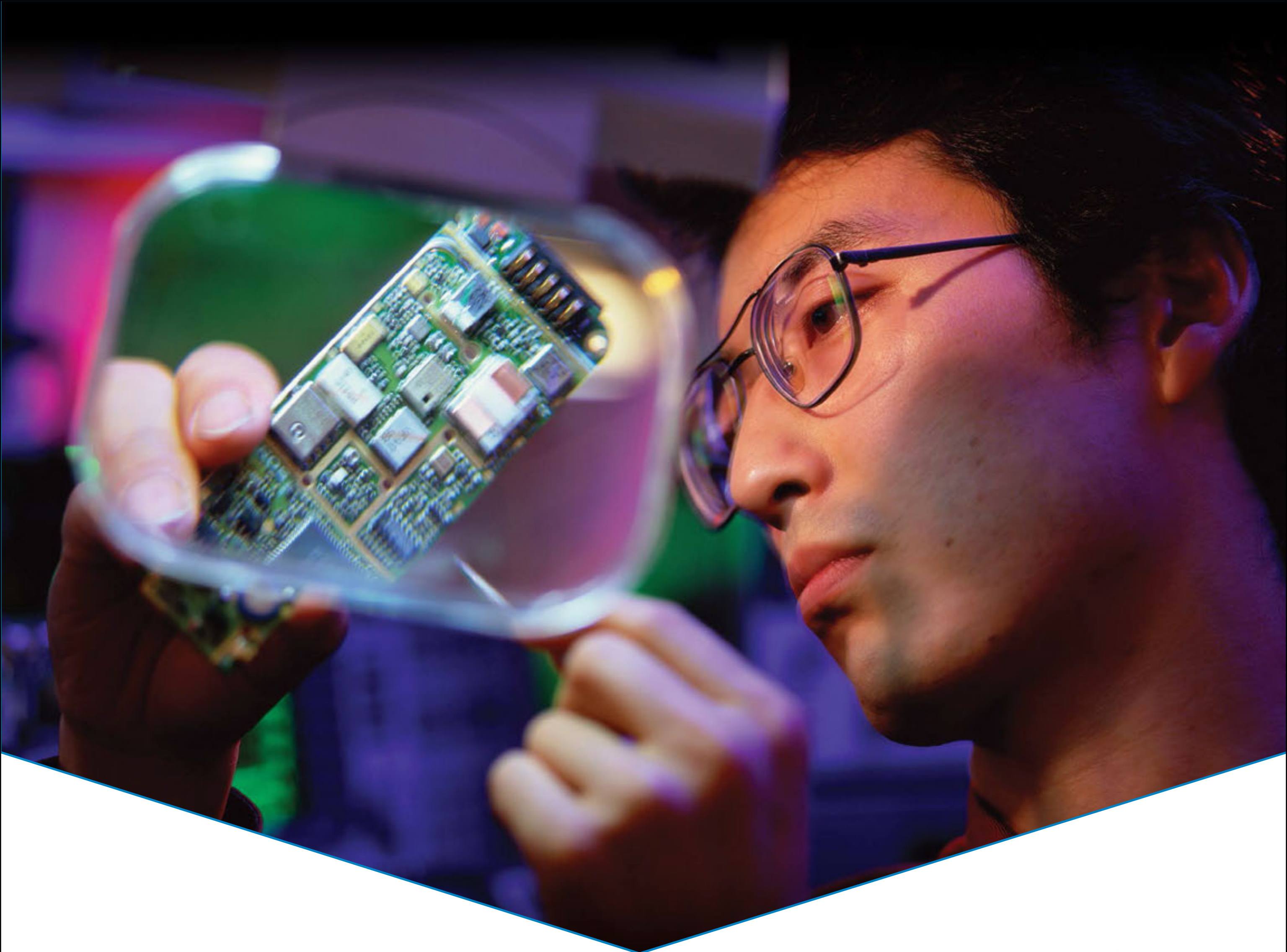
www.isola-group.com/RF

JULY 2015
VOLUME 4
NUMBER 7
thepcbdesignmagazine.com

THE PCB DESIGN MAGAZINE™
an I-Connect007 publication

THE OPTIMUM MAGAZINE DEDICATED TO PCB DESIGN

CONTENTS



The PCB Design Supply Chain

by Andy Shaughnessy
I-CONNECT007

It's not a simple task providing you all with information you can use. Most of you are really good at your job, and you could design just about any type of board that your boss throws at you. And you should be good; most of you are—ahem—seasoned veterans of the PCB design world. At this point in your career, you have this stuff down to a science.

You're probably not going to discover a brand-new way to design boards, but you know that there's always something to learn in this job: new design tips and tricks, changes in PCB technology, market trends, and much more. As the ol' saying goes, "You learn something new every day." We certainly do!

So, we occasionally look at other design topics that are not directly related to PCB design and layout techniques—markets, legislation, materials, education, etc. Is there designer interest in this topic? This is what happened when we surveyed our readers a few months ago.

We asked readers of *The PCB Design Magazine* if their supply chain was a problem for them. Almost two-thirds of respondents said no, but a solid 37% said yes. And surprisingly, for many it was an emphatic "yes." Navigating the supply chain is a huge challenge for some of our leading companies.

What sort of bugaboos were designers referring to? Long lead times, parts obsolescence, multiple component changes and ECOs…you name it. Some designers said the parts selection process on the front end is the worst part of the design cycle, and their companies have had to stock more and more hard-to-find parts. Redesigning boards with obsolete parts ranked high too.

Even worse, some respondents said that the supply chain was the one thing they could not influence, much less control. Comments such as "We're dealing with idiots" were typical. How would you like to work in a situation like that? Maybe you are right now. You have my sympathy.

So, this issue of *The PCB Design Magazine* focuses on supply chain management from the PCB designer's perspective. The cover story by Honeywell's David Ledger-Thomas takes a look at a typical OEM's PCB design supply chain, and discusses how processes can lead to successful outcomes—or not. Next, Barry Matties talks with Rogers Corporation's John Pavlak about customer complaints of long delivery times of Rogers RF laminates, and what Rogers has done to turn this trend around. We also have an interview Barry conducted with Gary Ferrari of Firan Technology Group, who discusses strategies for designing better boards and keeping designers provided with the most important part of the supply chain—information.

Columnist Barry Olney of In-Circuit Design Pty Ltd. continues his excellent series on stackup planning techniques with an in-depth look at a variety of stackup configurations. Contributor Bob Sadowski of UK-based Quadra Solutions explains the 10 considerations to keep in mind when thinking about outsourcing a PCB design. And Karel Tavernier of Ucamco provides an excerpt from his upcoming guide "PCB Fabrication Data: Design-to-Fabrication Data Transfer" by clarifying exactly what happens to your design data when it reaches the fabricator.

Now that summer is here, you're probably going to be headed to the beach. Well, we have some great summer reading for you! Don't forget to check out our other publications, *The PCB Magazine* and *SMT Magazine*. **PCBDESIGN**

Andy Shaughnessy is managing editor of *The PCB Design Magazine.* He has been covering PCB design for 15 years. He can be reached by clicking here.

by David Ledger-Thomas
HONEYWELL AEROSPACE

Let's start by defining exactly what a supply chain is. It's not a nautical term for an anchor chain, or a dynamic part of a chainsaw that never runs out of chain.

No, it's a key term used in the organization of resources which may form a system between entities. Now this sounds a little closer to something that may be utilized in producing an electronic end-item, in our case, a printed circuit board.

Some prepared formal definitions supplied by a couple of Internet sources:

Wikipedia: "A supply chain is a system of organizations, people, activities, information, and resources involved in moving a product or service from supplier to customer."

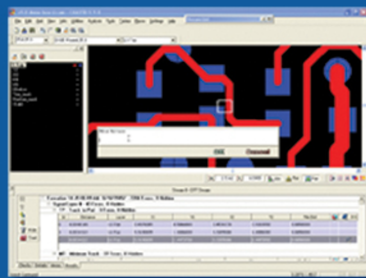
Investopedia: "The network created amongst different companies producing, handling and/or distributing a specific product. Specifically, the supply chain encompasses the steps it takes to get a good or service from the supplier to the customer. Quite often, many people confuse the term logistics with supply chain. In general, logistics refers to the distribution process within the company whereas the supply chain includes multiple companies such as suppliers, manufacturers, and the retailers."

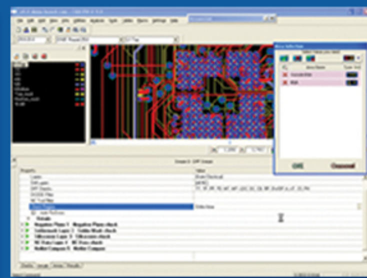
Those formal words are rather nice, but let's reel this in a bit to focus on PCB design. What does supply chain management have to do with PCB design?

Solutions to Ensure PCB Manufacturing Success!

CAM350

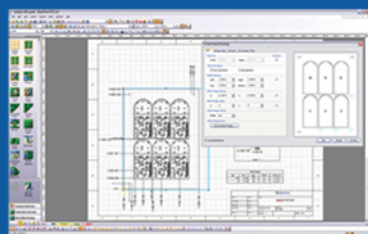
Verify and Optimize PCB Designs for Successful Manufacturing.

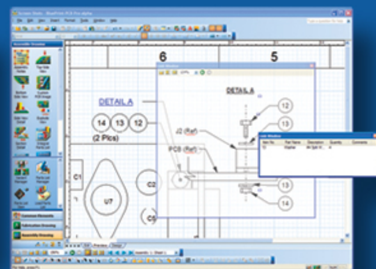




BluePrint·PCB

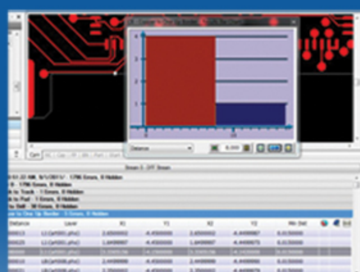
Create comprehensive documentation to drive PCB fabrication, assembly and inspection.





DFMStream™

Verify Design and Manufacturing Rules Any Time During the PCB Design Cycle.



At DownStream Technologies we are committed to delivering comprehensive solutions and workflow process improvements for all your PCB post processing needs.

For more information please visit downstreamtech.com or call 800-535-3226.

DownStream Technologies

Just so we are all in a similar frame of mind, PCB design data contributes to the fabrication of a printed circuit board. The PCB mechanically supports and electrically connects electronic components using conductive (usually copper) traces, pads, vias and other features etched from copper sheets laminated onto a non-conductive substrate or dielectric material.

First, let's take a look at the supply chain from the point of view of PCB design as its own organization or function. Who would be the suppliers and customers, and what is required for success? In this gander, PCB design and layout supplies only board software outputs and drawings.

But is that all there is? No, there is much more interaction required to have a successfully completed, working PCB assembly ready to be installed in the end-product.

A fundamental design flow is shown in Figure 1. The details of design dynamics and iterations of processes are either assumed (not shown), irrelevant or just not itemized by this depiction of PCB design.

Is Figure 1 a true representation of PCB design? In the real world, can a product be output from this simple process? Yes, of course, and in a high percentage of cases quite successfully. But in this brief graphic there is much more going on out of sight. Behind each block there may be a multitude of steps, ones that are transparent to the non-discerning eye. What defines the PCB design dynamic is the way a company (or design resource) has set up its PCB design department (i.e., a small business, a design service bureau, or an OEM's design department).

The view in Figure 2 is an expansion of the simple design process in Figure 1, with a supply chain beginning to show itself, yet still not completely defined in the process. The non-discerning eye is beginning to open a bit, seeing that it takes more than the PCB design office to produce a final product for delivery to the customer (in-house or out-of-house). Figure 2 shows a bit more definition to a few more items in the supply chain.

The basic design-to-product premise is basically the same. Figure 2 shows a basic design-to-product process which could comprise a one-man show or multitudes of offices, functions, and additional steps that are part of a design-to-product process. A duly placed footnote: When more functions and offices are involved, there should be a management style/

Figure 1: A fundamental design flow.

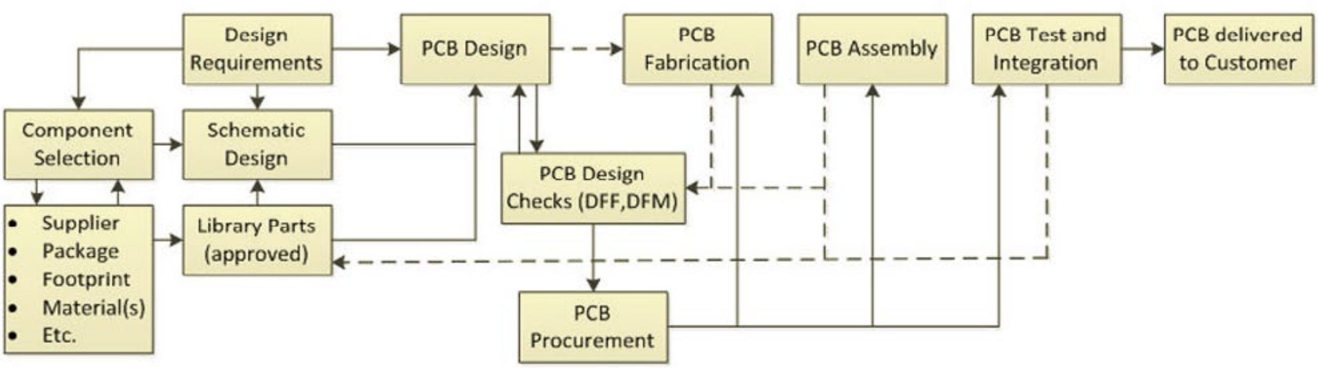
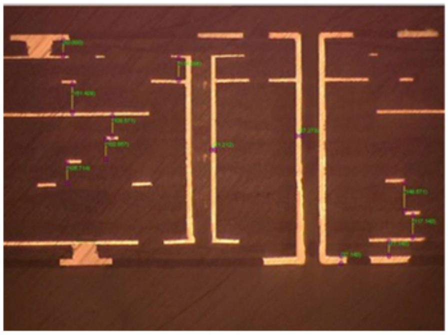


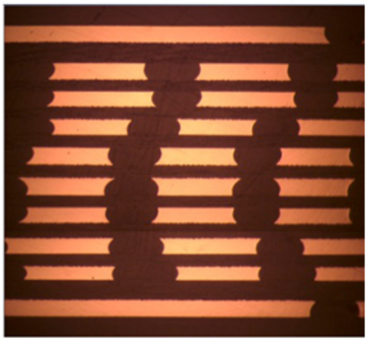
Figure 2: A basic design-to-product supply chain process chart.

EXTRAORDINARY

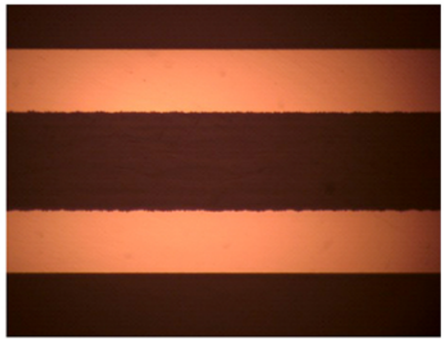
Heavy copper PCB – Microsection



Via in Pad Technology (Micro-via)

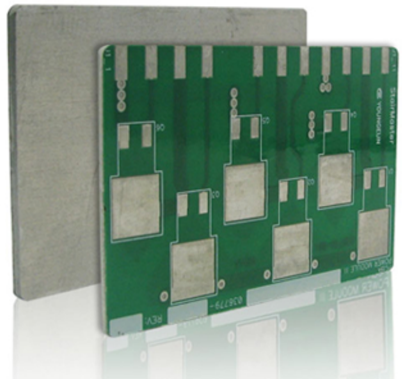


10 Layers: 140µm copper finished in each layer (4oz cu) with Buried via (L2- L9) and PTH (L1- L10)

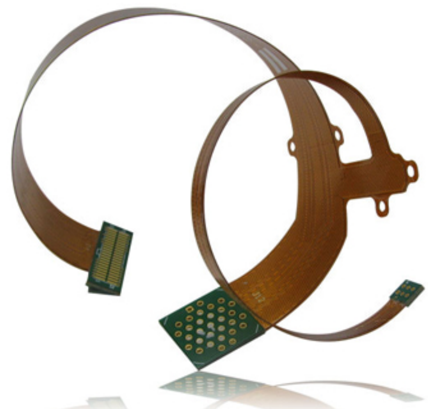


2 Layers: 525µm copper finished in each layer (15oz cu)

With our strengths in R&D, we are capable of turning out PCBs that meet your specific requirements promptly at competitive prices. ANS is devoted to technological innovation and providing our customers with highly satisfactory and speedy service.



Multi-layered aluminum board
lead-free HAL



8 layer rigid-flex
30mm flexible finished thickness

ANS

www.anspcb.hk

Head office:
ANS Industrial Co. Ltd.
Kowloon, Hong Kong
Tel: (852) 2757 1882

Factory:
Remapp PCB (Foshan) Co.,Ltd.
Guangdong Province, China
Tel: (86) 757-8862 3888

Email: sales@anspcb.hk

UL IPC RoHS

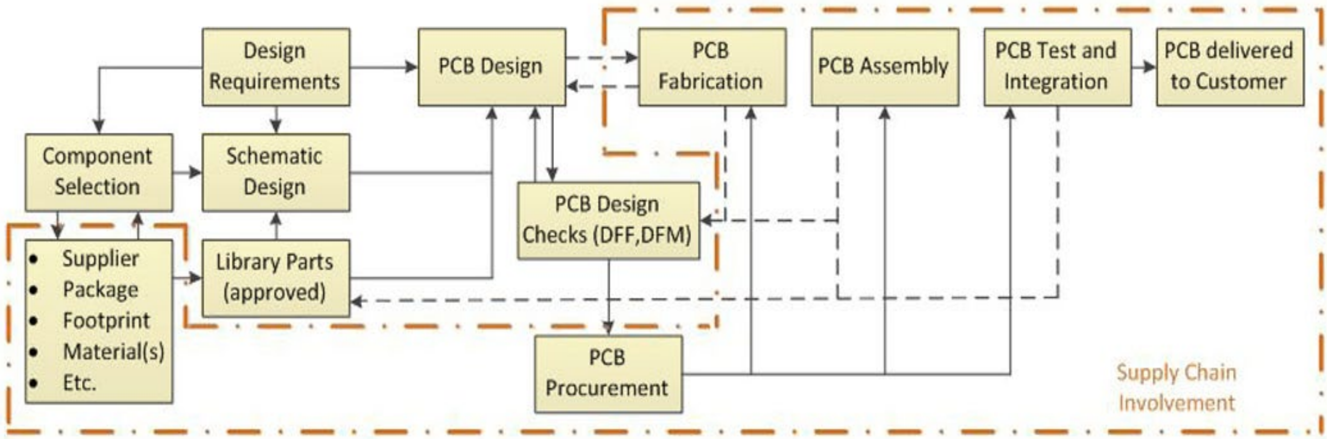


Figure 3: A more complete chart of a design-to-product supply chain.

system in place to set the arena for managing the supply chain.

If we define the PCB design supply chain as resources involved in moving from concept or design requirements to PCB delivered to end-customer, we see a better graphical representation of the supply chain in Figure 3.

Figure 1 may be highly simplified, while Figure 3 might be over-complicated, depending on your view. But they all depict examples of a PCB design supply chain, with a design-to-product process that is loosely the same, differing only in amounts of detail. What happens in between these charts may be contributed to design complexity, company infrastructure, design processes and a myriad other possible aspects that impact how PCB designs flow from design-to-product. A variety of mega-overhead, non-design oriented offices are involved in the process of PCB design.

To be successful in today's time-to-market metrics-driven design environment, many companies have discerned that the supply chain for the design-to-product process must be tightly managed. If implemented well, supply chain management can be a game-changer in the positive aspect.

A well-managed supply chain leads to a variety of positive outcomes, such as feedback from the fabricators and assembly folks. Any errors within a design are transmitted back into the design process to be fixed and to stop the repeat offenders.

Some other issues that a managed supply chain can preclude:

1. Fab notes that are ambiguous, causing fab delay awaiting clarification.
2. New fab processes used in a design without fabricator interface and input, causing delays.
3. Fab data anomalies, errors and bad application of design parameters.
4. Less-than-desired component placement or land pattern.
5. No process called for special assemblage.

Reducing these items would be a definite plus for reduction in time-to-market. There can also be some advantages upfront in the design process:

1. Standardizing parts and suppliers.
2. Board design rules by organization or company interest.
3. Design rules for DFF/DFM and different technologies.
4. Having a central library with active procurement data to minimize obsolete/high price point part selection.

There can be some very positive impacts with a managed supply chain. But a supply chain that is poorly implemented could be a bit of a hindrance to the design-to-product process.

This could be represented by:

1. Implementing out-of-the-box supply chain management software, with a total disregard of current practices and design concepts causing process conflicts up and down the process track.
2. Adopting a metrics-oriented process not based in reality in supporting design-to product, resulting in poor quality product deliveries.
3. Software interface that is user-unfriendly causing delays in product movement and/or misrepresented product descriptions (wrong product delivered).
4. Taking an extended time to complete a task, so it is not even realistic or possible to utilize particular supply chain management functions.

The above, poorly implemented examples are the result of process deprivation in PCB design. The visibility, knowledge and understanding of the impact to a process are at most times not apparent to the PCB designer. On the other hand, upper-level company integration could be just grand. With visibility to available stock, managers may be better able to manage inventory and second- and third-tier organizations to leverage company designs.

But supply chain management can be difficult for longer-term realized functions (i.e., design for future product, small batch prototypes, and engineering development). This is mainly due to processes that are impractical to utilize or too time-consuming to be effective.

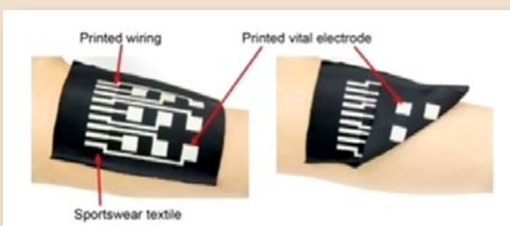
Supply chain management can be a bit of an allusive process if it is not well defined with high involvement at all levels within the organization. This would include equal say of all levels and functions of the process. An additional few bits would be; an environment for learning and an integrated education experience for the participants could become a positive aspect. It would seem that a well-managed supply chain includes two-way communication, with integration through all aspects of development, implementation, maintenance and cross-functional usage.

A clearly defined supply chain may not be very apparent to the designer. But there is, in fact, a supply chain for almost all PCB design processes, and successful PCB designers should embrace and utilize a solid supply chain management regime. The best approach is to place the supply chain management into your toolbox, and know what the process can offer to support the specifics of the PCB design-to-product process. Be aware of some the pitfalls, and communicate and educate for success. **PCBDESIGN**

David Ledger-Thomas is a PCB design engineer with Honeywell Aerospace.

Ink Paves Way for Printable Elastic Conductors

University of Tokyo researchers have developed a new ink that can be printed on textiles in a single step to form highly conductive, stretchable connections. This new functional ink will enable electronic apparel such as sportswear and underwear incorporating sensing devices for measuring a range of biological indicators such as heart rate and muscle contraction.



Now, Professor Takao Someya's research group at the University of Tokyo has developed an elastic conducting ink. The ink exhibited high conductivity even when it was stretched to more than three times its original length, which marks the highest value reported for stretchable conductors that can be extended to more than two and a half times their original length.

An Update on the **Rogers Material Supply Line**

by Barry Matties
I-CONNECT007

As part of a recent I-Connect007 supply chain survey, we found that RF laminate material can be very difficult to obtain. Rogers Corporation was named specifically in our survey as one supplier with a limited amount of material available. In fact, their delivery time was reported as being as high as 55 days for some materials at one point. In an industry where quick turnaround time is critical, this is one supply line that killed any hope of being quick.

Because Rogers was noted by name in our findings, we decided to go to them to learn about the current situation, and the short answer is that there has been improvement on delivery lead time with the promise of continued improvement.

The following interview is with John Pavlak, the director of global operations at Rogers Corporation.

Barry Matties: John, can you please give us an idea of what happened to your supply line and explain what you are doing to improve.

Pavlak: I can give you the story as I know it from my two years so far at Rogers. When I started in mid-2013, we had a plan to increase capacity because our market intelligence said that there was significant demand coming and it was primarily tied to the China 4G roll-out. The only difference between our plan and what actually happened, the 4G roll-out actually came in sooner than our original marketing intelligence. Fortunately, we had already kicked off projects to increase our global capacity. The challenge was that those were long lead-time and very expensive projects. We've invested more than $30 million in the past three years into global capacity projects.

We had ourselves locked in tightly with different OEMs and fabricators, but the wave of demand came sooner than expected, so we spent the latter half of 2013, and almost all of 2014, working very hard to increase capacity in other ways before that additional capacity came online. Every one of our locations in the global regions added capacity through internal improvements on throughput and figured out how to get more lamination press loads per week. All of our teams pushed very hard and each of the regions were able to demonstrate at least another 20% of additional capacity, even without the new equipment.

We did everything we could internally to turn the screws to give ourselves more capacity. In the past, we would let our different regions control how they would prioritize orders to our customers, but because we were capacity-constrained and our volumes were above capacity, and we knew that was going to extend for a period of time, we actually went into a capacity allocation mode. We decided on how to allocate globally, so we improved our response time to a lot of our largest customers by prioritizing and setting aside lamination press loads for them, so they knew they had those to count on. Even

though we extended lead times, our customers knew they had a defined amount of capacity reserved for them. It was not a fun time for our customers or us because the global demand exceeded our global capacity.

In the midst of that period, in Arizona alone, we were able to produce 36% more laminate in 2014 than we did in 2013. In China it was approximately 25%, and about 18% more out of our Belgium factory. When you aggregate it all together, it was well over 25% improved global throughput. Now we have the new capacity online; we added a thermoset treater in China for the RO4000 product line. It's a treater for creating the dielectric material we call pre-preg that goes into the laminate we make. We added 50% additional global pre-preg capacity when we brought it online in China, and it's up and running at full production speeds. On this treater we started the internal qualification in the latter half of 2013, and in 2014 we ramped it up from one shift, to three shifts, and we've moved that pre-preg production into China where the majority of our customers are. Now we have much significantly more pre-preg capacity than we need globally, so we are back ahead of the demand growth curve and are probably good for the next three years when we believe we'll need to introduce some additional pre-preg capacity.

From the lamination side, we had two large expansion projects in lamination going on, as far as new capital equipment. We have two presses we are adding in Suzhou. The first press we were able to expedite and we did get it online and approved in March 2015. We added that capacity and that press is primarily for the RO4000 product. That added about 10–15% more capacity globally; we also took our Belgium operation from six to seven days in lamination, and that added another 10%. Right now we are at 25% more capacity than we had if you turn the clock back six months ago.

The other thing that has happened in the midst of this 18-month journey is the acquisition of Arlon, and what came with that purchase are not just the different sets of products and customers, but also some available capacity. In the Arlon factories we are presently looking at the individual types of machinery, what products are run there, and we are identifying what capacity we can extract out of the factory to possibly run Rogers' legacy products. We are going through that analysis now and we are positioned very well for the next wave of increased demand.

> "We've been able to bring our lead times down significantly, although I think the roughest period was when we had to take our lead times up to 55 business days on RO4000 materials. That probably caused the most angst in the customer base. I'm glad to report we have done two lead time reductions since late March."

We've been able to bring our lead times down significantly, although I think the roughest period was when we had to take our lead times up to 55 business days on RO4000 materials. That probably caused the most angst in the customer base. I'm glad to report we have done two lead time reductions since late March. We took 10 days out in late March, and another 10 days out in May and are down to 35 days. Our plan is to get it under 15 days by the beginning of the fourth quarter and we are ahead of schedule for that lead time improvement. We've also brought lead times down on RO3000 materials as well.

In addition to the lead time reductions, we are creating some strategic stocking programs at the master sheet level so that when we do get a customer order on those parts we can turn the order in less than five working days. I think we have a pretty solid plan going forward on all product lines. The entire period of the last 12 months we went through was very demanding for everybody; it was seen but not expected to come that early and so we were unfortunately behind the demand wave. It's like a guy in the ocean who is going to ride the wave; it just so happened the wave came sooner than we ex-

pected so we had to do a course correction.

We are in very good shape now. Our goal is to get all of our lead times down to industry parity or better before the end of the year, and we are working our way down that glide path right now.

When looking at lead time, we want to be as good as anybody else in industry. So if the best in class is five days in a certain product line, we want to be at five days. If the best in class is 15 days, we want to be at least as good as them at 15 days. We want to be on parity when our customers want to place an order for Rogers' material versus a competitor; they know they can get it in at least the same time frame.

That's probably been the biggest source of irritation for our customers, when we had to balloon out the lead times, but at the time there was no other apparent solution that was quick. We had to create a little bit of breathing room so that we could try to get as much internal capacity turned around as we could. Overall, our team did a great job with that and with managing the customers through some difficult situations. The positive side you can make from the customer's view, we didn't really impact any customer that significantly. We didn't shut anybody down; we didn't create havoc at an OEM or anything like that. We were able to stay ahead of that, but it was not without pain.

Matties: I don't know that you can say that with certainty, though, because maybe the major OEMs were allocated for, but perhaps not all customers.

Pavlak: Right, for some of the customers there were bumps along the road. We put a new expedite process in and that expedite process didn't handle the smaller orders, so we created an escalation process; if they had a really critical order, we got it in the escalation process and gave them an answer within two to three days for how quickly we could pull it in. We did some displacements to try to help them through that period. With many customers, we didn't affect any big programs out there…

Matties: Where the rocket ship didn't launch.

Pavlak: Exactly. It's been an interesting journey for everybody. It's unfortunate that the capacity that we had planned didn't come on line sooner, but some of those projects you can't make go any faster without sacrificing something else. We did pay expedite cost for those capital expansion projects to try to bring them on faster, but I think the global operations team did a really good job of driving more internal improvements to make up much of the difference between demand and capacity.

Matties: That's what is really interesting: your internal process improvement strategy and the increased performance that you found. However, this is just good business practice in any case, whether you are on time or trying to catch up, to find best practices to try reduce waste and improve yields.

Pavlak: What it did was heighten the need to accelerate things that were already in progress. We really put a big effort on that to make sure that was number one in the operations playbook for everybody who was an engineer or technician. It is good business practice but it was like DEFCON 4, the highest sense of urgency. This was on top of everyone's list—no matter what you did when you come in today, here are the top two or three things you need to work on. We really stressed that mindset every week making sure we were focused on all the right things.

There are a lot of common challenges that happen in our industry that you have to overcome. You have to make sure all your suppliers are ready to go. There were some issues with copper availability for a short period of time, and we had a lot of pressure on our copper suppliers to deliver copper. We weren't the only ones suffering from copper shortages; it was across the industry—you mentioned some names earlier of other laminate suppliers having the same problems. We had to press that supply base really hard, too. In Q3 of last year, copper was probably at the tightest with supply it has been in recent times. Now we have some additional sources of copper for our materials, so we have flexibility that we didn't have before. From a raw materials standpoint, we are in really good shape going forward. It did press us into making some pretty tough decisions with some suppliers, as far as getting them moving with us in support of our customers.

Matties: You have your own supply lines issues.

Pavlak: The focus was from the suppliers all the way through the factories. We started at the factory level and said, "Let's look at what we can control and what we can push," and then we worked our way up to the supply base and made sure that we were good there.

We also introduced a slightly different concept into the operation called SMART planning. It's a rigorous S&OP process and we were able to utilize that process to drive more output through the plant without sacrificing quality. We took a much finer and granular look at our demand and who it's for and when it was needed, so we peeled back the onion on a lot of things and started looking at the demand data more closely to make sure we are taking care of our customers in a balanced manner. This improved S&OP process has created more rigor in our global operation —in how we plan, how we execute the production schedule—and that rigor is still there and improving every monthly planning cycle. It took us internally to another level of performance on how we work with our plants in the different regions, how we look at customer demand, prioritizing orders, and things of that nature. We have a lot better system now than we did before.

"The focus was from the suppliers all the way through the factories. We started at the factory level and said, "Let's look at what we can control and what we can push," and then we worked our way up to the supply base and made sure that we were good there."

Matties: Your strategy has really been reaching into the OEM and getting spec'd at that level, not necessarily focused on selling to circuit fabricators, per se. It sounds like once these OEMs have spec'd you in, they don't want to go through another process of bringing in somebody else to replace it. They've been pretty patient with the delays and the delivery times.

Pavlak: Actually, we have been working very closely with both OEMs and fabricators in selecting the best materials for particular applications and then supplying to them timely through our global manufacturing and supply chain network. Yes, our customers including OEMs and fabricators are very patient with us, but primarily because we have been working very closely with them on material selection, technical support, new products promotion, and various measures we have taken to improve the supply situation. I think the fact that we are spec'd in helped us through that period of time, and we spent a lot of time with those customers and the fabricators to make sure they understood that there was an end to this and we have an aggressive plan coming to fruition;

just stick with us for a few more months. They didn't walk away and a lot of that is based on trust that we were telling them the truth and they understood what we were doing.

We did a lot of communication with the fabricators and a lot of communication with the OEMs to make sure that they understood there was a plan to get out of this; we laid out a rough timeline so they knew when the additional capacity would be available. Every time we made a milestone in that plan, we did our best to communicate to key fabricators and the OEMs that we made this milestone and here is where we are now on output. We made sure that they were connected with us. We did not want to leave them hanging out there, so we put a lot of work into the communication with the customers.

Another key point with the demand being so high was making sure that the demand we were receiving was real, and to make sure there was no over-buying going on by somebody. So we had checks and balances built in between what the fabricators were ordering and what the OEMs' volumes were, which we call the "zero schedule," and we made sure that we had the right allocations set based on the OEMs feedback.

Communication was the key through all of it. We had to stay tight with both the fabricator and OEM to make sure that everybody was singing out of the same book, and that allowed us to get through the period. We feel we came through it okay, only the customers can tell you how they feel about it, but from our perspective we thought we did a good job of managing through this tough period with them.

***Matties:** Right, so now you started with the lead times over 55 days, and what are you currently at?*

Pavlak: All RO4000 laminates had peaked at 55 days. We were in that for over six months and then we reduced it to 45 days a month ago. Recently we reduced it to 35 and we have plans to get it down under 15 days by the beginning of Q4. Actually, we are ahead of that schedule right now, based on what we are seeing. We think we can get there quicker, so we are making a big drive to bring down the lead times.

***Matties:** The message then to the industry is, "We're almost there."*

Pavlak: Yes, we're almost there. There is another planned RO4000 lead time reduction for early Q3.

> "Every time we made a milestone in that plan, we did our best to communicate to key fabricators and the OEMs that we made this milestone and here is where we are now on output. We made sure that they were connected with us."

***Matties:** You'll be on par with pretty much everyone else by Q4?*

Pavlak: Yes, and the new capacity we are putting in, all the new equipment, is designed to take us out to the next three years of demand as we see it. We stepped up lamination, we stepped up the pre-preg production for RO4000 lamination and we are also bringing on a brand new coating line in Arizona for the RO3000 product line that is being released for an OEM application at the end of June. That new line is going to give us 100% more capacity on dielectric paper for RO3000 materials. Now we'll be discussing the next wave of capacity we want to put in place, when and where.

***Matties:** You don't want to find yourselves in the same position six months down the road.*

Pavlak: Exactly. We are going through another market evaluation cycle in the next couple of months, saying, "Where is the next capacity we need to be at, at what location on the globe, and what product lines do we need to provide additional capacity?" We are also taking a look

at the former Arlon products, understanding that we are past the due diligence phase with the purchase and now we are into the integration phase. We are trying to understand what capacity we can squeeze out of the former Arlon plants to help with the overall future demands.

Matties: Was the acquisition of Arlon capacity-driven or was it market-driven?

Pavlak: It was really primarily market-driven. It got Rogers into markets that we weren't into before. One of the strengths of Arlon is that they are a very strong player in the antenna market. The acquisition brought some new customers to us. It also provided some new product breadth and feel that this is a growth area, therefore it was more of a market-driven acquisition. Some additional operational capacity also came along with the acquisition. I call it "the pants come with the suit." The factory in Suzhou has some open floor space for additional presses and a coating line providing some additional benefits for our operations with the acquisition.

The nice part about the Arlon acquisition is we now have two plants in Suzhou; from one plant to the other is about a 45-minute drive. We call the locations Suzhou East and Suzhou West. We have the opportunity to really leverage the fact that it's in the same city and not that far apart. We can leverage the expertise across the locations and we can look at the capacity across locations. I think that because of the short distance between locations there is a synergy there that's going to help us in our China market.

Matties: And China is a booming market.

Pavlak: Yes, a majority of our global laminate sales goes to the Asia region.

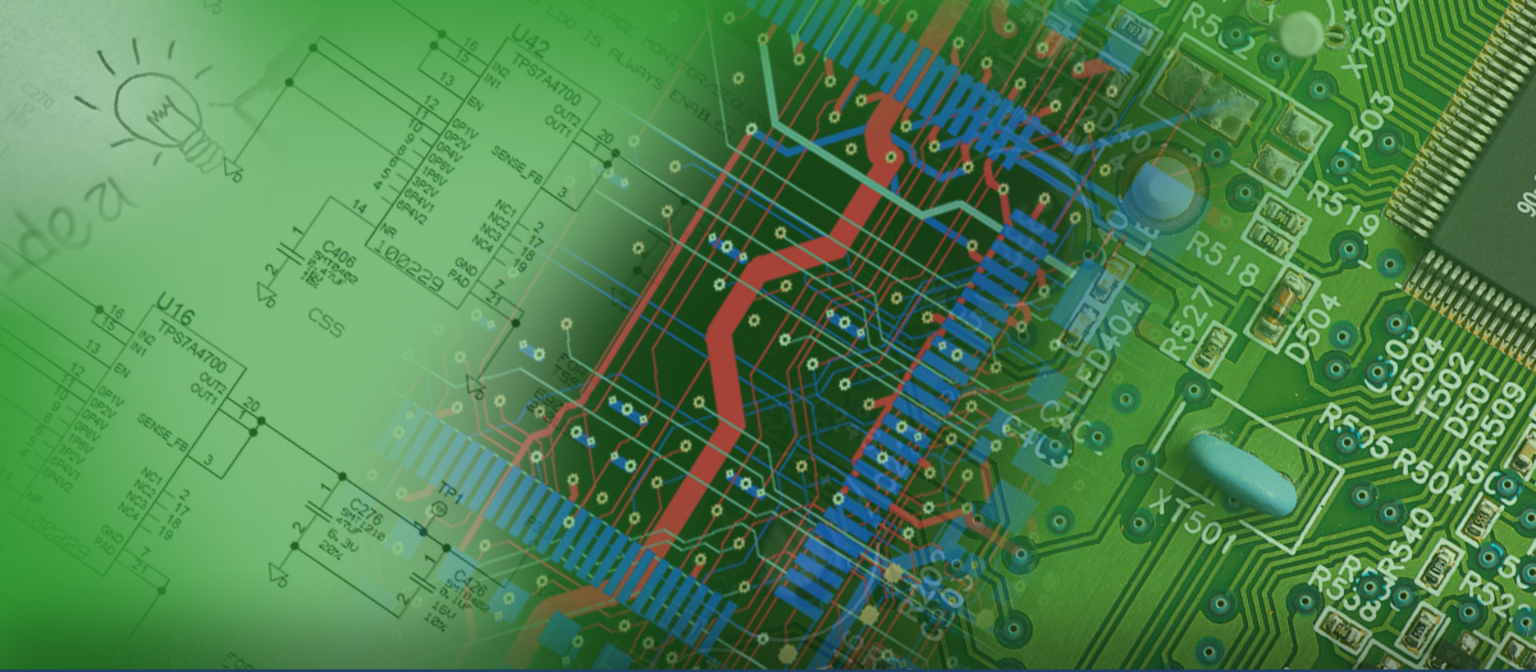
Matties: Arlon did a phenomenal job in China when the markets were dropping and China brought on 3G; it was just a great time for them.

Pavlak: They had a very strong year in 2014 as well. Of course, you always get into a lot of discussions when you are purchasing a company and it's all about "what can you do for us right now?" The Arlon acquisition has been very good for Rogers and for Arlon. The former Arlon employees are very positive about the fact that Rogers bought them. They were part of a private equity group for a period of time and one of the comments I've heard from several people is, "I'm glad that somebody in the industry bought us who understands our wants and needs and what we need to do in the plant to improve." I think everything I've seen so far has been extremely positive from that standpoint. To my knowledge there is nobody in Arlon upset about the acquisition; I think they're actually relieved that it's over and it was acquired within the industry.

Matties: John, thanks for sitting down with me today. PCBDESIGN

About John Pavlak

John Pavlak started in 1979 with General Motors in the Packard Electric Division and worked for General Motors in Delphi for more than a quarter-century. Starting out as a product design engineer, Pavlak then became a senior reliability engineer, quality manager, and eventually worked his way through many operational assignments. He ran two plants for General Motors/Delphi in Ohio before moving to Alabama, where he worked for eight years and again managed two plants, capping off more than 26 years with GM/Delphi. After leaving General Motors/Delphi, Pavlak went to Lennox HVAC, which provided a change of pace away from the automotive industry, and into the manufacturing of commercial rooftop air conditioning units. For two years he ran their HVAC commercial product factory in Stuggart, Arkansas, and then accepted an opportunity with Stoneridge Electronics, based in Juarez, Mexico. After running their global factories for wiring harnesses and electronics manufacturing for more than six years, in mid-2013, John joined Rogers Corporation, where he now serves as the director of global operations.



Gary Ferrari Shares His Thoughts on PCB Design and More

by Barry Matties
I-CONNECT007

Recently, I had a chance to talk with Gary Ferrari, the director of technical support at Firan Technology Group Corporation (FTG), on numerous topics related to PCB design. Our conversation ranged from CID training to the need for reaching high school students as a way of introducing more young people to career opportunities in our industry. We also covered strategies for helping customers design and build better product, and keeping designers provided with the most critical part of their supply chain—information.

***Barry Matties:** Gary, please begin by telling us a little bit about FTG.*

Gary Ferrari: FTG is a circuit board manufacturer—that's the circuits division. Then we have an aerospace division and their products are keyboards, bezels, and panels for the avionics industry. When you look at all the gauges on the dashboard of a cockpit, you are looking at the items we manufacture.

***Matties:** Is it more for the military application?*

Ferrari: It's for both commercial and military avionics. Our products are on helicopters and military aircraft. We're also on many of the commercial aircraft. We're on the new C919 from China. Actually, we have a plant to build those units over there. We have an aerospace and a circuits division in Toronto, as well as an aerospace and circuits division in Chatsworth, California.

***Matties:** What is your position at FTG?*

Ferrari: I am the director of technical support—that means that I get into the customer prior to the design or during the design phase and I help them with their design issues. I look at manufacturability from a fabrication standpoint, as well as assembly, DFM, test, etc. I try to help them design a good product in the long run. That's where I serve best.

***Matties:** When you talk about working with customers, is that at the pre-circuit level when they are just imagining this stuff or is it beyond that?*

Ferrari: They have already done the imagining, and now they need to bring it to the physical world. Sometimes they just don't understand the tolerances or the processes, the sequences in building a stack-up, material characteristics and combinations, buried vias and all those little details. I work with them so that they understand how to build something that's manufacturable and cost-effective. Anybody can design something, but when it comes down to getting fabricated, it could end up being a million dollar design when it could have been maybe a thousand-dollar design, but they messed it all up. We try to help them out with that, and that's what my job is.

***Matties:** We recently ran a survey of the circuit board designers and looked at some of their greatest challenges. It's interesting that you're talking about this particular topic because designers say one of the greatest challenges is lack of information; typically things are thrown over the wall and they say, "Design this; we need a board that does this." The other issue they are really challenged with is space utilization. Not enough real estate for the functionality being asked for by the purchaser, be it an OEM, EMS or whomever.*

Ferrari: It's interesting you asked that question. A customer came to me who had been to one of my seminars, and he said, "Gary, you said that we could take this many components and we could fit them on a board this big; how do we do that?" So I introduced them to new technologies, such as planar capacitance in this particular case. We went through all the simulations, which removed some of the components, and sure enough, we were able to get it shrunk down to the form that they wanted, plus they added even more technology to it. They were a happy customer.

***Matties:** How do we get more people in the position to come to the designers before they just design it? How does this process work? Someone designs a circuit board and then they say "build it," or do they come to an expert designer? What is that process?*

Ferrari: All of the above. Basically what they do is, design it themselves, and then they have to build it. Sometimes it's great and sometimes we have to go back and say, "Listen, this is not manufacturable, the annular ring is not right, and it needs to be fixed." Then we go through the explanation of the process. Other times they may not have an internal design group so they'll outsource that to a service bureau. Or they come to me and ask, "Do you know a service bureau that will help us with this thing?" I then put them in contact with a service bureau that we work with, on either coast. That's one way to get it done.

The other item that would help them is what I'm very much involved with, and that is designer certification. It just amazes me how many designers out there have never had any kind of formal design education. Actually, that kind of education doesn't exist, but we are seeing a big explosion in interest, all of a sudden.

"The other item that would help them is what I'm very much involved with, and that is designer certification. It just amazes me how many designers out there have never had any kind of formal design education. Actually, that kind of education doesn't exist, but we are seeing a big explosion in interest, all of a sudden."

***Matties:** You are referring to IPC Certified Interconnect Designer (CID)*

Ferrari: Yes, CID and CID+. The reason why we're seeing that is because the companies they work for have now realized that the average age of the designer is climbing like crazy. Most of the designers are in their 50s and 60s. The companies are going to start losing their expertise and they having nothing in the food chain coming up.

Now all of a sudden we're getting bombarded with requests for design certification. Certification is just a well-rounded course. It's not everything. It's not really detailed, but at least it gets them a start.

Matties: What was the driving force for the creation of CID?

Ferrari: Dieter Bergman asked me, "What can we do for the design community?" Obviously, there is very little for the design community. The electrical engineers have IEEE and there's ASME for mechanical, etc. All the designers have is sore eyes from looking at the tube all day; they have nothing. I said, "What we need to have is a society where the designers can learn." We formed it as a chapter-based society within IPC; it went worldwide. The chapters brought in guest speakers and went on plant tours. That was the start.

Then we said, "What else can we do?" The designers came back and said, "We'd like to have an education system because the engineer has a degree, this guy has that, and we don't have a formal degree. We'd like to have something that proves that we're not just connect-the-dots people, that we know manufacturing, assembly, test, etc." Engineers just have to know the electronics part of it. The designerscreated the CID and CID+ programs. We like to call it For designers, by designers. We went through a long process but I won't go into that. It was a certification that is supported worldwide, and all of those years IPC took a financial loss. We just broke even on the workshops, but not on the membership and any of the benefits. I can't say that it was a moneymaking endeavor.

Recently, in 2014, they outsourced it to licensed training centers. Now their only income from the program is from the materials that they sell. The cost of the materials went up. Some designers say, "Gee whiz, that's an expensive course." It is, but it isn't, because if you compare it against the cost for doing assembly training, or 6012 training, and other similar courses, it's a lot less expensive.

Matties: Or the cost of a poor design.

Ferrari: Or the cost of a poor design, exactly. It gives them enough information, I say, to be dangerous. They learn a tremendous amount about fabrication, assembly, and test. At least it gets them started in the right direction and most important, they know when to ask a question.

Matties: There are two issues in the design community. One is the aging population. Two is the lack of new entrants coming into this.

Ferrari: We're seeing that grow. I have attended several workshops and I see a lot of younger people coming in.

Matties: How are we attracting younger people? What's the magnet, if you will, to pull them in?

Ferrari: That's the sad part. There's no real formal thing to point at. When I worked with IPC, I used to make the rounds to the colleges, community colleges and all the designers council chapters, and that helped jazz it up. That element no longer takes place. I think what's driv-

ing it is that the companies are now realizing that they need to bring some new blood in. You can't just expect designers to understand all these new technologies by osmosis—it doesn't happen. They have to go someplace to get something going. Like I said, this year we're starting to see more. I hired three new instructors this year, and now all of a sudden we're out of instructors again. This is good. Demand is getting too high, so now I'm looking to add some more instructors. It's encouraging though; I've dedicated my whole career to getting this designer stuff going. I'm pleased to see that it is starting to take off again.

***Matties:** Do you think that part of the drive is reshoring? There's just this new competitive attitude in America in manufacturing that we are starting to see in the field.*

Ferrari: I think every little piece is part of it. It's not just one item when you think about it, but there has to be some of that. What many companies did when it was outsourced was maybe keep a department of one designer to coordinate with overseas. As the overseas workers see the benefits that everyone else is getting, they want more and more. The company costs start to go up to the point that in some market segments it costs as much to get it there as it does over here. Now the companies are saying, "Let's just bring it back." You can bring it back, but you laid everybody off and now you have to start hiring again. But now the technology has changed so much, it's just phenomenal where it is. The designer needs to know so much more, so you just don't hire somebody off the street and say, "Listen, you're a designer, and next month you better be designing these big complex boards." It just doesn't happen.

***Matties:** The CID program is great, but as you're saying, it's broad, it's not very specific. If there are designers struggling, according to our survey, with space issues, layout, and functionality, how do we solve that? What's the solution?*

Ferrari: The program does introduce new technologies such as planar capacitance for instance—getting rid of most of the decoupling capacitors that you just can't fit on the board. All the speeds are going up much higher, so we talk more about embedded components and embedded devices. We just can't possibly cover them in depth in the short period of time that they have there [in a workshop]. We originally worked on what we called focus modules to address specific technologies. I don't know if those will ever come out of IPC, but many of us offline, have decided that we may create some. Let's face it—we cannot update a course as fast as the technology changes. All the instructors tend to add some information about the latest and greatest when addressing the class.

***Matties:** I'm really interested in this…our readers are saying they want to know more about how to address this issue, and I think there's a responsibility that we have to go out and help answer those questions.*

Ferrari: Our biggest issue in all of the years since we started this thing has been that it's all based on volunteers. We're rewriting both of the two courses and it's all done with volunteers. These

people have full-time jobs and sometimes overtime and I've got them working every night and weekends. I get on the telephone with the West Coast at 8:00 p.m. and I'm on until like 11 p.m. or midnight with them working on it. They're working Saturdays, Sundays, holidays. They are doing all kinds of stuff and they're not even getting paid for it. They're doing it out of the dedication to make new things happen. So when you say you'd like to help do something, any help we can get, to get out or develop a course, we would take it.

Matties: How do we get more designers to go out and learn manufacturing?

Ferrari: I used to market to upper management. I showed the cost implication of a designer doing it incorrectly and how it affected their bottom line. The more they learned about that, the more they learned to appreciate the need to keep that designer up to date on technology. They need to invest some dollars to educate them. Let them go to conferences, let them go to chapter meetings where they get tours manufacturing facilities. When you say manufacturing, that's a broad term—that's assembly, test, field service, and board fabrication to name a few. All of those are items that the chapters attempt to cover. They'll bring in speakers. I speak around, Rick Hartley speaks around, and they'll program specific topics. The more education that they can do, the better off they are.

I encourage designers that whenever they go out and get any kind of education, whether they attend a seminar or whatever it is, that they bring back a little write-up on what they learned and how it can benefit the company. If they just say they learned this, that and the other, the next thing you know the company is saying, "Don't go there anymore because someone is going to steal you." If the designer comes back and says, "We could have used this here, we could have used that there," that kind of thing, then they become very valuable and they're justifying the company letting them go to the next one. That kind of addresses some of what we're talking about. The last thing to address, and it's something dear to my heart, is how do you attract new kids into this?

I'm going to tell you about the research I did when I was with IPC. As I mentioned, I went to colleges. I found out that all of the major colleges were not interested unless you provided an instructor, the curriculum, and the students. The students won't add anything to their schedule unless it is required. On the other hand, we had good successes with several community colleges.

Now it really hits home. I don't know if you have any kids in high school, but go to a high school, and go to the guidance counselor, and ask them "What listings do you have for career opportunities in printed wiring board design?" They come back and say "Oh, mechanical drawing." "No, printed wiring board." Printed wiring board design doesn't exist. How do you attract someone into a field that doesn't exist?

Matties: If you don't catch them at that age it's not even a thought.

Ferrari: You have to get them at the high school level, maybe get involved with some of the schools on some little projects or something like that. Several of the chapters used to run career days and they'd invite fabricators, assemblers and other support companies, just a little tabletop thing for the high school students to come in and be exposed to manufacturing in a field that the high school doesn't even say exists. These career days were very successful for both the potential designers and those exhibiting. From my perspective, we need to start at that level to show them that it exists. The kids are sitting there on tablets and computers, learning all kinds of things. They know you program that, they understand that, but they don't know that somebody had to manufacture it.

Matties: Gary, thank you so much.

Ferrari: Thank you. **PCBDESIGN**

PCB007 Highlights

EIPC Summer Conference, Berlin: Day 1

Berlin, Germany was the venue for the 2015 EIPC Summer Conference, which attracted delegates from sixteen countries, including Russia, Hong Kong, Japan, Israel, USA and Canada, as well as the European Union, to experience a programme of 21 technical presentations over two days. Also included was a visit to the Berlin laboratories of Fraunhofer Institute, Europe's largest application-oriented research organisation.

An Interview with Gardien Group's Jason Fraser

Gardien Group CEO Jason Fraser talks to Dan Beaulieu about some of the latest developments at the company, some of the technology trends driving the company's strategies, as well as provides his outlook as to where the PCB industry is headed.

Atotech Increases Equipment Manufacturing Capacity with New Facility in China

Atotech, a global leader in specialty plating chemicals, equipment and services today announced the inauguration of its second equipment manufacturing facility in Guangzhou, China to meet the growing demand for its plating equipment. Globally this is Atotech's third facility dedicated to manufacturing production equipment for the electronics industry.

HDPUG Demonstrates Benefits of Cooperative R&D

The High Density Packaging User Group (HDPUG) is a member-driven, non-profit, project-oriented industry consortium that addresses the integration of new electronics component packaging and interconnection technologies into the supply chains of its member companies.

American Standard Circuits' Unique Offerings Contribute to Long-term Success

At the recent IMS RF and microwave show in Phoenix, Arizona, Anaya Vardya, CEO of American Standard Circuits, sat down with I-Connect007's Barry Matties to discuss the current market trends, the company's recent equipment investments, and where American Standard Circuits' growth will likely come from.

Being Flexible in a Rigid World

With double-digit growth in the foreseeable future, flexible printed circuits (FPC), have found a tremendous niche as an enabler for various electronic applications. This trend is expected to drive the need to increase productivity while improving performance and reducing costs. Of course, in order to sell flex, one must tackle the unenviable task of metalizing these often difficult-to-plate materials.

Papers Sought for IPC APEX EXPO 2016

IPC is inviting all industry professionals to submit an abstract for one of the industry's premier technical conferences, or provide a course proposal for one of its largest educational events.

Amphenol Invotec Accepted into the SiG Programme

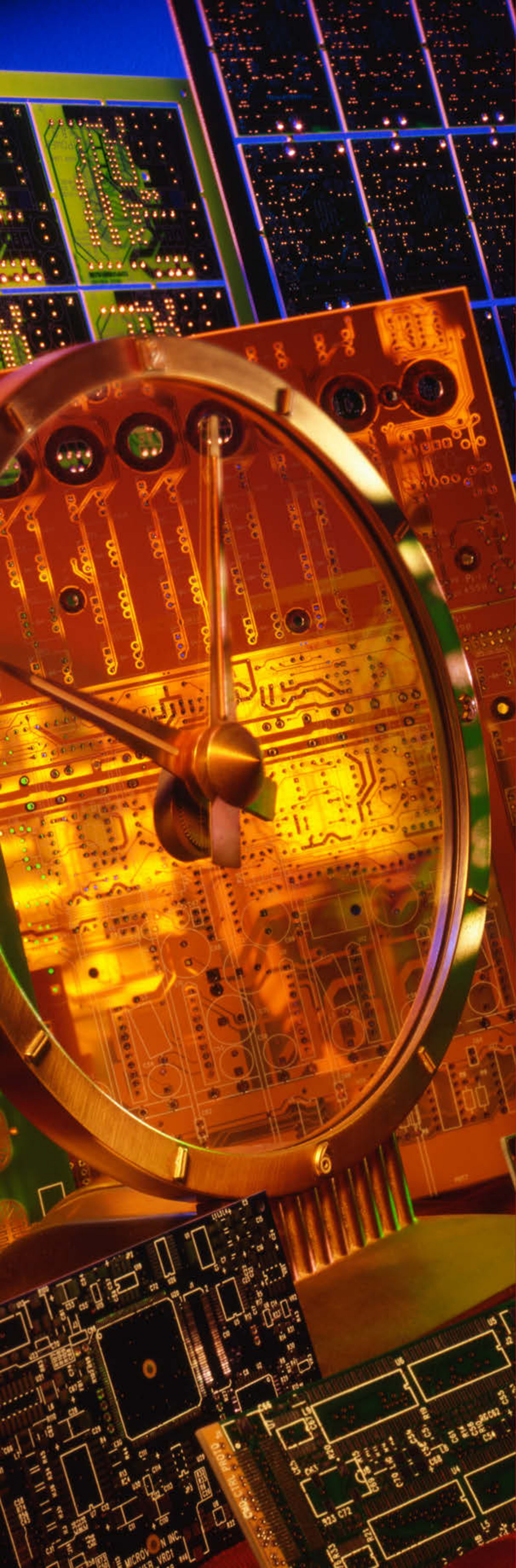
Europe's leading manufacturer of PCBs for critical applications, is delighted to announce that it has been fully approved to be a beneficiary on the Government-backed Sharing in Growth (SiG) programme following a rigorous three-month diagnostic process.

IPC Releases N.A. PCB Industry Results for April

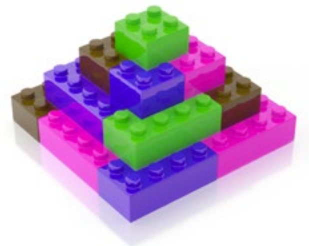
"North American PCB sales continued slightly below last year's level, and orders, which rallied in the fourth quarter of 2014 and remained strong in recent months, also fell below last year's level in April," said Sharon Starr, IPC's director of market research.

New IPC Report Details How PCB Makers Address Tech Trends

The survey-based study shows how PCB manufacturers are meeting today's technology demands and looks at the changes expected by 2019 that will affect PCB fabricators and their suppliers of materials and equipment.



Stackup Planning, Part 2



by Barry Olney
IN-CIRCUIT DESIGN PTY LTD

In Part 1 of the Stackup Planner series, I looked at how the stackup is built, the materials used in construction and the lamination process. And I set out some basic rules to follow for high-speed design. It is important keep return paths, crosstalk and EMI in mind during the design process. Part 2 follows on from this with definitions of basic stackups starting with four and six layers. Of course this methodology can be used for higher layer count boards—36, 72 layers and beyond.

Four-Layer Stackup

A four-layer board is probably not the most practical configuration for high-speed design. Although it does have the advantage, over double sided boards, of using planes for the distribution of power and ground and the planes also act as the return current path for signals. Microstrip traces tend to radiate emissions and this configuration should be avoided in preference to embedding the critical signals using a higher layer count configuration. On the other hand, if you intend mounting the four-layer board, completely shielded, in a metal box then this configuration may be acceptable.

Since all (most) stackups are symmetrical, then it is best to just work on just the top half of the stackup to begin with—this halves the construction time. Stackups are generally symmetrical, with even amounts of copper about the center, in order to prevent the board from warping during fabrication and during the reflow process. If one half has more copper, then it will cool at a slower rate thus warping the board.

The layer selection process is a follows:

1. With four layers, two planes are placed in the center of the substrate. One ounce (1.4 mil) copper is typical for plane layers.
2. Prepreg material separates the signal layers from the planes and this should be as thin as possible in order to achieve close coupling.
3. Solder mask is generally added to the outer layers. This will reduce the impedance by a few ohms.

In Figure 1, the virtual materials yield a 54.44 ohm signal ended and 96.82 ohm differential impedance. This is close enough (to 50 ohms). Now let's fine-tune this stackup by inserting real dielectric materials and then adjust the variables to get the desired impedance.

The ICD Stackup Planner's Dielectric Materials Library contains over 16,700 commonly used core, prepreg and solder mask materials, arguably the most comprehensive list of material properties ever compiled. Using the exact materials that are stocked by your fabricator can increase accuracy by up to 5%. So before you start this process, it is best to consult your fab

2 Layer | 4 Layer | 6 Layer | 8 Layer | 10 Layer | 12 Layer | 14 Layer | 16 Layer | 18 Layer

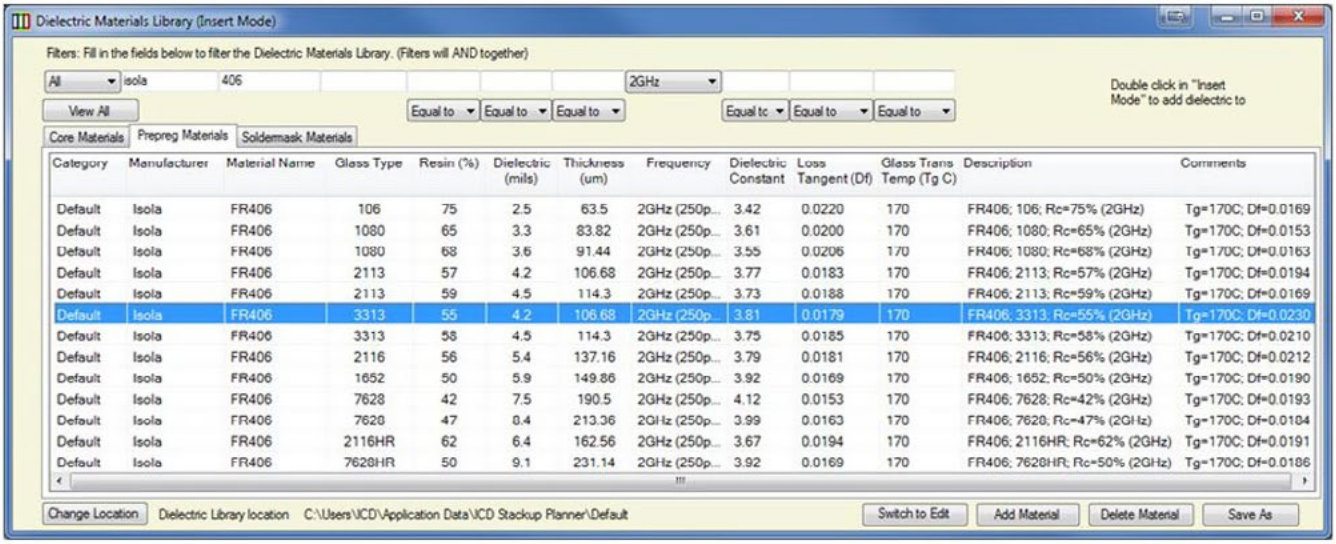
UNITS: mil — 6/1/2015 — Total Board Thickness: 10.8 mil

Differential Pairs > 50/100 ohms

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		Dielectric	3.3	0.5							
1	8	Signal	Top	Conductive			1.4	12	12	0.69	54.44	96.82	
		Prepreg		Dielectric	4.3	8							
2		Plane	GND	Conductive			1.4						

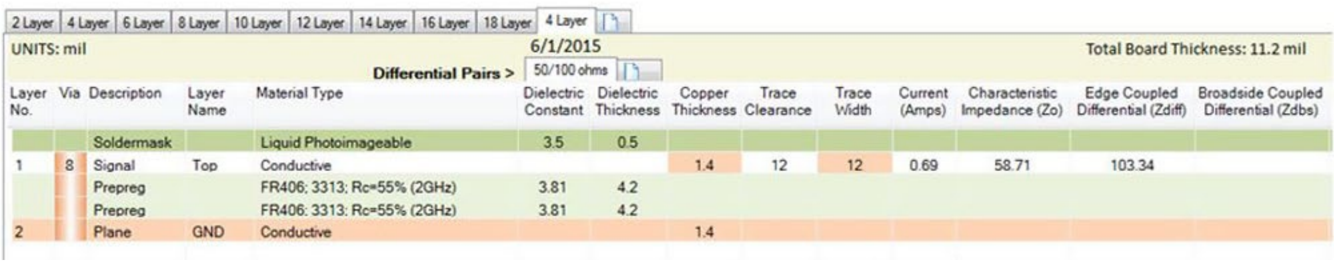
Figure 1: The top half of the four-layer stackup using virtual materials.

STACKUP PLANNING, PART 2 *continues*



Category	Manufacturer	Material Name	Glass Type	Resin (%)	Dielectric (mils)	Thickness (um)	Frequency	Dielectric Constant	Loss Tangent (Df)	Glass Trans Temp (Tg C)	Description	Comments
Default	Isola	FR406	106	75	2.5	63.5	2GHz (250p...	3.42	0.0220	170	FR406; 106; Rc=75% (2GHz)	Tg=170C; Df=0.0169
Default	Isola	FR406	1080	65	3.3	83.82	2GHz (250p...	3.61	0.0200	170	FR406; 1080; Rc=65% (2GHz)	Tg=170C; Df=0.0153
Default	Isola	FR406	1080	68	3.6	91.44	2GHz (250p...	3.55	0.0206	170	FR406; 1080; Rc=68% (2GHz)	Tg=170C; Df=0.0163
Default	Isola	FR406	2113	57	4.2	106.68	2GHz (250p...	3.77	0.0183	170	FR406; 2113; Rc=57% (2GHz)	Tg=170C; Df=0.0194
Default	Isola	FR406	2113	59	4.5	114.3	2GHz (250p...	3.73	0.0188	170	FR406; 2113; Rc=59% (2GHz)	Tg=170C; Df=0.0169
Default	Isola	FR406	3313	55	4.2	106.68	2GHz (250p...	3.81	0.0179	170	FR406; 3313; Rc=55% (2GHz)	Tg=170C; Df=0.0230
Default	Isola	FR406	3313	58	4.5	114.3	2GHz (250p...	3.75	0.0185	170	FR406; 3313; Rc=58% (2GHz)	Tg=170C; Df=0.0210
Default	Isola	FR406	2116	56	5.4	137.16	2GHz (250p...	3.79	0.0181	170	FR406; 2116; Rc=56% (2GHz)	Tg=170C; Df=0.0212
Default	Isola	FR406	1652	50	5.9	149.86	2GHz (250p...	3.92	0.0169	170	FR406; 1652; Rc=50% (2GHz)	Tg=170C; Df=0.0190
Default	Isola	FR406	7628	42	7.5	190.5	2GHz (250p...	4.12	0.0153	170	FR406; 7628; Rc=42% (2GHz)	Tg=170C; Df=0.0193
Default	Isola	FR406	7628	47	8.4	213.36	2GHz (250p...	3.99	0.0163	170	FR406; 7628; Rc=47% (2GHz)	Tg=170C; Df=0.0184
Default	Isola	FR406	2116HR	62	6.4	162.56	2GHz (250p...	3.67	0.0194	170	FR406; 2116HR; Rc=62% (2GHz)	Tg=170C; Df=0.0191
Default	Isola	FR406	7628HR	50	9.1	231.14	2GHz (250p...	3.92	0.0169	170	FR406; 7628HR; Rc=50% (2GHz)	Tg=170C; Df=0.0186

Figure 2 : Isola FR406 4.2 mil prepreg.



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		Liquid Photoimageable	3.5	0.5							
1	8	Signal	Top	Conductive			1.4	12	12	0.69	58.71	103.34	
		Prepreg		FR406; 3313; Rc=55% (2GHz)	3.81	4.2							
		Prepreg		FR406; 3313; Rc=55% (2GHz)	3.81	4.2							
2		Plane	GND	Conductive			1.4						

Figure 3: Virtual materials replaced by Isola FR406 material.

shop and ask what materials they have in stock. For this example, I will use Isola FR406 which is a common low-cost material.

1. Select an 8 mil prepreg. Since 8 mil is not available in the Isola FR406 range, I will select 4.2 mil and add two sheets together. Multiple prepregs are often combined to achieve the desired thickness.
2. Edit the solder mask and select a typical liquid photoimageable material.

Now that all the virtual materials have been replaced with the stocked items, you can see that the single ended impedance is a bit out. The differential is 103.34 ohms.

1. Adjust the trace width to get closer to 50 ohms, then adjust the trace clearance to get 100

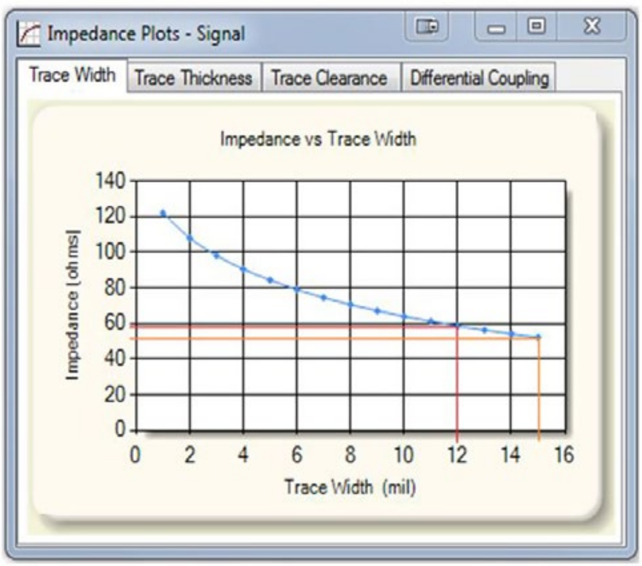
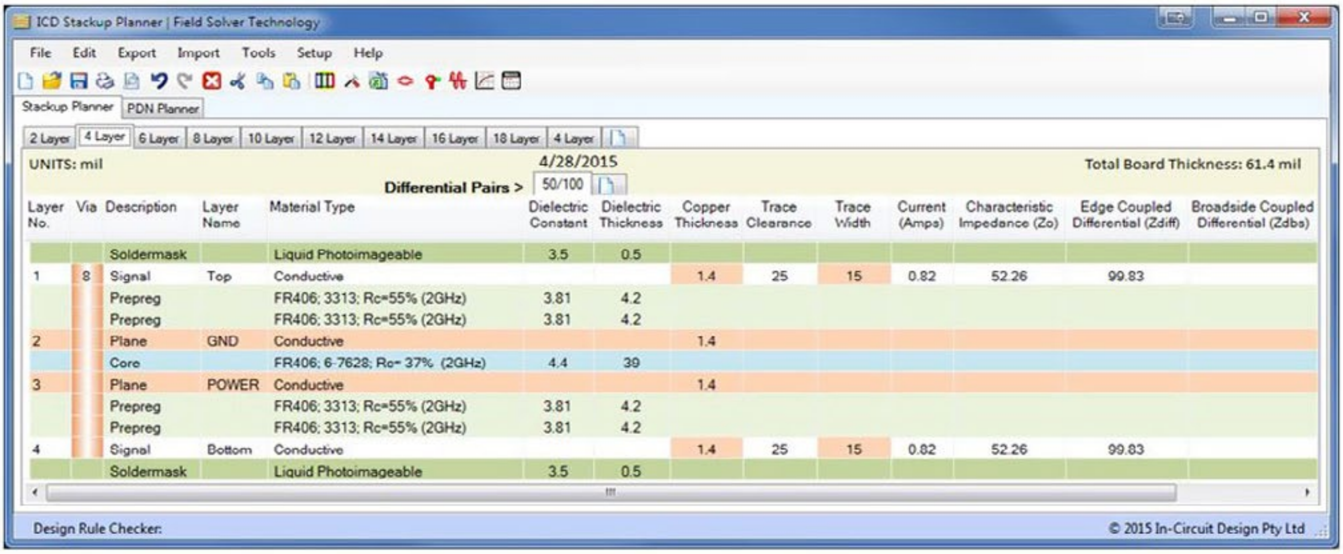


Figure 4: Impedance plots are used to determine correct trace width for 50 ohms.



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		Liquid Photoimageable	3.5	0.5							
1	8	Signal	Top	Conductive			1.4	25	15	0.82	52.26	99.83	
		Prepreg		FR406; 3313; Rc=55% (2GHz)	3.81	4.2							
		Prepreg		FR406; 3313; Rc=55% (2GHz)	3.81	4.2							
2		Plane	GND	Conductive			1.4						
		Core		FR406; 6-7628; Rc= 37% (2GHz)	4.4	39							
3		Plane	POWER	Conductive			1.4						
		Prepreg		FR406; 3313; Rc=55% (2GHz)	3.81	4.2							
		Prepreg		FR406; 3313; Rc=55% (2GHz)	3.81	4.2							
4		Signal	Bottom	Conductive			1.4	25	15	0.82	52.26	99.83	
		Soldermask		Liquid Photoimageable	3.5	0.5							

Figure 5: Completed four-layer stackup with ~ 50/100 ohms impedance.

ohms differential. In the Figure 4 impedance plot, you can see that a ~15 mil trace width is required to achieve ~50 ohms impedance.

2. Since the stackup is symmetrical, this top section of the stackup can be mirrored to create the entire stackup of Figure 5. You need to find a center core of appropriate thickness to beef-up the total board thickness to about 62 mils. A 39 mil core is selected from the Isola FR406 range to achieve this.

Now in this case, I was not able to tightly couple the ground and power planes because the center core had to be used to beef-up the total board thickness. However, to improve the EMC performance of a four-layer board, it is best to space the signal layers as close as possible to the planes, and use a large core between the power and ground plane keeping the overall thickness of the substrate to ~ 62 mils. For high-speed applications, you could reduce the trace width to say 4 mils and prepreg thickness to ~3 mils as shown in the six-layer stackup of Figure 7. This is the most cost-effective and most overlooked way to improve the performance of a four-layer PCB.

There are three advantages to this configuration:

1. The signal loop areas are smaller and therefore produce less differential mode radiation. Tight coupling between the signal and reference planes can amount to ~10 dB reduction in the trace loop radiation compared a stackup with equally spaced layers.
2. This also reduces the plane impedance (inductance) hence reduces the common-mode radiation from the cables connected to the board.
3. And, tight coupling will also decrease the crosstalk between traces.

Six Layer Stackup

Generally a six-layer board is created by adding two more signal layers between the planes of the standard four layer configuration. This has a huge advantage, in that embedding high-speed signals between the planes can reduce electromagnetic radiation by up to 10 dB. Embedding signals between the planes also reduces susceptibility to radiation, as well as providing ESD protection. So, not only do we prevent noise from being radiated, but we also reduce the possibility of being affected by an external source.

Figure 6, illustrates the dramatic difference of radiated emissions from high-speed signals routed on the outer microstrip layers (left) compared to those routed on the embedded inner stripline layers (right). You can see a notable 10

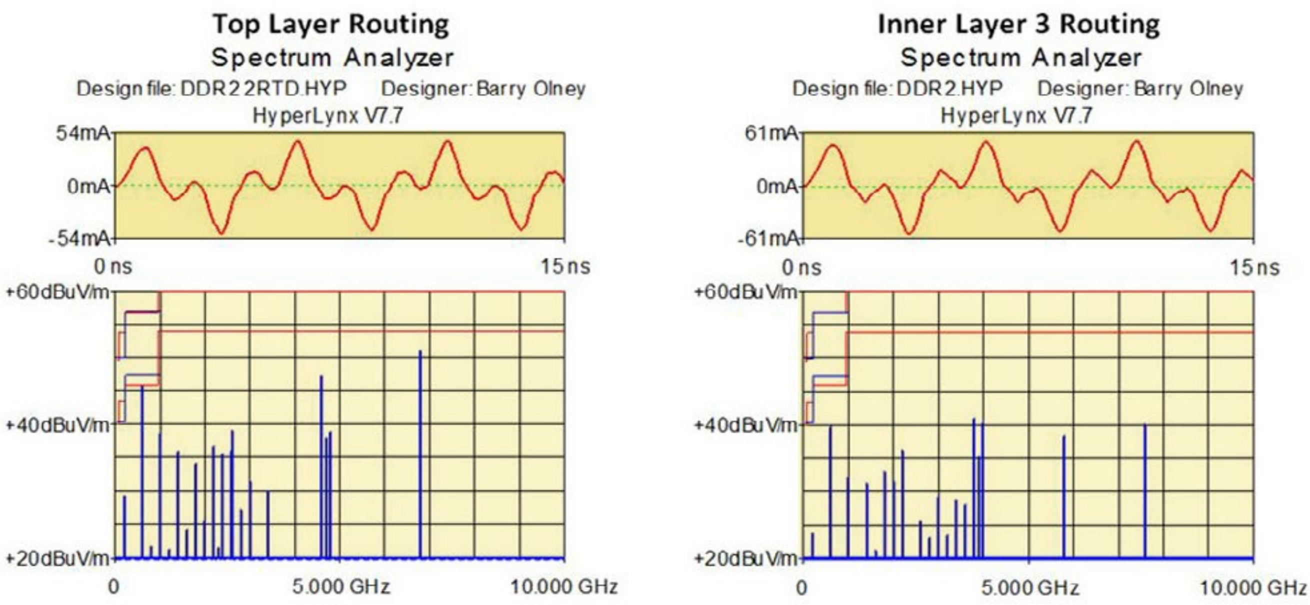


Figure 6: Comparison of radiated emissions from outer and inner layer routing.

dB reduction of emissions. This is enough, in this case, to get the board past the FCC Class B, electromagnetic compliancy.

There are four constraints to keep in mind:

1. Keep the mark-to-space ratio of the waveform equal, as this eliminates all the even harmonics.
2. Route high-speed signals out from the center of the board, where possible, as any radiation will be in the opposite direction and will tend to cancel.
3. Route high-speed signals between the planes, fanout out close to the driver (200 mils) dropping to an inner plane and route back up to the load again with a short fanout.
4. Use the same reference plane for the return signals as this reduces the loop area and hence radiation.

Figure 7 illustrates a six-layer stackup using Isola 370HR 2GHz material which is a commonly used product. This stackup configuration provides many advantages:

1. 1080 glass style prepreg is used for the microstrip outer layers. With just 2.8 mil thickness, this material provides close coupling between the outer signal and ground (GND).
2. The GND plane is used for the common reference—return path—for layers 1 & 3 and VCC for layers 4 & 6.
3. The signal loop areas are small and therefore produce less differential mode radiation.
4. The center is beefed-up by combining five sheets of 7628 material. This provides a total of 40 mils separation between the inner signal layers which reduces any broadside coupling that may occur. Also, routing these layers orthogonally will help reduce coupling.
5. EMI is reduced by routing the high-speed signal on layers 3 & 4 between the planes.

One minor disadvantage of this (and the previous four-layer) configuration is that there is not significant planar capacitance as the planes are separated by beefing-up the center core/prepreg. Therefore, the decoupling must be carefully selected to overcome this limitation. This is where a PDN Planner comes into play—a prelayout PDN analysis can import the stackup and evaluate the capacitance of the planes prior

STACKUP PLANNING, PART 2 *continues*

2 Layer | 4 Layer | 6 Layer 370 hr | 8 Layer | 10 Layer | 12 Layer | 14 Layer | 16 Layer | 18 Layer | 4 Layer

UNITS: mil — 6/1/2015 — Total Board Thickness: 59.2 mil

Differential Pairs > 50/100 ohms

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		Dielectric	3.3	0.5							
1	8	Signal	Top	Conductive			1.4	8	4	0.31	53.95	100.85	
		Prepreg		370HR; 1080; Rc= 64% (2GHz)	3.89	2.8							
2		Plane	GND	Conductive			1.4						
		Core		370HR; 1-2116; Rc=47% (2GHz)	4.2	4							
3		Signal	Inner 3	Conductive			1.4	12	4	0.31	55.27	100.16	107.94
		Prepreg		370HR; 7628; Rc= 50% (2GHz)	4.16	8							
		Prepreg		370HR; 7628; Rc= 50% (2GHz)	4.16	8							
		Prepreg		370HR; 7628; Rc= 50% (2GHz)	4.16	8							
		Prepreg		370HR; 7628; Rc= 50% (2GHz)	4.16	8							
		Prepreg		370HR; 7628; Rc= 50% (2GHz)	4.16	8							
4		Signal	Inner 4	Conductive			1.4	12	4	0.31	55.27	100.16	107.94
		Core		370HR; 1-2116; Rc=47% (2GHz)	4.2	4							
5		Plane	VCC	Conductive			1.4						
		Prepreg		370HR; 1080; Rc= 64% (2GHz)	3.89	2.8							
6		Signal	Bottom	Conductive			1.4	8	4	0.31	53.95	100.85	
		Soldermask		Dielectric	3.3	0.5							

Figure 7: Six-layer stackup using Isola 370HR 2GHz material.

to fabrication. From this, one can approximate the required decoupling and adjustments can then be made to compensate.

In Figure 8, you can see that the lack of planar capacitance creates high AC impedance (0.38 ohms) at 275 MHz. In this case, more decoupling is required around 400 MHz. It may just be a matter of exchanging the 10nF capacitors for 1nF in order to bring the "V" curve, created by the capacitors, up in frequency. The plane area could also be adjusted to move the curve up or down in frequency in order to position the resonant frequency of the planes spot on the 400 MHz fundamental—this is a trial-and-error process that should be done before layout.

Over the years of analyzing customer boards, I have seen many variations of the four and six layer configurations in order to try and achieve the most cost-effective design. But realistically, adding a couple more layers is not expensive and provides more real estate for routing which will in turn yield a high-quality product with less crosstalk, and better EMC. Next month, I will look at higher layer-count boards, particularly 8-layer and 10-layer stackups that are preferred for high-speed applications.

Points to Remember

- A four-layer board has the advantage of using planes for the distribution of power and ground (GND) to the ICs on both sides of the board and the planes also act as the return current path for signals.
- Four-layer configuration boards should be avoided (unless completely shielded) as the microstrip traces tend to radiate.
- Since all stackups are symmetrical, then it is best to work on just the top half of the stackup to begin with—this halves the construction time.
- Six-layer boards have a huge advantage, over four layers, in that embedding high-speed signals between the planes can reduce electromagnetic radiation by up to 10dB.
- One minor disadvantage of four and six layer configurations is that there is no significant planar capacitance as the planes are separate by beefing up the center core/prepreg.
- A prelayout PDN analysis can import the stackup and evaluate the capacitance of the planes prior to fabrication. Adjustments can then be made to the decoupling to compensate.

VALOR PCB MANUFACTURING SOFTWARE | Move into the fast lane with a free test drive of the latest manufacturing technology from Valor®. Get hands-on access to our lean NPI design verification, process preparation, and production planning tools from virtually anywhere. Complete your trial lap, and you'll be ready to cross the finish line with higher efficiency, performance, and productivity throughout your PCB manufacturing processes. Go ahead, take the wheel at www.mentor.com/drive

Mentor Graphics®

THE EDA TECHNOLOGY LEADER

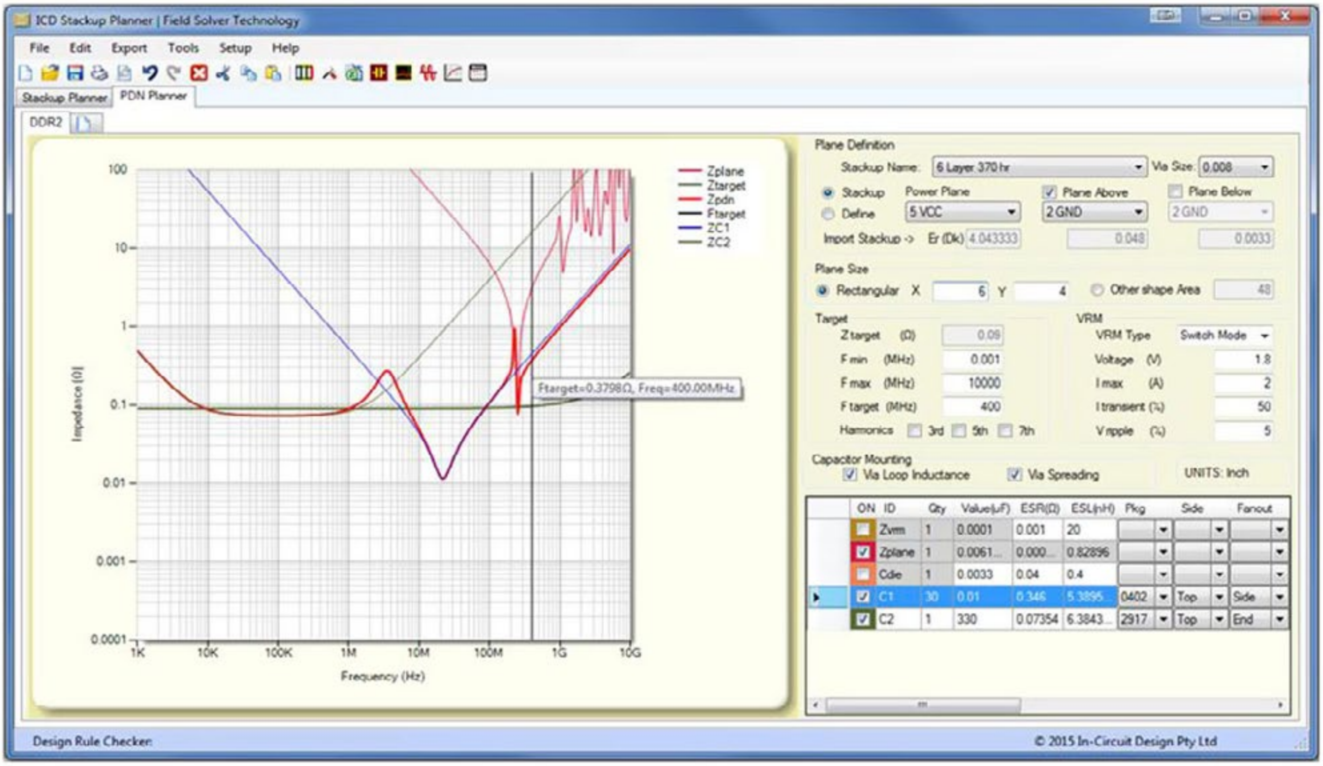


Figure 8: PDN analysis of the six-layer Isola 370HR stackup with decoupling.

References

1. Barry Olney Beyond Design columns: Material Selection for SERDES Design, Material Selection for Digital Design, The Perfect Stackup for High-Speed Design, and Embedded Signal Routing.

2. Henry Ott: Electromagnetic Compatibility Engineering.

3. The ICD Stackup and PDN Planner, visit www.icd.com.au.

Barry Olney is managing director of In-Circuit Design Pty Ltd (ICD), Australia. The company developed the ICD Stackup Planner and ICD PDN Planner software, is a PCB Design Service Bureau and specializes in board level simulation. To read past columns, or to contact Olney, click here.

New Lithium Ion Battery is Safer, Tougher, and More Powerful

South Korean researchers at the Center for Self-assembly and Complexity, Institute for Basic Science, have created a new lithion ion battery from a porous solid which greatly improves its performance.

The new battery is built from pumpkin-shaped molecules called cucurbit[6]uril (CB[6]) which are organized in a honeycomb-like structure.

What makes this new technique exciting is that it is a new method of preparing a solid lithium electrolyte which starts as a liquid, but no post-synthetic modification or chemical treatment is needed.

10 Considerations for Outsourcing PCB Designs

by Steve Dobson
QUADRA SOLUTIONS

Outsourcing your design work is a big deal. How do you know that the end-result will be as you envisaged? Will you have full control of your design? Will it be done to the quality you expect and within the time frame required?

Outsourcing can pose some fairly scary questions, so what are the key things to consider and what are the pitfalls to avoid?

1. Will Outsourcing Work for You?

Outsourcing of PCB design work is done by a wide variety of companies, in numerous industries ranging from one man operations to large design bureaus and equally there is no blueprint for the type of company that outsources or the reasons given.

Many small organisations may not have the capacity to employ PCB professionals or have sufficient design work to allow them to employ someone full-time. However, when new product development occurs, the need for these skills increases and, in order to innovate and adapt, these skills suddenly become a hot commodity. In these situations the ability to have these skills on-demand becomes a critical part of the potential success or failure of a project and outsourcing this crucial element can prove a worrying issue for many companies. However, many of the design bureaus and freelance designers can provide a vital lifeline in getting the product to market on time, keeping costs down and enabling small organisations to be lean and responsive to customer/client needs.

But outsourcing isn't just for those organisations without in-house personnel. Often, outside PCB design support is sought by organisations because the skills required are beyond that of the in-house electronic design professionals. It is also the case that sometimes the in-house professionals require additional support and advice especially if they have had limited exposure to the latest industry developments in design, software or manufacturing. High-speed signals, matching track lengths and signal propagation varieties are just a few of the elements making PCB design more challenging. The complexity is often driven by the require-

ment for smaller and faster technology.

PCB design bureaus are also able to step in during periods of high turnover, multiple projects or periods of staff absence, holidays or during interim periods between full time staff. Whatever the reason behind considering PCB outsourcing, the reasons are neither new nor insurmountable. PCB design bureaus are accustomed to working to tight deadlines and fulfilling the required project specifications catering to a range of industries and criteria.

2. Hello, is it PCB You're Looking For?

First thing to understand is exactly where the sticking point lies, and what elements you actually need to consider outsourcing. Is it schematic entry and library work, component placement, track routing or is it the complete end-to-end process from design through to manufacture and testing?

When considering outsourcing it is important to consider how much work is required and the process. The five stages of design are: initial design review and integration; mechanical definition; placement and review; routing; and pre-delivery and review. All are all critical to the success of the job and these stages need to be understood by you and your PCB designer/bureau so that timescales, costs and design time can be factored in.

Getting a full understanding of the project and work required before outsourcing your project is vital and often ensures that the work can be carried out more efficiently. There are some great articles out there about this that can help you understand more about the design process and range of reputable design bureaus that are happy to answer any queries or concerns you may have. Make sure the process allows for your input every step of the way. Many larger organisations use screen-sharing options that allow you to gain a full understanding of the work being undertaken as the project develops. It also enables you to view your live design on screen so that you can have the appropriate discussions to move your design forward.

"Make sure the process allows for your input every step of the way. Many larger organisations use screen-sharing options that allow you to gain a full understanding of the work being undertaken as the project develops."

3. Fail to Prepare or Prepare to Fail

Data preparation and library management are time-consuming activities, but is a crucial part of the process and if not done correctly it can severely impair the success of the project. From the inclusion of library parts to the collation of data sheets for each item, the importance of appropriate data preparation and library management cannot be overstated.

Some PCB design bureaus will offer this service, but some won't so it's important to understand what is required before work commences. Some companies will expect full CAD-generated schematics, but many can also work from hand-drawn sketches, if required. This initial planning plays a very important part especially when it comes to communicating your plans. This can create added complexities when locating appropriate elements and parts for the design.

4. The Project Management Triangle

Never heard of it? Neither have many people. Before you decide to outsource your PCB design work one of the questions you ought to ask yourself is, "What is important to the success of this project?"

An easy question, you may think. It needs to be on time, it needs to be within budget and it needs to be good. However, the one thing people often forget is that all of these three parameters are variable and without all three elements in harmony one facet of the project will always be lacking.

The project management triangle looks like this (Figure 1).

Obviously, the best possible place to be is right in the centre where time, quality and

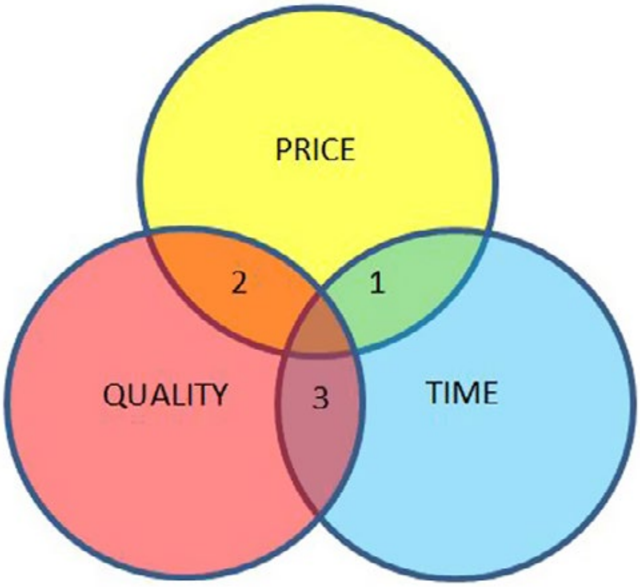


Figure 1: A typical project management triangle.

price targets are all met, but this isn't always possible. And therefore you need to consider your options as different outsourcing options often lay within those different sections of the circles

Segment 1 is often seen as the easy route. The work will be done on time and will be fairly well priced in terms of what needs doing but may have reliability and quality issues. Cheap and cheerful outsourcing is great as long as the design is simple and there is little to go wrong. However, if the design requires multiple layers, high-speed routing and other more complex layout issues, the time and cost saved could be expended by having to do additional re-work and testing.

Segment 2 is most likely to be independent professionals. They are often able to provide a high-quality service and be relatively well priced, due to their low overhead, but because of their nature as independent contractors, they will often, if they are good at what they do, have a number of jobs on at one time and require more time to complete the work. The challenge comes however when multiple jobs arrive at once and the individual has no additional support staff to adjust and adapt to their clients' needs.

Segment 3 usually consists of PCB design bureaus—a group of designers working on a variety of designs usually with extensive knowledge and experience of PCB design. There are a number of these across the UK all with different specialities and with different levels of experience and training. This option is often seen as the most reliable and efficient but sometimes comes at an increased cost. The increased cost can have a big effect on the overall outcome of the project, but ultimately there is a choice to be made in terms of its importance if the potential alternative is that the project runs late or has functionality issues.

Obviously, when searching for a suitable PCB outsourcing option, you want to be as close to the centre of the diagram as possible; make sure you have considered the consequences of falling outside of your target and which elements that are most critical to the success of your project. Don't forget that the initial costs quoted are just that: initial costs. You get what you pay for and often additional costs can be accumulated and added to the final bill, dramatically affecting your bottom line and manufacturing costs.

5. Quality, Quality, Quality

Seems like an obvious thing to say, but if you're looking to outsource, reliability will be a key factor in your decision making process. This doesn't just involve whether they are physically at the other end of the phone when you need them or whether they deliver the project in a timely fashion. These are all important but there are other areas of reliability that need to be considered.

The quality of the design work is crucial. When re-work and costly mistakes are something you can't afford, an unreliable contractor can severely cost a project time, money and effort. Ensuring you have a competent addition to your team is essential, but how can you tell that they are what they say they are?

Well the first thing to look for is whether they have the right accreditations, such ISO9000. The design professionals working on your job should have their IPC CID or CID+ certification. Many PCB design professional have years of professional training and whilst a good level of education is no substitute for experience, ensuring that the PCB design professional working on your job is up-to-date with all the latest international electronics and design standards is crucial to ensuring that the job you require to be done is completed to the right standard and quality.

On the subject of reliability it is also important to consider where and with whom you're information is being sent to. Many PCB subcontractors are happy to sign non-disclosure agreements, which ensures that your patent pending products or closely guarded product information is protected. It is also worth considering how safe your information is in the hands of that outsourcing partner and for some organisations the need to seek confirmation from government bodies in order to outsource their PCB design work is mandatory.

> "On the subject of reliability it is also important to consider where and with whom you're information is being sent to. Many PCB subcontractors are happy to sign non-disclosure agreements, which ensures that your patent pending products or closely guarded product information is protected."

6. Communication

A good and competent communication process for any outsourcing is essential. Knowing what a customer is looking for and how to deliver it can be challenging especially when there are two people looking at the same product, but with different ideas and different images of how it should be completed. Often this can be where the biggest errors occur. Getting the initial schematics correct is critical to the longer-term success of the project.

With this in mind, having the right methods of communication are also vital. Many companies use online web chats which not only enable companies to work collaboratively, but also allow them to use screen-sharing technology to ensure that changes being made are in agreement with both parties. Once the design is complete it can be signed off and agreed by both parties.

The increase of cloud technology has also helped with sharing of large, data heavy documents and means that the work can be shared, adapted and amended, speeding up the decision making and delivery times. With that in mind, geographical distance is less and less of an issue and companies are sourcing designers from far and wide, based upon their work and quality produced as opposed to their location.

7. Long-term Commitment

Often when someone decides to go down the route of outsourcing their PCB design, they will go back to that organisation again (obviously subject to satisfactory competition of the first job). Check their website for case studies, recommendations and online reviews of the company. This can be a useful tool when searching out an appropriate outsourcing partner. If other companies are happy and consistently returning there is more likelihood that they will be a sound partner to work with.

Long-term partnerships are also advised, as an outsourcing partner is in a better position to know instinctively what additional things to note within the design or manufacture. Once a subcontractor is familiar with your procedures and design requirements, they will be accustomed to the challenges and methods required to make those design successful.

8. Outsourcing the Outsourcing?

Quality is a huge concern for many when considering outsourcing and the last thing you want is to go through the process of finding the right designer for you, checking their accreditations, working with the organisation only to find out that the work they have promised to do has been outsourced to another organisation.

This can be a fairly common situation and even though the organisation may trust the company they are outsourcing to, the question is whether their standards will be maintained if they are using multiple designers and companies. This isn't always a problem, but an addi-

tional person in the communication chain can prove challenging which could ultimately create additional costs and challenges.

9. Value for Money

This doesn't just refer to whether you receive the right product for the right price. Instead it refers to the number and the frequency you require PCBs developed. For some organisations, PCBs are required on a regular basis and as a product takes shape and grows in its development (e.g., new product ranges or advances in technology), changes to PCBs will be required. In this situation, in-house design capabilities and software can be seen as a good investment.

However, for some companies, PCB designs are done infrequently or on an ad-hoc basis. In this situation it can often be cheaper and easier to outsource. Ultimately the return on investment needs to stack up.

10. Improved Expertise

Undoubtedly one of the biggest reasons to choose to outsource your PCB design work is the knowledge and experience that can be utilised and harnessed by working with an organisation that regularly undertakes this sort of design work.

Professional PCB designers will work on a wide range of designs daily, so their familiarity with the software is unparalleled. They also regularly undertake training and development, enabling them to be at the forefront of the developments within the industry. Their ability to react and adapt to the needs of the customer and bring additional ideas and solutions to the table helps support and enhance your project.

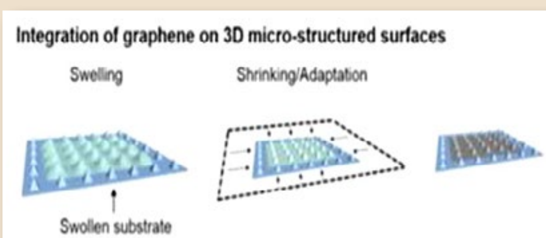
A good design team will have tackled hundreds of projects from a wide range of industries and can play a vital role in bridging the gap between product development and manufacturers by providing an instant skill set. **PCBDESIGN**

Steve Dobson is a PCB design and computer-aided design solutions specialist with Quadra Solutions.

Building 3D Objects from Graphene Sheets

Researchers from the University of Illinois at Urbana-Champaign have developed a new approach for forming 3D shapes from flat, 2D sheets of graphene, paving the way for future integrated systems of graphene-MEMS hybrid devices and flexible electronics.

"To the best of our knowledge, this study is the first to demonstrate graphene integration to a variety of different microstructured geometries, including pyramids, pillars, domes, inverted pyramids, and the 3D integration of gold nanoparticles (AuNPs)/graphene hybrid structures," explained SungWoo Nam, an assistant professor of mechanical science and engineering at Illinois. "The flexibility and 3D nature of our structures will enable intimate biosensing devices which can be conformed to the shape and characteristics of human skin and other biological systems. The 3D protruding micro-structures can also achieve enhanced sensitivity by maximizing the effective contact area between the sensors and non-flat surfaces."



Graphene, a two-dimensional honeycomb lattice of sp2-bonded carbon atoms, has been widely studied due to its high carrier mobility, chemical inertness, and biocompatibility. To date, various reported methods of graphene transfer have been mostly limited to planar or curvilinear surfaces due to the challenges associated with fractures from local stress during transfer onto 3D microstructured surfaces.

Detailed scanning electron microscopy, atomic force microscopy, Raman spectroscopy, and electrical resistance measurement studies show that the amount of substrate swelling, as well as the flexural rigidities of the transfer film, affect the integration yield and quality of the integrated graphene.

MilAero007 Highlights

Enigma Interconnect Now AS9100 Aerospace Certified

Enigma Interconnect is now fully certified to the AS9100 Aerospace Standards. Enigma's consistent implementation of best business practices in quality assurance, major investments in technology and exceptional customer service have been recognized as reaching the high standards demanded by such organizations as the Department of Defense, NASA, and the Federal Aviation Administration.

American Standard Circuits Attending the 2015 International Paris Air Show

American Standard Circuits CEO Anaya Vardya will be part of a group of companies from Illinois attending this year's International Paris Air show to be held from Monday, June 15 to Sunday, June 21 at Le Bourget Exhibition Center outside of Paris.

Flexible Circuits and UAV Applications

The utility offered by flexible circuits in UAVs mimics the advantages that make it popular in other portable electronic applications: lightweight, thin, highly reliable, flexible during use, and possessing an ability to electrically connect across multiple layers as part of a complete packaging interconnect solution.

Wearable Technology and Flexible Circuits

Flexible circuits are an ideal fit for wearable technology. Wearable electronics need to be light, dense and bendable. While what is currently considered standard flexible circuit technology is more than adequate for many the wearable products, there are requirements that may be pushing the boundaries a bit.

Flexible Circuit Materials for High-Temperature Applications

To meet the increasing needs for flexible circuit materials for high-temperature applications, new test methods will need to be developed. These new methods will assign new ratings that are consistent with actual performance.

TTM and Viasystems Receive FTC Clearance for Acquisition

The United States Federal Trade Commission has closed its investigation into TTM's proposed acquisition of Viasystems and the parties are free to complete the transaction.

AT&S Receives Wiener Börse Award

AT&S AG is the winner of the "Small& Mid Cap" category of the Wiener Börse Award 2015. The specialist jury awarded AT&S the first place in this category because of its outstanding performance in 2014, its excellent investor support and its high liquidity.

Dynamic & Proto Circuits Names Matrix Electronics Supplier of the Year

Dynamic & Proto Circuits Inc. (DAPC) named Matrix Electronics Limited as their Supplier of the Year. Each year, this award is granted to the supplier who demonstrates the most excellent track record for product quality, delivery and supply chain resourcefulness.

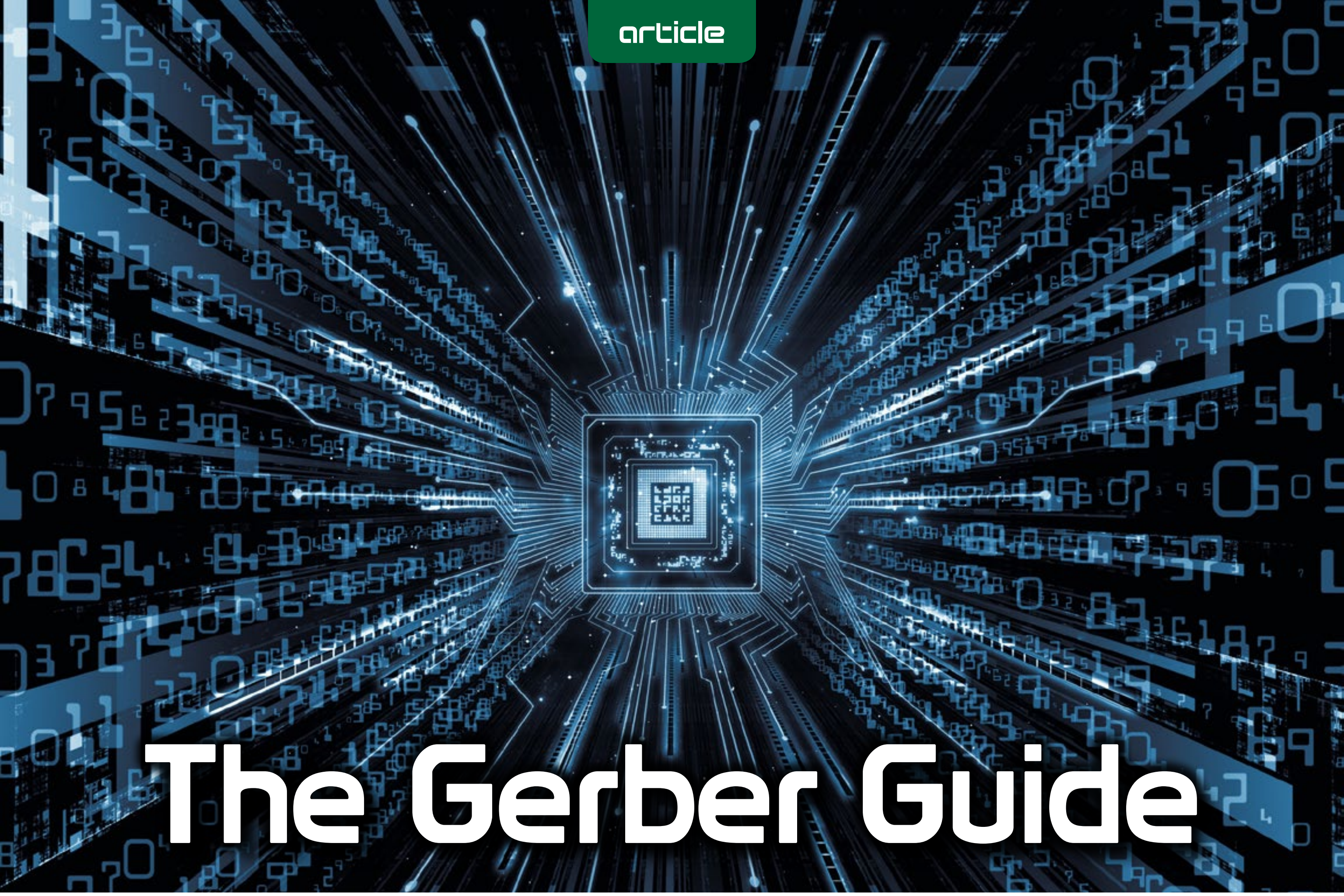
Prototron Circuits' Tucson Facility Purchases New Chemcut Etcher

Prototron Circuits recently purchased a Chemcut XLI-30 Alkaline Etch with Resist & Tin Strip modules for its Tucson facility. According to Kim O'Neil, general manager, this etcher will allow Prototron to produce fine line etching process materials as thin as 2 mil core inner-layers and will allow the company to continue its path towards very fine lines and spaces as well as complex RF geometries.

Multilayer Technology Passes AS9100 Surveillance Audit

Multilayer Technology has passed its AS9100C surveillance audits. The audit consisted of 2.5 days of intense auditing by a registrar. All departments were completely audited to ensure compliance to the AS9100C Certification. This marks the five-year anniversary that Multilayer has maintained this aerospace certification.

article



The Gerber Guide

by Karel Tavernier
UCAMCO

It is clearly possible to fabricate PCBs from the fabrication data sets currently being used—it's being done innumerable times every day all over the globe. But is it being done in an efficient, reliable, automated and standardized manner? At this moment in time, the honest answer is no, because there is plenty of room for improvement in the way in which PCB fabrication data is currently transferred from design to fabrication.

This is not about the format, which for over 90% of the world's PCB production is Gerber: There are very rarely problems with Gerber files themselves. They allow images to be transferred without a hitch. In fact, the Gerber format is part of the solution, given that it is the most reliable option in this field. The problems actually lie in which images are transferred, how the format is used and—more often—in how it is not used.

In this monthly column, Karel Tavernier explains in detail how to use the newly revised Gerber data format to communicate with your fabrication partners clearly and simply, using an unequivocal yet versatile language that enables you and them to get the very best out of your design data. Each month we will look at a different aspect of the design-to-fabrication data transfer process.

This column has been excerpted from the guide, PCB Fabrication Data: Design-to-Fabrication Data Transfer.

Chapter 1: How PCB Design Data is used by the Fabricator

In this first article of the series, we'll be looking at what happens to the designer's data once it reaches the fabricator. This is not just a nice add-on, because for designers to construct truly valid PCB data sets, they must have a clear understanding of how their data is used. This, more than anything else, clarifies how it should be prepared.

We will not look at how to design PCBs for easy fabrication, which is completely outside the remit of the developer of the Gerber format and a matter for the PCB fabricators themselves.

2015 Programs

September 9–11
IPC India Pavilion at electronica India / productronica India 2015
New Delhi, India

September 26–October 1
IPC Fall Standards Development Committee Meetings
Rosemont, IL, USA
Co-located with SMTA International

September 28
IPC EMS Management Meeting
Rosemont, IL, USA

October 13
IPC Conference on Government Regulation
Essen, Germany
Discussion with international experts on regulatory issues

October 13–15
IPC Europe Forum: Innovation for Reliability
Essen, Germany
Practical applications for meeting reliability challenges like tin whiskers, with special focus on military aerospace and automotive sectors

October 26–27
IPC Technical Education
Minneapolis, MN, USA
Professional development courses for engineering staff and managers:

- DFX-Design For Excellence (DFM, DFA, DFR and more)
- Best Practices in Assembly
- Advanced PCB Troubleshooting
- SMT Problem Solving

October 28–30
IPC Flexible Circuits-HDI Conference
Minneapolis, MN, USA
Presentations will address Flex and HDI challenges in methodology, materials, and technology.

November 2–6
IPC EMS Program Management Training and Certification
Chicago, IL, USA

November 4
PCB Carolina 2015
Raleigh, NC, USA

December 2–3
IPC Technical Education
Raleigh, NC, USA
Professional development courses for engineering staff and managers:

- DFX-Design For Excellence (DFM, DFA, DFR and more)
- Best Practices in Assembly
- Advanced PCB Troubleshooting
- SMT Problem Solving

December 2–4
International Printed Circuit and APEX South China Fair (HKPCA & IPC Show)
Shenzhen, China

December 7–11
IPC EMS Program Management Training & Certification
San Jose, CA, USA

Questions? Contact IPC registration staff at registration@ipc.org or +1 847-597-2861

What does a PCB fabricator do with CAD fabrication data?

PCBs are typically fabricated in about 22 steps, many of which are digitally controlled and require dedicated data modules called production tools.

Some designers believe that their PCB fabrication data will drive the fabricator's production machines directly; that the Gerber files will be used directly on the PCB fabricator's photoplotter; that Excellon drill files will go straight onto the fabricator's drilling machines; and that IPC-D-356A netlist will go right into electrical test machines.

Not so. Fabricators never use the Gerber or Excellon files directly on their equipment.

There are many reasons for this, the simplest of which is panelization. Even though the designer's data describes a single PCB or maybe an array, the job is never manufactured as such. It is always put on a production panel, which will typically have multiple jobs on it, as well as a border for plating, test coupons, etc. This is illustrated in Figures 1–3.

It follows that the fabricator can do nothing with production tools for a single job: he needs films and drill files for panel production. Another reason is that deviations are inevitably introduced by the fabrication processes, such as layer distortion during lamination and line width reduction during etching. These must be compensated for prior to manufacture. A third reason is that the production tools driving the fabricator's equipment must fit the fabricator's specific requirements so must often be converted to a proprietary format associated with the machine.

For all these reasons, the production tools that will drive the fabricator's equipment are generated by the fabricator's CAM department. PCBs cannot be professionally fabricated without this step: no CAM, no fabrication. It's as simple as that.

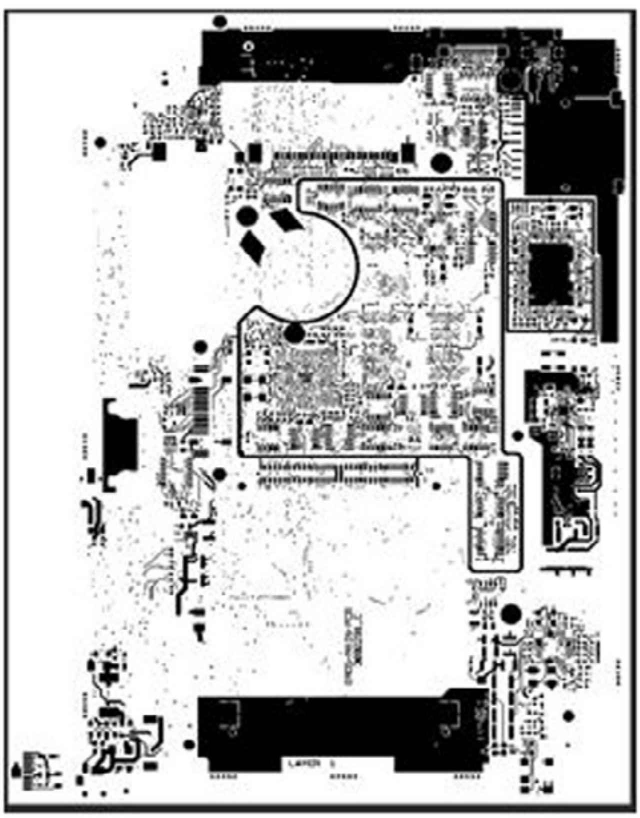


Figure 1: A single PCB.

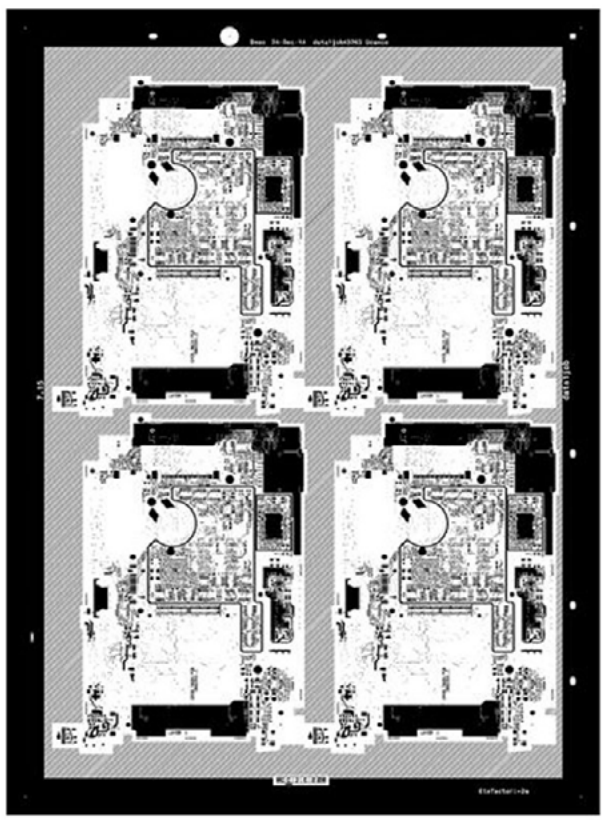


Figure 2: A panel with multiple PCBs.

The PCB CAM system typically performs the following steps (Figure 4):

- Input digital data (Gerber, Excellon, etc.)
- Reconstruct a physical model of the PCB. This may require the manual addition of information that has been "lost in translation."
- Check for errors and if necessary communicate with the designer.
- Send product information to ERP to support quoting.
- Send product information to the engineering department who will decide how and on which equipment the PCB will be made, produce the travelers etc.
- Optionally tweak the design for DfM and if necessary communicate with the designer. At this stage CAM has a model of the single PCB as it will be delivered.
- Create the customer panel (a.k.a. array or shipment panel).

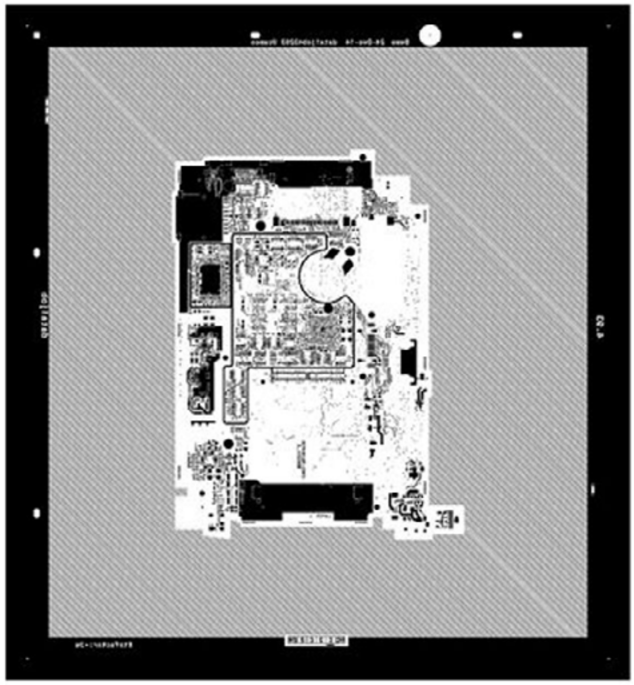


Figure 3: A panel for a prototype PCB.

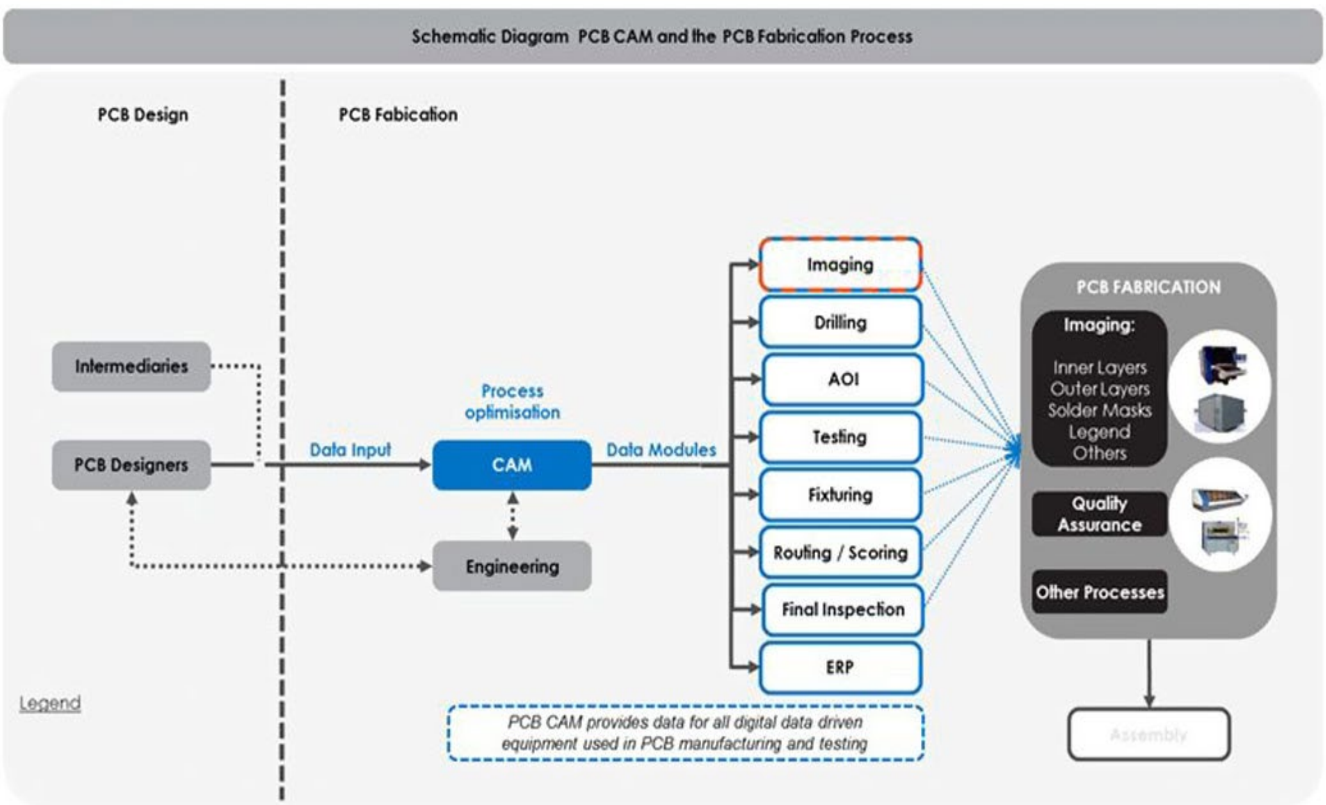


Figure 4: Schematic of the PCB CAM and fabrication process.

- Create the production panel (a.k.a. working panel).
- Compensate for deviations in the manufacturing processes (e.g., scaling to compensate for distortions during lamination). At this stage CAM has a model of the production panel as it will be manufactured.
- Send fabrication data to ERP.
- Output dedicated digital production tools to drive the NC fabrication equipment (photoplotting, direct imaging, legend printing, drilling, routing, scoring, AOI, electrical test files, AVI). The content of these production tools is very specific to the fabricator's and customer's setups, as the information needed and format used are often proprietary and specific to the equipment brand and model, while the equipment must "know" customer-specific details such as the location of the fiducials for registration on the direct imager and "don't care" zones for the AOI system.

"To generate the production tools, the CAM operator needs a perfect physical model of the bare PCB."

To generate the production tools, the CAM operator needs a perfect physical model of the bare PCB. As we have said, the client's incoming fabrication data is used as digital data to reconstruct that physical model.

It may at first sight seem strange that machine files such Gerber, Excellon and IPC-D-56A are used for this, but it actually makes perfect sense. The Gerber format has evolved way beyond its origins as a photoplotter driver to become the perfect vehicle with which to transfer digital image and drill information from CAD to CAM. And it is precisely because of those origins that Gerber files are so perfectly suited to their current role of accurately representing where copper and other materials are. Similarly, Excellon drill files correctly specify where the drill holes are. Which makes these formats capable of describing a PCB.

So the incoming Gerber, Excellon and other data is always read into the fabricator's CAM system which analyses, reworks and transforms the image and drill information into production tools. This is a very different proposition from using the designer's datasets directly as production tools in two aspects:

- The data files are not treated as standalone items, but must be viewed as an interconnected dataset that, together, describes a PCB.
- PCB CAM needs to "know" more about a PCB than just the image. CAM needs to know, for example, which pads are edge connectors, because these need to be gold plated; CAM needs to know which drill holes are vias, because the solder mask is treated differently for via pads than for component pads, and so on.

These two points affect the way in which the designer's PCB fabrication data should be structured. The designer need not worry whether his or her files will be able to drive NC production machines; it is the job of the fabricator's CAM to manipulate the design data so that it will run on real production equipment. The designer's focus should be on specifying the end product accurately, completely and unequivocally.

Remember: CAD output is CAM input and not machine input. When creating fabrication data from CAD, do not ask, "What can I do to create better production tools?"

Instead, ask, "What can I do to create better CAM input?"

Part 2 of this series will be published next month. **PCBDESIGN**

Karel Tavernier is managing director of Ucamco. Karel has 30 years' experience in software and imaging equipment for the PCB and electronic printed packaging industry, including sales, service and R&D at Barco, Belgium.

Recent Highlights from PCBDesign007

1 I-Connect007 Seeks Experienced Editor

I-Connect007 is seeking an experienced editor to help us take our publications to the next level. "We work hard, but we have a lot of fun," said Andy Shaughnessy, managing editor of The PCB Design Magazine and PCBDesign007. "The right candidate will have industry experience and enjoy working independently and with teams in a systems-based environment."

2 EDA Analyst Gary Smith Dead at 74

Longtime EDA analyst Gary Smith died July 3 in Flagstaff, Arizona at the age of 74. He had been ill with pneumonia. He is survived by his wife, Lori Kate, his son, Casey, and granddaughters Rachel and Shannon. A memorial service is set for Sunday, July 12 at 11 am at the DoubleTree San Jose in San Jose, California.

3 Broadcom PCB Design: Miniaturization on the Cutting Edge

I recently attended the Orange County Designer's Council "Lunch and Learn" meeting, held at the Broadcom offices on the campus of the University of California, Irvine. Afterward, I sat down with Scott Davis, CID, the senior manager of PC board design at Broadcom, to discuss the company's savvy PCB design department and their approach to PCB design.

4 Beyond Design: Stackup Planning, Part 1

The PCB substrate that physically supports the components, links them together via high-speed interconnects and also distributes high-current power to the ICs is the most critical component of the electronics assembly. The PCB is so fundamental that we often forget that it is a component that must be selected based on specifications to achieve the best possible performance of the product.

5 The Do's and Don'ts of Signal Routing for Controlled Impedance

In this column, Mark Thompson once again focuses on controlled impedance structures, both from the layout side and the simulation side. He breaks them down into the sub-categories of the models they represent and the important points to remember when using the various models. He also asks questions such as, "Why would a fabricator ask for a larger impedance tolerance?"

6 The Composite Properties of Rigid vs. Multilayer PCBs

Data sheets often provide data in IPC formats, which may look at properties in rigid laminates rather than thin lams and prepregs used for multilayer PWBs. Plus, some product lines are manufactured with consistent resin content for all product thicknesses, thus maintaining dielectric properties while letting the mechanical properties be dictated by the constructions and resin content.

7 Kelly Dack Discusses his Recent Move

I've known the Prototron staff for years, and worked with them for a number of years as well. So, when I heard they had hired Kelly Dack, a longtime PCB designer and guest editor for PCB-Design007, I wasted no time meeting with Kelly to talk about his new position, the future of PCB design, and the nascent interest millennials are showing in the PCB industry.

8 Freedom CAD Named Official Altium Service Bureau

North American design and engineering firm Freedom CAD Services has recently been named an official Altium Service Bureau. Altium's service bureaus are independent service providers that have been qualified to use Altium products and solutions to service their customers and are listed on Altium's website as an official service bureau.

9 Polar Talks Impedance Control and Insertion Loss Testing

During IPC APEX EXPO, I sat down with Polar Instruments product specialists Michael Bode and Geoffrey Hazlett to talk about the company and its products and solutions. We also discussed some of the signal integrity technologies being enabled by the company, including controlled impedance and insertion loss testing.

10 ICD Releases 2015 Version of the ICD Stackup and PDN Planner

In-Circuit Design Pty Ltd (ICD), Australia, developer of the ICD Stackup and PDN Planner software, has released the 2015 version of their popular software. "The impedance plots are simulated by multiple passes of the field solver (in the background) to create heads-up graphs of how to adjust the particular variables to achieve the target impedance," said Barry Olney, CEO.

events

For the IPC Calendar of Events, click here.

For the SMTA Calendar of Events, click here.

For a complete listing, check out *The PCB Design Magazine's* event calendar.

SEMICON West
July 14–16, 2015
San Francisco, California, USA

Ohio Expo & Tech Forum
July 16, 2015
Cleveland, Ohio, USA

7th Annual SMTA Vendor Show
August 21, 2015
Penang, Malaysia

NEPCON South China 2015
Aug 25–27, 2015
Shenzhen, China

electronica India
September 9–11, 2015
New Delhi, India

PCB West 2015
September 15-17, 2015
Santa Clara, California, USA

SMTA International 2015
September 27–October 1, 2015
Rosemont, Illinois, USA

TPCA Show 2015
October 23, 2015
Taipei, Taiwan

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

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

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

EDITORIAL:

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

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

MAGAZINE PRODUCTION CREW:

PRODUCTION MANAGER: **MIKE RADOGNA**
mike@iconnect007.com

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

INNOVATIVE TECHNOLOGY: **BRYSON MATTIES**

COVER: **SHELLY STEIN**

The PCB Design Magazine® is published by BR Publishing, Inc., PO Box 50, Seaside, OR 97138

July 2015, Volume 4, Number 7 • The PCB Design Magazine© is published monthly, by BR Publishing, Inc.

ADVERTISER INDEX

Coming Soon to *The PCB Design Magazine:*

August:
The War on Process Failure

September:
Cars: A Driving Force in the Electronics Industry