

THE **pcb** **design** MAGAZINE

May 2014

What is DFM, Really?
by Mark Thompson p.22

**Closing the Gap Between
Design and Fab**
by Ben Jordan p.30

**Consult With Your
Fabricator—Especially
for Hybrid Designs**
by Amit Bahl p.46

AN  I-CONNECT  PUBLICATION

Design For Manufacturing



efficiency

The LEAN NPI Flow: All in One, and Good for All

by Michael Ford, page 12

isola
The base for innovation

the pcb list

THE best way to find a PCB fabricator, anywhere.

GRAPHIC PLC

Crediton, Devon, United Kingdom



GRAPHIC

Overview

Contact

Specs

About

Videos

Photos

Brochures

News

Graphic is one of the most technically competent manufacturers of high-technology printed circuit boards. With headquarters located in the heart of the English countryside, Graphic focuses on leading edge technology released to the most demanding international quality standards. Constantly looking at innovative solutions to customer requirements,

Markets: Communication, Industrial, Medical, Military/Aerospace

Board Types: Multilayer, Rigid-Flex Other: HDI, SBU

Mfg Volumes: Prototype, Small, Medium, Large

Specialties: Blind/buried vias, Embedded Components, Filled/plugged vias, HDI, Sequential lamination

Certifications: AS-9100, ISO 14000, MIL-PRF-38532, NADCAP, UL




Click here to see a demo

Why YOU should Showcase:

- Capabilities listing for advanced search functionality
- Specialties and certifications listing
- Ability to upload brochures and videos
- Quick and easy "Contact" and "RFQ" buttons
- News, web and contact links

Click to see a partial list of registered OEMs!

www.thepcblist.com



**Why did you
choose
THAT
fabricator?**

Designers: when you choose a fabricator, what guarantee do you have that they won't deliver scrap?

We can connect you with manufacturers that we have audited and are qualified to deliver the product that you need, on time and on budget.

How we qualify PCB manufacturers

CLICK TO VIEW 



(360) 531-3495
www.nextlevelpcb.com

This Issue: DESIGN FOR MANUFACTURING

FEATURED CONTENT

A PCB design isn't worth much if it can't be manufactured, or if it gives your fabricator a migraine. This month, our expert contributors Michael Ford, Mark Thompson, Ben Jordan, and Amit Bahl discuss what it takes to have good DFM practices, and why communication with your fabricator is so critical.

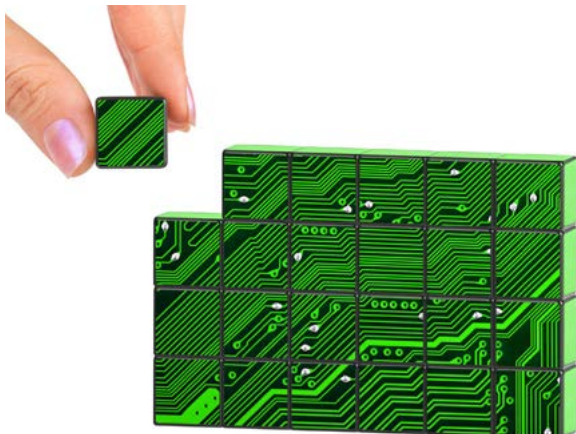
- 12 The Lean NPI Flow:
All in One, and Good for All**
by Michael Ford



- 30 Closing the Gap Between
Design and Fab**
by Ben Jordan

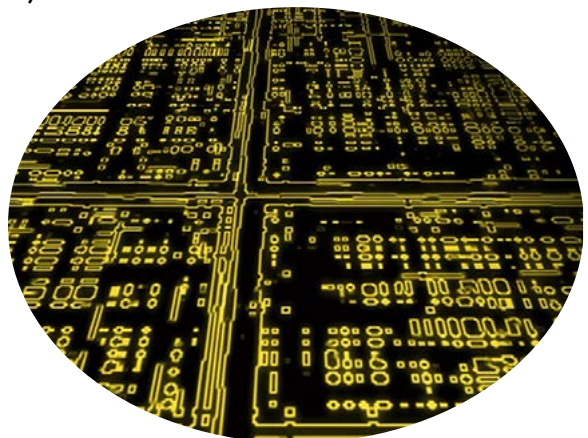


- 22 What is DFM, Really?**
by Mark Thompson



FEATURE COLUMN

- 46 Consult With Your Fabricator—
Especially for Hybrid Designs**
by Amit Bahl



Discover the Best-in-Class Laminate for High Voltage, Tight-pitch PCB Designs

I-Speed® High Speed Digital Materials

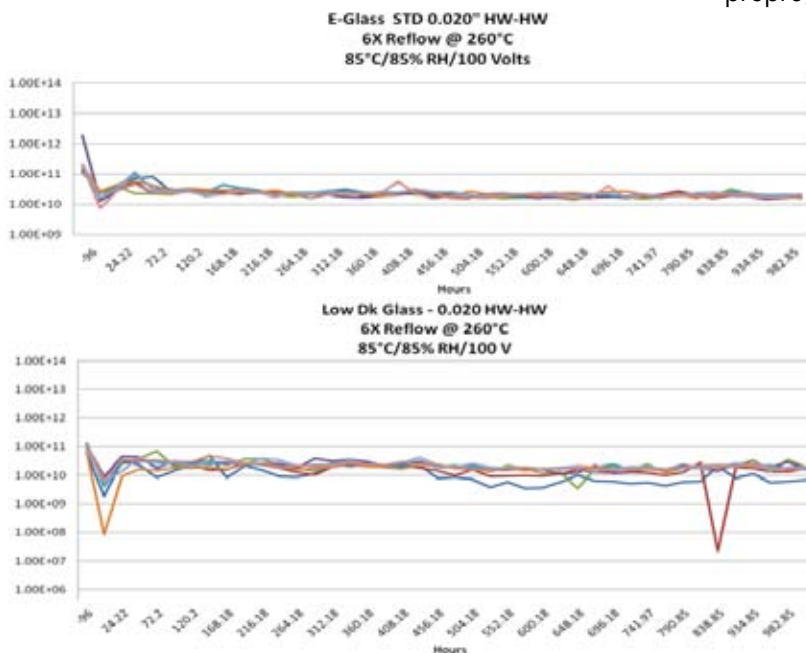
I-Speed laminate and prepreg products are manufactured with Isola's patented high-performance multifunctional resin system, reinforced with electrical grade (E-glass) glass fabric. This system delivers a 15% improvement in Z-axis expansion and offers 25% more electrical bandwidth (lower loss) than competitive products in this space. These properties coupled with superior moisture resistance at reflow, result in a product that bridges the gap from both a thermal and electrical perspective.

I-Speed CAF Test Vehicle Results

- Passed: 85°C/85% RH/100V after 1,000 hours at 0.65 and 0.75 mm pitch
- Passed: 35°C/85% RH/10V after 500 hours at 1.0 mm pitch

I-Speed Features

- Global constructions available in all regions
- Optimized constructions to improve lead-free performance
- Improved Z-axis CTE 2.70%
- IPC 4101 Rev. C /21 /24 /121 /124 /129
- A low Df product with a low cost of ownership
- VLP-2 (2 micron Rz copper) standard offering
- Offer spread and square weave glass styles (1035, 1067, 1078, 1086, 3313) for laminates and prepreps



isola
The Base for Innovation

www.isola-group.com/i-speed

CONTENTS

COLUMNS

- 8 Couch Time at SMTA Atlanta**
by Andy Shaughnessy



- 38 Transmission Lines: From Barbed Wire to High-Speed Interconnect**
by Barry Olney



- 50 HDI Design Education: Is Anyone Interested?**
by Dan Smith



- 56 Customer Support: Not Just for Customers Anymore**
by Tim Haag



NEWS HIGHLIGHTS

- 28 PCB007**
54 Mil/Aero007
60 PCBDesign007



VIDEO INTERVIEWS

- 26 Gary Ferrari Talks Design Certification**



- 44 PCB Design From a Turkish Perspective**



- 59 Always Providing Extra Value to Meet Customers' Needs**



SHORTS

- 20 Engineers Take "Flexible" to the Molecular Level**
49 Nanotube-Infused Clothing Protects Against Chemical Weapons

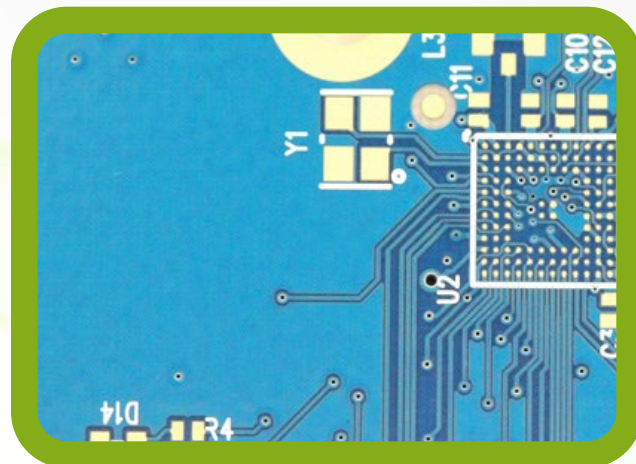


EXTRAS

- 62 Events Calendar**
63 Advertiser Index & Masthead



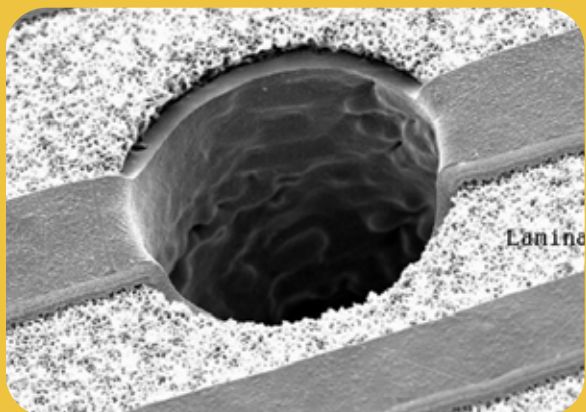
We deliver Advanced PCB Solutions!



Fine line PCB, 2 mil line /space

We have developed a unique set of processes based on electro-depositable liquid photoresist.

[CLICK TO LEARN MORE](#)

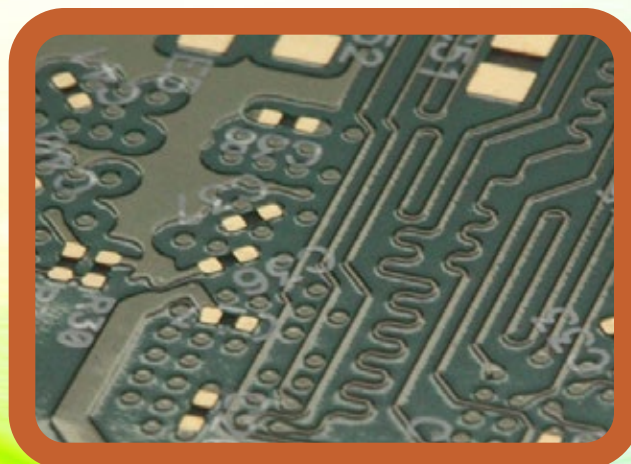


Landless Via Technology

We're the only manufacturer in the world to have perfected a liquid photo-imageable resist (PiP) in electrolytic form, used for all our production.

[CLICK TO LEARN MORE](#)

Our unique processes enable us to create and deliver Advanced PCB Solutions that meet the needs of the demanding market at competitive prices!



Impedance Tolerance <5%!

With our panel plating process combined with continuous foil lamination, our impedance tolerances are second to none!

[CLICK TO LEARN MORE](#)



CANDOR INDUSTRIES, INC.
ADVANCED PCB SOLUTIONS

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

Couch Time at SMTA Atlanta

by Andy Shaughnessy

I-CONNECT007

Every spring, a few months after DesignCon and IPC APEX EXPO have wrapped, I trek across town to the Gwinnett Civic Center for SMTA Atlanta. These regional tabletop trade shows are a great alternative (or a supplement) to the megashows; exhibitors almost can't afford not to attend. How else can you get your company in front of customers and potential customers for less than \$400? And in Atlanta, we offer that famous Southern hospitality, even for visitors who "talk funny."

The exhibitors at SMTA Atlanta just about filled up the room. Everyone I spoke with was in good spirits. Some companies were in hiring mode, and a few had just experienced great quarters, but no one was predicting wild revenue growth this year. Like the electronics industry in general, they were planning for steady, incremental growth. That's better than no growth at all.

But my main mission was to moderate the Designers' Roundtable. This annual event draws designers from all over Atlanta, particularly those who were part of the "Great Scientific Atlanta Diaspora" and scattered to the four cor-

ners of metro Atlanta after SA was acquired by Cisco in 2005.

Local designer Albert Gaines has dubbed the Designers' Roundtable "couch time" for PCB designers, because all designers are borderline crazy and could use a trip to the psychiatrist every so often. Maybe we could get some real couches next year!

This year's roundtable drew about a dozen attendees. It was good to see UP Media Group President Pete Waddell looking good after his health scare last year. He joked about how he quit smoking eight months ago: "All it took was a heart attack!"

This year's attendees represented Cisco, Siemens and a few other smaller companies, including some small (one-man) design bureaus. We also welcomed two fabricators: Rick Kincaid, owner of K&F Electronics, and Paul Handler, North American sales manager for Schoeller Electronics. Someone asked Rick why he was at a designer session, and he said, "I'm here to yell at all of you!" But it wound up being the other way around, though without the yelling.

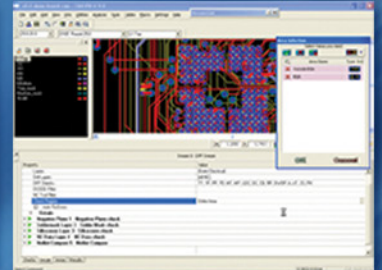
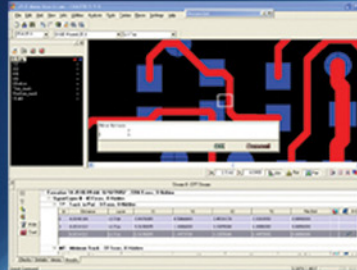


Solutions to Ensure PCB Manufacturing Success!



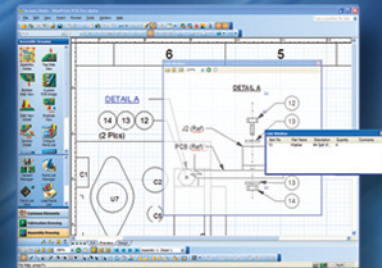
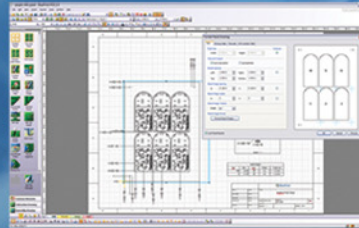
CAM350

Verify and Optimize PCB Designs for Successful Manufacturing.



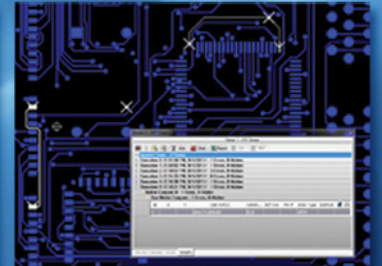
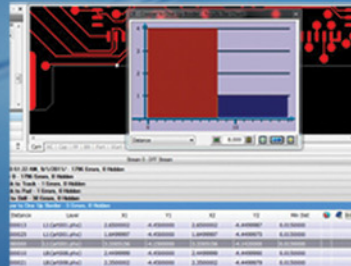
BluePrint-PCB

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.

COUCH TIME AT SMTA ATLANTA *continues*

We also had one brand new newbie, a wide-eyed college-aged kid who described himself as a hobbyist. I wonder if we scared him away from PCB design?

So, I put on my imaginary doctor's coat and asked the designers to tell me what was bothering them. One designer asked Rick why it was so difficult to get fabricators to build smaller lines and spaces.

"Is 4/4 the limit? Have we hit the wall?" he asked. "Come on, it's 2014!"

(Remember all of the predictions and roadmaps in the late nineties that 3/3 and 2/2 would be routine by now? And HDI and embedded passives would be the norm?)

Rick said fabricators would go below 4/4, but it might be expensive, because of the costs the fabricators incur acquiring new, cutting-edge equipment. Rick said his CAM department is happy to consult with the designer, but many times, the OEM is not even aware that his company is fabricating their board. He said he spent years building PCBs for one big OEM, before they even realized he was their fabricator.

That got a laugh, and many of the designers nodded. One designer pointed out, "We never know which board shop is going to build our boards. They hold an auction every three months!"

Does this describe your company?

PCB design instructors, columnists and feature writers, here and at other publications, constantly advocate communication between designer and fabricator, but often that's just not possible.

Another big punch line: A designer asked, "Whatever happened to loyalty?"

The Survey Said

One of the best parts of these roundtables is getting to survey the whole group. I asked how many designers routinely used 3/3 spaces and traces, and one hand went up.

I asked what types of EDA tools they all used, and it was primarily a Cadence/OrCAD crowd, with one Zuken and one Altium user. There were no Mentor Graphics users in the room. One designer laughed, "If it was 10 years ago, it would have been the other way around."

No one seemed truly happy with their design software. They complained that EDA companies pretend to be interested in what the designer needs, but it seems that the EDA company managers market the tools to the OEMs' executives. And they believe EDA tool companies continue to add features that designers don't need and will never use, because they look cool during the presentation. When your CFO is about to scratch out a check for six or seven figures, a cool presentation might seal the deal.

I asked if anyone supervises foreign PCB design teams, and a few hands went up. One designer said, yes, designers based in "cost-effective" countries can design a board correctly. But it takes a lot of time and effort to get them trained. And sometimes they get really good and quit to go work for a competitor.

Then I asked, "What do you wish your EDA tools would do that they don't do now?" Some answers:

- "I wish they would stop changing the things I like about the tool."
- "I wish my tool was stable." (That was the Altium user.)
- "I wish my tool would slap the engineer."

No, there wasn't much love for EEs at the Designers' Roundtable, nor for EDA companies, or ignorant customers. One bureau owner said customers often make mutually exclusive demands, such as having boards comply with a certain IPC class, but using vendor-supplied footprints. Fortunately, designers are all about educating the customer. Right?

UPMG Marketing Manager Frances Stewart topped it off by bringing us chocolate candy, which is always a great way to bribe designers and journalists.

I hope the designers enjoyed "Couch Time" at SMTA Atlanta. See you next year! **PCBDESIGN**



Andy Shaughnessy is managing editor of *The PCB Design Magazine*. He has been covering PCB design for 13 years. He can be reached by clicking [here](#).



“ Providing *Solutions* to Board Fabrication Challenges ”

FROM **CONCEPT** TO **COMPLETION**

Solutions for Every Complex Situation

At Multilayer Technology we have the skills and the knowledge to be able to say “Yes We Can!” to your most complex design requirements.

We specialize in High-Speed Digital and RF Design constraints. In addition, we offer the following solution-based services:

- Extensive Exotic Material Processing
- Pre-DFM Services Available
- State-of-the-Art Industry Leading Processes
- Space-Based Reliability Requirements Standard

REQUEST A QUOTE

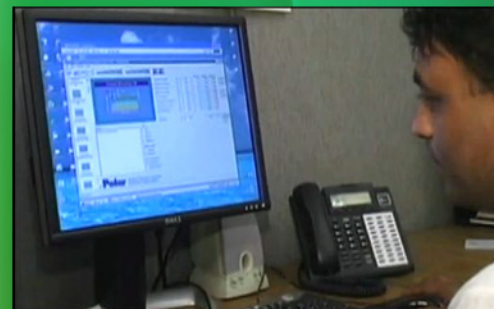
WWW.MULTILAYER.COM



Mil-PRF-55110



AS 9100



Multilayer Technology
3835 Conflans Rd
Irving, TX 75061-3914

(972) 790-0062

efficiency

The Lean NPI Flow: All in One, and Good for All

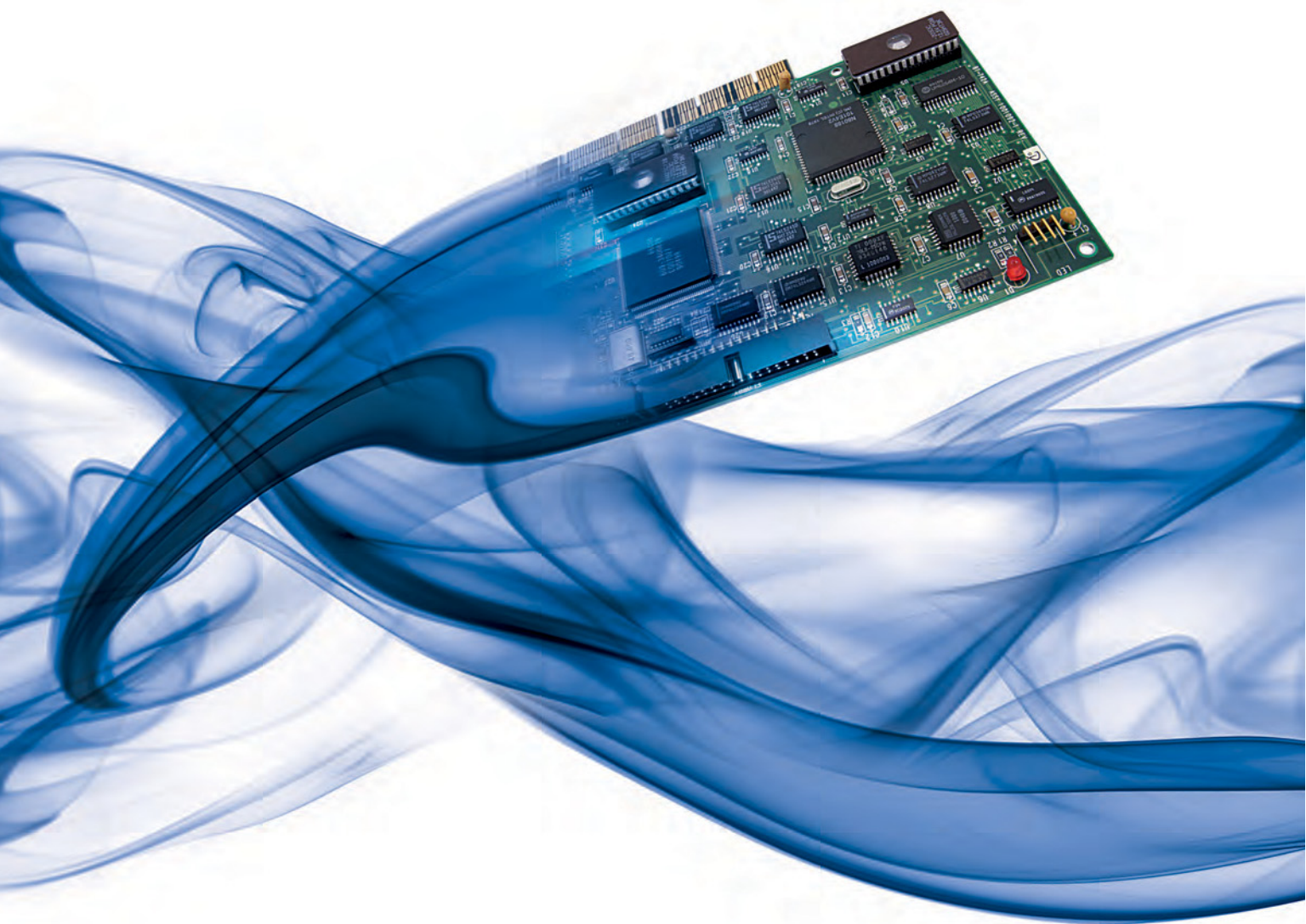
by **Michael Ford**
MENTOR GRAPHICS

Electronics are part of almost everything that we do today. Supporting this huge industry are many different types of electronics manufacturing companies: OEMs (original equipment manufacturers) ODM (original design manufacturers), and EMS (electronic manufacturing services providers). The scale of manufacturing spans products in quantities of tens of millions where each may cost only a few cents, up to a single satellite controller that may cost millions of dollars. Electronics industry sectors are also diverse: consumer, industrial, telecoms, automotive, aerospace, and military.

Geographically, the manufacturing operation also exhibits significant variation, from tightly controlled operations in the West to more seat-of-your-pants operations in the East.

The move to large contract manufacturers has further muddled the water as each of these companies may operate across several sectors and tiers, as well as design and build their own products in the case of ODM companies. In essence, the electronics manufacturing industry is a mess of different operations. No wonder that those in each electronics manufacturing operation think that they are unique, that they are special, that no-one else does things the way they do. Although this is to some extent the reality of what is generally going on inside electronics manufacturing, this situation is simply the consequence of evolution.

Despite the variation in electronics manufacturing companies, they still have significant areas of commonality, against which definitions of best practice can be made. One of the most important is the area of new product introduction (NPI). Even though companies have dif-



Advancing the applications of conformal coatings

Our knowledge of the wider industry, combined with our ever advancing approaches to research and development, have created a range of Conformal Coatings with the necessary high performance and reliability required for the military, aerospace and commercial industries.

Although every electro-chemical solution is in our spectrum, our specialist approach to problems has established us as a key provider and developer to the technical electrical industry, both commercial and domestic.

Isn't it time you discovered out how Electrolube can serve you?
Simply call, or visit our website.

Tel: 888-501-9203

www.electrolube.com

Scan the code to discover
our full spectrum of superior
Conformal Coatings and
electro-chemical solutions.



ELECTROLUBE
THE SOLUTIONS PEOPLE

Electronic & General
Purpose Cleaning

Conformal
Coatings

Encapsulation
Resins

Thermal Management
Solutions

Contact
Lubricants

Maintenance
& Service Aids

THE LEAN NPI FLOW *continues*

ferent products, markets, brands and locations, there is a lot of common ground when it comes to taking an electronics-based product from conception, through design, manufacturing, and to the market to become profitable, no matter who, where or what it is.

The Lean NPI flow is a modern best-practice efficient approach that spans PCB design all the way to manufacturing, reducing the time to market, and eliminating data reconstruction at manufacturing, as well as reducing design respins and revisions. New design software manages the setting of rules that model manufacturing process capabilities and configurations in a way that creates simple rules for the layout process. This allows a designer to avoid manufacturing issues early on, even without any manufacturing process knowledge.

New PCB fabrication tools can be used to immediately set up the necessary fabrication processes. Assembly tools allow the manufacturing processes to be set up directly, without the

need for data reconstruction, and without the need to manage different formats and libraries of data across machines of different types and from different vendors.

The Lean NPI flow is not an ideal that serves only the elite few. It comprises key principles and tools for a common best practice that allows companies to create their products in whatever way they choose and take them to market in whatever way they choose, while eliminating mistakes, waste, and errors so that the end goal is reached much faster, more reliably, with significantly reduced cost and improved quality.

The Lean NPI flow can be applied in virtually any circumstance. As an illustration, let's take a look at two diverse theoretical companies that are in the process of bringing a critical product to market. They each use the Lean NPI flow and tools effectively, bringing differentiation and success to their product launch. The wide gamut of companies in the market fall in between and around these sample cases, and

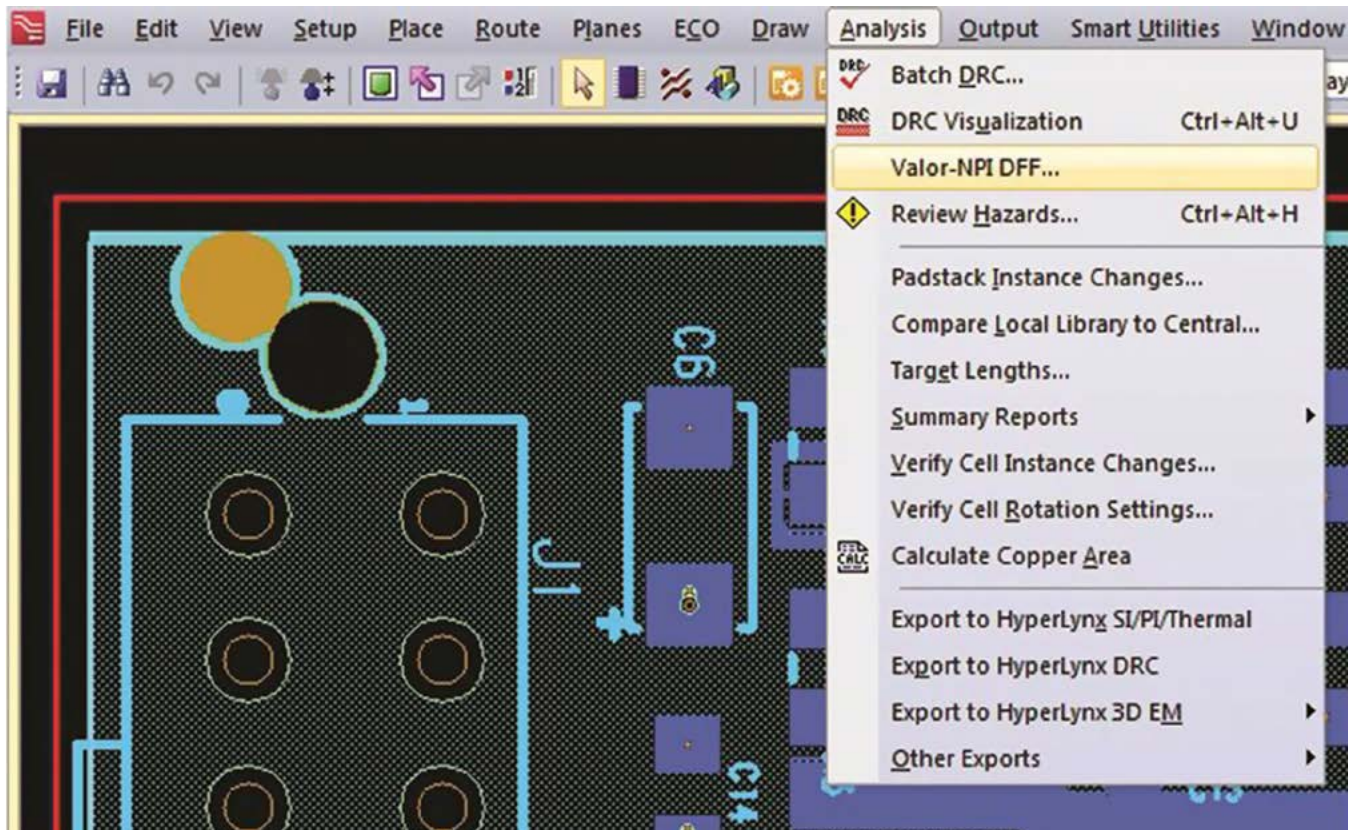


Figure 1: Lean NPI flow as seen in the Xpedition PCB Layout tool.

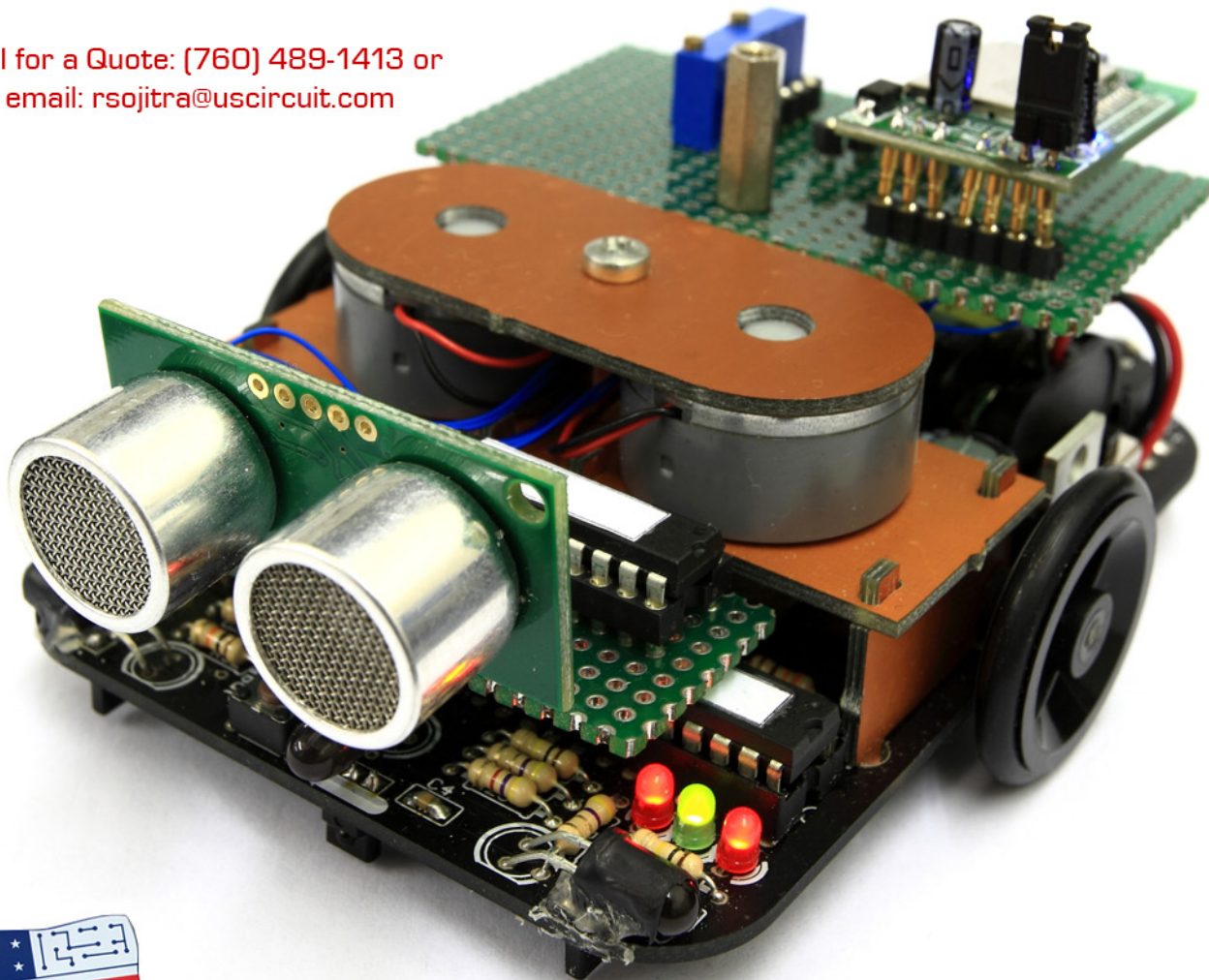


We Deliver...A 5-Star Performance

At U.S. Circuit, we are UL Certified to build **hybrid construction** with Rogers/FR408 family & Rogers/370HR as well as Isola I-Speed materials.



Call for a Quote: (760) 489-1413 or
email: rsojitra@uscircuit.com



U.S. CIRCUIT

Manufacturer of
Printed Circuit Boards

U. S. Circuit, Incorporated

2071 Wineridge Place

Escondido, CA 92029

Ph: (760) 489-1413

Fax: (760) 489-2965

www.uscircuit.com

Superior Quality
Superior Service



ITAR Registered



ISO 9001:2008



MIL-PRF-55110G

THE LEAN NPI FLOW *continues*

they are all capable of achieving similar benefits from the use of these same Lean NPI flow and tools, no matter how mainstream or niche their business may be.

Introduction

The first company, which we'll call "ABC Electronics," is an OEM company in the automotive segment in Europe, with just one main manufacturing site. The company's core mission statement revolves around brand image and quality, with a focus on elimination of safety-critical issues. Demand for increased flexibility from their main customer, a luxury car maker, is putting significant pressure on internal operations, with more product variations, and shorter lead-times. There is an increasing risk as new players are coming into the market, driven by the growth of electronics manufacturing in the automotive sector. The plan is to expand and take on the challenge of new technologies, creating a potential doubling of capacity over the next two years. The question is how to effectively advance in terms of size and flexibility, but still be in control of quality, to avoid any potential disastrous recall scenarios. This has driven the need to look at removing the expensive overhead and burden of new product introduction, while eliminating as much of the seemingly random and human errors as possible.

The second company that we will look at, "Wang Electronics," is an example of a third-tier mainstream contract manufacturer (EMS provider) with three manufacturing sites across China. This company prides itself on being able to make anything for anybody, anytime, if the deal is right. The core mission statement is to act quickly and strongly to win opportunities

away from the competition, so as to grow the business into becoming a Tier 2 company within three years. Responsiveness is critical to be able to provide a price and delivery schedule in a very short time back to the customer. Doing

business in this way, however, has already exposed unexpected costs and risks based on issues from incomplete or inadequate design in terms of capability or compatibility issues in production.

Legacy New Product Introduction

For ABC Electronics, the introduction of a new product follows a well-established internal flow. First, the product is designed, electrically and mechanically, to meet the required specification, followed by the layout of the circuit board, including selection of materials. The emphasis is on quality, cost, reliability, and performance. Manufacturing optimization is a secondary consideration, even though issues with designs have caused significantly increased costs for production.

Although design and manufacturing in this company are physically close, the gap in time and technology means that useful communication between design and manufacturing has been difficult to achieve. The PCB layout designer is a specialist at what he does and often has no knowledge or experience in manufacturing. He is measured on the performance of products and time-based goals related to completing the designs. The designer just follows standard design rules within his layout tool, which are thought to be as much as can be done to make the design a success. This process doesn't, however, include significant qualification for manufacturing. The company's legacy NPI process flow includes

There is an increasing risk as new players are coming into the market, driven by the growth of electronics manufacturing in the automotive sector.

The plan is to expand and take on the challenge of new technologies, creating a potential doubling of capacity over the next two years.

The question is how to effectively advance in terms of size and flexibility, but still be in control of quality, to avoid any potential disastrous recall scenarios.

INTRODUCING COOLSPAN® TECA

thermally & electrically conductive adhesive

Rogers can help by being your reliable conductive adhesive film source

Get the heat out of those high-power PCBs. COOLSPAN® Thermally & Electrically Conductive Adhesive (TECA) Films are ideal for dissipating heat in high-frequency circuits. COOLSPAN adhesives feature outstanding thermal conductivity (6 W/m/K) and reliable thermal stability. Keep things cool, with Rogers and COOLSPAN TECA film.

CONTACT YOUR
SUPPORT TEAM
TODAY



MEET YOUR COOLSPAN® TECA FILM SUPPORT TEAM

Leading the way in...

• Support • Service • Knowledge • Reputation

SCAN THE CODE TO GET OUR CONTACT INFO.



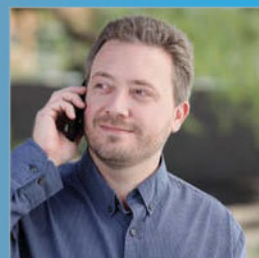
Greg Bull
Applications
Development
Manager
Northern
Territory
(U.S.) &
Canada



Dale Doyle
Applications
Development
Manager
Western
Territory
(U.S.)



John Dobrick
Applications
Development
Manager
Southern
Territory
(U.S.) & South
America



Scott Kennedy
Applications
Development
Manager
Eastern
Territory (U.S.)



John Hendricks
European Sales
Manager



Kent Yeung
Regional Sales
Director Asia

If you are unable to scan a QR code please visit our
Support Team website at www.rogerscorp.com/coolspan



www.rogerscorp.com

THE LEAN NPI FLOW *continues*

many issues originating in design that result in re-spins, delays, cost, and call-backs. This annoys the designer who, at the time a particular design hits manufacturing, is already working to a deadline for the next design, or even the one after.

For Wang Electronics, the NPI process starts with an over-enthusiastic sales team member bringing a USB stick of files, a physical folder full of drawings, and sometimes a sample product, often just a prototype which very roughly illustrates what the product is. The race then starts with the sales team pushing for an “immediate” response as to how much the production costs would be and when production could start.

Opening up the files on the USB stick quickly reveals what information is available: Gerber files normally, a graphical representation of the product, but no actual data. Other files contain parts lists, a simple bill of materials. With luck, it will include placement locations. There can be some form of native CAD data included, which would be a pain to have to translate, but normally, just the drawings provided will serve to illustrate the layout of the PCB. Significant work is required to piece together all of the different pieces of information, to work out what can be understood, and then to start filling in the gaps, either by looking at the prototype, or asking the customer some questions. Going back to the customer is discouraged by the sales team, as they want to maintain their “we can do anything” stance. With the intense time pressure, it invariably means that significant guess-work is done for a reply made back to the sales team.

Wang has a significant success rate to win deals. The bloated engineering team, however,

with many specialists to cover the various different types of machines from different vendors have to spend significant amounts of time to complete the preparation of the data for manufacturing, during which significant data reconstruction and trial and error will take place before the designated production line can start work. Often, unforeseen issues come up, too late to be addressed with the customer or with the sales team, that simply have to be managed. Many additional production operators are ready to deal with any situation. Quality and reliability of delivery is often threatened, with very high scrap rates and barely acceptable quality levels.

“**Significant work is required to piece together all of the different pieces of information, to work out what can be understood, and then to start filling in the gaps, either by looking at the prototype, or asking the customer some questions. Going back to the customer is discouraged by the sales team, as they want to maintain their “we can do anything” stance.**”

The Compelling Event

The people at ABC Electronics know they have to embrace new market demands. They will introduce more products with higher technology, at a greater mix, with shorter lead times. Quality and pricing will be more critical than ever. There is no room in the NPI flow now for so many design re-spins or costs due to issues in manufacturing originating from design. A high-tech vertical OEM has the opportunity to link much more closely the design and manufacturing, and this must be the key differentiator to retain and grow the business. All information in the flow must be available to everyone at any time. The Lean NPI flow tools are chosen to make that happen.

For Wang Electronics, the compelling event was a series of disastrous orders which caused the company significant loss. Top management became very concerned that overall growth was being slowed because the sales team lost confidence, as did their customers. The company's reputation was suffering. The engineering team

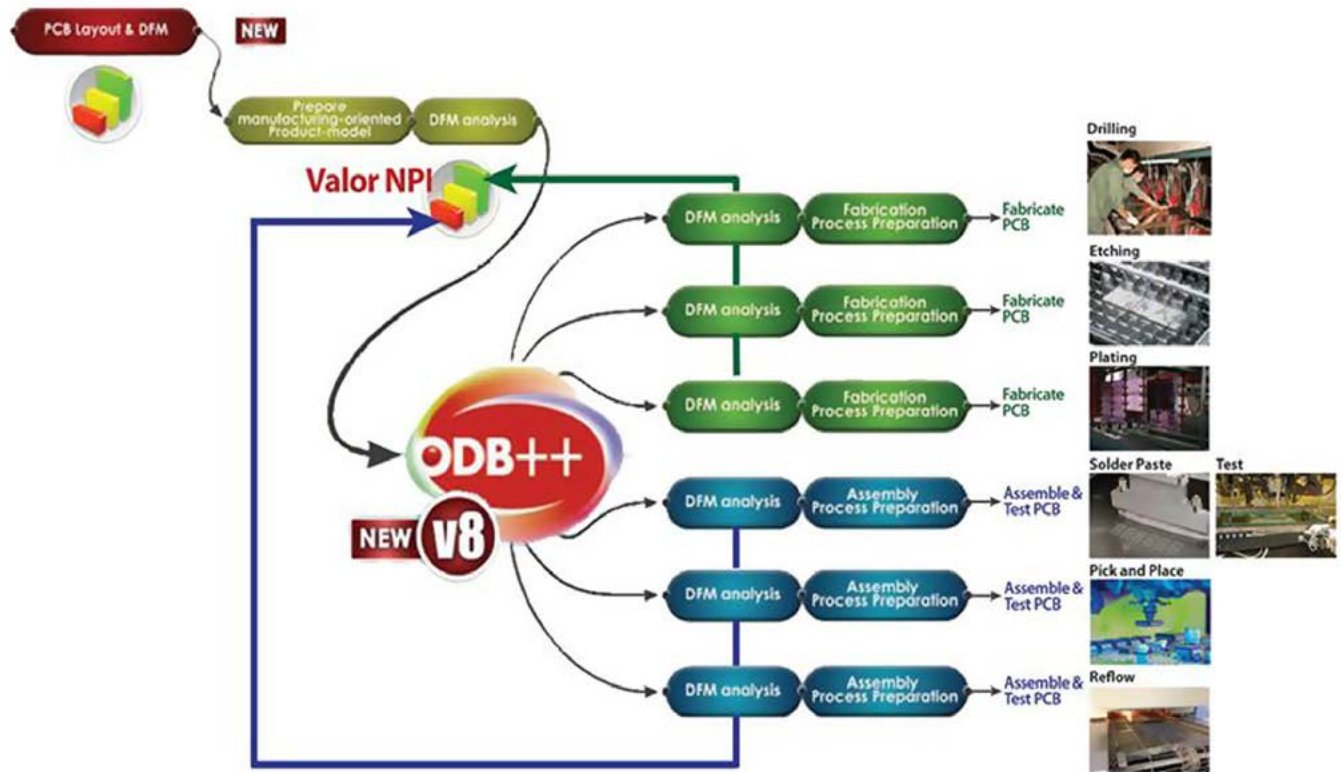


Figure 2: The Lean NPI flow using the latest DFM software tools.

had missed a key issue in one order, where a whole new set of testing procedures were required, at great expense, which had only been recognized after the contract had been signed. A second issue closely followed where materials had been placed incorrectly. There had been a mistake in the engineering checks, which had resulted in products that appeared to work correctly, and even passed testing, but in the market they would fail to work in many customer use-cases. The decision to take control of the engineering data, to increase the ability of the engineering team to spot issues, and give them time to completely analyse the product and eliminate the guesswork was clearly needed if the company was going to expand successfully. Adoption of Lean NPI tools could make that happen.

The Lean NPI Flow

Designers at ABC Electronics are now working with the Lean NPI flow tools. They can now see opportunities to improve the design based on DFM rules created by manufacturing. From

within their design tool, without any knowledge of manufacturing, designers can now be confident that there will be very few if any issues for manufacturing. A few extra mouse clicks have replaced seemingly endless complaints, call-backs, and design re-spins requests. Designers now can quickly move on to the next design and easily hit the time goals expected of them.

The manufacturing teams for PCB fabrication and assembly each receive from design a single ODB++ file that contains all of the information needed for manufacturing the qualified product model. Feeding just this file into In-CAM fabrication tools allows immediate setup production. For assembly, multiple production configurations can be created, giving choices to the planning team to optimize the factory efficiency. Programs, data, and documentation sent to the machines and processes are created based on simulation of the operation, virtually reducing the line trial and error down-time. The data for the machines is complete and fully detailed, including parts libraries in native ma-

THE LEAN NPI FLOW *continues*

chine format ready to load and run, all generated from a single library.

For Wang Electronics, all data received from potential customers goes straight into the Valor NPI tool, which makes the analysis of all of the data, exposing potential issues. As each issue is quantified, production resource and hence, cost can be accurately assessed. In some cases, a definitive report can be sent back to the customer to allow them to change the design to make a considerable cost saving and quality enhancement. This is quickly recognized by the sales team as being an additional service and value for the customer, another positive differentiation for the company. In production, once orders have been won, there is little additional work to get production up and running right the first time.

The Result

From a business standpoint, ABC Electronics now is able to move products from the design and product teams into the market faster than ever before. The company has been able to take on the expected increase of product variations, meeting all of the demands from the customer. Quality has been enhanced, reducing the cost associated with mistakes and building confidence with the brand. Production costs have actually decreased, even though the product mix has increased. A far greater degree of control and flexibility has been achieved, without

compromise to performance of either cost or quality. The expansion of the business has been a success, and the company has become a great role model in the industry.

Business is also booming at Wang Electronics, where the turnaround times to customer requests for quotation have been slashed, while at the same time, profitability and quality performance levels have increased because unexpected issues are now a thing of the past. The company has also started to attract higher value business opportunities, for higher quality more critical electronic products. Margins are much more controlled and predictable, giving confidence to top management. Growth is on-track.

The best practice Lean NPI flow works for almost all companies, across the whole industry. Opportunities continue to grow as OEMs work with CEMs using the Lean NPI flow as a standard platform and communication ability, each company helping each other be in control, and successful.

The Lean NPI flow is good for everyone.

PCBDESIGN



Michael Ford is senior market development manager, SDD Valor Marketing, Mentor Graphics.

Engineers Take “Flexible” to the Molecular Level

Nanoengineers at UC San Diego are asking what might be possible if semiconductor materials were flexible and stretchable without sacrificing electronic function.

UC San Diego Jacobs School of Engineering professor Darren Lipomi compared the difference between flexible and stretchable electronics to what would happen if you tried to wrap a basketball with either a sheet of paper or a thin sheet of rubber. The paper would wrinkle, while the rubber would conform to the surface of the ball.

While flexible electronics based on thin-film semi-

conductors are nearing commercialization, stretchable electronic materials and devices are in their infancy. One of the chief applications envisioned by Lipomi is a low-cost “solar tarp” that can be folded up for packaging and stretched back out to supply low-cost energy to rural villages, disaster relief operations and the military operating in remote locations.

Lipomi’s team has also created a high-performance, “low-bandgap” elastic semiconducting polymer using a new synthetic strategy the team invented. Solid polymers are partially crystalline, which gives them good electrical properties, but also makes the polymer material stiff and brittle. By introducing randomness in the molecular structure of the polymer, Lipomi’s lab increased its elasticity by a factor of two without decreasing the electronic performance of the material.



www.5pcb.com

- ☒ Convenient
- ☒ Fast delivery
- ☒ High quality
- ☒ Low prices

With our online system, getting quotes and placing orders is fast, simple and convenient. In fact every step of the process can be managed online—even payment! And our quality is guaranteed.

All you need to get started is to set up your account.

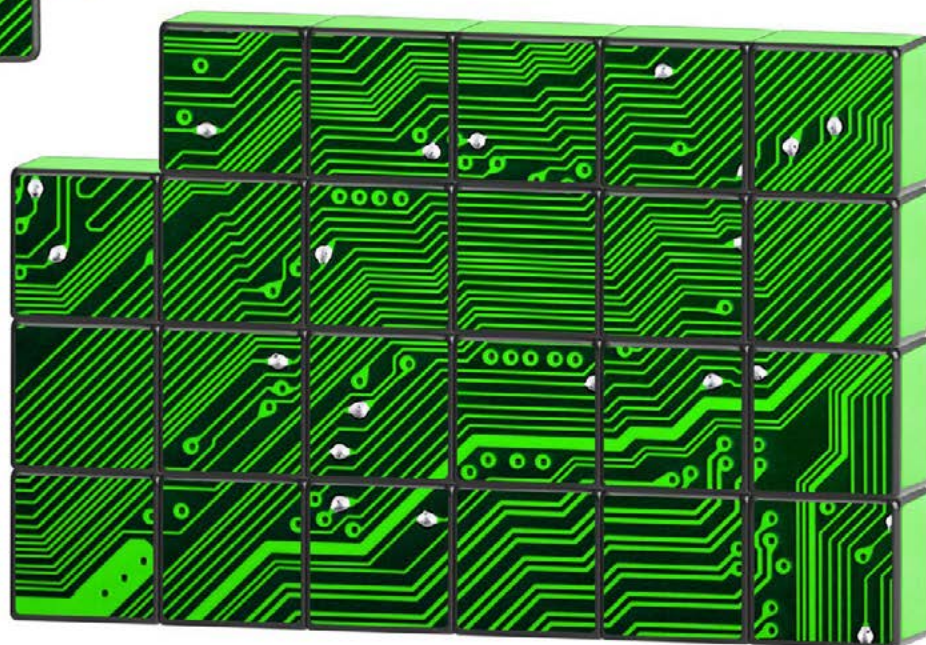
Create free account



D.M. Electronic International Co., Ltd.

What is DFM, Really?

by Mark Thompson, CID
PROTOTRON CIRCUITS



OK, so what is DFM, really? The term ‘design for manufacturability’ has been used for many years now but does everyone really understand this concept?

For instance, do you design for 10%? Do you design for a specific manufacturer’s capabilities, therefore making you less likely to seek alternative fabricators? How are your drawings worded?

In this article, I will be discussing the reality of DFM and what benefits you, the end-user, by embracing these practices.

Why Design For Manufacturability at All?

Good question. Even if you only buy your boards from a single source—if you have qualified the company already and feel you can expect certain press parameters and dielectric constants based on what they have provided you—it is STILL a good idea to at least design with some latitude. If your design is .1 mm lines and spaces there is not a whole lot of room to either expand or decrease the traces to achieve certain impedances. Clearly, when you have to ingress and egress out of tight-pitch components and your design takes you down to .003”/.003” there is NO ROOM at all for an etch compensation,

so you are typically quoted by manufacturers as quarter-ounce foil start. This foil is so thin that we need not compensate for a loss at the etcher like the other copper weights.

Again, as I have mentioned before in my [columns](#), the general rule of thumb is that for every half-ounce of starting copper, you give all the metal features an etch compensation of half a mil. Asking for 1 oz. starting copper, for instance, with .003”/.003” will normally be a no-bid as fabricators would be hard-pressed to be able to run with .002” spaces at Image prior to etch. (Attempting to compensate the .003” traces for 1 oz. copper with 1 mil will result in .002” spaces at Image prior to etch.) So, .003”/.003” is usually the limit.

While we are on the topic of etch compensations let’s talk about drill compensations. I cannot over-emphasize this: So many times lately we have issues at CAM with insufficient annular rings. This is because many customers still do not realize that, in order to keep a plated hole at a nominal size, it has to be drilled larger (this is what we mean by drill compensation) to be able to plate back down to the nominal size and have continuity. When we talk to customers about drill compensations, they typically hear only the first part: That we need to drill approximately .004”–.005” over the FINISHED nominal hole size to plate back down to NOMINAL. This

It's About Time DDR Sign Off Got Easier

Unique timing closure solution

Integration

There is a solution to your DDR sign off headaches

You can now sign off on your entire
DDR interface with total confidence.
View our free on-demand webinar
to learn how.

View Webinar ►

WHAT IS DFM, REALLY? *continues*

does not allow for any annular ring at all and most fabricators would like to see a minimum of .002" per side annular ring pad to drill compensated hole size.

Normally, what we fabricators recommend is .010" over the finished hole size for all pads and lands with holes, and approximately .015" over for anti-pads (internal plane clearances, from drilled hole to adjacent copper plane). This allows for a drill compensation of .004"–.005" plus .005" for a minimum annular ring of .002" after our plating manipulation for drills.

Speaking of Plating...

Let's talk about some of the many assumptions about plating and surface finishes. So many times we see jobs that are externally impedance-controlled, but the calculations from the customer do not match the reality of board fabrication. Remember, the holes and surfaces need to be plated to achieve the through-hole continuity to make the part function.

Many customers specify .0014" or 1 oz. finish and use that number for the purpose of calculations. First off, in order to meet IPC with a minimum of 8/10 of a mil of plated copper in the hole, normally this means fabricators will plate up in whole-ounce increments, so we have to start on a base copper whether it be copper foil or copper clad core material. So, starting on half-ounce copper and plating up an ounce results in 1.5 oz. or 2.1 mils finish.

What does this mean to you at the design stage? It means if you are calculating your impedances with a lighter finished copper weight and not considering the additional plate-up for continuity, your trace sizes will end up having to be thinner than designed. If you are already at .0039"/.0039" (.1mm/.1mm), this half a mil can mean the difference between a shop asking for +/-15% for the impedances vs. +/-10% or worse, getting no bid at all.

Design for manufacturing has so many different implications. At the design level, for in-

stance, it implies you have already done a design review, so things like differential pair spacing or matched lengths are scrutinized and drawings are updated. But sometimes things change.

An engineer may ask for additional layers for electrical reasons. Or changes to the various impedance needs may result in new revisions with all kinds of unforeseen issues. The engineer, for instance, may choose to remove some non-essential impedances or add some new ones; many times, drawings are not updated and you find your fabricator saying, "But there are no .005" traces on layer 3." Sometimes, as is the case with additional layers, the reference plane scenarios change, making the impedances impossible. Take the time to review the drawing for any notes pertaining to things that may have changed between revisions.

OK, so we have talked about etch compensations, drill compensations, impedances and drawing notes. Now, let's talk about DFM as it relates to profitability. Yes, that is another reason to design for manufacturability. Not taking the time to do a thorough design review prior to release to a manufacturer may result in a phone call from the fabricator and lost time. If you are on a particularly tight time schedule, this impact can reach thousands or even hundreds of thousands of dollars!

But I know what you're thinking; after spending years in CAM, I can almost read your mind! You're probably thinking, "With all the fabricators in the world that perform an analysis to air out any issues prior to quoting my job, why should I take the time?" Because NOT taking the time limits the type of fabricators who can do your job. (Not all fabricators can perform a full review to catch all possible CAM anomalies prior to quoting your job.) Some fabricators do a cursory review to make sure the basics are covered: The impedances are possible, and the drill files match the drawings. Additionally, should the CAM department catch something in pre-analysis, it will still need to be fixed.

"With all the fabricators in the world that perform an analysis to air out any issues prior to quoting my job, why should I take the time?" Because NOT taking the time limits the type of fabricators who can do your job.

ventec



8 1/2 Reasons You **WIN** with Ventec



Show Me



#1



#2



#3



#4



#5



#6



#7



#8



#8 1/2

WHAT IS DFM, REALLY? *continues***What Else Should be Considered for DFM?**

What other issues should we focus on for DFM? Well, the IPC netlist is a good one. Again, let's say you have made a revision to a part and added layers, but have neglected to run a new IPC netlist. The fabricator will be calling you, again, resulting in lost time.

Common netlist mismatches that require clarification from the fabricator are issues like these:

a) AGND to DGND short. Many times an intentional short is by design.

b) Castellated holes (plated half holes at a part edge). Here they clearly make a connection to a post sometime later in their lifecycle. From a fab standpoint, we come up with erroneous "broken" or open nets.

c) Non-plated holes defined as net points.

d) Undrilled surface mounts defined as net points. Time can be saved by making note of any netlist issues on a read me or even on the drawing.

In conclusion, why should you embrace DFM? Ultimately, designing for manufacturing and allowing process tolerances will enable your design to be built by more fabricators, allowing for more quotes from more fabricators, increasing your supply chain and saving you money.

Additionally, carefully reviewing designs for ease of manufacturing also means less time spent on the phone with your fabricator resolving these issues, which also saves time and money.

As always, I appreciate your time. If you have any comments or questions, feel free to contact me. **PCBDESIGN**



Mark Thompson is in engineering support at Prototron Circuits. His column, *The Bare (Board) Truth*, appears bi-monthly in The PCB Design Magazine. To read past columns, or to contact Thompson, [click here](#), or phone 425-823-7000, ext. 239.

video interview**Gary Ferrari Talks Design Certification**

*by Real Time with...
Designers Forum*



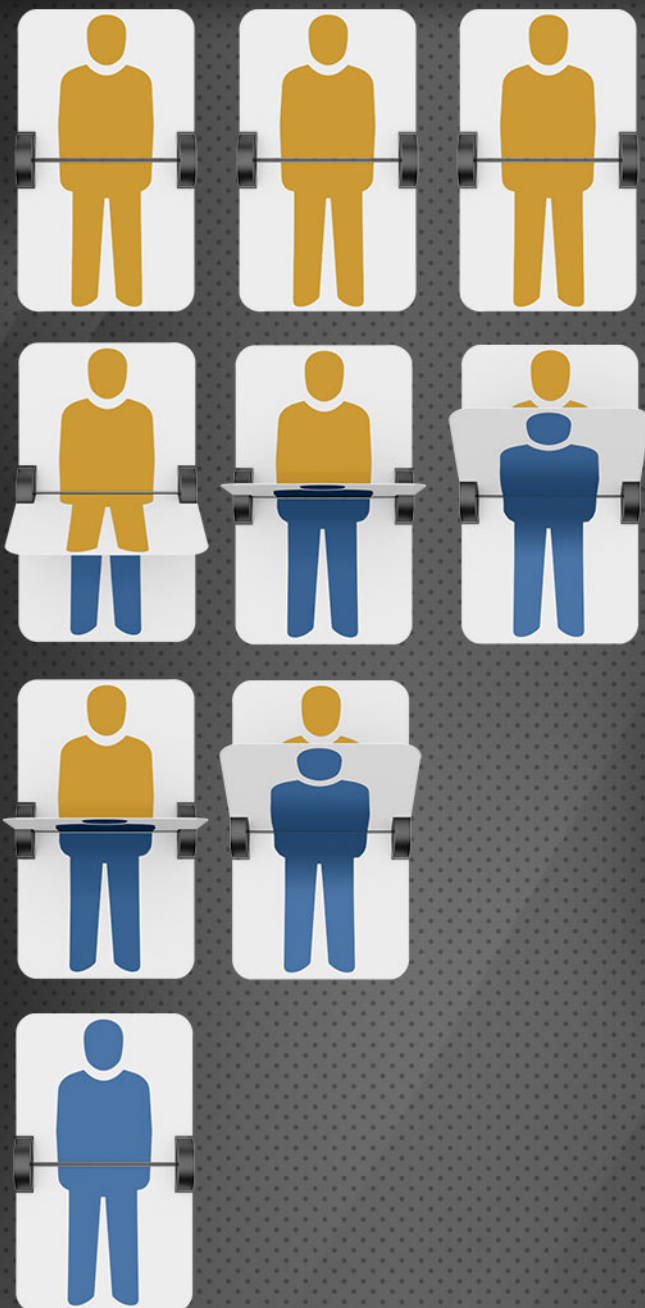
Gary Ferrari of FTG discusses his efforts with IPC's Certified Interconnect Designer training program during IPC APEX EXPO, as well as his work on IPC standards committees covering topics such as HDI.



realtimewith.com



More and More PADS Users are Switching to **Altium Designer**



Upgrade to the Next Generation

Switching to Altium Designer is Easy



Get the Benefits of Switching Today

PCB007

News Highlights



Q1 North American PCB Market Snapshot

More than 370 industry professionals answered the call to take I-Connect007's first-ever quarterly market survey. This one-minute survey is designed to provide a snapshot of the current state of the PCB and EMS industries. Based on the results so far, we believe, with your help, we've been able to capture a fairly accurate picture of the current state the market.

IPC Honors Volunteers for Contributions to the Industry

IPC presented Committee Leadership, Distinguished Committee Service, and Special Recognition Awards at IPC APEX EXPO at Mandalay Bay Convention Center in Las Vegas, Nevada. The awards were presented to individuals who made significant contributions to IPC and the industry through IPC committee service.

Advanced Circuits Gears Up with Orbotech's LDI System

Advanced Circuits has purchased Orbotech's Paragon-Xpress LDI system for use in the manufacture of key components for the commercial PCB arena as well as the high reliability, military/aerospace/defense and high-technology marketplace.

Replace Complex Wiring with Rigid-flex PCBs

Complex wiring needs space, which is often not available. The solution has arrived: Rigid-flex PCBs. They are the result of a creative interdisciplinary collaboration between the drive manufacturer WIT-TENSTEIN in Igersheim and Würth Elektronik in Niedernhall.

FTG's Circuits Segment Sales Up \$1.6M in 1Q14

"The momentum from the end of 2013 has continued into the start of 2014 with strong results across the company, particularly at our two new aerospace facilities in Tianjin and Chatsworth," stated Brad Bourne, president and CEO. He added, "Our established Circuits facilities both performed well in the quarter and we are working hard to get our Circuits Joint Venture through its start-up and customer qualification phase so it too can contribute to our success in the future."

Viasystems Expects 10% Sales Increase in 1Q14

Estimated net sales for the quarter ended March 31, 2014 are expected to be in the range of \$290–\$300 million, representing a year-over-year increase range of 6% to 10% compared to the first quarter 2013, and representing a seasonal sequential decrease from the \$303 million net sales reported for the preceding quarter.

Sunstone Circuits Among Top Places to Work in Oregon

"[We are] thrilled to be one of the Top Places to Work in Oregon," said Rocky Catt, COO. "We want Sunstone employees to feel they are part of an organization that's moving forward in the right direction, that their everyday contributions are valued by management and that the work they do is a part of something meaningful."

PCI Acquires Microcraft Flying Probe Test Machine

The Microcraft flying probe machine will enhance PCI's technical capabilities for higher density flexible circuits, and at the same time will improve lead time and lower cost on smaller quick-turn builds. Eliminating the cost and time to assemble fixtures is a major improvement for the quick-turn prototyping environment.

Global PCB Manufacturing Market: \$74.31B in 2018

The global PCB manufacturing market is expected to increase its market size from approximately \$62.3 billion in 2013 to near \$74.31 billion in 2018, growing at a CAGR of 3.6%. The market volume is also expected to increase to 32 billion units and 3.92 million tons by 2018, growing at a rate of 3.8% and 5%, respectively.

TTM Trims Q1 Forecast on Foreign Exchange Loss

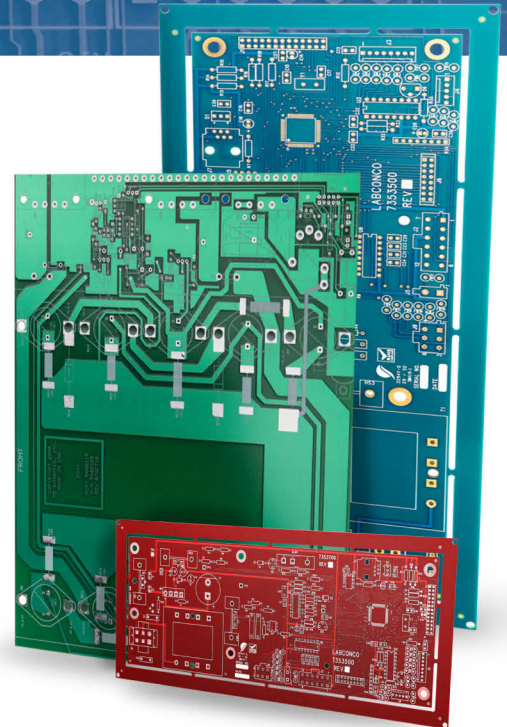
While operating performance was consistent with its guidance, the company expects to incur an unrealized, non-cash foreign exchange loss of approximately \$3.6 million, (\$0.03) per diluted share, in the quarter due to the rapid depreciation of the Chinese RMB against the U.S. dollar.

Quality PCBs from the **established** industry leader

With over 40 years of experience delivering high quality PCB prototypes, Sunstone Circuits® is committed to improving the prototyping process for the design engineer from quote to delivery.

We lead the industry with an on-time delivery rate of over 99%. Plus, our on-site technical support is available every day of the year (24/7/365), giving Sunstone unparalleled customer service.

**Get a quote instantly
at Sunstone.com**



- Live customer support 24/7/365
- Over 99% on-time delivery
- Best overall quality & value in industry
- In business for over 40 years
- Online quote & order
- Free 25-point design review
- RF / exotic materials
- Flex / Rigid-Flex boards
- RoHS compliant finishes
- Free shipping & no NREs
- PCB123® design software
- Controlled impedance testing



Questions? Sunstone Technical Support is always open: 1-800-228-8198 - support@sunstone.com

CLOSING THE GAP BETWEEN DESIGN AND FAB



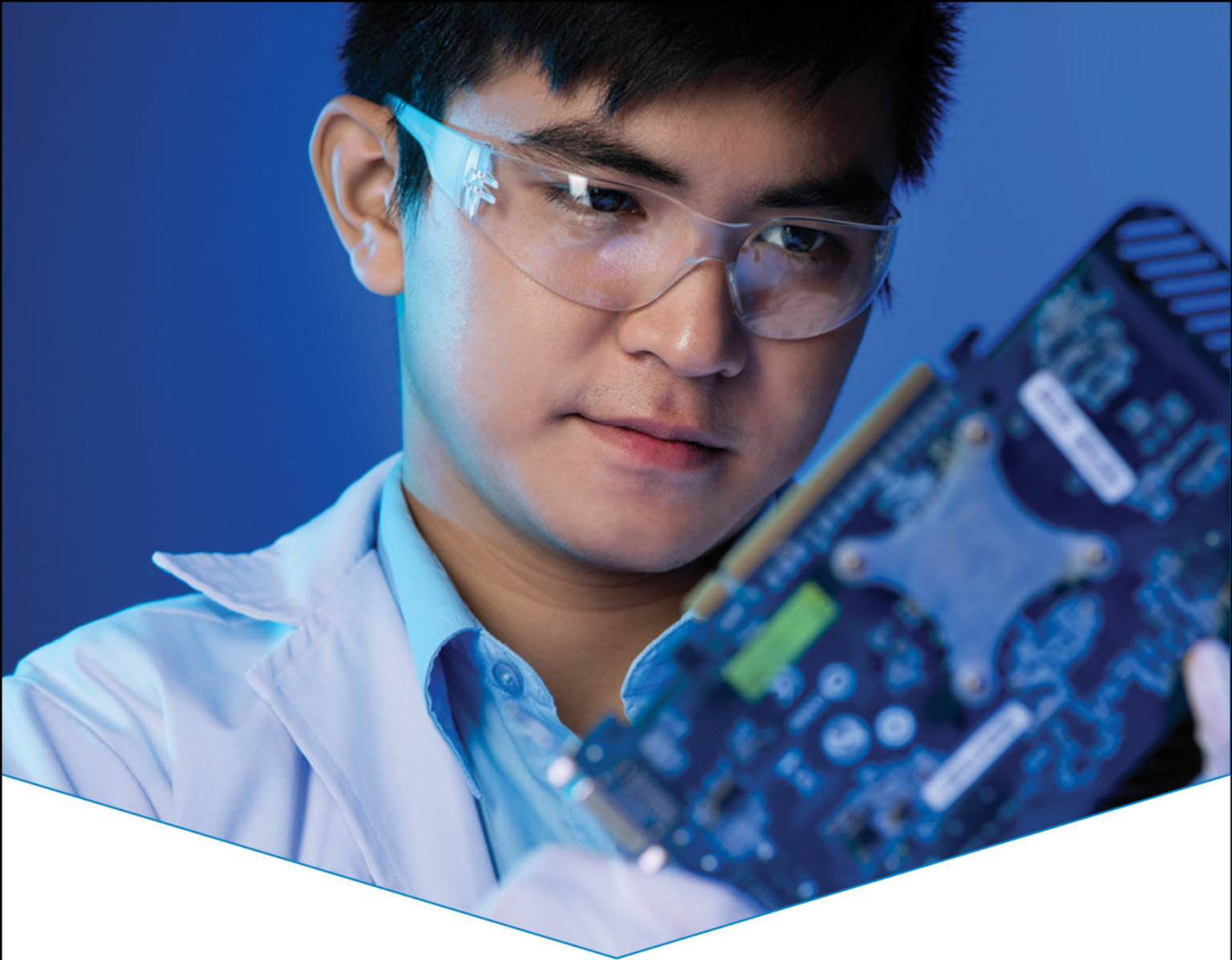
by Ben Jordan
ALTIUM

Recently, I've been doing a bit of investigation to find out what really needs to happen to get a good turnaround on designs—from idea to a working board, that is. There is a more pressing need than ever for ECAD vendors to do some close work with PCB fabricators, but it seems that ECAD vendors have a complete disconnect. I think that CAD tool vendors in general have done a fantastic job of making design easier and faster, but there is still a huge gap between what a PCB designer intends, thinks, and lays out in CAD and what a PCB fabricator needs in order to build and assemble the boards with reasonable yields. It's one (good) thing to support universal data transfer formats, but that's not an automatic fix-all, either. What I'm really talking about is DFM.

Over the years, IPC has made efforts to fix this disconnect. This is why the current CID and CID+ programs have an emphasis on DFX. For those unfamiliar, DFX means “design for

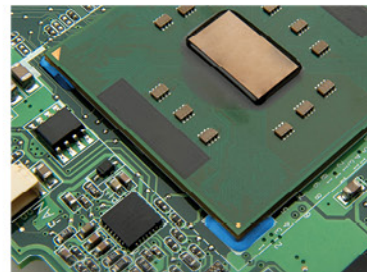
x,” where x = manufacturing, assembly, and test—together we call it Design for eXcellence. The thought process here is that if PCB designers are trained to know the materials, processes, steps and limitations of a PCB fab, then they will inherently know how to design the PCB to be “manufacturable.” Part of this also is the classification of PCBs based on complexity and producibility levels.

Of course, the designer then has a new job as a kind of mediator between engineering management, product marketing, and the fab: having to laboriously explain why the product they are working on cannot be designed the way marketing wants it, or why moving a button one inch to the left will cause the product assembly costs to significantly rise. It's a big ask for designers to have to bear in mind all the fabrication and assembly processes, and then have to turn around and put these practical limitations up against the wants of other stakeholders. If I were hiring designers, I would limit my search to CID-certified people for this reason alone. Yet, in a sense, this is like having a li-



IT'S HIGH-PERFORMANCE RUGGEDIZATION. ONLY WITH MUCH HIGHER EFFICIENCY.

Dymax 9309-SC is the only light-curable edgebond material with patented See-Cure technology, giving you the speed and efficiency of visual cure validation with outstanding shock and impact resistance. And by holding its shape, 9309-SC minimizes capillary action and prevents material leaks. All this, backed by the Dymax Edge...an integrated offering of oligomers, customized adhesives and coatings, cure and dispense equipment, and application expertise to deliver the best solution possible. Visit www.dymax.com/ruggedize to request a product sample.



CLOSING THE GAP BETWEEN DESIGN AND FAB *continues*

cense to drive with little or no actual driving experience.

It counts a lot to actually go to a PCB fab and take a tour, and fortunately most fabs with oil in their lamps are willing to oblige if you give them enough notice. I have done this myself, recently spending a fair amount of time with a few fabs near me in California. And taking tours of real PCB fabs and interrogating them has confirmed my suspicions.

I can illustrate the sorts of problems that can occur between design and manufacturing with a tangible real-world example. On a visit to Sierra Circuits in Sunnyvale (also known as ProtoExpress.com) I got to walk through the entire process of preparing and laminating a rigid-flex panel “book.” It was an eye-opening experience—not only for getting to see how a rigid-flex board is made, but just as much to be made aware of the fabrication process and its limitations. This was a particularly interesting example, because the PCB in question was very

small and had to have thin flexible sections—about 3mm in width—between the rigid sections, each about the size of a U.S. quarter. The final board prototype is shown in Figure 1.

The big deal with this board was that with such narrow flex-circuit sections, it was not possible to use “bikini” coverlay for the flex. This meant that polyimide coverlay film had to extend through the entire lamination of the board, which in turn meant that acrylic adhesive layers had to be used to adhere the polyimide coverlay layers to the rigid cores. This may not seem like a problem, except that the PCB is four layers in the rigid areas, and with such densely populated components on top and bottom layers, the vias were on the risky side of smallness. Why? Because the adhesive layers are known to expand in the Z-axis during solder reflow.

In other words, the fab already knew that this board was going to have lowered yields in assembly caused by cracking vias, in turn caused by adhesive expansion, caused by the need to

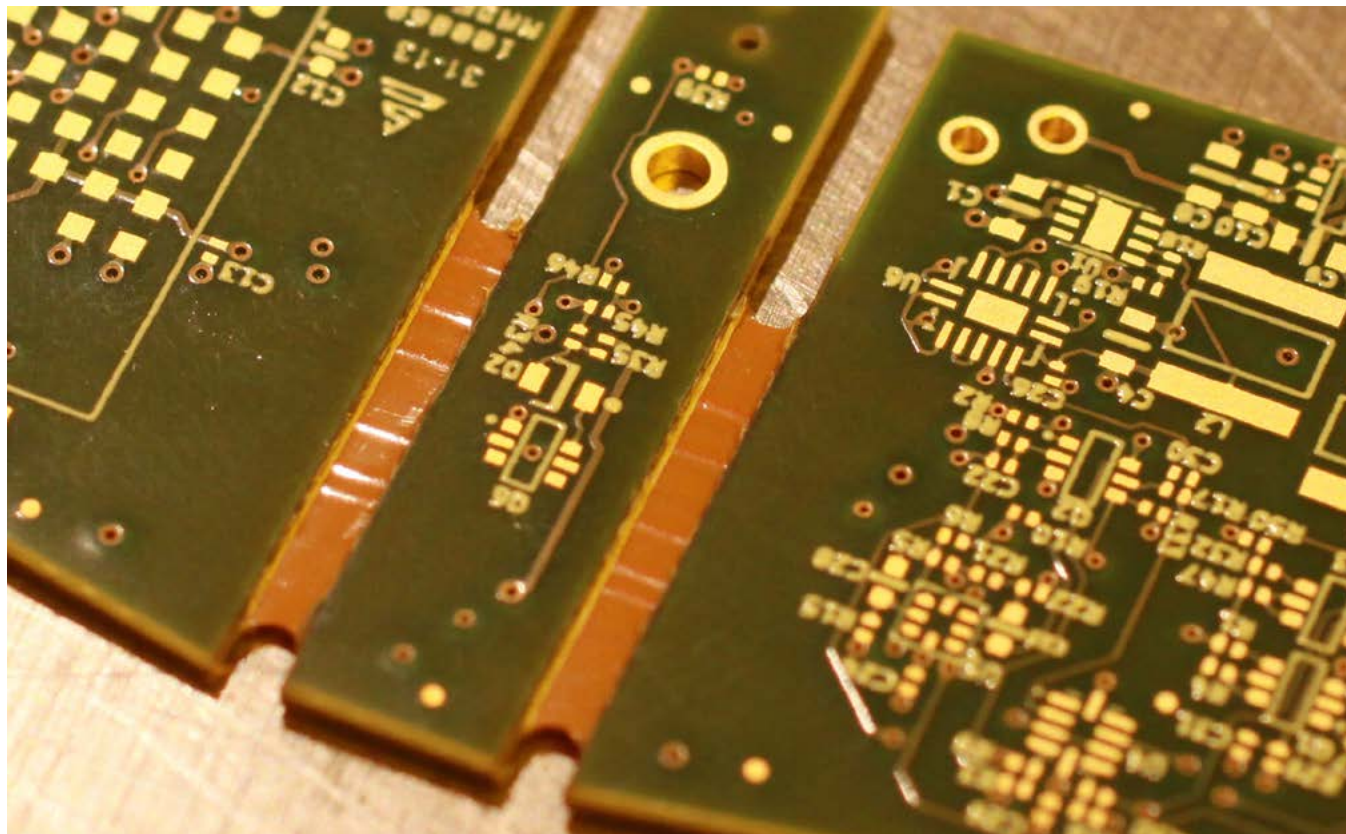


Figure 1: Difficult-to-make rigid-flex board.



Engineering And Providing Balanced Interconnect Solutions



Flexible Printed Circuits • Rigid Flex • Wire Harnesses • Complete Assemblies

Our products and services are specifically designed to empower our customers to produce a product with superior performance and quality, as well as lower design, assembly, and system installation costs.

Partnering With Your Engineering Team To Deliver The Best Solution

...connecting technology

**Miraco, Inc. • 102 Maple Street, Manchester, NH 03103 • miracoinc.com
603-665-9449 • Fax: 603-665-9459**

CLOSING THE GAP BETWEEN DESIGN AND FAB *continues*

Figure 2: White acrylic adhesive layers are added to the lamination book.

have coverlay film within rigid sections of the board, caused by the size constraints of the design and its mechanical form factor. They actually advised the designer about the problems, and the risk, but in this case the designer was out of time and really could not easily redesign for better assembly yield and maintain the necessary product form factor. This falls into the age-old trade-off between form (a largely marketing driven design aspect) and manufacturing cost (i.e., scrap). In this case, the prototypes were needed quickly, so the designer requested production in spite of the associated risk.

Sierra is an example of what I call an educator fab—these are the guys who will tell you what is wrong with your design so that you can try to get it right next time. But there is still a level of frustration for them, as it seems like the designers rarely perform comprehensive DFM checks on their designs. I'm told that it is still most common that designs are at first rejected for DFM issues when the fab runs their own

DFM checks using their own CAM/DFM tools.

Another type of fab, such as Hughes, in San Marcos, California, is just as likely to request the original source documents from the PCB designer in the native CAD format. I refer to these guys as a fix-it fab. You could say part of their service is to make the changes necessary to your design to improve the manufacturing yield and lower costs—optimizing your design to their processes and equipment. Most high-volume fabs would want to work with the designer in similar ways to ensure low waste, but if you're working with fabs on distant shores this may not be desirable. As an example, I asked Hughes how they need masks to be generated in the CAD software; their response was a firm, "Don't; let us do that for you in CAM."

From all the PCB fabs I've talked with, there is an overwhelmingly common message: Even though PCB designers have access to DFM checks and tools, there's no easy industry standard way for the PCB designer to fully run DFM

checks before hand-off. This common complaint from the fabs is that designers still largely don't hand over manufacturing-ready designs.

In response, it would be easy to ask, "What about constraint-driven design?" It's a good question. I think that a designer having a good understanding of the constraints and then being able to run design rule checks against the fabrication constraints is an essential part of the solution. The problem with this, however, is that you often have one design, but multiple fabricators who may use different equipment. Of course, the answer to that problem appears to be well-accepted transfer data formats such as IPC-2581 or ODB++. IPC-2581 has promise for streamlining the data handoff, but it still doesn't address the basic problem that, from the outset, designs often begin with too little awareness of how the boards will actually be made once the design is finished.

When discussing these issues with Sierra, Hughes and others, I received a surprisingly overwhelming response that "ECAD tools don't have proper DFM checks in them." Frankly, the first time I heard that I was taken aback. Really? Yet when I probed deeper, I discovered that the DFM checks referred to are available in the ECAD tool's design rule check engine. The real problem is actually that the designer either does not apply them, or applies them without properly defining the constraints according to their fab's capabilities.

For example, let's consider silkscreen ink being allowed to print over bare copper. As a test, I sent a design that had a small bit of silkscreen designator text over an untented via (a via with no soldermask over the copper lands) to three different fabs: an educator fab, a fix-it fab, and a broker for offshore fabs. The educator fab immediately informed me that I had silkscreen over a mask opening and it had to be fixed in my design. The fix-it fab asked for the files in the native (Altium Designer or Protel) format, and the broker simply sent the Gerbers to a fab, which eventually emailed me informing me that some silkscreen items would be removed because of overlap with mask openings. In all three cases, at a minimum, the production date would be delayed. So what are the options for a solution?

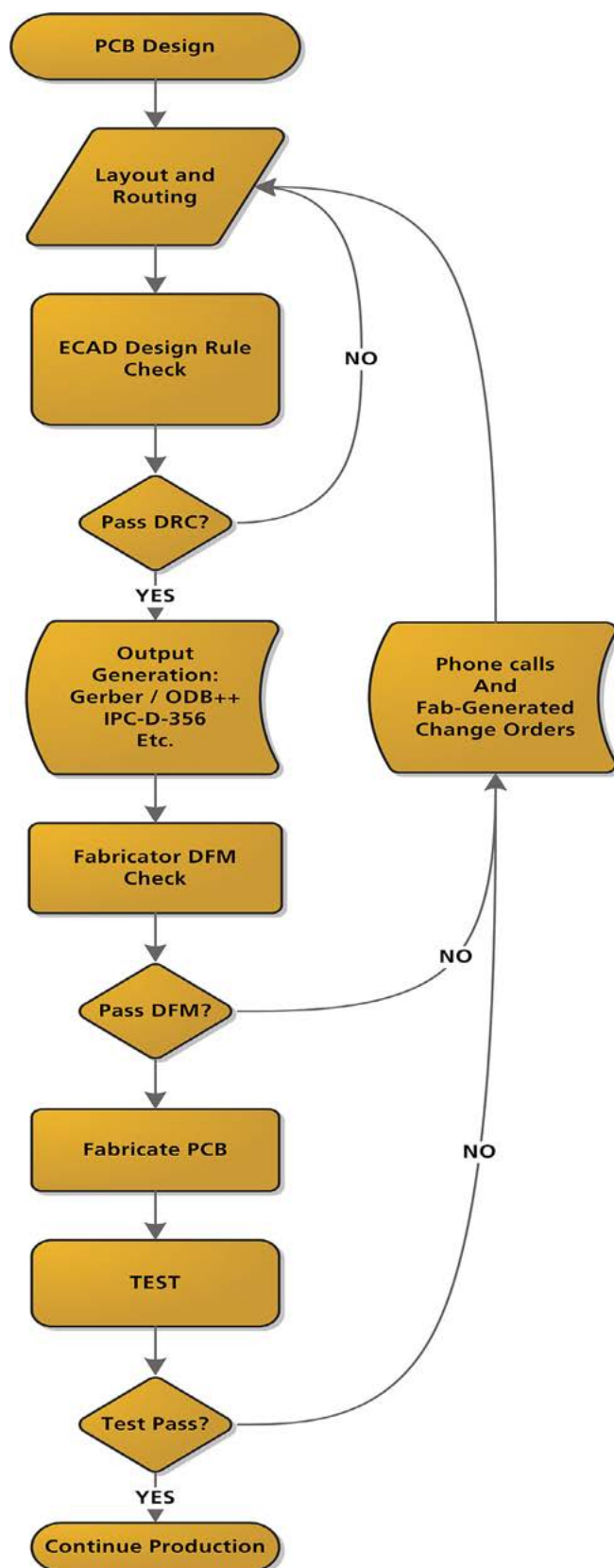


Figure 3: The usual PCB design-to-fab flow looks connected, but there's plenty of room for error.

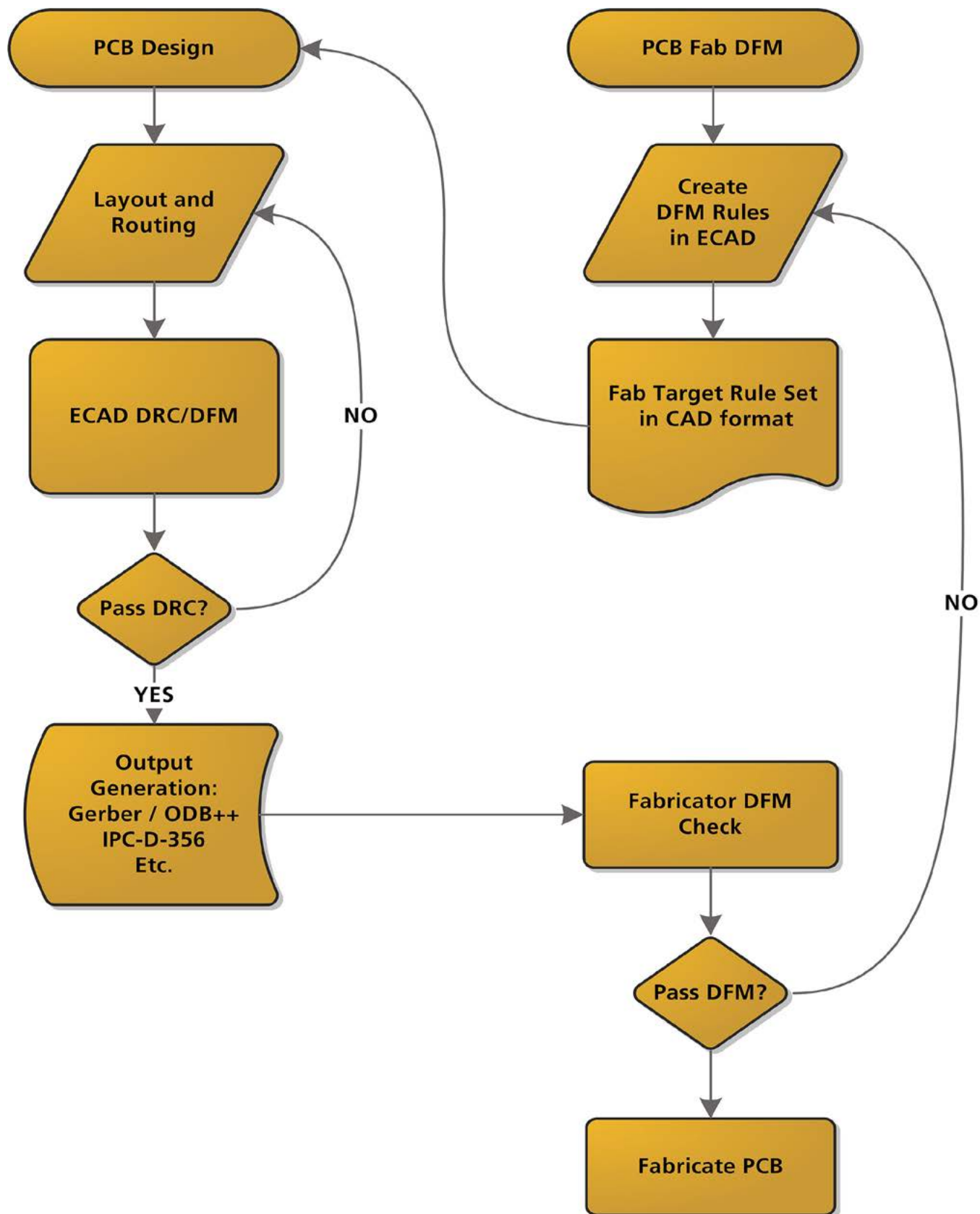
CLOSING THE GAP BETWEEN DESIGN AND FAB *continues*

Figure 4: A better way: Involve the fabricator early with DFM.

One option is to develop some standard, basic DFM checks which all designers can handle for simple to moderate (Level 1 and 2 producibility) designs. In this case, an industry body could publish a list of DFM checks and their tolerances which could easily be ported into any ECAD design rule check engine, so designers could efficiently apply them and have confidence that most fabs would successfully produce the boards. This sounds good, but it's actually quite a difficult thing to achieve—the number of constraints that would have to be thought of ahead of time is staggering.

Another option is for the fabricators to be more involved in ECAD tool development. While some have actually used free PCB design tools as a loss leader, it is a very difficult thing to make tools which can handle cutting edge or even some mainstream designs efficiently. I would go as far as to say it could financially detract too much from the core business of the fabricator. Personally, I would prefer the fabs to put money into expanding their production lines with newer equipment to tackle the next generation of products. But I do think for DFM, we might actually be onto something here...

A better solution space for this problem then, the best in my opinion, is for fabricators to have technical staff actually involved in design planning and in setting up the rules and constraints as early as possible in the design process. The difficulty on the face of this solution is that you may have to know up front who your fab is. Most companies tend to use the same fabs over the long haul so it's not such a big deal. This can be further eased by the fabs. The fabs have a great opportunity to improve business here and I can see at least a few ways of making this easy for designers to really get it right the first time:

At the simplest level, the fabs need to get into each ECAD toolset, and produce a pre-packaged set of DFM rules (or constraints) for each production line or process complexity level they support. Then, those DFM rule sets can be published or made as a download from the web for the customers who are working with that fab. In turn, the designer can then run a DFM check which gives a high confidence when everything passes, that the fab they are using will be able to get good yields with their design.

Going a step further, PCB fabs could offer whole template projects for various toolsets as free downloads for their registered customers, which in a “shrink wrap” include the various DFM checks for their production lines.

Take this one step further again: ECAD tool vendors like Altium could make the API for the software available to the fab, who can build their own quoting and DFM check systems into the ECAD tool, so the designer who intends to use that fab can make sure their design is good with the click of a button and even receive a quote from the fab for prototyping.

There's a lot that can be done here. The third solution requires the fabricator to have the necessary software infrastructure in place and secure links between ECAD and online DFM servers. However it adds two distinct advantages. The first is that the PCB fabricator can run full DFM without exposure of their processes, and the other is that the designer can run DFM checks against a real fab's processes without other humans interacting with their design source documents.

For these problems to be solved in a way that best suits the designer, integration is really needed. Closing the gap between design and fabrication needs reliable and universal data transfer, yes, but above all, it needs good collaboration between all parties involved. Having good file formats is important, but it is not the complete solution. While work has been done with ODB++ and IPC-2581, the uptake is slow—ODB++ has less than 20% use according to the fabs, and IPC-2581 is in its infancy. But if we attack the problem with a one-two punch of good data format combined with early, integrated and direct involvement from fabricators themselves, then lots of wasted time and materials can be avoided. **PCBDESIGN**

This article originally ran in the [February issue of The PCB Magazine](#).



Ben Jordan is a senior manager at Altium. To contact him, [click here](#).

Transmission Lines: From Barbed Wire to High-Speed Interconnect

by Barry Olney

IN-CIRCUIT DESIGN PTY LTD

Long before Facebook and Twitter, there was a more primitive type of social network. Hailing from the Old West, it allowed distant communities to meet remotely to share music, spread news and to just gossip. The non-proprietary, ad hoc network was an unwitting model of democracy and free speech. Unfortunately however, it collapsed—overwhelmed by commercial pressure. The long-forgotten social revolution, and extremely basic technology, was built on barbed wire fences.

Barbed wired fences appeared in the United States in the 1860s and their success, in the control of cattle, swept across the North. However, the South, fearing that the product may harm their cattle, was hesitant to buy the fad at first but ultimately succumbed. Ironically, the barbed wired fence is associated with the cowboy, but unfortunately, it also sounded their end.

As with all communications systems, getting connected, particularly in remote areas, is always a challenge. In the early 1900s, the Bell Telephone company was focusing all efforts on connecting urban areas and like the telephone companies of today, had little interest in connecting remote communities, due to the cost of the infrastructure.

However, an enterprising rancher figured that the West was already sprawled with wire—barbed wire—and discovered that if you hooked two Sears or Monkey Ward telephone sets to a barbed wire fence, he could talk between the telephones as easily as between two city telephones connected via an operator's switchboard. A rural telephone system that had no operators, no bills—and no long-distance charges—was born.

But that lack of broader connectivity eventually doomed the ad hoc network. The commercial phone system's ubiquity, and especially their coveted connection to distant cities, eventually dominated. By the 1920s, the barbed wire telephones and the networks they helped spawn had disappeared.

By comparison, today's long-distance digital communications operate over various types of media including coax and twisted pair cables. And multilayer PCBs also transfer data via microstrip or stripline transmission lines. Surprisingly, all of these different types of media can be modeled using the same equations. The Telegrapher's Equations accurately model



Figure 1: Barbed wire fence (courtesy of fenceworkshop.com).

We are Committed to Delivering Outstanding Quality and Service, On-Time.

AND OUR CUSTOMERS AGREE!

Dear Prototron Team


Thanks for your continued performance and quality!! We have had 35 engineering designs fabricated by Prototron since 2012 and we are very happy. Your personal contact, knowledgeable staff and commitment to on time delivery continues to allow us to meet our commitments to our customers. Prototron's engineering department has been extremely helpful on recommending improvements to new PCB designs. You guys are great!!

Thanks,

Fred Sievert
Aeroflex Colorado Springs

[**Click to Learn More**](#)



Prototron
 **Circuits**
Quality printed circuit boards

FASTEST
America's Board Shop
www.prototron.com

TRANSMISSION LINES *continues*

Figure 2: Broadcom Fast Ethernet 100Base-T4 over barbed wire (courtesy Broadcom).

the propagation of signals along the media providing the following:

1. A well-defined uniform path that exists for the flow of both signal and return current.
2. Conductors that are closely spaced by comparison to the wavelength of the signals.
3. Conductors that have a long length, compared to the space between the conductors.

In 1995 at the Interop Expo, Broadcom engineers demonstrated their T4 Ethernet chipset operating over the worst possible cable—barbed wire. During the show, 100Mbps of data was successfully transferred through rusty barbed wire. Figure 2 shows the setup. Wideband Corporation later effectively demonstrated 1Gbps over barbed wire with their transceiver. Providing the impedance and delay are constant along the length, the dielectric loss is low and the crosstalk is low (due to the large space between the pairs), the performance is not impaired. I wonder if razor wire would make it perform faster, as it is more cutting-edge.

You may wonder how this actually works, since it is obvious that two uninsulated twisted wires would touch. The trick is that they used rusty barbed wire; the ferric-oxide coating acts as an insulator and the twisted pair looks like

100 ohms of impedance. So, if you try this at home, don't use new wire.

Telegrapher's equations are a pair of linear differential equations that describe the voltage and current present on a transmission line relative to distance and time. Oliver Heaviside developed the transmission line model in the 1880s. Remarkably, the theory still applies to transmission lines, of all frequencies, including high-frequency transmission lines, as seen in the multilayer PCB. The telegrapher's equations have many derivatives and the math employed is beyond the scope of this column, not to mention beyond the ability of the author.

Contrary to common belief, the transmission line does not carry the signal itself but rather guides electromagnetic energy from one point to another. It is the movement of the electromagnetic field or energy, not voltage or current that transfers the signal. The voltage and current exist in the conductor, but only as a consequence of the field being present as it moves past.

The ICD Stackup Planner^[5], in Figure 3, illustrates the three most common transmission line structures of a multilayer PCB. For embedded microstrip (solder mask coated microstrip), the electromagnetic field propagates partially in the dielectric material (Isola 370HR), the solder mask and the air. Whereas, in both stripline structures, the electromagnetic field propagates in the dielectric material sandwiched between the planes. The traces simply guide the signals

UNITS: mil				5/6/2014				Total Board Thickness: 62.3 mil					
				Differential Pairs >		50/100	USB						
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	3.8	0.5							
1	8	Signal	Top	Conductive			1.4	15	5	0.37	49.97	97.35	Embedded Microstrip
		Prepreg		370HR : 1080 : Rc= 66% (5GHz)	3.72	2.9							
2		Plane	GND	Conductive			0.7						
		Core		370HR : 1-1652 : Rc=51% (5GHz)	4.0	5.3							Symmetric Stripline
3		Signal	Inner 3	Conductive			0.7	12	4	0.19	49.75	98.29	
		Prepreg		370HR : 1652 : Rc= 48% (5GHz)	4.09	5.3							
4		Plane	VDD	Conductive			0.7						
		Prepreg		370HR : 2116 : Rc= 56% (5GHz)	3.92	4.8							
5		Signal	Inner 6	Conductive			0.7	12	4	0.19	51.22	100.76	Dual Asymmetric Stripline
		Core		370HR : 1-2113 : Rc=46% (5GHz)	4.1	3							
6		Signal	Inner 7	Conductive			0.7	12	4	0.19	48.83	96.25	Stripline
		Prepreg		370HR : 3313 : Rc= 59% (5GHz)	3.86	4.1							
7		Plane	GND	Conductive			0.7						

Figure 3: Embedded microstrip, symmetric and dual asymmetric stripline configurations.

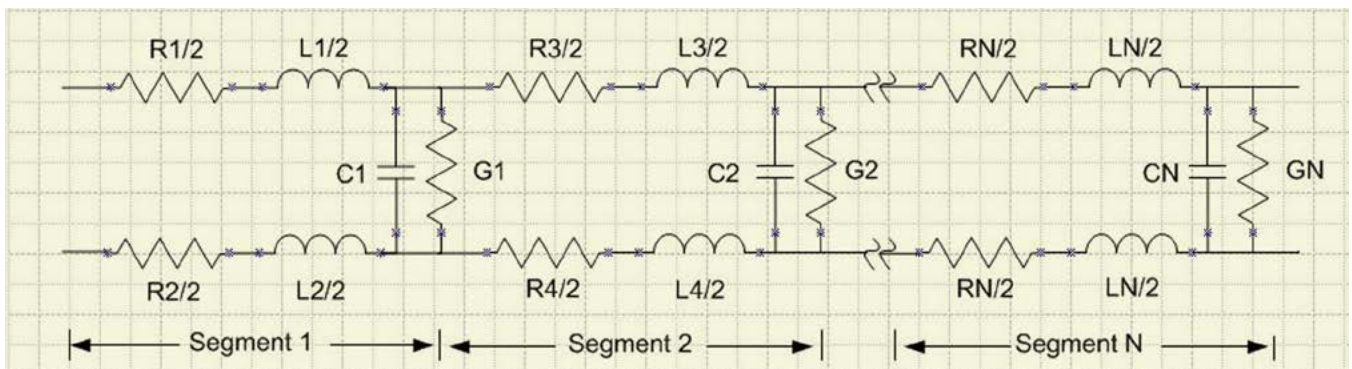


Figure 4: Transmission line represented by a series of R-L-C-G elements.

wave as the electromagnetic energy propagates in the dielectric material.

So the dielectric material determines the velocity (v) of propagation of the electromagnetic energy:

$$v = \frac{c}{\sqrt{Er}} \quad \text{Equation 1}$$

Remember that c is the speed of light (in free space) and Er is the dielectric constant of the material (FR-4 is ~4.0). By contrast, the Er of air is 1. Therefore, the velocity of propagation in FR-4 is about half the speed of light, or 6 inches per ns. The important concept is that it is the electromagnetic energy that propagates down the transmission line—not electron flow. Electrons flow at about 0.4 inches per second, a snail's pace in comparison.

A transmission line can be represented by an infinite number of segments, incorporating series resistive (R) and inductive (L) elements with shunt capacitive (C) and conductive (G) elements, as in Figure 4. And because of the restricted velocity of propagation in the media, the signal does not know what the termination is at the end of the line. It can only see the impedance of the line. The impedance of the line can be represented by:

$$Z_o = \sqrt{\frac{R + j\omega L}{G + j\omega C}} \quad \text{Equation 2}$$

$$\omega = 2\pi f$$

If we assume that the transmission line is lossless—which occurs at frequencies below a few hundred MHz—then the R (conductor loss)

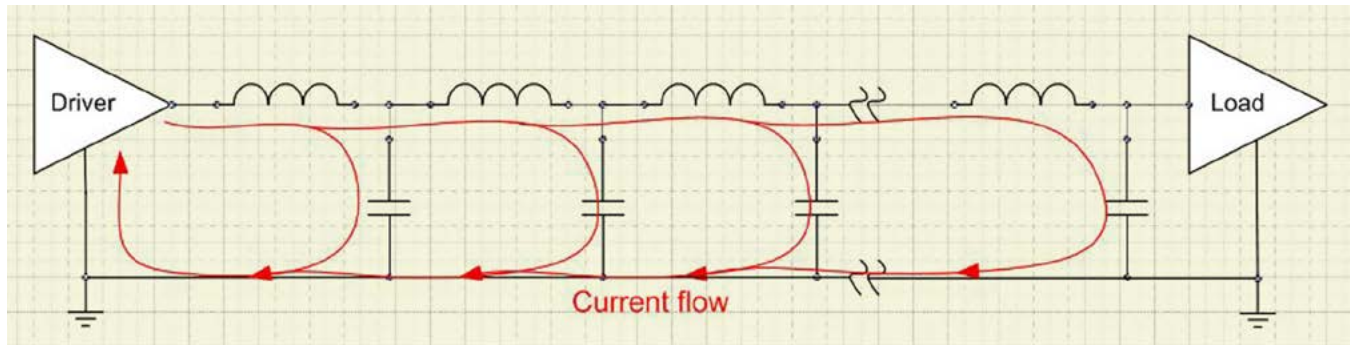
TRANSMISSION LINES *continues*

Figure 5: Model of a lossless transmission line indicating current flow.

and G (dielectric loss) terms can be assumed to be zero simplifying the equation to:

$$Z_0 = \sqrt{\frac{L}{C}} \quad \text{Equation 3}$$

The characteristic impedance (Z_0), of an ideal transmission line, remains constant at all frequencies. It has no imaginary part and is not frequency dependant. As the electromagnetic energy propagates down a transmission line, current is induced into the conductors as illustrated in Figure 5. The current flows along the conductors, charging the first section's parasitic capacitance and then flows back, on the return conductor (reference plane), to the source. The current exists only on the rising edge of the propagating wave and thus charges each section's capacitance as it moves down the line with the return current, from each section, flowing back to the source. Then as the pulse passes, the falling edge discharges each section's capacitance in turn. By the time the signal wave reaches the load, it has established multiple paths of return current along the PCB planes.

Current always flows in a loop. But it does not flow along the transmission line to the end (load) and then return to the source as one would expect with DC. But instead, the signal charges each section in turn as the rising edge propagates along the transmission line on the dielectric material.

Although a lossless model is a good representation of a typical low-frequency transmission line, at high frequencies, the conductor and dielectric losses need to be considered. For this reason, the closed-form equations shown here are approximations only and a 2D field solver, such as that integrated into the ICD Stackup Planner is required to accurately model the impedance of a multilayer PCB transmission line.

Points to Remember

- In the early 1900s, barbed wire fences were used as an ad hoc, rural telephone system that had no operators, no bills, and no long-distance charges.
- Telegrapher's equations still accurately model the propagation of signals along transmission lines.
- In 1995, Broadcom demonstrated its 100Base-T4 Ethernet chipset operating over barbed wire.
- Wideband Corporation later successfully demonstrated 1Gbps over barbed wire.
- A transmission line is a series of conductors that guide electromagnetic energy from one point to another. It is the movement of an electromagnetic field or energy—not voltage or current.
- The three common transmission line structures of a multilayer PCB are: embedded microstrip, symmetric, and dual asymmetric stripline configurations.
- The velocity of propagation in FR-4 is about half the speed of light, or 6 inches per ns.

There's a whole lot of quality under this roof.

We're proud of our commitment to quality. Delivering the best design, fabrication, assembly and test is what we're all about.

Certifications

Facility

- ITAR Registered
- ISO AS-9100
- ISO 9001
- NADCAP Certified, bare board mfg and assembly
- ISO Medical 13485 Certified
- RoHS & Lead Free Compliant
- FOD Program
- Lean manufacturing and 5S Program
- COMSEC & KLIF account

Printed Circuit Board Manufacturing

- IPC Industry Standards, Class 3
- MIL-PRF-55110G, approved for FR-4 & Polyimide
- MIL-P-50884E Flex Rigid/Flex approved

Circuit Card Assemblies

- J-Std-001 & IPC-610 Class 3 Certified Operators
- ESD Training Program
- ESD Controlled

Awards

- 2011 Raytheon SAS Recognition Award
- 2011 Sandia Plata award
- 2011 IDS 4-star award
- 2012 SAS 3-star award
- 2012 IDS 5-star award



Murrietta Circuits

The Integrated Solution

sales@murrietta.com | www.murrietta.com | (714) 970-2430

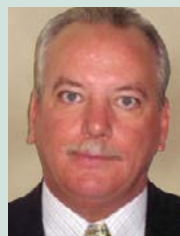
TRANSMISSION LINES *continues*

- It is the electromagnetic energy that propagates down the transmission line—not electron flow.

- As the electromagnetic energy propagates along a transmission line, current is induced into the conductors. This current flows along the conductors, charging the first section's parasitic capacitance and then flows back on the return conductor (reference plane) to the source.

- Current always flows in a loop. But it does not flow along the transmission line to the load and then return to the source. The propagating signal charges up each section, in turn, as the rising edge propagates along the transmission line.

- Closed-form equations are approximations only and a 2D field solver is required to accurately model impedance of multilayer PCB traces. **PCBDESIGN**



Barry Olney is managing director of In-Circuit Design Pty Ltd (ICD), Australia. This PCB design service bureau specializes in board-level simulation, and has developed the ICD Stackup Planner and ICD PDN Planner software. To read past columns, or to contact Olney, [click here](#).

References

1. Henry Ott, Electromagnetic Compatibility Engineering
2. Howard Johnson, High-speed Signal Propagation
3. Bob Holmes, Bush Telegraph and New Scientist Magazine
4. Telegrapher's equations, Wikipedia
5. The ICD Stackup Planner and PDN Planner can be downloaded from www.icd.com.au

video interview**PCB Design From a Turkish Perspective**

*by Real Time with...
Designers Forum*



Halil Ibrahim Yaslak probably traveled a lot farther than most Designers Forum attendees. Halil is a PCB designer with the Turkish company Aselsan; you may remember that Halil won a Mentor Technology Leadership Award in the milaero category in 2012. He and Editor Kelly Dack discuss his company's cutting-edge work and the different challenges faced by designers worldwide.

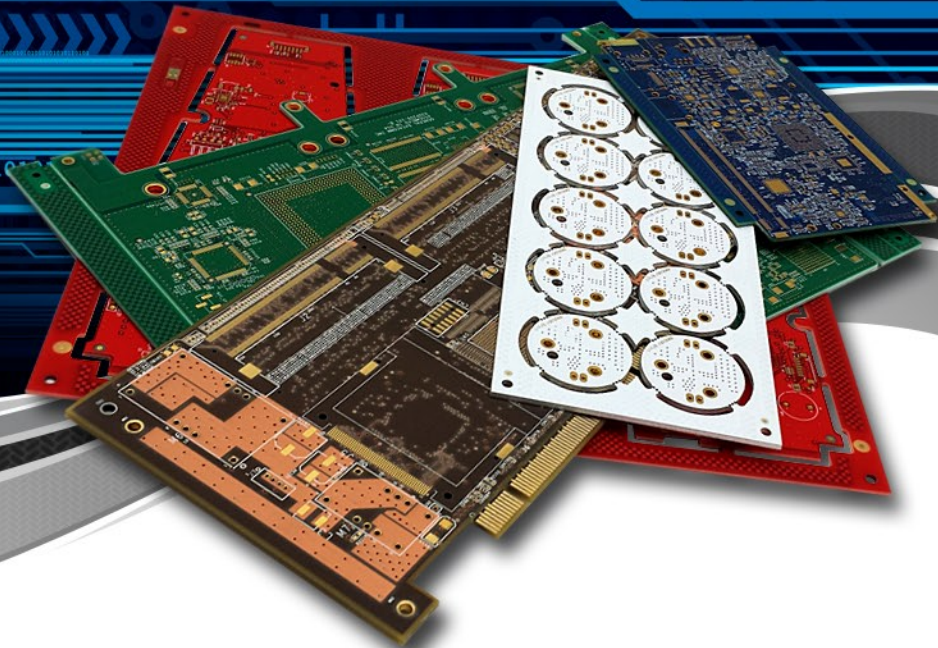


realtimewith.com



The Absolute Best Value in High Technology Printed Circuit Boards

101001 011010
001101 010011
010010 000100



Since 1979, Eagle Electronics Inc. has provided our Customers with the highest quality Printed Circuit Boards at fair and competitive prices. We are committed to exceeding our Customers' expectations and requirements, achieving **total customer satisfaction** on each and every job. It's just the way we do business!



MANUFACTURERS OF QUALITY PRINTED CIRCUIT BOARDS

With Eagle, you can expect:

- Rapid Response to Quote Requests
- Fair and Competitive Pricing/Costs
- 100% Quality Performance
- 100% On-Time Delivery Performance
- Flexibility Scheduling
- Stock/Consigned Inventory Programs
- Thorough follow-up after job completion
- Total Satisfaction!

**click here for a
virtual tour of our facility!**

www.eagle-elec.com

Consult With Your Fabricator—Especially for Hybrid Designs

by Amit Bahl
SIERRA CIRCUITS

For many years, the demand for polyimide and PTFE materials was very low, and very specialized. Demand for polyimides was confined almost exclusively to high-temperature or high-voltage applications and in the case of the PTFE laminates, such as the Rogers duroids, microwave circuits.

Designers are increasingly turning to those laminates, perhaps for good reasons based on the thermal or electrical attributes listed in data sheets, but without regard to certain manufacturing characteristics that set them apart from the fabrication processes for conventional FR-4. Many of those manufacturing characteristics are not apparent from data sheets.

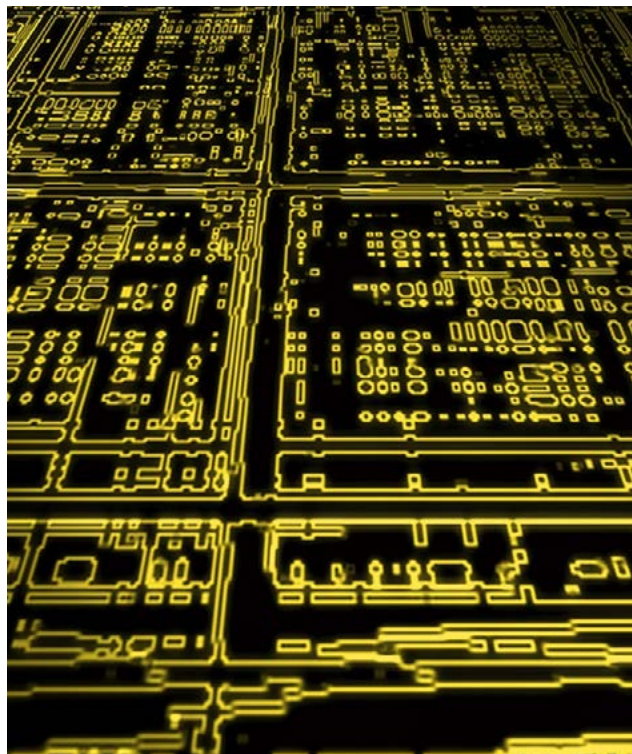
Wise designers consult manufacturers before developing hybrid stackups, because combining laminates with dissimilar mechanical properties can complicate fabrication and therefore, bear on cost, especially with respect to yield. By the time a design is ready to prototype, it's often too late and too expensive to recast in a way that would achieve the design objectives and yet be easier to build. My company, which is devoted to prototype manufacture and up to medium scale production, often takes on challenging projects that might have been better architected had the designers reached out to us at the stackup stage.

Not Covered in Data Sheets

Let me focus on one aspect of hybrid builds brought to mind by a recent conversation with the engineer who supervises drilling operations. "The problem is," he emphasized, "there can be very different feed and speed requirements for drilling one material compared to another." Several designs we recently built involved three, four, even five different materials. For example, we had a project with two different polyimide materials: FR-4, and flex material, combined.

The in-feed setting for the drills (how quickly they descend) and their spindle speed for drilling the polyimides is completely different than the in-feed and spindle speed for the flex material. Polyimide laminates are hard materials that fracture easily. Therefore, they must be drilled at a relatively low in-feed rate and a high spindle speed. Flex material is just the opposite, requiring a high in-feed rate and a slow spindle speed, because the slower the in-feed and the higher the spindle speed, the more heat that will be generated.

"It's difficult when those materials are combined, because the polyimide can't be drilled using the flex parameters, or vice versa, or the boards will be compromised," the engineer pointed out. "The softer the material—the duroids and flex materials are



Exclusive Publications for the PCB Design Community



THE **pcb**
design™
MAGAZINE

INSIDE DESIGN
WEEKLY NEWSLETTER



SUBSCRIBE NOW!

 **I-CONNECT** *007*
GOOD FOR THE INDUSTRY

CONSULT WITH YOUR FABRICATOR—ESPECIALLY FOR HYBRID DESIGNS *continues*

about the same—the easier it is to drill, but the more susceptible it is to heat. So if you also have a harder material in combination to drill, it's easy to distort the softer material if you go in at too high an in-feed. The drill can essentially pull the material out of the hole wall and then the material snaps back, but not all the way, leaving what appears to be negative etchback in the hole wall of the softer material."

What we are doing in such rigid-flex cases is "peck" drilling. We control the machines to drill a just certain distance and then withdraw the drills to let them cool, and then drill further. We are guided by the drilling characteristics of the most-sensitive materials.

Unlike the duroids, Rogers 4000 materials have drilling characteristics that are relatively close to FR-4. Some defects may result because the 4000 materials must be drilled more slowly and generate more heat. There is a tendency for the interconnects to smear a little bit in the holes—so-called nailheading—though that's typically not cause for rejecting a board. Slowing down the feed rate for a given material in a stack runs the risk of causing some defects in the region of the hole where that material is located. Polyimides and the ceramic-filled materials have slower in-feed rates and higher spindle speeds because they are hard, and therefore less material is removed per revolution of the drill. The combination of feed rate and drill speed is sometimes referred to as chip load. A harder material necessitates a lower chip load; that is, a lower feed rate and a higher spindle speed. When you have a combination of materials, you have to adjust the drilling parameters to meet the requirements for the hardest material, or distortion inside the hole can result and that can interfere with plating the hole. But when you have two materials with very different drilling parameters, the settings are a compromise.

When you have a combination of materials, you have to adjust the drilling parameters to meet the requirements for the hardest material, or distortion inside the hole can result and that can interfere with plating the hole.

For manufacturers, there's yet another consideration besides chip load and that's drill hit count; the sharper the drill, the fewer the issues that will be encountered. For FR-4 the drill hit count typically is around 800, but for a hard material, drills have to be changed after 400 or so hits, and that affects project cost.

Still another consideration internal to manufacturers is which of three entry materials will be used on top of the board stack for drilling support. There's a coated aluminum material that's best for drilling small holes; there's an aluminum material with a paper core that's used for most other drilling needs; and there's phenolic material, which provides the most surface support and would usually be used when a soft material, such as a Rogers duroid, is the top layer of a board. The phenolic material is the worst of the three entry materials for drilling accuracy; it is hard and drills can skate when they start. If the drill diameters involved are not less than 10 mils, accuracy is not compromised by the phenolic material.

If the drill diameters are much smaller, the coated aluminum material must be used or the drills will snap. However, in most cases, the amount of burring, the debris left in small holes as a result of using the coated aluminum material, is negligible.

There is considerable work among laminate suppliers to provide alternatives to polyimide and PTFE materials whose manufacturing characteristics are closer to those of FR-4.

Switching Gears

Let me turn to a different design consideration that influences manufacturing and cost as well as electrical and mechanical performance: surface finish. For example, if HASL (hot-air solder leveling) is selected, the PCB design must not include any fine-pitch components because a HASL surface finish will be too uneven to

CONSULT WITH YOUR FABRICATOR—ESPECIALLY FOR HYBRID DESIGNS *continues*

ensure uniform contact bonds for such parts. Immersion silver or immersion gold finishes are better alternatives if devices with tight contact pitches are involved. The immersion gold finish is called ENIG (electroless nickel, immersion gold).

Immersion silver does not require a layer of nickel underneath as does the immersion gold finish. The nickel is used as a barrier layer to prevent copper migration into the gold over time, which can result in increased contact resistance if boards are left unassembled too long. Both immersion silver and ENIG result in even surfaces that are much flatter than can be obtained with HASL; moreover, both are more electrically conductive. The overall thickness of the immersion silver finish can be held to tighter tolerance than that for ENIG, so silver is preferred if there are press-fit connectors in the design. However, silver tarnishes quickly and assembly must therefore be completed soon after board fabrication. ENIG is the better choice for designs with very fine traces. ENIG is also better for thin boards because it is a relatively low-temperature process.

Both electrolytic soft gold, which would be a choice for designs that involve wire bonds

in assembly, and electrolytic hard gold, which also supports wirebonding and has advantages for sliding contacts, have downsides. To accomplish either finish requires the addition of buss bars on panels to electrically interconnect the copper features during the plating process, which afterward must be severed from the PCB circuits. Moreover, copper can remain exposed on trace sidewalls.

Electroless nickel, electroless palladium, immersion gold (ENEPIG) is a somewhat more expensive surface finish that has nearly universal advantages. Nonetheless, my advice regarding surface finishes jibes with my advice regarding hybrid stackups: Consult your prospective manufacturer at the outset of your project, to make sure you don't become invested too deeply to improve your design decisions. **PCBDESIGN**



Amit Bahl directs sales and marketing at Sierra Circuits, a PCB manufacturer in Sunnyvale, California. He can be reached by [clicking here](#).

Nanotube-Infused Clothing Protects Against Chemical Weapons

Nerve agents are among the world's most feared chemical weapons, but scientists at the National Institute of Standards and Technology (NIST) have demonstrated a way to engineer carbon nanotubes to dismantle the molecules of a major class of these chemicals.

The team's experiments show that nanotubes can be combined with a copper-based catalyst able to break apart a key chemical bond in the class of nerve agents that includes Sarin.

The team developed a way to attach the catalyst molecule to the nanotubes and then tested the effectiveness of the tube-catalyst complex to break the bonds. To perform the test, the complex was deposited onto a small sheet of paper and put into a solution containing the mimic molecule. For comparison, the catalyst without nanotubes

was tested simultaneously in a different solution. Then it was a simple matter of stirring and watching chemistry in action.

Principal investigator Angela Hight Walker says that several questions will need to be addressed before catalytic nanotubes start showing up in clothing, such as whether it is better to add the catalyst to the nanotubes before or after they are woven into the fabric.

Nanoengineers at the University of California, San Diego, are asking what might be possible if semiconductor materials were flexible and stretchable without sacrificing electronic function.

Today's flexible electronics are already enabling a new generation of wearable sensors and other mobile electronic devices. But these flexible electronics, in which very thin semiconductor materials are applied to a thin, flexible substrate in wavy patterns and then applied to a deformable surface such as skin or fabric, are still built around hard composite materials that limit their elasticity.



HDI Design Education: Is Anyone Interested?

by Dan Smith
RAYTHEON

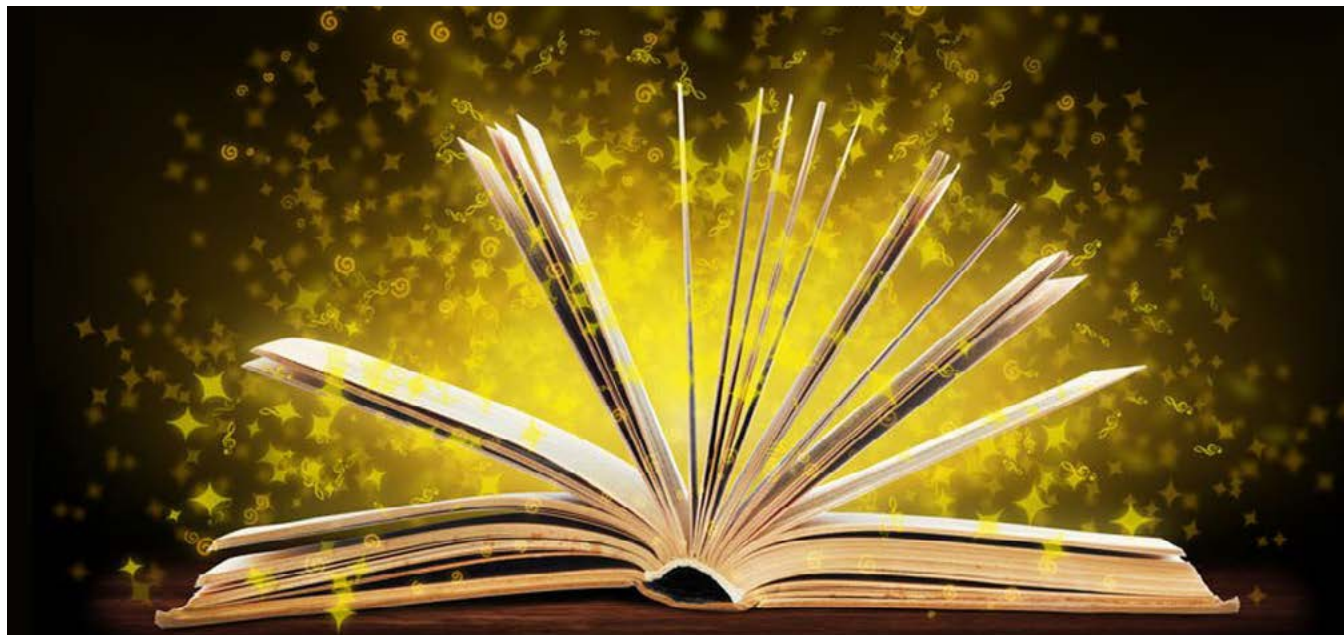
I first became aware of high-density interconnection (HDI) design methodology in 1997 when a co-worker of mine came back from a training class that was taught by Happy Holden. “Little Dave” (since our department had two employees named David W.) came back excited to try this new design knowledge on a multilayer board stackup that only used blind and buried vias, with no through-hole vias. His excitement and dedication paid off, as he completed the design with an autorouter exactly as he planned. Through Dave’s success, our department then started to create other designs using HDI stackups that were taught in Happy’s class and our autorouter tool.

I did a few complex designs that fit into the IPC Type III category. One of the more complex designs I attempted could only autoroute up to 98% completion after trying many setup files (with great assistance from “Big Dave”) and our most powerful computer available within our

group. I finished the design manually, and the board was fabricated and worked as designed.

About six months later, when we upgraded our department’s autorouter computer, I restored the design from archive, and re-ran the last autorouter configuration file on that 98% completed design. This time, the design was now 100% completely autorouted, and took about a third less time than on the previous “most powerful” computer. This one little re-investigation became the catalyst for my full immersion into HDI design research, reviewing the existing designs and now trying newer autorouting approaches with a variety of more challenging stackups, via spans, and via sizes. Each new redesign attempt created more curiosity to push autorouters and stackup methodologies to their absolute limits.

Over the next few years, any article written on HDI that crossed my path became assimilated into my mental database. I met Mike Fitts



Privately Held Global Printed Circuit Board Fabrication

LOCAL SUPPORT



and a **WORLD** of Circuit Board Manufacturing Options

USA

GERMANY

JORDAN

TAIWAN

CHINA

Manufacturing at ANY Volume, both On and Offshore
Quick Turn, Prototypes, DFM, Global Management
Specialties like Rogers, Aluminum, Hybrid Materials
Logistics including Consignment, VMI, JIT & More
24-7 Local Engineering, Service, and Support

[Download Digital Brochure](#)

[Download Our Capabilities](#)



GLOBAL CIRCUITS

sales@htglobalcircuits.com

| (727) 327-6236

| www.htglobalcircuits.com

HDI DESIGN EDUCATION: IS ANYONE INTERESTED? *continues*

lecturing on HDI at a design conference, and I asked him to clarify my repository of HDI redesign questions. When Mike and I met at another design conference, he introduced me to the very person who inspired Little Dave: Happy Holden.

Over the next few years, I had several conversations with Happy until we finally worked together at the same company.

Around the same time, Mentor Graphics' Charles Pfeil (whom I also worked under over a few years) released [BGA Breakouts and Routing](#), which was well received by the PCB design community. Happy later released his [HDI Handbook](#), a collection of industry experts' perspectives, which included HDI design processes and methodologies. Happy used to cheer the number of free downloads as it went from the hundreds into the thousands.

Happy and I later developed a four-day complete HDI training workshop, and then co-taught together.

In late 2008, Happy gave me over 400 gigabytes of his 20+ years of history teaching, writings, and personal research. He passed his torch to me, and I am truly grateful for this brain-dump to his "grasshopper" (thanks to Eric Bogatin for that nickname).

When we parted to work for other employers, Happy and I continued teaching HDI for the next couple of years. I created a 100+ page hands-on workshop where designers could try to complete an IPC I, II, or III design, requiring only that the user choose different via sizes, and then let the autorouter attempt to complete the design based on their choices. The course was made available to the public in June 2011.

I then contacted all of the major EDA vendors about sponsoring me to write a complete HDI course that would feature their tools in creating a HDI design from start to finish. At that time, no one from the EDA vendor community was interested in having me create such a course. I was also approached by a major industry standard group in mid-2010 to write a

HDI certification course, but was told to just wait until they thought it was time to start writing it. My last contact by this group about this topic is now four years old.

So, this is why I have been sitting on the sidelines since June 2011. Like the traditional musician (yes, I am the Porch Dawg pianist) who waits by the phone for that important call,

I came to the realization I needed to move on to other challenges and opportunities that were not related to HDI.

Other PCB design industry experts have taught HDI specialty courses at conferences, but the focus and the audience attendance on this design process has truly lost momentum. Happy, Charles, and even Mike have gone on to lecture on different non-HDI-related topics. With these key industry innovators/lecturers of the 2000–2010 time period no longer working on anything related to

HDI design education, what does the PCB design user community think of this reduction of HDI education?

What do YOU want or need regarding your HDI education? We want to know! Yes, it's time for another survey, one focused solely on HDI.

[Click here](#) to take this short HDI survey, and share your thoughts with the design community. The results will be published in future columns.

Thank you in advance for your time and feedback. I am sure both Big Dave and Little Dave will be curious about your input, too.

PCBDESIGN

“

Other PCB design industry experts have taught HDI specialty courses at conferences, but the focus and the audience attendance on this design process has truly lost momentum.

”



Daniel J. Smith is a principal technologist for Raytheon Missile Systems. He has taught multiple aspects of the PCB design process internationally, and he has authored several PCB-related patents, articles, and standards over the past 30+ years. To contact Smith, [click here](#).

The Tools You Need for GREAT DESIGN

IPC Designer Certification (CID and CID+)

Gain **technical training** that provides a solid **foundation** for meeting today's challenges, like **Design For Manufacture**.

www.ipc.org/designer-certification

IPC Standards — the framework for manufacturing throughout the electronics industry

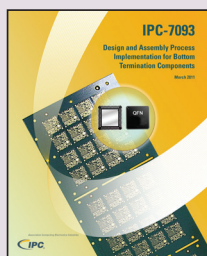
IPC-2220 Family of Design Documents

- IPC-2221, *Generic Standard on Printed Board Design*
- IPC-2222 on design for **rigid organic** printed boards
- IPC-2223 on design for **flexible** printed boards
- IPC-2224 on design of PWBs for **PC cards**
- IPC-2225 on design for **organic multichip** modules (MCM-L) and MCM-L assemblies
- IPC-2226 on design for **high density interconnect** (HDI) printed boards

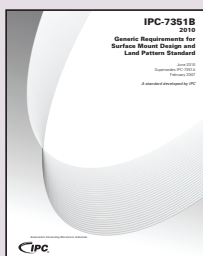
COMING SOON

DFX: Design For Excellence — Guidelines for

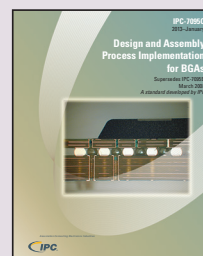
- DFM (Design For Manufacture),
- DFR (Design For Reliability),
- DFA (Design For Assembly) and more



IPC-7093, *Design and Assembly Process Implementation for Bottom Termination Components*



IPC-7351B, *Generic Requirements for Surface Mount Design and Land Pattern Standard*



IPC-7095C, *Design and Assembly Process Implementation for BGAs*

Association Connecting Electronics Industries



www.ipc.org

Mil/Aero007

News Highlights



Viasystems Receives Supplier Award from Rockwell Collins

Viasystems Group, Inc. received the 2014 Printed Circuit Boards and Electro-Mechanical Solutions Build to Print Supplier of the Year Award at the Rockwell Collins Annual Supplier Conference. The award is an acknowledgement of significant contributions made during the year by suppliers and is based upon quality, delivery, total cost of ownership, lead time, and customer service.

IPC Releases Conflict Minerals Data Exchange Standard

"IPC-1755 is XML-schema based, which allows for more efficient communication and quicker application of data across companies, supply chain levels and industries," said John Plyler, chairman of the 2-18h Conflict Minerals Data Exchange Committee. "IPC-1755 will be compatible with several software tools and Version 3.0 of the CFSI Conflict Minerals Reporting Template."

Dragon Circuits Diversifies with Instagram and Bitcoin

Progressive circuit board manufacturer Dragon Circuits announces the implementation of Instagram to track the process of their drone department, Dragon Drones. The company is also partnering with Coinbase to accept payment via Bitcoin.

Innovative Circuits Acquires Orbotech Inkjet Printer

This next-generation Orbotech Sprint 120 Inkjet printer features DotStream Technology and UV LED curing while delivering consistent top-quality printing at high speeds. Registration accuracies of 35µm are achieved through automatic measurements and scaling.

Military Communications Market at \$30.12B by 2019

The report "Military Communications Market—(SATCOM Terminals, Commercial off The Shelf (COTS), Military Communication Security, Tetra Radio, JTRS, Software Defined Radio, Radio Communication, Tactical Communication)—Worldwide Forecasts & Analysis (2014–2019)" is now available.

DARPA Selects Boeing Phantom Swift for X-Plane Program

Phantom Swift, a prototype Boeing initially built in less than a month, has been accepted to be part of the Defense Advanced Research Project Agency (DARPA) Vertical Takeoff and Landing X-plane program. DARPA is trying to mature a new aircraft configuration capable of both efficient hover and high-speed cruise.

Paving the Way for Unmanned Ships of the Future

Ships of the future will soon be steered across the seven seas—unmanned. A new simulator is helping propel these plans forward. Partners from five different countries engineered the design of the autonomous freighter. "In Europe, making a career in shipping is no longer a popular choice," explains Project Coordinator Hans-Christoph Burmeister. "This industry has successor problems."

DARPA Awards PARC \$2M to Develop Vanishing Electronics

PARC, a Xerox company, has signed an up to \$2 million contract with the Defense Advanced Research Projects Agency (DARPA) to develop and demonstrate PARC's disappearing electronics platform (called DUST), with intriguing implications for a variety of military, ecological, and commercial interests.

SOMACIS Intros R&D Project for Signal & Power Bus Bar

Engaged in the SAPBB project, SOMACIS, along with two other Italian companies, is developing an advanced technology bus bar type solution for automotive applications. The circuit board will enable both power and signal distribution between key components of the automotive power system.

M2M SatCom Market to Reach \$4.76B in 2019

Increasing data communication need, rising M2M applications, and fast return on investments is leading to the creation of more and more avenues for the M2M satellite market. The global market is expected to grow from \$2.98 billion in 2014 to \$4.76 billion by 2019, at an estimated compound annual growth rate (CAGR) of 9.8% from 2014 to 2019.



Living proof that flying dragons are real!

Dragon Circuits is proud to be at the forefront in unmanned aviation.

Interested in unmanned aerial or ground vehicles? Dragon Circuits is developing the newest technology to enhance vehicle run-time, performance and durability to put your company at the top of the industry. Proprietary electronics, built in the USA with ITAR, AS9100 and MIL-SPEC standards. Contact us for more information on a range of services from design, manufacturing, and even flight school!

LEARN MORE »



www.dragoncircuits.com | 972.790.7610

Customer Support: Not Just for Customers Anymore

by Tim Haag

INTERCEPT TECHNOLOGY

A couple of months ago, I went into my local electronics shop to purchase some resistors for a hobby project. This store sells many things besides electronic components, so their employees are trained to upsell whenever possible. As I made my way to the back of the store for my resistors, I noticed that the clerk behind the counter was trying to engage me. When I passed her, she practically leapt over the cash register in her efforts to reach me. She recklessly rounded the aisle where I was, lost her balance, and almost took me down with the nearby display of merchandise.

She was still not to be deterred, however, even after that near miss. A few moments later at the cash register, with my paltry purchase of a few resistors, she asked me how my cell phone quality was. I was still reeling from the near miss in the aisle and with my project waiting for me back at home, the last thing on my mind was my cell phone. So I told her, "Everything is fine." While skilled at taking my cash, she

seemed unable to take my hint, and asked me if I needed a new cell phone battery. "No... everything is fine," I reiterated as I collected my change and mentally rehearsed my escape from the store.

I have thought about this encounter a few times since then, and I have come to realize that there is an important lesson to be learned here. Well, two lessons actually. But the first one is pretty obvious; don't go into these types of stores without sufficient armor to protect you from flying clerks. So let's move onto the second and more important lesson.

This clerk was more intent upon making a larger sale than she was in providing good customer support. Now, I don't want to demonize this person; she was just doing the job that she was trained to do. As I pointed out earlier, she definitely put everything she had into it with "flying" gusto. But in her efforts to upsell me into a new phone or phone battery, she missed the opportunity to sell me what I really





***THE* best way to find a PCB fabricator, anywhere.**



Quick Search



Advanced Search



Try it now!

ThePCBList.com

CUSTOMER SUPPORT *continues*

needed. What if the conversation had instead gone something like this?

Clerk: *"What are you building with these resistors, sir?"*

Me: *"I'm adding LED lighting to a scale model that I'm building."*

Clerk: *"That sounds interesting. How are you wiring them all up?"*

Me: *"I've already got the wiring diagrammed out, but come to think of it, I am very low on solder."*

Clerk: *"I've got that right here on this shelf; how much would you like?"*

But as it was, I was in such a hurry to get out of her store due to her brashness that I forgot about my need for solder and I ended up buying it elsewhere the next day. Not only was I frustrated by my experience at her store, but she missed the opportunity to sell me the additional supplies that I really needed. And more importantly, she failed to build my trust and loyalty and keep me as a returning customer.

In my role as the customer support manager for my company, I have seen plenty of examples of customer support and I am proud of how we serve the needs of our customers. But my point here is not to focus on customer support as a function of a support technician. Instead, I want to explore the concept of how we should all strive to provide the best level of customer support in our jobs, no matter what we do—whether we are actual support technicians or not.

Our customers should not just be the end-users of the products or services that our com-

panies provide. Our customers include our co-workers, bosses, cross-department relationships, strategic partnerships and even upper-level management. Any time we interact with someone professionally—and yes, even personally—we should treat this as a potential customer support role. Take the scenario for instance of a co-worker who comes to you with a question on how to do a certain function of the job. You could give them the simple answer and leave it at that, or you could take some extra time and help them to make sure that they understand what you've told them. You could then ensure that once they take this information back to their station that they are successful with it. In other words, you could support them.

This kind of relationship between people working together helps to build trust, loyalty, and camaraderie. The benefits to your company of supporting each other like this are immediate; the tasks needing to be done are accomplished more efficiently and productivity is increased. But there are long-term benefits to this synergy as well.

Usually a group of people who have developed trust and loyalty among each other will achieve better results than a group of people who are working towards the same goals as individuals. Increased creativity is another benefit of trust, and when creativity is encouraged, a wealth of new ideas and processes become available. When you know that you have co-workers who are willing to go the extra mile for you, that they "have your back," you tend to step out with more confidence in your own job. This kind of confidence can be seen as risky, and therefore people who feel isolated in their jobs may not be willing to take those

kinds of risks. But when people feel supported by their co-workers they are usually more will-

Our customers should not just be the end-users of the products or services that our companies provide. Our customers include our co-workers, bosses, cross-department relationships, strategic partnerships and even upper-level management. Any time we interact with someone professionally—and yes, even personally—we should treat this as a potential customer support role.

ing to try something new, which may help put their department or company ahead of the game.

Obviously, there are some dangers here. Support can become co-dependent and destructive. You need to make sure that those you are supporting are growing from your efforts and not just sucking the life from you. I have seen many instances where the same people need the same support over and over again. They are in a place where it is easier for them to get you to do their work for them instead of taking your initial help and growing on their own. These kinds of destructive relationships need to be avoided. But the majority of your co-workers will flourish in a supportive environment where you are helping one another.

Often it is seen as safer to work as an isolated person, but it's definitely not better for you or your company. Yes, we can continue to work with our heads down and give quick answers to questions, ward off supportive teamwork and stay in our narrow comfort zones. Or we can

reach out to our co-workers, bosses, and others in our company and support their efforts in order to be more productive. When others come to us for help we can view them as an annoyance to our immediate goal, or we can re-visualize our goal to understand that working together and supporting each other will ultimately provide the best results for our company.

The benefits of adopting a customer support type of thinking in our professional and personal relationships allow us to grow not only as productive workers, but also as better people and stronger members of our community.

PCBDESIGN



Tim Haag is customer support and training manager for Intercept Technology.

video interview

Always Providing Extra Value to Meet Customers' Needs

by Real Time with...
IPC APEX EXPO



Prototron's Mark Thompson is the guardian at the gates, so to speak. In the CAM department, he sees potential DFM issues with your design data before they become big problems. He and Editor Dan Beaulieu discuss the extra services Mark provides customers, and some common mistakes that become evident after data handoff.



realtimewith.com



TOP TEN

PCBDesign007
News

News Highlights from PCBDesign007 this Month

① DownStream Releases DFMStream Version 11

DFMStream 11.0 introduces over 70 additional DFM analysis checks to support both the PCB fabrication and assembly processes. The analysis expansion was designed to satisfy requirements for advanced vias, back drilling, fine pitch SMD components, tighter tolerances for solder and paste masks and other advanced technologies.

② Zuken's Contribution to IPC-2581 Recognized

Zuken's Humair Mandavia and Steve Watt have been honored by IPC for their contribution to the cross-industry IPC-2581 consortium. Zuken was the first EDA vendor consortium member to support IPC-2581.

③ PCB Design Instructor Glenn Wells Has Died

Glenn Wells, a PCB designer, instructor, and tireless advocate for improving design education, died in March. He recognized the need for getting PCB design instruction into colleges and universities. Glenn once said that his goal was to make every high school guidance counselor aware of the PCB designer career.

④ AWR Releases NI AWR Design Environment V11

"AWR's V11 release of the NI AWR Design Environment supports our ongoing commitment to enabling our customers to spend more time focusing on their design challenges and less time on driving the software," said Sherry Hess, vice president of marketing.

5 Mentor Graphics Unveils New PADS Web Site

The Mentor Graphics PADS team is inviting PCB designers to experience the all-new PADS.com. PADS.com is built specifically for designers to learn more about how to improve their PCB designs.

6 Cadence Strengthens Portfolio in Q1; Revenue at \$379M

Lip-Bu Tan, president and CEO, said, "We advanced our functional verification platform by releasing our new Incisive vManager solution and acquiring Forte Design Systems. And we announced that we entered into a definitive agreement to acquire Jasper Design Automation, Inc., which will meaningfully add to our verification capabilities."

7 Mouser Sponsors Design Contest for Engineers & Students

The grand prize winner will receive global recognition and a cash prize of \$20,000 for an innovative product that benefits society and the economy. Previous contests have produced more than 9,000 design ideas from engineers, educators, and students in more than 100 countries. Entries are being accepted now through July 1, 2014.

8 Zuken Intros CADSTAR 15 with New Routing Features

With high-speed interfaces now almost universal, easing their implementation is vital. CADSTAR's P.R. Editor now supports impedance balanced routing that simplifies the implementation of high-speed interfaces. Engineers can easily route to JEDEC standards and meet DDR3 performance specifications. This reduces design iterations by helping designers optimize circuits for the highest clock speeds.

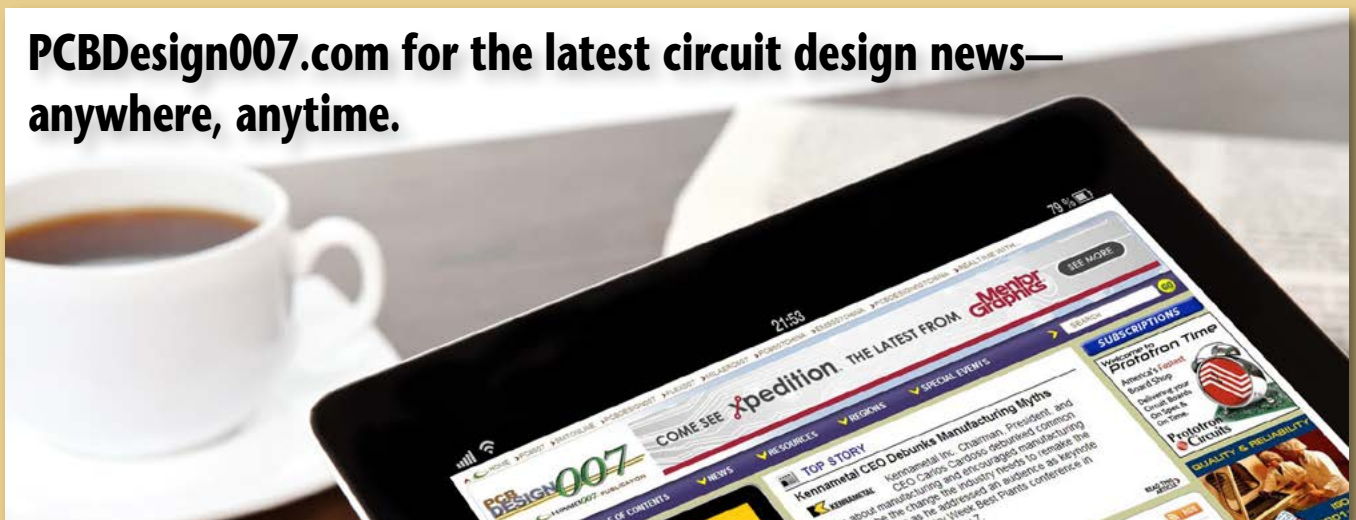
9 Prasad to Lecture on SMT, BGA Design, & Manufacturing

The course material is based on Ray's textbook "Surface Mount Technology: Principles and Practice, 2nd Edition," IPC -7095 Design and Assembly Process Implementation for BGA, chaired by Ray and IPC-7093 "Design and Assembly Process Implementation for Bottom Terminations surface mount Components (BTCs) such as QFN, DFN and MLE, also chaired by Prasad.

10 Altium Implements Growth Strategy; Relocates Units

The relocation to San Diego, where Altium has had a long-established sales and operational presence, represents a natural next-step in the implementation of the company's renewed growth strategy. It is a result of the focus on PCB design tools and solutions for the PCB design market.

PCBDesign007.com for the latest circuit design news—anywhere, anytime.



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](#).

International Conference on Soldering and Reliability

May 13–15, 2014
Toronto, Ontario, Canada

Internet of Things

May 14–15, 2014
Milwaukee, Wisconsin, USA

Toronto SMTA Expo & Tech Forum

May 15, 2014
Toronto, Ontario, Canada

2014 Technology & Standards Spring Forum

May 19–22, 2014
Seattle, Washington, USA

12th Annual MEPTEC MEMS Technology Symposium

May 22, 2014
San Jose, California, USA

IPC Southeast Asia High-Reliability Conference

May 28, 2014
Singapore

DAC 2014

June 1–5, 2014
San Francisco, California, USA

RAPID Conference & Exposition

June 9–13, 2014
Detroit, Michigan, USA

IPC SE Asia Workshop on Soldering of Electronics Assemblies

June 9, 2014
Penang, Malaysia

IEEE ICC 2014

June 10–14, 2014
Sydney, Australia

CES Unveiled Warsaw

June 17, 2014
Warsaw, Poland

Upper Midwest Expo & Tech Forum

June 18, 2014
Bloomington, Minnesota, USA

CE Week

June 23–27, 2014
New York City, New York, USA

Symposium on Counterfeit Electronic Parts and Electronic Supply Chain

June 24–26, 2014
College Park, Maryland, USA



PUBLISHER: **BARRY MATTIES**

barry@iconnect007.com

PUBLISHER: **RAY RASMUSSEN**

(916) 337-4402; ray@iconnect007.com

SALES MANAGER: **BARB HOCKADAY**

(916) 608-0660; barb@iconnect007.com

MARKETING SERVICES: **TOBEY MARSICOVETERE**

(916) 266-9160; tobey@iconnect007.com

EDITORIAL:

GROUP EDITORIAL DIRECTOR: **RAY RASMUSSEN**

(916) 337-4402; ray@iconnect007.com

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**

INNOVATIVE TECHNOLOGY: **BRYSON MATTIES**

COVER ART: **SHELLY STEIN**



The PCB Design Magazine® is published by BR Publishing, Inc., PO Box 50, Seaside, OR 97138
©2014 BR Publishing, Inc. does not assume and hereby disclaims any liability to any person for loss or damage caused by errors or omissions in the material contained within this publication, regardless of whether such errors or omissions are caused accidentally, from negligence or any other cause.

May 2014, Volume 3, Number 5 • The PCB Design Magazine© is published monthly, by BR Publishing, Inc

ADVERTISER INDEX

Altium.....	27
Candor.....	7
DM Electronic International.....	21
Downstream Technologies.....	9
Dragon Circuits.....	55
Dymax.....	31
Eagle Electronics.....	45
Electrolube.....	13
EMA/EDA Design Automation.....	23
H&T Global.....	51
I-Connect007.....	47
IPC.....	53
Isola.....	5
Miraco Inc.....	33
Multilayer Technology.....	11
Murrietta Circuits.....	43
Next Level PCB.....	3
Prototron Circuits.....	39
Rogers.....	17
Sunstone Circuits.....	29
The PCB List.....	2, 57
US Circuit.....	15
Ventec.....	25

**Coming Soon to
The PCB Design
Magazine:**

**June:
Designing
Flex Circuits**

**July:
Concurrent
Design**

**August:
Materials**