PCB007 PRESENTS

January 2012

the pcb magazine

AN I-CONNECT007 PUBLICATION

GOOD FORTUNE?

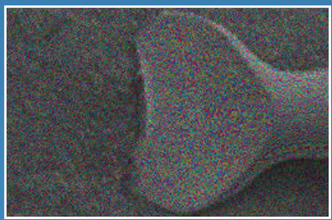
WHAT THE FUTURE HOLDS FOR THE INDUSTRY

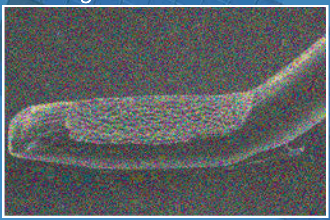
Electronics

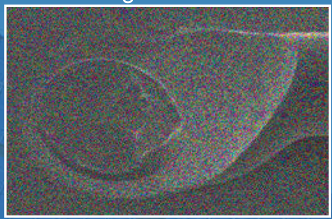
Final Finishing

Universal Finish SolderBond (ENEPIG and ENEP) for Soldering, Au-Wire and Cu-Wire Bonding

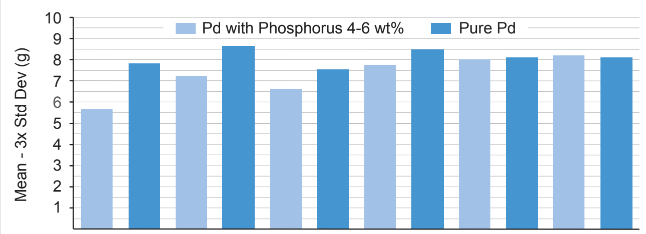
Soldering


Au-wire wedge


Al-wire wedge


Cu-wire wedge

Gold Wire Bond Pull Test



Process	ENEPIG	UFSB	ENEPIG	UFSB	ENEPIG	UFSB	ENEPIG	UFSB	ENEPIG	UFSB	ENEPIG	UFSB
Pd Layer	PdP	pure Pd	PdP	pure Pd	PdP	pure Pd	PdP	pure Pd	PdP	pure Pd	PdP	pure Pd
Nickel µm	6.0	6.0	6.0	6.0	7.0	7.0	7.0	7.0	7.0	7.0	7.0	7.0
Pd µm	0.05	0.05	0.15	0.15	0.25	0.25	0.05	0.05	0.15	0.15	0.25	0.25
Au µm	0.04	0.04	0.04	0.04	0.04	0.04	0.15	0.15	0.15	0.15	0.25	0.15

Universal Finish SolderBond provides also with low Au-thickness excellent results, as pure Pd is 50% softer than Palladium-Phosphor (4-6wt%).

Process Flow

- **Cleaner** – Pro Select UC
- **Micro Etch** – MicroEtch C
- **Dip** – Aurodip (Option)
- **Pre Dip**
- **Activator** – Aurotech Activator 1000
- **Fine Line Post Dip** – Aurotech FL Plus Post Dip (Option)
- **Electroless Ni** – Aurotech CNN mod.
- **Electroless Pd** – PD-Tech
- **Immersion Au** – Aurotech SF Plus

	NiPdAu (ENEPIG)	NiPd (ENEP)
Deposit Structure		
Electroless Nickel	4.0 µm - 6.0 µm	4.0 µm - 6.0 µm
Electroless Palladium	0.05 µm - 0.3 µm	0.2 µm - 0.3 µm
Immersion Au	0.02 µm - 0.06 µm	--
Functionality		
Soldering		
Eutectic Sn/Pb	Good	Good
Sn-Ag-Cu Alloy	Excellent	Excellent
Contact Switching	Yes	--
Al Wire Bonding	Yes	--
Au Wire Bonding	Yes	--
Cu Wire Bonding	--	Yes

Features and Bene ts

- Three finishes (ENIG, ENEPIG and ENEP) in one process
- ENEPIG is Au-wire and Al-wire bondable
- ENEP is Cu-wire bondable
- Wider Au-wire bond process window compared to ENEPIG with PdP
- Pure Pd-layer without phosphorus co-deposition
- Pure Pd is 50% softer than PdP
- Superior solder joint reliability
- Excellent reliability regarding thermocycling, hot storage test and humidity
- For wirebond processes of today and tomorrow

Atotech Headquarters · P.O. Box 21 07 80 · 10507 Berlin · Germany · www.atotech.com
Tel. +49 (0) 30-349 85-0 · Fax +49 (0) 30-349 85-777

TOTAL ATOTECH

GOOD FORTUNE?

WHAT THE FUTURE HOLDS FOR THE INDUSTRY

Let the January Issue of The PCB Magazine help raise your awareness — of 2012!

Welcome to our first issue of 2012 — with feature articles, interviews and columns by industry veterans who have some pretty interesting ideas about what's on the horizon for the PCB and electronics industries in 2012.

Not yet a subscriber to The PCB Magazine? As the Zen master says, click here now (or is that, *be* here now?) to get it delivered straight to your inbox every month.

January Feature Articles

This month, **DownStream Technologies** takes a look at three key areas of interest for the coming year and **Mentor Graphics** scopes the state of PCB design technology best practices and how it relates to the economy's gradual recovery. Other contributors include a handful of regular columnists who've taken their opinions to feature status: **Gray McQuarrie** *(Solving DAM Problems)* provides commentary on the second chance for making our manufacturing the best in the world. Regular PCB007 columnist **Martyn Gaudion** *(The Pulse)* presents his unique perspective on what's to come from a signal integrity point of view and **Steve Williams** *(POV)* explains how 2012 may be the "year of retrenching." Other feature contributors include **PentaLogix** and **Databeans,** who offer plenty of insight into what to expect in 2012.

"Thank you for your routing tips for DDR2... they helped a lot and the board now works like a dream. I will continue to follow your articles in The PCB Magazine. It's people like you who make life so much easier for the rest of us."

January Columns

We've got the usual suspects helping us ring in the New Year and their thoughts on just about everything PCB-related. First, we're excited to welcome back **Dale Smith** *(Lean, Mean PCB Specialist)* who begins a brand new series on rigid-flex. Also on hand this month are **Barry Olney** *(Beyond Design),* **Mike Carano** *(Trouble in Your Tank),* **Karl Dietz** *(Karl's Tech Talk),* and I-Connect Publishers **Ray Rasmussen** *(The Way I See It)* and **Barry Matties** *(The Sales Cycle),* who help us keep a long view of things.

What else?

There's still plenty more to come! In our Articles Department, we've got veteran I-Connect Tech Editors, **Harvey Miller** and **Pete Starkey.** Miller, with his expert awareness of the domestic PCB industry, offers a comprehensive look at the state — and chances for survival — for the U.S. PCB shop. Fresh from productronica, Starkey shares an interview he conducted with Rainbow Technologies Founder Jonathon Kennett, who explains his take on primary imaging, Rainbow style. We'll also have a fresh handful of productronica videos and our industry highlights sections covering milaero, flex, new products, events and more.

2012 is here, folks, so be aware…be very aware — with the help of The PCB Magazine!

In last month's The PCB Magazine, an unfortunate omission was made during the editing of Patrick Valentine's article, *Understanding, Measuring and Tracking Phosphorous in an Electroless Nickel Bath.* The article in its corrected form will be posted at www.pcb007.com in the coming week.

CONTENTS

PCB007 PRESENTS

JANUARY 2012

VOLUME 2

NUMBER 1

THE DEFINITIVE INTERACTIVE MAGAZINE DEDICATED TO THE GLOBAL PCB INDUSTRY

thepcbmagazine.com

the pcb magazine

AN I-CONNECT007 PUBLICATION

GOOD FORTUNE?

WHAT THE FUTURE HOLDS


THE Future OF
Surface Finish
SPF™ (Semblant Plasma Finish)
Replaces or Enhances Gold/Silver
Reduces Process Costs
Environmentally Responsible
Mitigate Creep Corrosion
Pb-Free Solderability
Flux Containment
Semblant
semblantglobal.com

THE WAY I SEE IT

Outlook for 2012

SUMMARY

Like anything else in life, being prepared is half the battle. Observe what the market is doing now, in order to prepare for what's to come. This includes business cycles and emerging technology. 2012 is shaping up to possess plenty of opportunity for those who pay attention.

by Ray Rasmussen
I-CONNECT007

At productronica last November, I asked everyone I interviewed what they thought 2012 would bring. Every single person felt good about the market in 2012, but all of them said that their customers seemed to be taking a "wait and see" approach. As a result, they had little or no visibility into their sales for 2012.

You can watch the interviews along with all the coverage from productronica, here.

Near the end of the show, I had a chance to interview Walt Custer and get his take on the global business conditions for our industry. It seems that the lack of visibility most claimed, the hesitancy of PCB, EMS and OEMs to pull the trigger on capital equipment purchases, is just the nature of our economy. Although most attached this market uncertainty to the debt crisis in Europe, Walt was a bit more matter-of-fact, pointing out that we're at the bottom of a typical business cycle. We can expect several more quarters of negative growth and in the second half of the year, we'll start to see things turn around. He and most of us who've been around this industry for more than a few years have been through these cycles many times. Of course, there are extraordinary events which affect this cycle, but whether we like it or not, the business cycle pretty much follows the same pattern.

The good news about the cycle is that we'll be coming out in 2012 with growth years in 2013 and 2014. And, if the global economy can get back on track during that time we might see some pretty good growth for the industry. I'm optimistic.

Everyone Needs Electronics

Some actually do need electronics as they integrate the latest technologies into their factories and businesses, increasing efficiencies or capabilities. It's the electronics that keep them at the top of their game, or at least keep them in the game. All industries these days rely on some type of electronics, which is good news for all of us building electronic products. Of course there are those who want the latest and greatest smartphone or laptop. We're sure seeing that with Apple products. Most people don't need the latest tech, but they just have to have it. This drives infrastructure growth, which provides greater capabilities for our electronic systems, which then pushes us to upgrade to the next service and technology. There's an exciting cycle of innovation going on at an ever-accelerating pace.

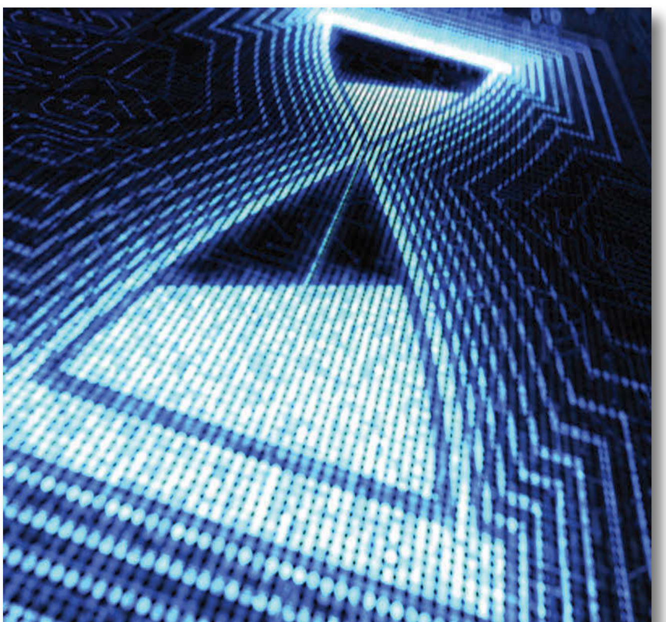
Recession-Proof

The market for electronics seemed to die with everything else in November 2008, but it was back on track within a few short months. Then, while the rest of the economy suffered,

electronic products had a pretty good year in 2009. 2010 was better yet and 2011 has been great for most companies until around September, when the business cycle peaked and orders began to drop off. We see this in the latest numbers from the IPC, where the orders have dropped significantly. So, on the whole, we might consider the industry to be recession proof, somewhat, but we aren't cycle proof. We'll likely have another chance to talk with Walt again at the IPC APEX EXPO at the end of February. We'll see what he says, then.

Managing Through the Cycle

To run our businesses effectively, we have to keep our eyes on a few things as we try to navigate the markets and prepare for the ups and downs, which is the nature of things. Of course there are those events that we don't have much control over, such as acts of God, economic calamities and new government policies. (The latter we have some control over if we're represented properly.) Still, one thing we do have some control over is the business cycle. It's fairly predictable, and we have plenty of time to adjust our business strategies to this cycle. There are leading indicators and people like Walt who can help us look out six months to a year. I don't know how many companies I've seen go out of business over the years as they initiate expansion plans at the top of a business cycle and ride the cycle down into insolvency. Pay attention. The trough we're in now (the bottom of the cycle) is why equipment manufacturers don't have any solid commitments, yet, for 2012. Companies are getting better at recognizing the business cycle and they're holding off until the leading indicators turn up.



If we can keep these in perspective, we'll do all right. Nobody likes to hear about the market declines, but if we get our houses in order, we can manage our businesses effectively in all phases of the cycle; if we're prepared properly, we can gain market share with every cycle and use it to our advantage. It really isn't rocket science. Again, we just need to watch those leading indicators, which Walt talks about extensively.

Electronics is a great business to be in. We enable just about everything new under the sun. If it's not a new technology or product, it's an older, established one being upgraded with new intelligence. Our challenge, as an industry, is to stay on top of economic changes as well as the changes in the way electronics are being made, which can and will have a dramatic effect on the way we run our businesses.

Game Changers

I'm sure most of us don't want to spend too much time trying to sort out the different drivers, which can make or break our businesses. Trade policies, new regulations and acts of God can turn our best-laid plans into mush in pretty short order. We need people like Walt and organizations like the IPC to watch out for us. Another potential game changer is innovation. Although this can be great for our businesses as new innovations in electronics take off, creating new markets for our services, there's a potential downside for those who have their heads in the sand. We have to keep our eyes and ears open to the opportunities, as well as the dangers all around us. The good thing about the cycle of innovation is that it usually takes time. It follows a logical pattern and, in most cases, we have time to prepare.

Printed Electronics Looks Hot

Something I've been talking about for years is printed electronics. This is a classic example of an innovative, enabling technology, which will transform many of the products we'll build for our customers in the years to come. Some see it as complementary, value-add to the products we build. Others see it as a game changer, able to ultimately replace the PCB and assembly industries as we know them.

The first week in December I attended the IDtechEx show in Santa Clara, California (Silicon Valley). This is my fourth year attending. Each year I've been impressed with how the show has grown in attendance and in the number of exhibitors by 20-30 percent. The amount of new materials and equipment is quite impressive. This industry will have a direct impact on PCBs and assemblies. It's certainly about to take off. It's close. Products are being built — lot's of them. E-readers and displays are some of the most obvious examples, but more are coming.

I'm starting to see more and more PCB guys walking the floor at the PE show, which is encouraging. The IPC held a PE standards meeting the day before the show, this year, which is a first, and Mary MacKinnon, IPC sales manager for IPC APEX EXPO, was walking the floor selling booth space for their new PE section at APEX this year.

So, what does 2012 hold for our businesses? We can expect little or no growth in most markets for the first half of the year, but then the market will recover in the fall. Now is a good time to get your house in order and prepare for the next uptick in the business cycle. **PCB**

Ray Rasmussen is the publisher and chief editor for I-Connect007 publications. He has worked in the industry since 1978 and is the former publisher and chief editor of *CircuiTree Magazine.* Ray can be contacted at: ray@iconnect007.com

---

ATOTECH EXPANDS FACILITY IN PODNART, SLOVENIA

SHORT

On December 6, Atotech inaugurated the expansion of its production site in Podnart, Slovenia. Approximately 500 guests joined the celebrations, including the President of Slovenia, Dr. Danilo Türk and Ciril Globocnik, Mayor of Radovljica. On the following day, a customer seminar for General Metal Finishing and Electronics was organized, "Green Technology for Green Environment." The seminar provided an outlook on the latest environmentally responsible solutions for the surface finishing industry, thereby offering the opportunity to discuss practical field experiences with Atotech's technologies at diverse customers' sites.

The new production facility of Atotech Slovenia covers a total area of approx. 3,200 m², hosting 29 mixing vessels for an annual production capacity of 15,000 tons, which can be further expanded to 20,000 tons/year. The new warehouse for raw materials and products occupies 2,067 m², providing a total warehouse capacity of 3,714 m². Additional 464 m² are dedicated to the laboratories, which are equipped with 20 cabinets for chemicals, a salt spray corrosion test chamber and several analytical tools for production quality control and customer support. Among the many capabilities of the new Slovenian laboratories are pH and density tests, titration, UV/VIS spectrophotometry, Atomic Absorption Spectrometry (AAS), chromatography (HPLC, IC), polarography, conductometry and Cyclic Voltammetric Stripping (CVS).

Atotech's new facilities in Slovenia are the latest addition to a constantly expanding global network, currently including three regional Headquarters, seven R&D centers, sixteen production sites and seventeen TechCenters. All Atotech sites around the world are equipped with state-of-the-art production technologies, cutting-edge scientific tools and advanced recycling and waste treatment plants, ensuring highest product quality and safety for workers and the environment.

Atotech is headquartered in Berlin, Germany and employs about 3,300 people in more than 40 countries.

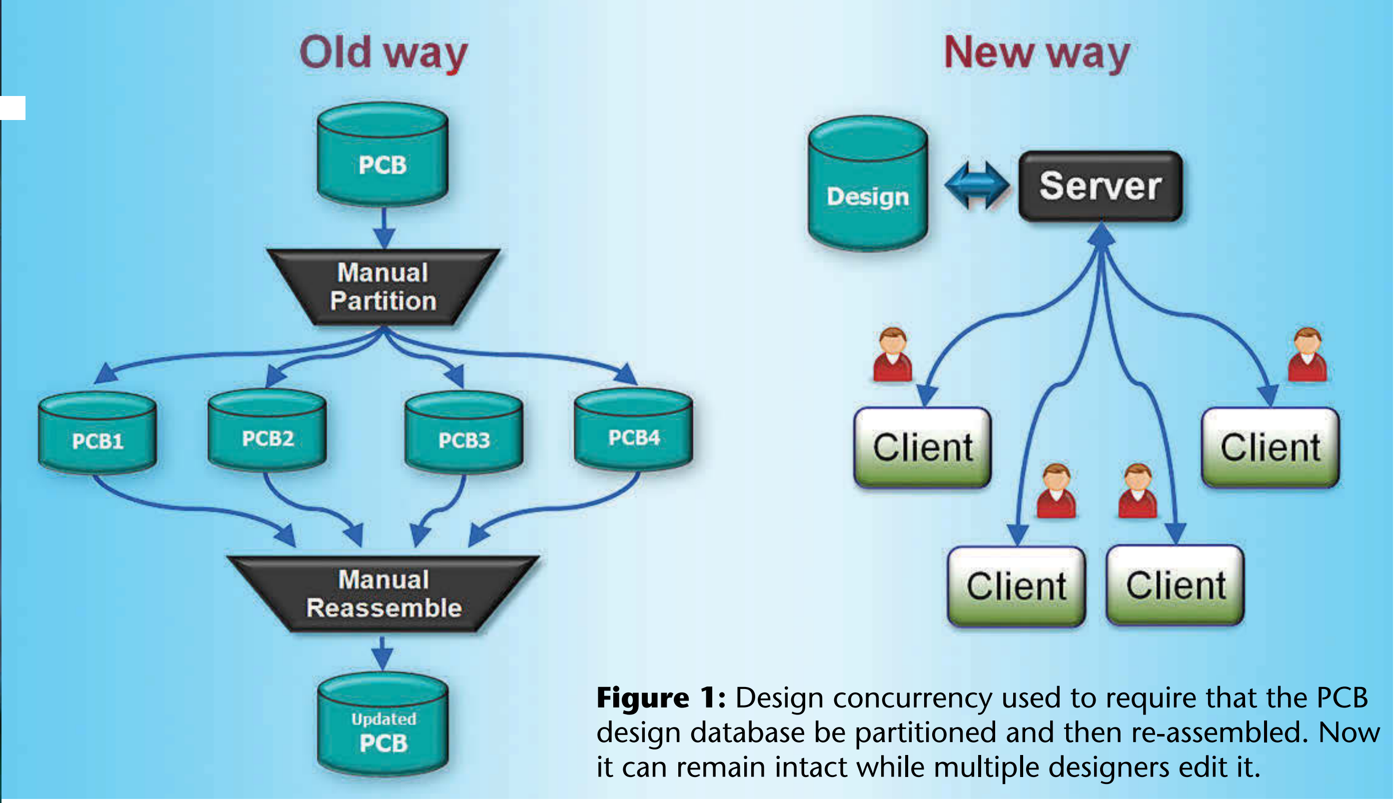


Figure 1: Design concurrency used to require that the PCB design database be partitioned and then re-assembled. Now it can remain intact while multiple designers edit it.

Working Smarter in 2012

by John Isaac
MENTOR GRAPHICS CORP.

SUMMARY

In a survey by Aberdeen Group, best-in-class electronics companies identified six design technology best practices that enable them to meet their aggressive business goals. These essential PCB technology practices are necessary components for companies in 2012, as the economy slowly improves.

As the electronics industry fights to recover from recession, there is an increasing pressure to enable complete, design-through-manufacturing process support. Electronics companies worldwide will have to deliver differentiated products to market faster and at lower costs, continuing a trend that has accelerated due to the soft economy. Even the China market for PCB systems design and manufacturing support tools was eager to capitalize on the latest technologies and increase their productivity.

In a survey by Aberdeen Group, best-in-class electronics companies identified six design technology best practices that enable them to meet their aggressive business goals. These essential PCB technology practices are necessary components for companies in 2012, as the economy slowly improves.

Here is an overview of those six key tactics that enable companies to work smarter, which we believe will be critical to execute in 2012.

TECHNOLOGY TACTIC 1: Product Development Process Concurrency

Concurrency is basically the ability to turn serial action into parallel and it comes in two flavors. The first is the ability to have multiple designers working on the same design process simultaneously. This is really nothing new, but the technology provided can make a tremendous difference in its effectiveness. Designers have always had the ability to take a PCB design database and partition it into pieces. Each designer then works on a section

> **“The answer for the future is to perform simulations in software, i.e., virtual prototyping.”**

of the design — but then the database has to be put back together. This merging is a time consuming and error-prone process, but the end result is a reduction of the design cycle time.

Now we have the ability for those same multiple designers to work simultaneously on the *same* database without partitioning. This applies to many processes in the PCB design including schematic entry, constraint (high-speed and manufacturing rules) entry and management, and physical layout. Each designer is also able to see the changes of their peers' real time. This significantly improves not only the design cycles times but also the productivity of the designers and the quality of the product. Some users of this technology report improvement in design times by 30 to 70%. To compete in 2012, companies will have to achieve similar design cycle improvements (Figure 1).

The second form of concurrency is the ability to run several different processes in parallel, versus serially. Schematic, constraints, layout and analysis can all be parallelized, thus further adding to the productivity of the designers and improvement in cycle times. However, this requires sophisticated design data management with versioning, synchronization, access and update controls as we will see later in this article.

TECHNOLOGY TACTIC 2:
Virtual Prototyping

Companies have traditionally verified their products by building and testing multiple prototypes. Design a PCB. Build a physical prototype. Test it in the lab. Determine what changes need to be made and repeat the process with design re-spins.

This methodology has several problems. First, it is very time consuming and expensive to build and test prototypes. When time-to-market is critical you may miss your market window. Secondly, testing in a lab may not highlight all possible errors. For example, if you want a product that is to be in the field for multiple years in harsh environments such as extreme vibration, "shake and bake" chamber testing may not run long enough to point out long-term problems. The same can apply to signal integrity. Corner case extremes may not be achievable in the lab.

The answer for the future is to perform simulations in software, i.e., virtual prototyping. These are performed during the PCB design process and can cover many possible domains: signal (digital, analog, RF) and power distribution network integrity; thermal and heat management at the IC, package, PCB and full-system simulation; vibration and shock (Figure 2); PCB fabrication and assembly practices; 3D mechanical interference and more. Performing these during and throughout the design process can ensure that the design proceeds without the need to back up and correct errors. Further, software can explore the corner cases as well as simulate long term problems in hours versus weeks and months in test chambers. Although designers love to get a piece of hardware in their hands quickly, and performing extensive virtual prototyping may delay this, the benefit is reduced cycle times and cost, as well as increased designer productivity and product quality/reliability.

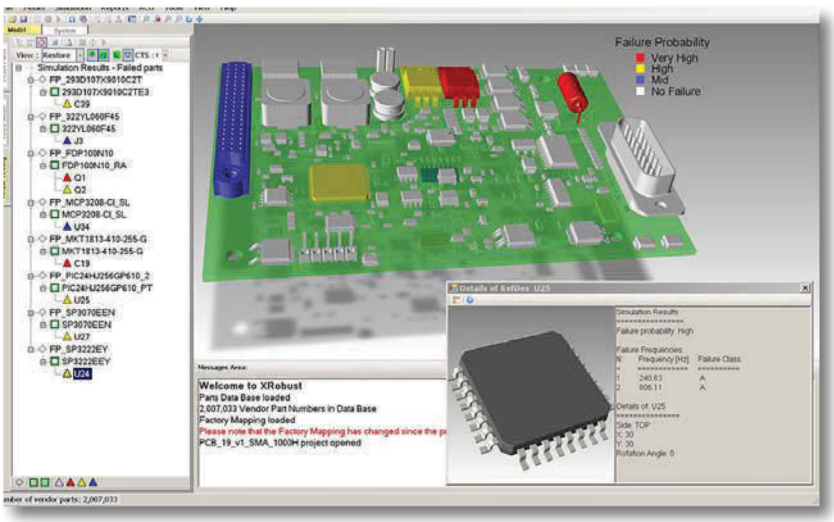


Figure 2: Vibration virtual prototyping can replace weeks and months of test chamber time with hours of software simulation, and highlight components (red, yellow, blue) with the possibility of failure.

TECHNOLOGY TACTIC 3:
Design-through-Manufacturing Process Support

Time-to-market and product cost are king in many industries. Even industries such as mil/aero and automotive, that in the past had long development times and/or high-cost limits, have changed to more aggressive goals. In addition, PCB designers must not forget that their responsibility does not end once data is passed to manufacturing. As well, from an EDA supplier point of view, it is important to provide support that does not end at design, does not enable the designer to easily fulfill their responsibility for a manufacturable product, and includes for the manufacturer to optimize their production lines to deliver the product at minimum cost.

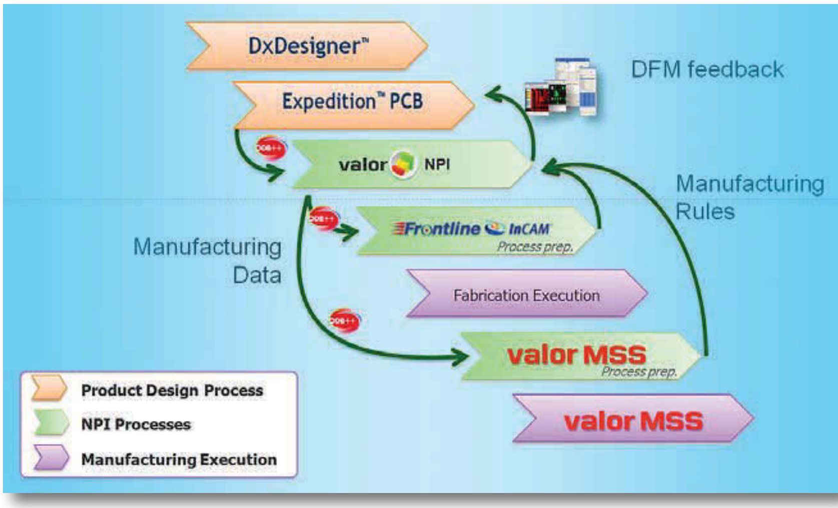


Figure 3: Complete design-through-manufacturing process support can guarantee high yields, high product reliability and lower production costs.

Figure 3 shows how the process flow is evolving. Support starts with the ability for the manufacturer to define rules and best practices that can positively affect the yields and reliability in the fabrication and assembly processes. These design for manufacturability (DFM) rules are then used by the PCB designer, during the design process. DFM software can be executed from the design environment, violations highlighted and then corrected by the designer. Coincidently, most manufacturers use the same set of rules and software to check incoming design data. This assures that once the design is passed to manufacturing production, it can proceed without the need for design re-spins.

Once the design reaches manufacturing via an intelligent interface like ODB++, the manufacturers have software that enables them to model their production lines and optimize its utilization. As the line is running, software continually monitors the line for just-in-time parts delivery, machine stoppages and product traceability. Even failures in quality assurance are tracked and can highlight a machine or process that has less than acceptable failure rates.

TECHNOLOGY TACTIC 4:
Complexity Management

For differentiated products that beat the competition, companies have to use the latest and most advanced technologies, push the limits of squeezing more functionality into less space, and still meet an aggressive market window; this will be even more important in 2012. Integrated circuit technology continues increasing density, high speeds, high pin counts at smaller spacing and high-power dissipations. Printed circuit board fabrication technology such as HDI/microvias can increase densities, yet are more complex to design. How do we maintain and improve the productivity of the designer in the face of this continuing complexity evolvement? The answer is to likewise increase the power of the design tools.

For example, not long ago, a typical design might have contained a few high-speed nets that had to follow length and adjacent rules. These few nets could be managed easily by the designer. Now, the majority of designs have greater than 50% high-speed nets and some have up to 90%. Another example is the increase of BGA pin counts and densities. This presents a challenge when trying to fan and break out on the PCB. These types of issues present a complexity problem that, without sophisticated design tools, would result in

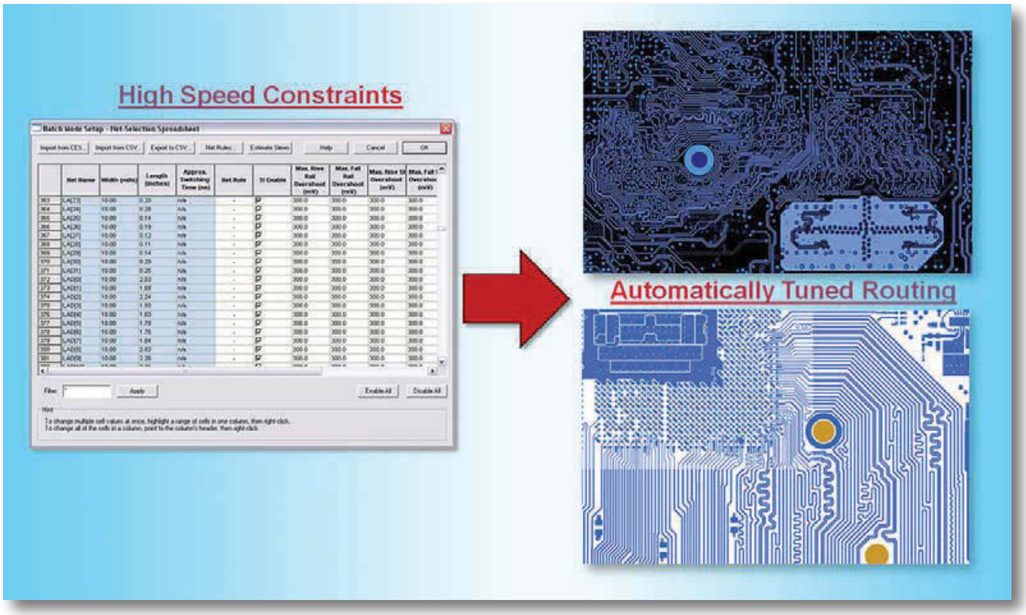


Figure 4: Routing automatically follows high-speed interconnect rules to meet performance goals.

severe reduction in productivity and increased time to market.

If we look at the first example of increasing high-speed net content, tools now exist that help the designer follow all of the delay and signal integrity rules. The engineer sets up these rules in terms of constraints and then the CAD designer routes the individual interconnects or bus structures. Automatic tuning of the high-speed nets then matches the lengths to the min/max constraints, as well as matches nets such as differential pairs. This task, if done manually, would be overwhelming.

TECHNOLOGY TACTIC 5:
Multidiscipline Collaboration

Several disciplines are required to develop and deliver a product. In the electronics domain we have IC, package, FPGA, RF, analog and digital specialists. We have engineers in the mechanical domain that design the enclosure and perform CAE analyses. We have procurement, supply chain and manufacturing. We have embedded software development. All of these need to collaborate efficiently throughout the development process. In the past, this was performed with paper and emails. Now it is done mostly electronically, but even then problems exist with en masse dumps of data going between team members. How do you effectively determine what has changed, how you, in your domain, should react and what controls you actually have? This problem must be dealt with in the near future.

It turns out than most interactions are actually a negotiation process. For example, if a mechanical engineer discovers that a component on the PCB is going to interfere with the physical product enclosure, the engineer might propose that the location of that component change. This would take the form of an incremental (only then change) proposal being communicated to the PCB designer — incremental as opposed to an en masse data dump which the PCB guy would have to sort through to determine the proposal. Incremental change capability was recently developed as a standard ("EDMD") by Mentor Graphics, PTC and users, and then approved by ProSTEP.

That proposal would be graphically represented to the PCB designer who could accept the proposal, reject it, or counter propose one that suits the PCB. This negotiation process would proceed until agreement is reached, at which point both the mechanical and electrical databases would be updated. This type of fully electronic collaboration is just one example of the many that exist today.

TECHNOLOGY TACTIC 6:
Intellectual Property (IP) Management

The last key to future success identified by best-in-class electronics companies is the management of their work-in-progress and library data. Design team members, whether local or dispersed around the globe, need to create access efficiently and control what the company considers one of their most valuable assets. Components are qualified and corporate librarians enter them into approved library management infrastructure where they can be accessed by the designers. Previously designed complete or portions of PCBs might be added to the library for reuse in later projects.

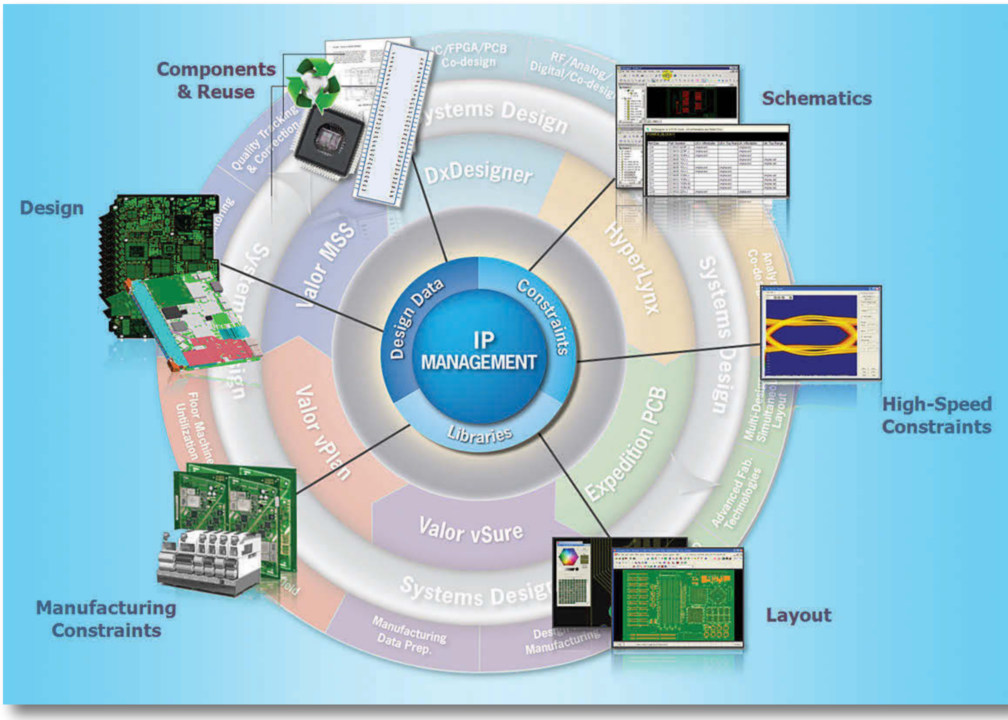


Figure 5: Management of electronic product IP is extremely complex and requires ECAD specialized software. Standard PLM systems cannot maintain the inter-relationships and synchronization.

As the PCB design proceeds, the team creates schematics, constraints and the PCB layout data. This work-in-progress data management is very complex and requires infrastructure systems specifically created to manage electronic IP. Data edited by different team members creates versioning and synchronization issues. The company might employ a service bureau to design part of the product and wants to share only that portion of IP. These are complexities (Figure 5) that preclude the use of standard PLM systems. That said, as the product matures, the finalized design data must be uploaded to the companies PLM, ERP, etc., systems for lifecycle management. So the ECAD supplier must provide not only the PCB specific IP management but also support standards that interface with corporate infrastructure systems.

2012 – Work Smarter

If we look back at 2011, we saw significant increase in the needs of the electronics companies as they strove to stay ahead of their competition in the face of the recession and its slow recovery. This was very evident in Mentor's Technology Leadership Awards program. Each year we ask companies around the world to submit their most aggressive designs and be judged by a set of independent industry specialists. The 2011 entries once again were extremely more complex and the methods and tools used in the design process more advanced. Using the design technology enablers described in this article in the future is no longer a luxury, but a necessity. **PCB**

John Isaac is director of market development at Mentor Graphics and currently responsible for worldwide market development for the Systems Design Division. He has worked in the Electronic Design Automation (EDA) industry with PCB and IC technology for more than forty years. Since joining Mentor Graphics, Isaac has held marketing positions in both PCB and IC product areas.

A New Slant on Matched-Length Routing

SUMMARY

This month, Barry Olney discusses the traditional serpentine routing for matched-length signals and looks at the octagonal spiral pattern, a potentially desireable alternative, especially if real estate is at a premium.

by Barry Olney
IN-CIRCUIT DESIGN PTY LTD, AUSTRALIA

For source-synchronous interfaces such as DDR memory, the data signals must arrive at the load at the same time as the clock or data strobe. This is done, for instance, by matching the lengths of all the signals between data lanes and strobes within a certain tolerance. For DDR2, this is 50 MIL within data lanes including the strobe. Address and command signals are not as critical with a tolerance of 200 MIL. However, matching the lengths does not mean that the propagation delay for each signal will be the same. Also, traces routed to length on different layers exhibit different delays and this is most evident when comparing microstrip (outer layers) to stripline (inner layers) and is due to the difference in dielectric materials surrounding the traces.

Typically, serpentine traces (or meander lines) are used to match the length of these critical signals assuming that the extra length of the serpentine pattern will be electrically the same as a straight trace and no parasitics are introduced. However, as technology advances, and demands for smaller traces with less clearance and faster rise times become more the norm, this assumption may no longer be valid.

Contrary to what you may believe, the propagation delay of a serpentine trace is less than the delay through an equivalent length straight trace. The signal is sped up

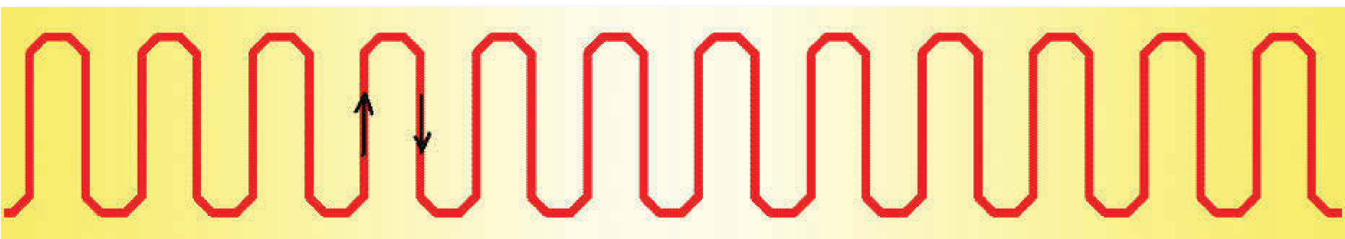


Figure 1: Serpentine trace with short-coupled U pattern and 5x trace width spacing

UNITS: MIL — ICD STACKUP PLANNER – www.icd.com.au — 12/1/2011 — Total Board Thickness: 62.6

Differential Pairs > USB | DDR2 | SATA | ETHERNET | LVDS

Layer Number	Layer Name	Material Type	Dielectric Constant	Dielectric Thickness	Copper Thickness	Trace Clearance	Trace Width	Current (Amps)	Impedance Characteristic(Zo)	Edge Coupled Differential(Zdiff)	Broadside Coupled Differential(Zdbs)	Description
		Dielectric	3.3	0.5								Soldermask
1	Top	Conductive			2.1	5	4	0.42	51.8	90.76		Signal
		Dielectric	4.3	3								Prepreg
2	GND	Conductive			1.4							Plane
		Dielectric	4.3	8								Core
3	Inner 3	Conductive			1.4	15	7	0.47	51.17	89.48		Signal
		Dielectric	4.3	12								Prepreg
4	VDD	Conductive			1.4							Plane
		Dielectric	4.3	4								Core
5	GND	Conductive			1.4							Plane
		Dielectric	4.3	12								Prepreg
6	Inner 6	Conductive			1.4	15	7	0.47	51.17	89.48		Signal
		Dielectric	4.3	8								Core
7	VCC	Conductive			1.4							Plane
		Dielectric	4.3	3								Prepreg
8	Bottom	Conductive			2.1	5	4	0.42	51.8	90.76		Signal
		Dielectric	3.3	0.5								Soldermask

Figure 2: Eight-layer stackup with 50 ohms characteristic impedance

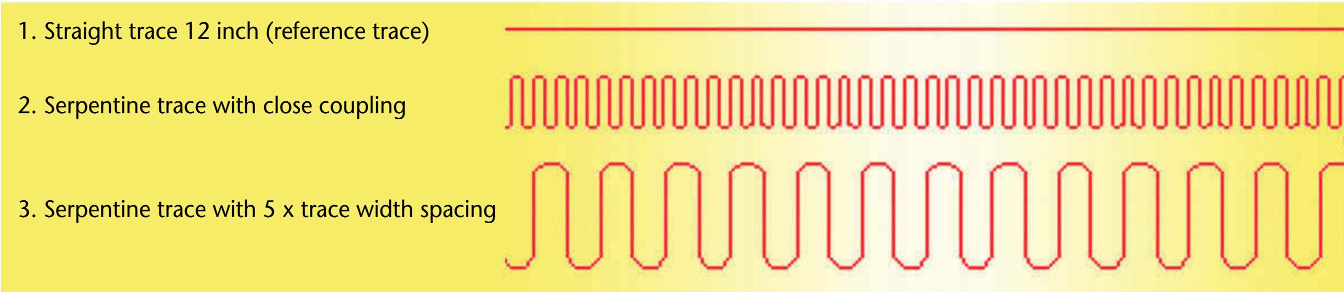


Figure 2a.

because a portion of the signal will propagate perpendicular to the serpentine. And, this also varies with the type of serpentine pattern used. For example, the serpentine pattern may have long parallel lengths spaced close together in the 'U' bend coupling the signal many times through the serpentine pattern. This self-coupling (forward and reverse crosstalk) shortens the electrical path.

Let's have a look at the effects of altering the patterns. First, we must build the stackup so that we have 50 ohms characteristic impedance on all signals so that we can compare apples to apples. Figure 2 was built with the ICD Stackup Planner (download from www.icd.com.au).

I first created a layout, placed three drivers and loads, then routed three different microstrip trace patterns with identical lengths and series terminators (Figure 2a).

All traces have the same attributes:

Driver:	CMOS, 3.3V, 300pS *
Trace length:	12000 +/- 10 MIL *
Trace width:	4 MIL
Impedance:	~ 50 ohm
Series Terminator:	43.8 ohm

As you can see, there is crosstalk between each segment of the closely coupled serpentine trace. The coupled region is actually the entire length of the serpentine trace although the

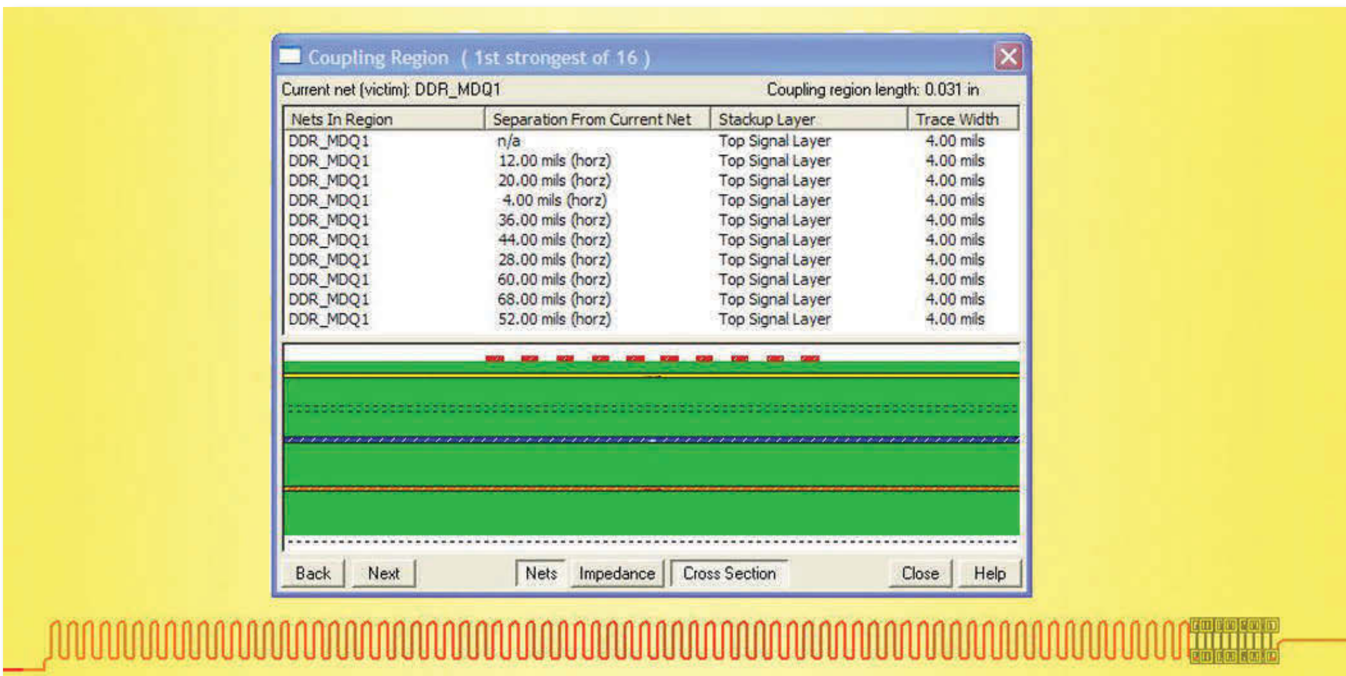
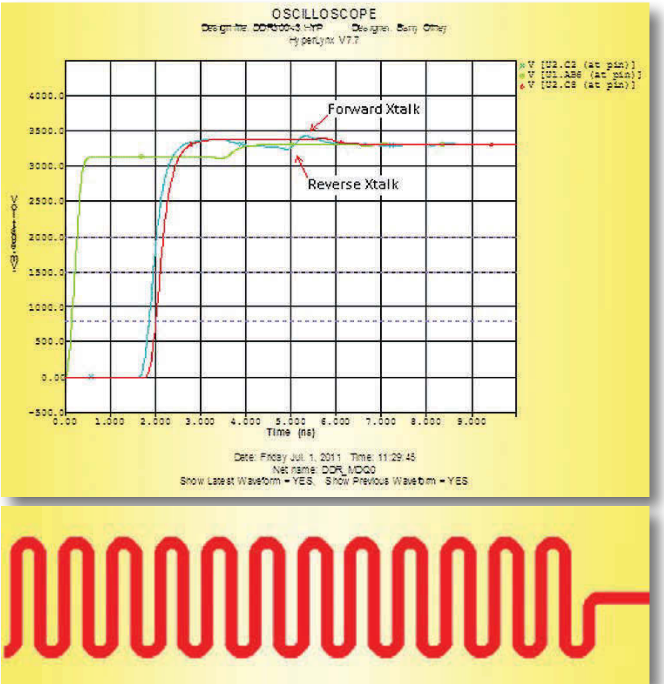


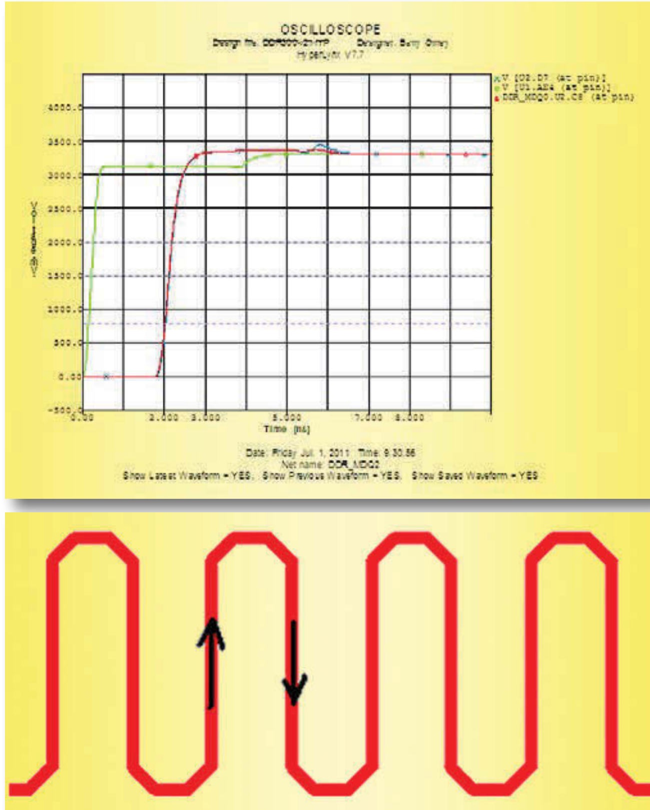
Figure 3: Serpentine trace with close coupling.



Green is driver
Red is straight (reference) trace.
Blue is the closely coupled serpentine trace which leads the reference trace by 150 pS.

The peaks in the blue serpentine trace (from 4 to 6nS) are the reverse and forward crosstalk respectively from the close coupling.

Figure 4: Closely-coupled serpentine trace vs. straight trace (microstrip)



Green is driver
Red is straight (reference) trace.
Blue is the 5 x trace width spacing serpentine trace which leads the reference trace by 15 pS.

The peak in the blue serpentine trace (around 5nS) is the forward crosstalk.

Figure 5. Serpentine trace with 5x trace spacing vs. straight trace (microstrip)

software only displays the first few segments.

* *This model and trace length were chosen to enable comparison to Howard Johnson's simulations* [4]

We can see from Figures 4 and 5 that the 5x trace width spacing serpentine trace has less noise and the delay matches the straight reference trace very closely.

For decades, RF engineers have been using printed circuit coils to create specialized inductors in their designs. Inductors may be etched directly into the PCB by laying out the trace in a spiral pattern. Some of these patterns are quite complex and are best left to the experts as many consider this a black art. In 1928, Wheeler first developed an equation to calculate the inductance of a planar octagonal spiral coil inductor.

$$L = \frac{a^{1.54} \,.\, n^{1.93}}{17.23 \,.\, c^{0.55}}$$

where n = number of turns
a = (Diameter max + Diameter min)/4
c = (Diameter max - Diameter min)/2

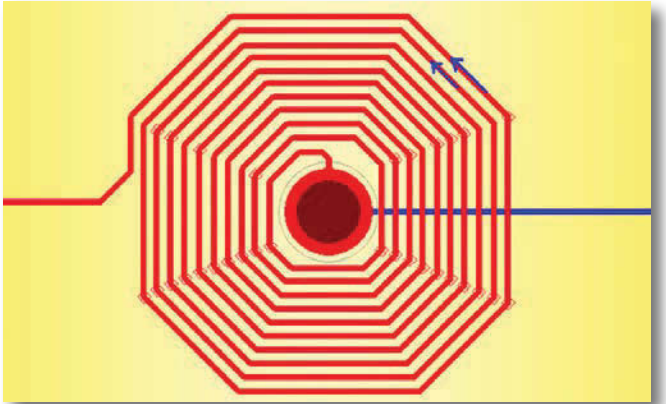


Figure 6: Planar octagonal spiral coil inductor.

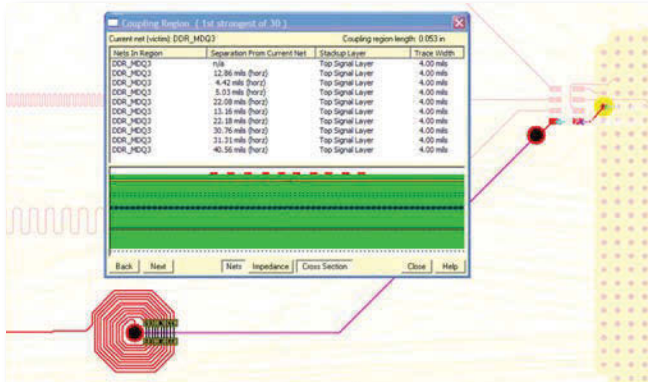


Figure 7: Octagonal spiral inductor forward crosstalk (microstrip)

The serpentine pattern was replaced by the octagonal spiral inductor in Figure 6. A trace was routed from the driver, through the series terminator, then to a via at the centre of the spiral. The pattern then spirals out using a 4-MIL trace with 4-MIL spacing to give 10.25 turns. Using Wheeler's equation above, the inductance is calculated at ~ 3nH.

Notice that the current flow in the coil is in the same direction. Therefore, there is no reverse crosstalk, but only forward crosstalk.

The coupled region is actually the entire spiral although the software only displays the first few segments. It is difficult to route with modern shape-based interactive routers as the trace tends to clean up behind the cursor, eliminating the spiral. With all glossing and post-route clean-up turned off, it can be completed quite quickly. The nice thing about the spiral pattern is that it is very compact.

Let's compare this to that of the serpentine trace, first simulating it on the outer (microstrip) layers.

Serpentine (blue) leads the straight reference trace (red) by 150 pS. Spiral (violet) is leading the reference trace by 22 pS.

As can be expected, the octagonal spiral inductor has a huge 500-mV peak from the effects of forward crosstalk. No reverse crosstalk is present.

Figure 8: Octagonal spiral inductor (violet) vs. the closely coupled serpentine trace (blue) – (microstrip)

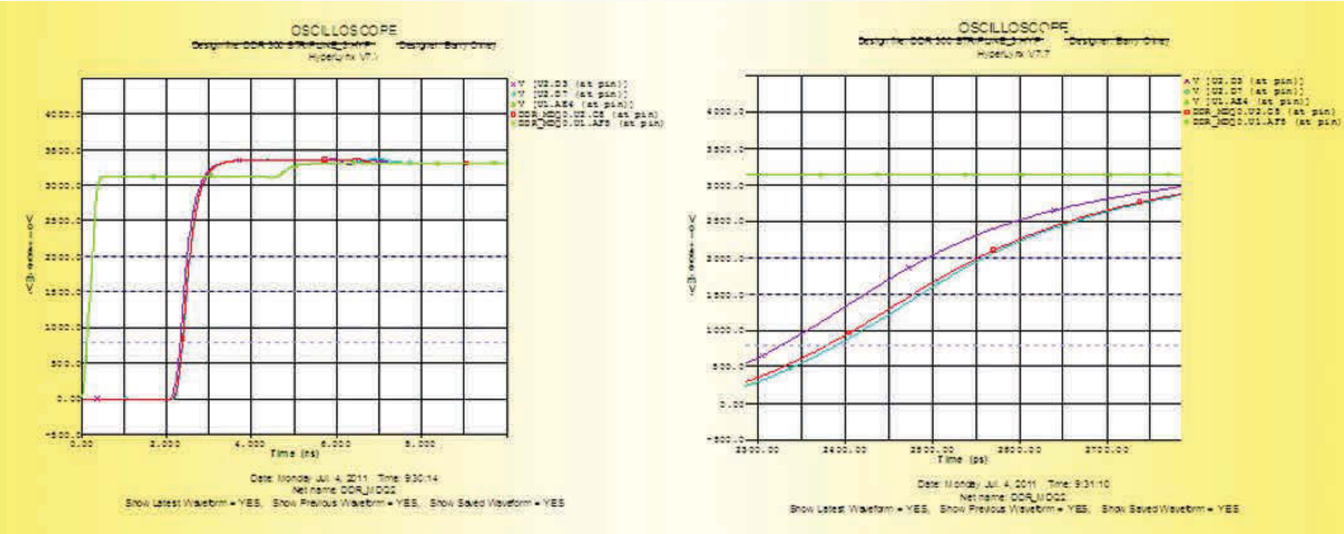


Figure 9: Serpentine (with 5x trace width spacing) vs. spiral pattern (stripline)

In this case, the spiral's delay is much less than the closely coupled serpentine trace. Since we would not normally have 12 inches of trace on a high-speed design, we can divide this by 6 to give us approximately 2 inches of length, which is more the norm. Therefore, the serpentine trace will be leading the straight reference trace by 25 pS and the spiral will be just 3.6 pS ahead of the reference trace. But, crosstalk has dramatically increased.

In theory, the forward crosstalk or far end crosstalk does not exist in the stripline configuration. So let's see how the spiral performs embedded between the planes in a stripline construction (Figure 9).

The straight reference trace (red), the

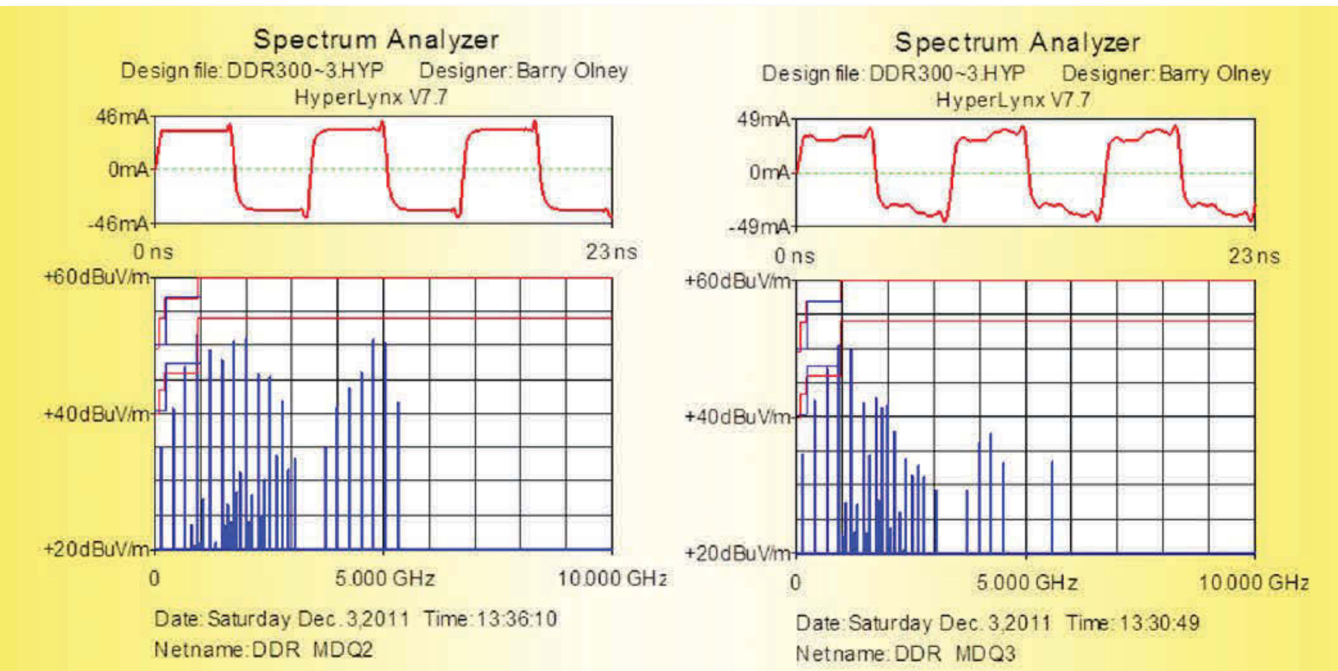


Figure 10: (left) EMI from the serpentine pattern (microstrip) 51.54 dB @ 952 MHz and 50.82 dB @ 4.76 GHz, (right) EMI from the spiral pattern (microstrip) 50.54 dB @ 952 MHz and 37.63 dB @ 4.24 GHz

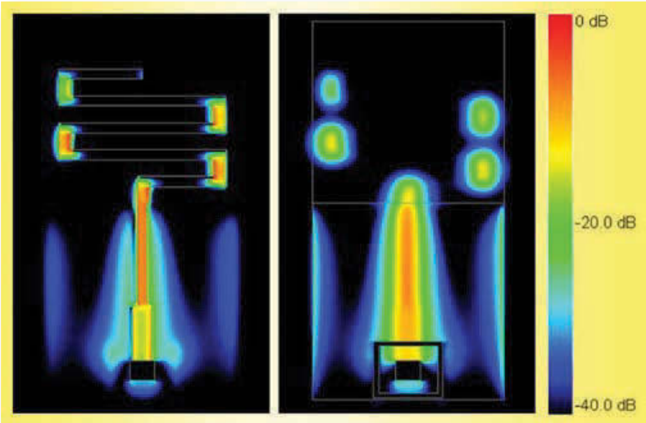


Figure 11: Current magnitude of a microstrip meander-line antenna [7] — meander trace (left) and the ground plane (right)

serpentine with 5x spacing (blue) and the spiral (violet) are now routed on layer 3 (stripline) and the forward crosstalk is eliminated as predicted. The spiral trace now leads the straight trace by ~ 60 pS and the serpentine lags by 10 pS. Dividing the results again by 6 (to bring us back to the real world) gives the relative delay at 2 inches: Spiral leads the straight line by 10 pS and the serpentine lags by 1.66 pS. So, with the negligible 10-pS lead, the spiral pattern has potential as an alternative to serpentine routing.

Normally, one would be concerned about a crosstalk level of 500 mV, which was exhibited on the microstrip spiral pattern (Figure 8), as it may well exceed the noise margin. But is this really crosstalk? Crosstalk is the unintentional electromagnetic coupling between traces on a PCB — normally between two or more different signals where one is an aggressor that imposes noise on the victims. But in this case, we have the same signal imposing noise on itself. So the peak that we have called forward crosstalk is actually just overshoot of 500 mV at the top on the rising edge. This overshoot does not affect timing and has little effect on the signal integrity. But does it create radiation (EMI)?

For the stripline configurations, where the waveforms are almost identical for both serpentine and spiral patterns, the EMI is also identical. But for microstrip configurations, the radiation is quite different. One would not normally route 12-inch traces, as previously stated on high-speed designs, so we can expect the level to be high in both cases.

It is interesting to note in Figure 10 that the radiation level below 3-GHz peaks at exactly the same frequency (51 dB @ 952 MHz) for both the serpentine and spiral, which is the 7th harmonic. However, at higher frequencies the radiation rolls off for the spiral pattern emitting 13 dB less radiation, with fewer harmonics, around 5 GHz. This is interesting because one would expect this to be the opposite since overshoot or ringing is generally a source of radiation.

The serpentine pattern on the microstrip layer acts as an antenna emitting radiation from the ends of the 'U' bends. A Meander-line Antenna (MLA) is a type of microstrip antenna first developed in 1953, but recently become very popular in mobile devices as it is extremely compact, low profile and easy to fabricate. The MLA can be a half wavelength dipole or a quarter wavelength ground plane format.

One can see (Fig. 11) the large magnitude of current on the end sections of the meander-

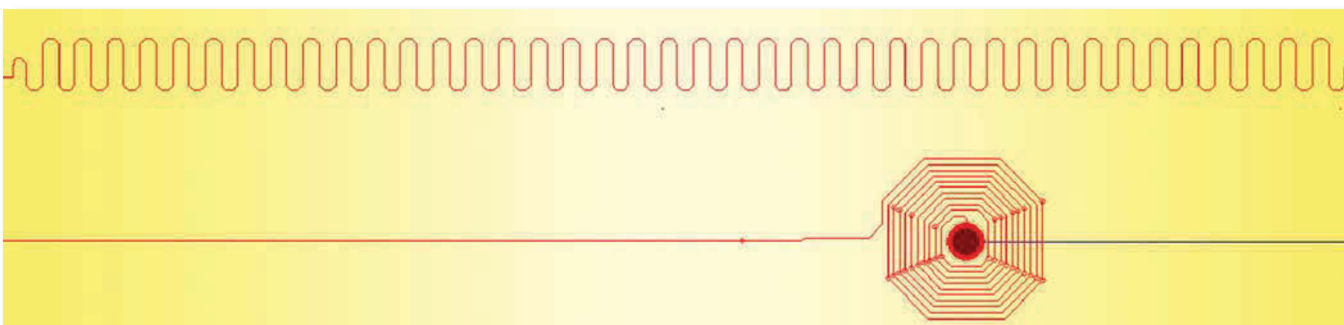


Figure 12: Area of the serpentine pattern (with 5x trace width spacing) vs. spiral pattern.

Reliability in Medical Electronics

by Real Time with...productronica 2011

Editor Ray Rasmussen sat down recently with Dr. Markus Riester of maris TechCon, who discusses some of the issues he addressed during a panel session on reliability in medical electronics at productronica 2011.

realtimewith.com

---

line. These are essentially the only source of radiation — the wave resonates between the microstrip antenna and the ground plane. The serpentine pattern may not be an efficient antenna but nevertheless is acting as a 50-ohm antenna, as the harmonics approach a multiple of a quarter wavelength, since it is driving a high impedance load.

In conclusion, serpentine traces are commonly used for matched length signals and supported by all popular EDA tools. It is therefore easier to follow tradition and use this pattern. It would be best using the serpentine pattern with at least 5x trace width spacing in a stripline configuration to reduce crosstalk and radiation.

However, as technology advances and demands for thinner traces with less clearance and faster rise times become more the norm, one may have to look at alternatives. As you can see from Figure 12 below, the octagonal spiral pattern takes up less than ¼ of the space (area) of the serpentine pattern. So, if real estate is at a premium then maybe it's time to give the old spiral a go. **PCB**

References

1. PCB Design Techniques for DDR, DDR2 & DDR3 — Barry Olney
2. Advanced Design for SMT — Barry Olney
3. Design for EMC — Barry Olney
4. High-Speed Signal Propagation, Advanced Black Magic – Howard Johnson
5. Method for Compressing Serpentine Trace Routing on Printed Circuit Boards – Bogusch, Ernest B
6. Transmission Line Design Handbook – Wadell
7. The Study and Implementation of Meander-line Antenna for an Integrated Transceiver Design — MinJie Ma, Kai Deng
8. The ICD Stackup Planner can be downloaded from www.icd.com.au

Barry Olney is Managing Director of In-Circuit Design Pty Ltd. (ICD), Australia, a PCB Design Service Bureau and Board Level Simulation Specialist. Among others through the years, ICD was awarded "Top 2005 Asian Distributor Marketing and "Top 2005 Worldwide Distributor Marketing by Mentor Graphics, Board System Division. For more information, contact Barry Olney at +61 4123 14441 or email at b.olney@icd.com.au.

Make Maximum Mileage

by Martyn Gaudion
POLAR INSTRUMENTS, LTD.

SUMMARY

2012 may just be the year when the industry will work to optimize existing technologies and maximise yields rather than go for anything ground-breaking. Reliability, durability and economy — not a bad place to be in tougher economic times.

I make no apologies for the non-metric unit's title for this first Pulse of 2012, since the metric "Make Maximum Liters/100km" doesn't alliterate quite so well, but for January, I will do as asked and use my crystal ball to look ahead at the next year from a PCB signal integrity perspective.

Just as toughening economic circumstances encourage people to make maximum gas mileage, so do companies look to make the maximum returns from existing technology when the market is uncertain or slowing. That doesn't mean background work is happening on new and breakthrough technology, but often the breakthroughs are held back until the market is in a more positive frame of mind.

So what might be happening on the technology front from a signal integrity perspective over the next 12 months? I would put my money on the industry getting the maximum from existing technologies, rather than seeking any risky breakthroughs, though rewards for breakthroughs could be significant, especially if they are able to deliver higher speeds and more storage, whilst keeping prices down.

Still, my money is on the fine-tuning and maximising yields of existing technologies whilst the underlying market conditions clarify; i.e., are we headed for another slowdown? Or, is this just a routine cyclical dip in the industry as predicted by Walt Custer. I, for one, hope that Walt is correct — and that as he predicts, we will see business start to rebound toward mid-2012.

One of the benefits of a market slowdown (you have to be optimistic to be in this business, don't you?) is the opportunity to fine-tune existing designs and maximise yields — hence the title of this column. When the market moves fast and one new technology seems to follow hot on the heels of another, there is little time for shakedown and optimisation of the existing technology. This is something that I often observe in the annual visit to Munich for productronica/ electronica; in the German market there seems to be a much stronger desire to fine tune and perfect the existing rather than relentlessly looking for the next great breakthrough. It is an interesting approach and one that contrasts

"When the market moves fast and one new technology seems to follow hot on the heels of another, there is little time for shakedown and optimisation of the existing technology."

with the American high-technology approach which seems to tend towards replacing one breakthrough product innovation with the next at the fastest possible product development cycle. An interesting effect of that cross-Atlantic cultural divide in thinking is that in 2011, the German foreign ministry[1] decided to migrate their PCs to Windows XP — yes, XP — not Vista or Windows 7.

A temporary slowdown in the market ensures that companies look to maximise yields and get the most from existing designs, and new design work is focused on areas which will yield tangible benefits — tangible enough to be economically justifiable even in a slowdown.

It is always of great interest to me how some technologies seem to take off rather quickly, and others last for decades, sometimes outliving and outperforming interim standards which come and go. Ethernet communication wiring and protocol (Figure 1) is a case in point.

Many pretenders to the throne have come along, and Ethernet seems to win in the end. Once in place and in widespread use, existing technologies, especially proven and reliable ones, become hard to displace.

One underlying trend seems to be to make communications, especially high-speed serial comms "adaptive," i.e., they are self-adjusting to the available channel bandwidth. This is a very clever way of making fault tolerant — or perhaps I should say "performance" tolerant — architecture. Traditionally a weak link in a comms channel could cause the channel communication to break down. With adaptive architecture the transceivers test the capacity of the channel and run the link up to the capability measured in its internal testing. This technique of being able to run the channel up to the capacity of the weakest link in the chain interests me as it does add a degree of future proofing into systems. Perhaps in 2012 this type of adaptive technology will become more widespread.

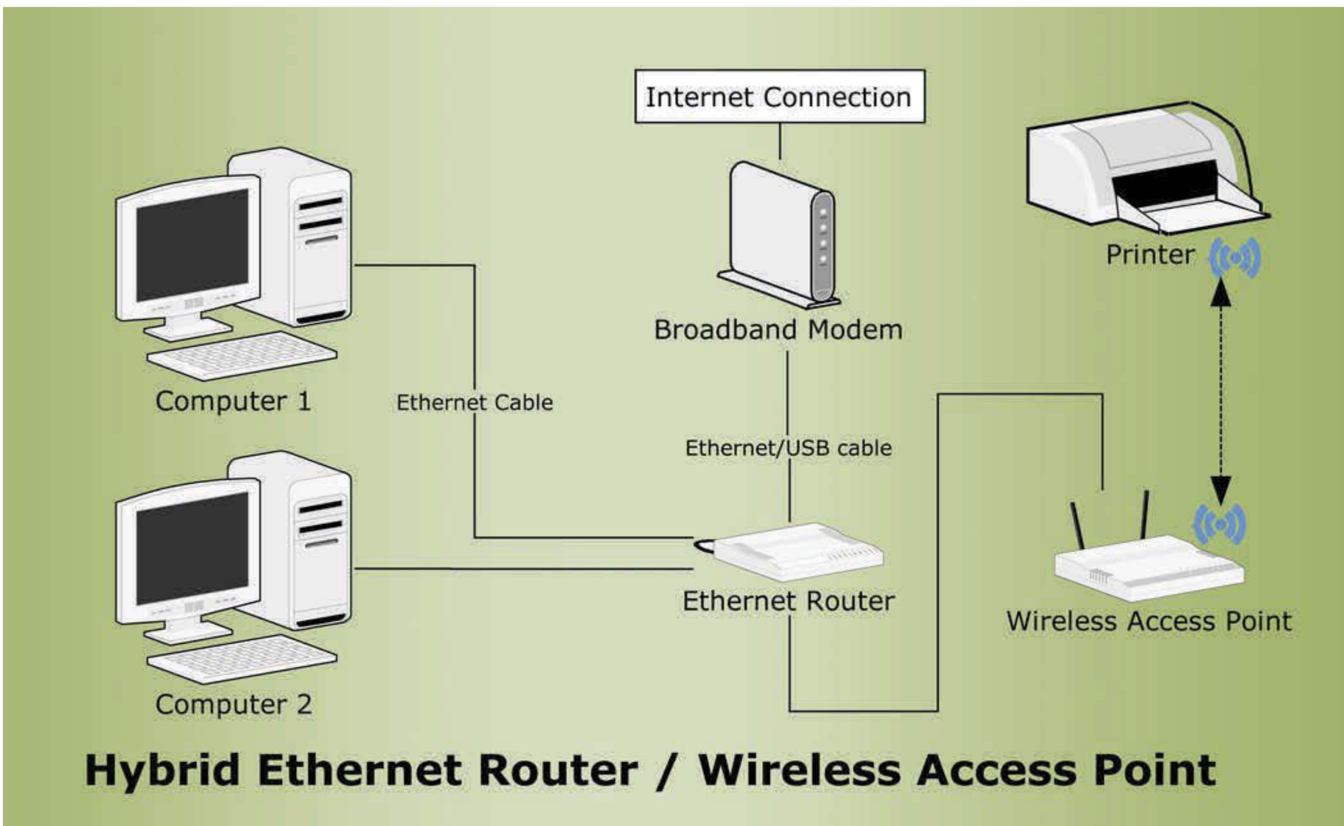


Figure 1: Typical Ethernet small office network.

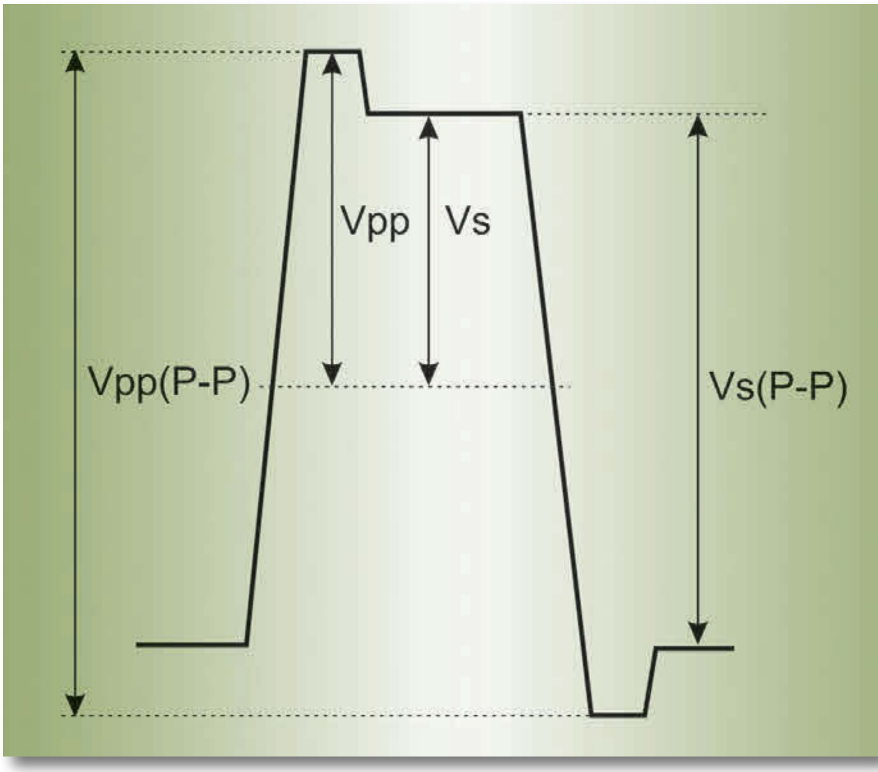


Figure 2: Pre-emphasis waveform with steady-state voltages boosted to VPP

It does allow for systems to be updated and then run faster as budgets dictate that the weak links are updated. ADSL internet connections are a good example of this — apart from the fact they are marketed the other way with the maximum speed advertised and the "up to" part in the small print! Nonetheless, when an adaptive system is updated and the weakest link is brought up to speed, all the users get an instant performance increase without the need to re-equip. Figure 2 shows typical waveform pre-emphasis — commonly employed to improve signal quality.

Tough economic times bring out the best in creative people, as in times of boom it is seemingly easy just to throw money at a problem. The lack of budgets mean that performance increases in systems need to be thought through and made as cost effective as possible, so in our PCB world it will not surprise me to see renewed emphasis placed on getting maximum performance out of economic base materials.

On the subject of technologies and the speed of adoption, the prize of optical communication on PCBs seems always "just around the corner;" and a pretty long corner at that, but maybe one day this will happen. I can recall in the late 1990s some leading PCB shops that were very excited about optical channels being fabricated in a similar method and embedded as layers on traditional PCBs by employing printed polymer layers as optical waveguides. What happened to that? Maybe one of the other PCB007 columnists can enlighten us to any progress in this direction.

Back to the main focus of Making Maximum Mileage, and just as investment regulators say "past performance is not always a prediction of future performance," I would hazard my best forecasting on 2012 being a year when the industry will work to optimize existing technologies and maximise yields rather than go for anything ground breaking. Reliability, durability and economy — not a bad place to be in tougher economic times. **PCB**

References

1. www.theregister.co.uk

Martyn Gaudion, CEO of Polar Instruments Ltd., began his career at Tektronix in the early 1980s. He joined Polar Instruments Ltd. in 1990 and was appointed CEO in 2010. Gaudion writes occasional articles for a number of PCB industry publications and regularly contributes to IPC high-speed, high-frequency standards development activities. Look for his regular column, The Pulse, appearing monthly at PCB007.

FLEX007 Highlights

All Flex Expands Flexible Heater Line
All Flex Flexible Circuits has expanded its flexible heater offerings. In addition to custom flexible heaters, All Flex now offers standard catalog-based heaters.

New Path to Flex and Stretch Electronics
Imprinting electronic circuitry on backplanes that are flexible and stretchable promises to revolutionize a number of industries and make "smart devices" nearly ubiquitous.

Happy Holden: Printed Electronics Won't Replace Flex
Happy Holden, "Mr. HDI," sits down with Guest Editor Michael Carano to discuss his keynote presentation at ECWC. Happy explains why printed electronics will never replace flexible circuits.

Canadian Circuits Completes Upgrade, Increases Capacity
Flex maker Canadian Circuits finished a complete upgrade of its facility in Surrey, British Columbia.

DuPont Materials Enable Hypothermia Prevention Gear
DuPont Electronics & Communications and North American Rescue launched a line of hypothermia prevention thermal wraps, built upon DuPont's Kapton polyimide film.

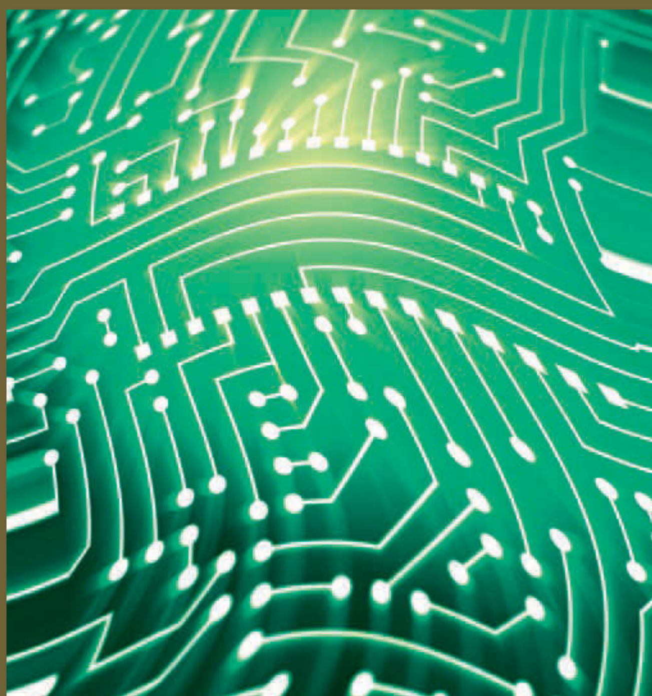
IPC Forms PE Management Council Steering Committee
IPC has created a new IPC Printed Electronics Management Council Steering Committee that will oversee the development of educational programs to support the printed electronics industry.

N.A. Flex Shipments, Bookings, B2B Down in October
Flexible circuit shipments in October 2011 were down 11.3% and bookings declined 10.3% compared to October 2010.

EPTE Newsletter from Japan: The Beginning of a Collapse?
Laminate manufacturers in Taiwan and Korea have unveiled plans to expand plants. Some newfound business stems from the devastating flood in Thailand, and the shift in the flexible circuit business from Japan. Will any of this business return to Japan?

TTM Wins Stoke's Supplier Excellence Award
TTM Technologies was named the 2011 winner of Stoke's new Supplier Excellence Award. TTM achieved a five-star supplier rating in the categories of technology participation, quality control, responsiveness, on-time delivery and continual improvements to costs.

The Way I See It: Printed Electronics in the Middle
There's an interesting convergence happening in the printed electronics industry, and the 2011 Specialty Graphic Imaging Association show, along with other recent news, illustrates where we are, and more importantly, where we're headed with PE. Hold on...here we go!



Color in Photoresists and Soldermasks

By Karl Dietz
KARL DIETZ CONSULTING, LLC

SUMMARY

Before entering the world of electronics, where I was surprised and puzzled by the presence and role of colorants in photopolymer formulations, it was my privilege to have worked in the field of dyestuffs and colored pigment research. Color in photoresists has been addressed in this column before, but it's time for a renewed discussion.

I always had a fascination with colorants, as a tool for artistic expression or the basis of a growing chemical industry in the 19th century, as the testing ground for an emerging patent system, or the gateway to new synthetic methods and chemical innovations. It was my privilege to have worked in the field of dyestuffs and colored pigment research before entering the world of electronics, where I was surprised and puzzled by the presence and role of colorants in photopolymer formulations. Since the functionality of color in this application was not readily apparent to a novice like me, I did some digging and came up with the following:

Color in Soldermasks and Coverlays

The traditional color for soldermasks has been green (Fig. 1), maybe for historical reasons only or because of the good contrast the color of copper (see discussion below). However, there have been other colors, notably black and white. One application for white soldermask is packaging of LEDs that emit white light. Non-white soldermask will affect the hue of the emitted light which is undesirable.

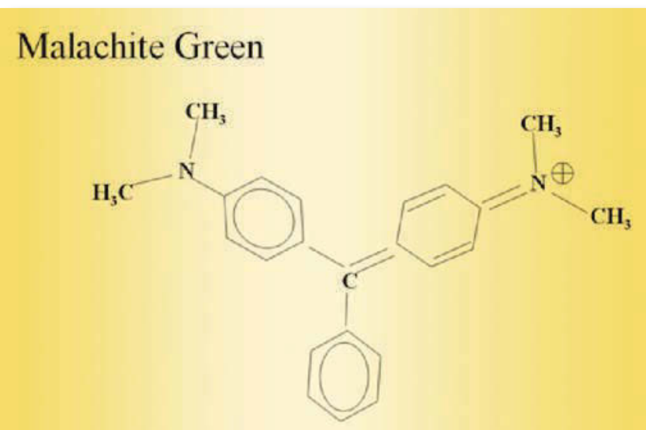


Figure 1: Example of a popular green dye stuff.

Circuit boards and components cover with black soldermask, coverlay, protective coating seem to appeal to Apple Inc. which perceives an aesthetic value in such appearance.

Photoresist Color as Visual Aid in the PWB Fabrication Process

Colorants have been used in photoresist formulations to assist the human eye in the resist application steps to:

- Better visually judge the uniformity and completeness of substrate coverage with liquid photoresists
- Make it easier to align dry film resist to a panel on a hot roll laminator
- Create an optical image counterpart ("print-out image") to the latent polym erization pattern for visual verification of image registration
- Have a good visual contrast of the developed resist pattern for easier inspection ("contrast to copper")

Early liquid formulations were either colored or had to be dipped into a colorant solution after the resist coating. The depth of color could be used as an indicator of resist thickness distribution, and coating defects, such as pinholes, were easier detected. This function of color was less important with dry-film photoresists, but color helped to judge the proper placement of the resist on the board and to make necessary adjustments.

First, there was a variety of photoresist colors in use, but blue and green shades have become most popular. The human eye perceives these hues to give the best contrast to the pinkish color of a copper surface. The pink-orange of copper and the bluish-green tints of


INTEGRAL TECHNOLOGY
Circuit boards in smartphones are under a lot of stress.
8:44
BEND
HEAT
DROP
Zeta® products relieve it.
ZETA®
NEXT GENERATION HDI
STOP PAD CRATERING NOW!
949.587.3370 www.HDIoptionstoday.com

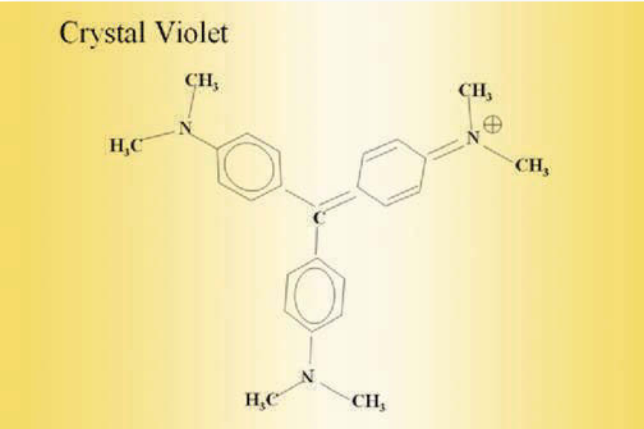


Figure 2: Crystal violet.

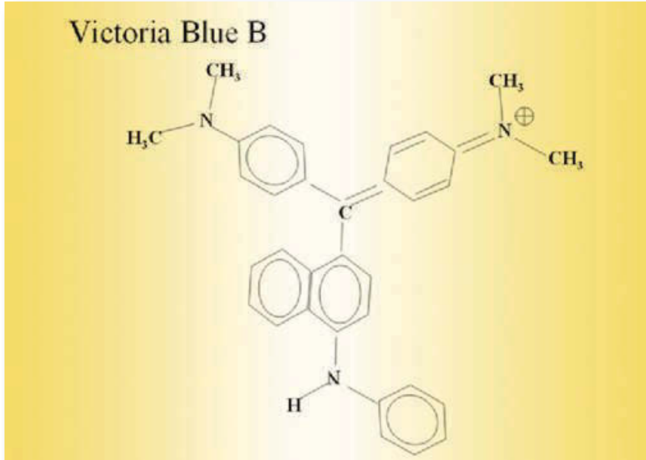


Figure 3: Victoria Blue.

the resist happen to be close to a pair of colors called complementary, a pair of colors which is perceived by the human brain to show good contrast. In addition, one could argue that the eye is most sensitive in the 550-nm (green) wavelength range. Typical colorants belong to the family of triarylmethane dyes or pigments. Pigments are less soluble than dyes and tend to have better light fastness. Examples of triarylmethane dyes are crystal violet and Victoria Blue (Figures 2 and 3).

After exposure, the resist has a "latent" image of polymerized and unpolymerized areas without visual contrast, unless the formulation contains ingredients that develop color or fade away upon exposure. The visual contrast of exposed areas to unexposed areas is helpful to production operators to verify that boards have been exposed, or to check the registration of the image relative to tooling holes or conductor vias. If exposed resist has a deeper color it is referred to as a print-out image. Conversely, if the exposed color is lighter, it is called a fade-out image. This fade-out characteristic is sometimes also referred to as photofugitive.

A typical color former or color precursor is a "leuco" (white) dye from the family of triarylmethane dyes. The leuco form may be a reduced, colorless compound (Fig. 4) which oxidizes when exposed to form the color. Leuco crystal violet and leuco malachite green are examples.

To form a fade-out image, a color or color precursor needs to be chemically altered during the exposure process to non-colored, or non-colorforming species. An example of such chemistry is found in the familiar diazo phototool chemistry: a stabilized diazonium salt, part of a color precursor package, loses its diazonium group (N2) during exposure to form a phenol derivative which is incapable of linking with a coupler molecule to form the azo dye in the ammoniacal developer (Fig. 5).

Both approaches, print-out and fade-out images, are being offered in commercial products, but the print-out image seems to be more prevalent. However, a fade-out image may have the advantage of avoiding or minimizing a potential problem in some applications: occasionally, colored exposed resist leaves an undesirable stain after stripping on the substrate. This is particularly objectionable in photochemical

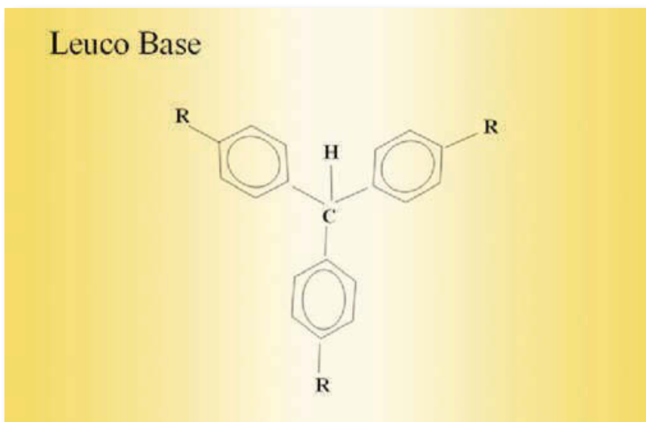


Figure 4: Example of a colorless dye precursor.

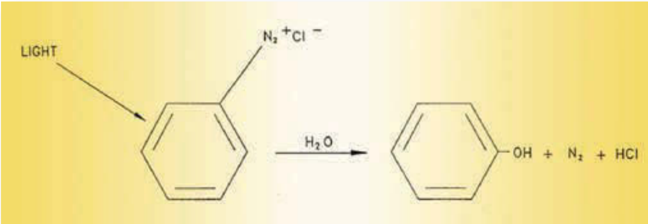


Figure 5: Exposure of the diazonium salt results in nitrogen loss and formation of phenol which cannot couple to form a dye.

machining applications and a fade-out image may be the answer to this problem.

It is important that dyes and pigments have a low-absorption "window" in the 350-410 nm region so that little radiation energy is taken away from the photoinitiation process of the radical polymerization. Since all dyes show some absorption in this UV region, they tend to detrimentally affect the photospeed of the resist, so that depth of color and print-out image may have to be traded for higher photospeed.

Some dyes actually help initiate the photopolymerization. In dye-sensitized systems, dyes can play a role in an electron transfer process, typically an electron transfer from an activator to a photoexcited dye. However, these systems are not commonly used in photoresists because they lack adequate shelf life. Dyes capable of such electron transfers include structures from the families of xanthene, acridinium, phenazine and thiazene dyes.

Photoresist Color in Automatic Exposure Systems and Laser Direct Imaging (LDI)

Automatic exposure systems which use CCD cameras (charged coupled devices) for board-to-photomask alignment, rely on a minimum transparency of the colored photoresist for this alignment process to function properly. First, yellow light shines through registration holes and is detected on the other side by CCD cameras. Based on the coordinate information of the registration hole, the phototool is then moved in the right position relative to the holes. This registration process is typically done in two steps, a rough pre-registration, followed by a precision adjustment. The registration hole also serves as reference point for front-to-back registration.

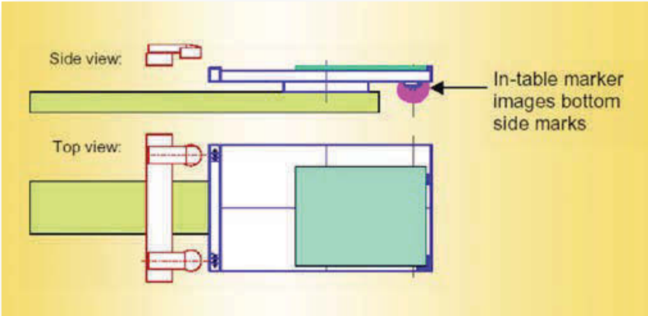


Figure 6: Marking the bottom-side registration target (courtesy: Orbotech).

When LDI exposure machines were first developed, tool suppliers quickly learned that predrilling boards for registrations purposes was not always acceptable to the fabricator. They prefer to position this relatively dirty process after the exposure, development and etching processes (post-etch-punch). An alternative was to use colored fiducials to aid front-to-back registration.

This is achieved through the "print-out image" characteristic of the photoresist, a color change of the resist during exposure that makes the latent polymerized features visible to the eye. The panel is placed on the machine's table for topside exposure. As vacuum is applied, the markers, typically non-laser light sources, placed inside the table start to image the targets on the bottom side of the

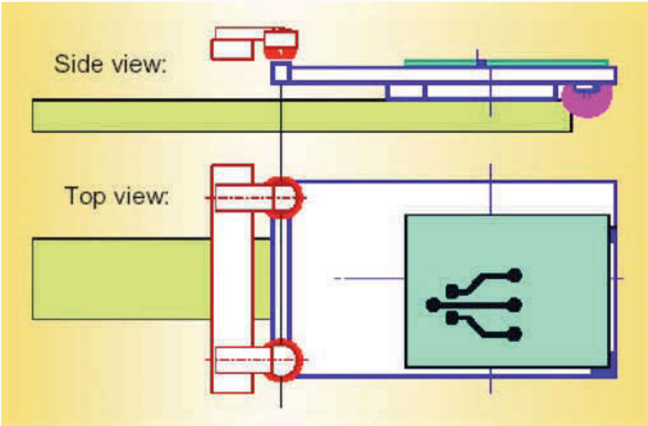


Figure 7: Imaging the top-side (courtesy: Orbotech).

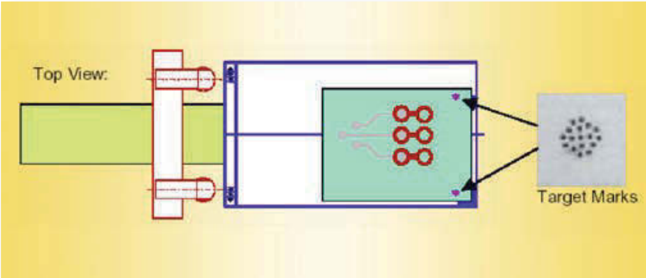


Figure 8: Locating the bottom-side targets, registration and bottom-side imaging (courtesy: Orbotech).

panel. CCD cameras then align the panel to the table as the panel moves in and the top-side is imaged during the reverse movement. After flipping the panel, CCD cameras locate the target marks on the bottom side (now the top side). Registration and imaging follow.

Photoresist Color and Automatic Optical Inspection (AOI)

Some PWB fabricators like the option of inspecting the developed resist image on the copper clad laminate before etching, a procedure that can easily cause resist pattern damage due to handling. Nevertheless, this inspection may be useful in a troubleshooting mode, and the resist color becomes an issue in such an inspection step. In this case one has to distinguish between reflectance type AOIs and (laser) fluorescence mode AOIs.

Regarding the reflective inspection, blue/green dry films which absorb strongly around 660 nm work well. The filter used in the AOI is orange, and the resist appears black to the machine while the copper appears bright. Red-colored photoresists are a rare species nowadays and are more difficult to inspect, but can be handled with a green filter in the AOI. One application challenge is in tent and etch board inspection. If the resist surface curves down into the hole and the resist surface is shiny, contrast degrades because the resist sends a reflection into the diffusive light gatherer, thus producing a signal relatively high versus the copper. Obviously, a matte resist surface will be less of a problem. Very thin resists also present a challenge because, for a given level of coloration, thinner resists absorb less than thicker resists (see Beer's Law). However, with some fine-tuning, reflectance mode AOIs seem to have no problems distinguishing resist from reflectant copper.

There is an interesting, special situation regarding AOI testing in the reflective mode in the presence of resist: the inspection of innerlayers after etching, with a permanent innerlayer resist remaining on top of the copper traces. To the best of our knowledge, there is no volume production experience for this scenario. Basically, the reflectance of the copper through the resist has to be high enough to provide sufficient contrast to the lower reflectance of the adjacent dielectric. If the coloration of the resists is low, e.g., to gain acceptable photospeed of the resist, this condition is generally met. However, what can interfere is a highly reflective, shiny resist surface.

Regarding the fluorescence mode AOI, the inspection of a resist image on solid copper, before etching, has not presented problems. Fluorescence is stimulated by a 442-nm (He-Cd) laser. The normal point of inspection is a board after etch and strip. The dielectric, sometimes formulated to include fluorescence enhancers, emits fluorescence in the 525-575 nm region. The copper, on the other hand, will reflect a good portion of the 442-nm light, and, to make the copper look black, the AOI's detector is filtering out this wavelength. For resist inspectability on copper, the resists have to absorb in the 442-nm region and fluoresce in the 525-575 nm range. Experience shows that they do. The "window dyes" of the photoresist have an appreciable absorption around 442 nm (and above) and may assist in this process or, at least, won't interfere. **PCB**

Karl Dietz is president of Karl Dietz Consulting LLC. He is offering consulting services and tutorials in the field of circuit board and substrate fabrication technology. Contact Dietz by e-mail at karldietz@earthlink.net or phone (001)- 919 870 6230.

Cost Savings of Copper Wire Bonding

by Real Time with...productronica 2011

Copper wire bonding offers a substantial cost saving over gold wire bonding in packaging applications. Atotech's Global Product Manager for Selective Finishing, Mustafa Ozkok, describes to Editor Pete Starkey how electroless palladium, direct on copper, offers an ideal substrate finish.

realtimewith.com

VIDEO INTERVIEW

EPEC MARKS 60TH ANNIVERSARY; PROUD TO BE AMERICA'S OLDEST

As the oldest production PCB company in North America, perhaps in the world, the 60-year story of Epec is connected to the development of the PCB industry.

Epec, through the merger of Electralab and Printed Electronics Corporation (EPEC), are proudly two of the five founding members of the IPC, the 2,900 member trade association supporting the $1.5 trillion global electronics industry.

In incubating new technologies, the Epec team continues to break new frontiers as a global leader in the engineering and manufacturing of customized built-to-print electronic products, including PCBs, flex and rigid-flex circuits, membrane switches, graphic overlays, silicone rubber keypads, custom battery packs, and EC fans and drives.

From pioneering innovation in the PCB industry with R&D, training and setting professional core values, the legacy of Epec has now passed to a new generation of very bright young people, but still continues the great tradition of imagination.

President Kendall Paradise shared, "I am extremely proud to be a part of our history and honorably recognize the leaders who first built our company: Al Hughes of Electralab and Richard Zens of President of Printed Electronics Corporation."

Founded in Massachusetts in 1952, Epec is celebrating the 60 year anniversary with events throughout 2012.

Epec has built the industry's leading supply chain platform and technology team, with Epec UL certified manufacturing operations in Shenzhen, Taipei, and North America, to manage the needs of customers' product life cycles.

With dedicated employees throughout North America and Asia, Epec Engineered Technologies has become one of the fastest growing companies in the Industry. For more information, visit www.epectec.com.

SHORT

2012: The Year of Retrenching?

by **Steve Williams**

SUMMARY

Will 2012 be the year that U.S. PCB manufacturers get back to basics? With U.S. consumer confidence dropping below 40%, businesses across all product lines are failing at unprecedented rates. Redefining business models and returning to core competencies may be the only way to hold off the steady decline of the American PCB footprint.

Change is Constant

When Asia started to threaten the North American PCB industry, it began with Japan, which was replaced by Taiwan, which was replaced by China as the current, dominant, low-cost solution for a broad range of technologies. One thing is certain, the evolution will continue as a new low-cost country (LCC) emerges in the next three to five years to replace China, and the cycle will perpetuate. To stay in the game, America needs to reshape, realign and refocus on supporting the niche pockets that LCCs cannot.

Global PCB Landscape

Global revenue for printed circuit manufacturing has shown a staggering 50% growth over the past five years, with sales predicted to exceed $76 billion in 2012. What makes this number so impressive is that it has come during a period of increasing technology, combined with overwhelming cost pressures. The just-published report *Printed Circuit Board: World Outlook* is forecasting that the APAC region will continue to lead the world in PCB manufacturing growth in 2012. There are currently 2,940 PCB manufacturing operations globally, with roughly 50% in China.

American Decline Continues

The news is not so positive for U.S. printed circuit manufacturers. While IPC has reported an increase in North American top-line PCB sales in recent years, this is an increasingly misleading metric. Much of reported N.A. PCB sales are generated by U.S. companies that are outsourcing manufacturing to China or other low-cost countries. The volume of PCB manufacturing that is actually performed in the U.S. continues to decline, as does the number of PCB operations domestically (Figure 1).

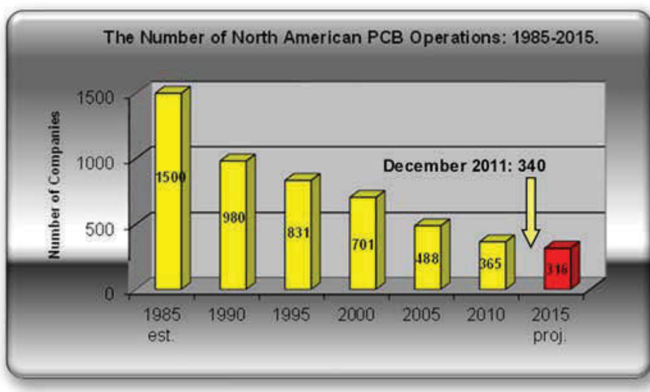


Figure 1: The decline of the North American PCB Industry. (Source: Harvey Miller-FabFile Online 2012. 2015 projection based on 3% reduction/year SUW 2012.)

Technology and Speed

Two niche areas where America has always maintained an advantage over LCCs are technology and speed. We must take steps to preserve these offerings and avoid complacency by believing that these are "untouchable." Case in point: When I ordered our first iPod, I did so on the Apple website because they offered free custom laser engraving on each unit (a side note on the customer experience: as I typed my message in the dialog box, I watched it magically appear on the back of an iPod displayed on my screen so that I could see exactly how it would look). I was told I would receive an e-mail with tracking information when my order was shipped. The next morning, I received an e-mail with the tracking number, not from Cupertino, CA, but from Shenzhen, China. Just 72 hours after placing the order, my custom, laser-engraved iPod arrived on my doorstep! Technology and speed…

Strategic Alliances

One strategic decision that is being made more frequently each year is for U.S. manufacturers to align in some fashion with a partner in a LCC. There are three basic ways to accomplish this: alliance (partnership), joint venture (partial equity position) and direct ownership (full equity position). Each will serve its purpose under the right circumstances, however, care must be taken to choose the right partner if going down the alliance path. Without an equity position there is little to stop a partner from eventually cutting out the middleman in this relationship (kind of like pulling your best customer from a salesperson and making it a house account). When properly executed, this strategy is extremely effective in allowing U.S. companies to capture business that would have been lost had they not had this option to offer their customers.

Embrace Lean

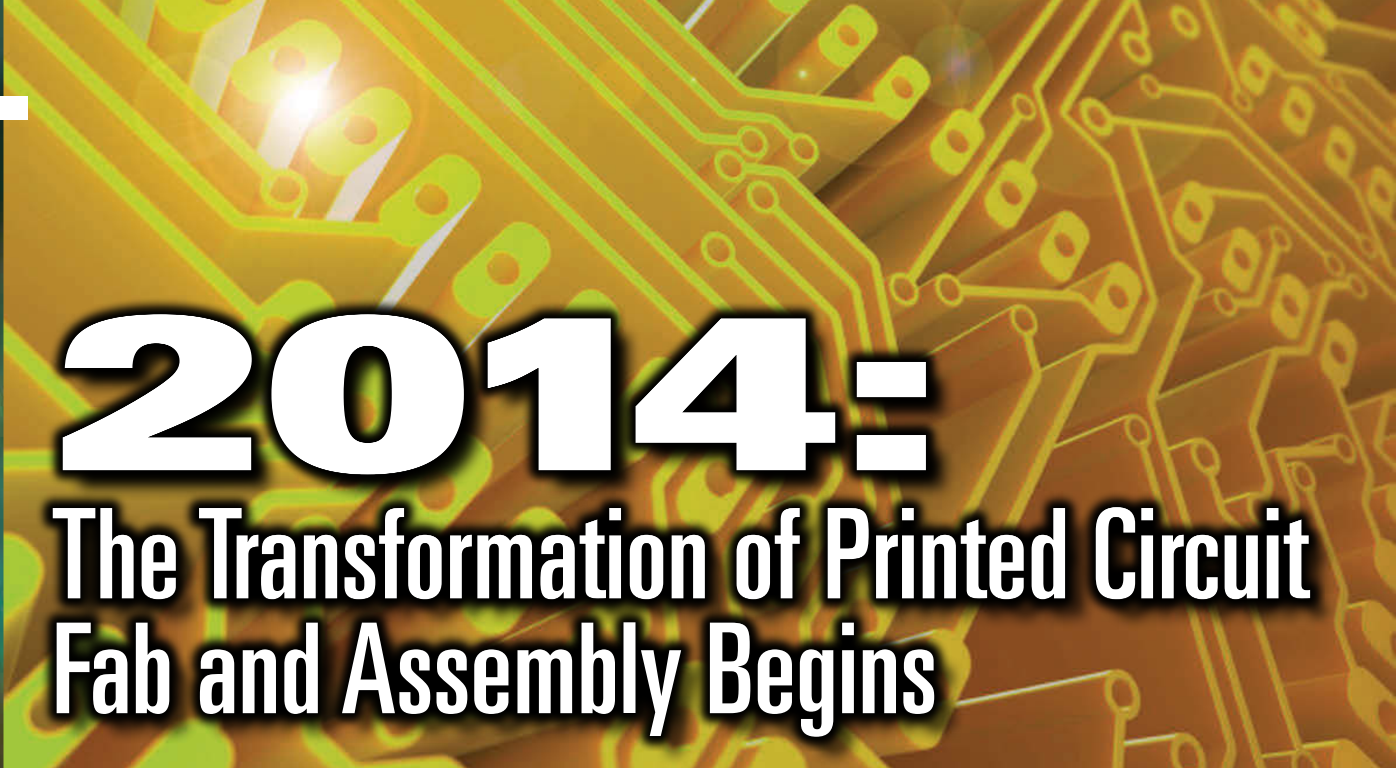
Only recently have American companies begun to get serious about Lean; not because they want to, but because their foreign competition is forcing them to. U.S. manufacturers as a whole have resisted the Lean manufacturing philosophy that Asia has embraced for decades, which has a direct correlation to our inability to remain globally competitive. It's always about the dollars, and the bottom line is that customers end up paying for any companies' inefficiencies in one way or another. In the highly dynamic environment that we all play in today, the major cost drivers are cost and flexibility, both of which can be improved dramatically by Lean practices.

Survival Is Not Mandatory

In the global economy that is today's business environment, there are no guarantees. In this environment, big-box retailers want to be your one-stop shop, where you can wander through stadium-sized warehouses purchasing anything from enormous high-definition TVs to equally enormous jars of pickles, and everything in between. In this environment, your small neighborhood grocer, drug store and gas station can no longer compete and are being pushed away by the giants of industry. In this environment, the big continue to get bigger through acquisition or elimination of the competition. Never before has American manufacturing had to look in our rearview mirror as we do now. With the exodus of American products, jobs and technology to LCCs accelerating at unprecedented rates, the threat has reached critical mass. Indeed, survival is not mandatory! **PCB**

Steven Williams is a 35-year veteran in the electronics industry and an authority on manufacturing and management. He is currently the commodity manager for a large global EMS provider, a distinguished faculty member at several universities and author of the book Survival Is Not Mandatory: 10 Things Every CEO Should Know About Lean (www.survivalisnotmandatory.com), and writes a monthly column, Point of View, for The PCB Magazine.

ARTICLE



2014: The Transformation of Printed Circuit Fab and Assembly Begins

by Harvey Miller
I-CONNECT007

SUMMARY

The 2011 Top 10 North American-based PCB fab companies exemplify the global scope of pathways to survival — the main thrust of this Part I — to that new day to come.

"China saved us"
— Tom Hicks, former head of Hicks, Muse, Furst and Tate, referring to Viasystems, in The Asian Wall Street Journal, December 3, 2003.

"The dynamic market demand such as request for smaller form factor, diverse features, lower voltage, higher electronic current, increased frequency, higher reliability and lower cost, all drive the technology development of the industry, including the responsive change of layout in embedded substrate. Whether active or passive components, resistor or capacitor, there are both advantages and drawbacks; as such, full comprehension and evaluation of those devices are a must."
— E. Jan Vardaman, president and founder of TechSearch International SiP at Global Summit, Taiwan, September 9, 2011 (SEMI).

Introduction

The former of the seemingly dissonant quotations above describes an important aspect of today's industry, one that preoccupies the attention of many. This first article of two will analyze the impact on the PCB industry.

The 2011 Top 10 North American-based PCB fab companies (Table 1), and the Top Ten of 2010 (Table 2), are rich models of the state of North American electronic manufacturing. Table 1 exemplifies the global scope of paths to survive to that new day to come, and Table 2 provides significant comparisons, showing just how the market leaders are "surviving."

Anyone who has lived in the dynamically changing electronics industry for any length of time has seen many technologies, and the companies that espoused them, become obsolescent and disappear. For survival's sake, it's wise to stay a little ahead of the curve. The problem is identifying tomorrow's problems and their potential solutions. What's behind that upcoming curve will be the focus of Part 2 — future opportunity, which is much needed in the beleaguered U.S. manufacturing sector. For now, Tom Hicks' exclamation above provides a good reference for understanding the present and how we arrived.

PCBs, since the emergence of integrated circuits in the 1960s, have been forced to meet the interconnection needs of their ever-increasing density.

“For survival’s sake, it’s wise to stay a little ahead of the curve. The problem is identifying tomorrow’s problems and their potential solutions.”

TOP 10 GLOBAL AND NORTH AMERICAN PCB FAB FOOTPRINTS--PRELIMINARY 2011
revised December 20, 2011

N.A. BASED COMPANIES' TOTAL PCB FAB CALENDAR 2011 SALES (pro forma estimates)	NON- N.A. FAB FACILITIES	NON- N.A ORIGIN $M	N.A. FACILITIES	N.A .FAB ORIGIN $M	TOTAl FAB $M	RANKINGS* GLOBAL	N.A.
TTM (including MEADVILLE GROUP HK--excl B/P assy)	8	870	6	449	1319	1	1
MULTEK (div of FLEXTRONICS -excl materials,displays)	7	845	1	45	890	2	8
VIASYSTEMS (excluding assy)	4	748	2	138	886	3	3
MFLEX (**excluding** assembly $510M est.= $860M total)	4	350	0	0	350	4	
SANMINA-SCI	3	226	3	120	346	5	4
DDi-DYNAMIC DETAILS	0	0	7	265	265	6	2
3M (Singapore and CA)	1	150	1	15	165	7	
ENDICOTT INTERCONNECT	0	0	1	85	85	8	5
ADVANCED CIRCUITS (COMPASS DIVERSIFIED)	0	0	3	80	80	9	6
AMPHENOL PCB OP'NS	1	20	1	54	74	10	7
TOTALS	**28**	**3209**	**25**	**1251**	**4460**		

Assembly excluded in all flexible circuit estimates

* for N.A.-based companies

Table 1.

Consider the imperative of Moore’s Law of Electronic Markets: “Transistor count must double every two years.” This enables new applications, creates new equipment markets and obsoletes old ones. PCBs, pushed by IC’s increasing complexity and pulled by electronic equipment demands for smaller, lighter, thinner, have gone down paths that worked — in the past. However, the future convergence of these forces of push and pull will require the interconnection industries to go back to the drawing board, sometime after 2014.

Back to the Present — How we got Here

North American PCB production declined from $11B in 2000 to $3.3B in 2010. Chinese PCB production increased from $4B in 2000 to $22B in 2010, or 47% of global PCB output, according to the IPC World Market Report.

Why? Mainly because those legions of chemical engineers, etc., have succeeded in creating a high labor cost PCB fabrication and assembly monster. (See Table 3, Cost model comparisons U.S. vs. China.) Of course supply chain proximity is another factor, but the migration of equipment production, too, was originally a result of lower labor cost.

As discrete transistors were replaced by ICs and as IC packages presented with increasing lead counts on smaller pitches, PCBs moved from 1-sided to 2-sided to multilayer to HDI. Assembly moved from through-hole to surface mount. I’m not criticizing those engineers. They were just doing their job, solving problems in ingenious ways — and creating new problems.

Before 1989, OEMs typically manufactured, as the acronym stands for original equipment

TOP 10 GLOBAL AND NORTH AMERICAN PCB FAB FOOTPRINTS--FINAL 2010
revised June 2, 2011

N.A. BASED COMPANIES' TOTAL PCB FAB CALENDAR 2010 SALES (pro forma estimates FINAL 2010)	NON- N.A. FAB FACILITIES	NON- N.A ORIGIN $M	N.A. FACILITIES	N.A .FAB ORIGIN $M	TOTAl FAB $M	RANKINGS* GLOBAL	N.A.
TTM (including MEADVILLE GROUP HK)	8	768	6	502	1270	1	1
MULTEK (div of FLEXTRONICS) -excl materials,displays	7	800	1	45	845	2	8
VIASYSTEMS (including MERIX)	4	658	2	148	806	3	3
MFLEX (**excluding**assembly $450M est.)	3	350	1	30	380	4	
SANMINA-SCI	3	218	3	122	340	5	4
DDi-DYNAMIC DETAILS (incl CORETEC)	0	0	7	268	268	6	2
3M (Singapore and CA)	1	150	1	15	165	7	
ENDICOTT INTERCONNECT	0	0	1	100	100	8	5
AMPHENOL PCB OP'NS	1	30	1	60	90	9	7
ADVANCED CIRCUITS (COMPASS DIVERSIFIED)	0	0	3	75	75	10	6
TOTALS	**27**	**2974**	**26**	**1365**	**4339**		

Assembly excluded in all flexible circuit estimates

* for N.A.-based companies

Table 2.

MANUFACTURING FABRICATION COST COMPARISON – U.S. vs. CHINA

RIGID PCBs

Pre-Ship Cost Savings 32%

CATEGORY	CATEGORY CONTRIBUTION TO TYPICAL MFR COST STRUCTURE	SAVINGS FROM MFR IN CHINA	RESULTANT COST SAVINGS
Materials	40%	20%	8%
Labor (Direct/Indirect)	25%	75%	19%
Overhead/ Equipment	22%	10%	2%
SG&A	8%	20%	2%
Other	5%	20%	1%
TOTAL	100%		

CAM Engineer makes $6K in Asia and $60K in the US

Utilities are very expensive with power being 3x what it is in the US, equipment costs the same for everyone.

Table 3.

manufacturer. That's a joke. IBM, HP, Cisco — you name it — do not manufacture much, except for wonderful designs and software to play great applications that have transformed our lives, with more and better to come. By the end of the 1980's, OEM managements realized that the increasing complexity of manufacturing was a diversion from their main business charter, better left to experts. Then the CFOs sharpened their pencils and looked at their spread sheets and you know where that manufacturing migrated.

Comparisons Do Matter

Now, let's look at a comparison of the first four entries of the Top Global and North American Fab Footprints, 2010 vs. 2011. These were selected to demonstrate the advantage of China operations at this point in PCB technology development. (Analyses of the remaining 6 will be included in Part 2.)

As for methodology, a few things to consider:

- Fiscal years are adjusted to calendar to allow for "orange to orange" comparisons
- Acquisitions are dated as if January 1 for pro forma figures
- Assembly, components, other products are netted out for fabrication only
- Since 2011, last calendar quarter is not complete at time of writing; all figures are estimates
- Since official figure derivations are not always transparent, estimates are used based on metrics, analyst meetings, etc.

TTM

I wouldn't say that China saved TTM, but the recent merger with Meadville by TTM was smart. It gave TTM an inside place in the high-growth, mobile electronics race as a true partner to a first class group of people. And, it goes beyond business opportunities.

An abstract from TTM China, Development of Electro-Optical PCBs with Polymer Waveguides for High-speed On-board Optical Interconnect, previously published at I-Connect007, serves as an example of one of the future developments that will replace today's interconnections, which will be discussed in Part 2, coming next month in The PCB Magazine.

TTM, like IBM, GE, DuPont, etc., knows that no nation has a monopoly on brains. Here's to rooting for the tide that lifts all boats.

Note that geographical diversification rewarded TTM with increased total sales, even though U.S. sales were adversely impacted by declining Mil/Aero procurements.

MULTEK

Parent Flextronics is a $32B EMS company, second only to Foxconn in that category.

It is not very transparent reporting its component division sales, in which Multek resides. Multek Flex competes in the tablet and smart phone businesses against MFLEX and TTM. In 2007, Multek Flex was #12 on Dr. Nakahara's NTI list with $770M. Now it is #14, with operations in Germany and Brazil, as well as China, along with the former Sheldahl operations in the U.S.

It is important, not only because of its size but because it combines PCB fabrication with assembly capabilities, a model appropriate for the embedded component future.

VIASYSTEMS

Tom Hicks' words of 2003 are an understatement applied to 2011. The existing automotive electronics stake augmented by that of the Merix acquisition, all in China, probably makes Viasystems the world's #2 supplier of PCBs to that resurgent industry with 41% of its sales. China leads the world in automobile sales (China H1 car output, sales rank world's first).

It can't be all bad, from the point of view of an American, for a U.S. based company to have that position. even if China operations are the source.

The former Merix operation also complements in backplane PCB production. These complex boards for the computer and communications end markets earned an alternate source position in an earlier time from Teradyne Connection Systems, a testimonial. Viasystems brings complementing assembly. Therefore, the combination competes with Amphenol, the new owner of TCS, and with Sanmina-SCI, in backplanes, with the advantage of a China operation.

Most of Viasystems' 2011 growth is due to the China operations.

MFLEX

The company has failed again in 2011 to reach that elusive billion dollar mark, because of RIM's problems, but they will make it in 2012, with the help of ZTE, the leading Chinese feature and smart phone manufacturer. By the way, the customers cited here are based on my opinion and not supplied by MFLEX.

There is one problem for MFLEX. Its flex circuit capacity is so large, with about one million square feet of factory space, that it can only serve 3 or 4 customers with very fast response, for very large orders. The sales concentration has led to temporary sales declines, but MFLEX always comes back, always increasing sales over time.

How many flexible circuit fabricators are big enough to supply an Apple iPad?

A final note to the discussion of the above companies: I share the concern of the loss of manufacturing jobs in the U.S. and Canada. (Mexico is doing OK.) The PCB fabrication and assembly industries are prime examples. This concern is tempered with two qualifications, as listed below.

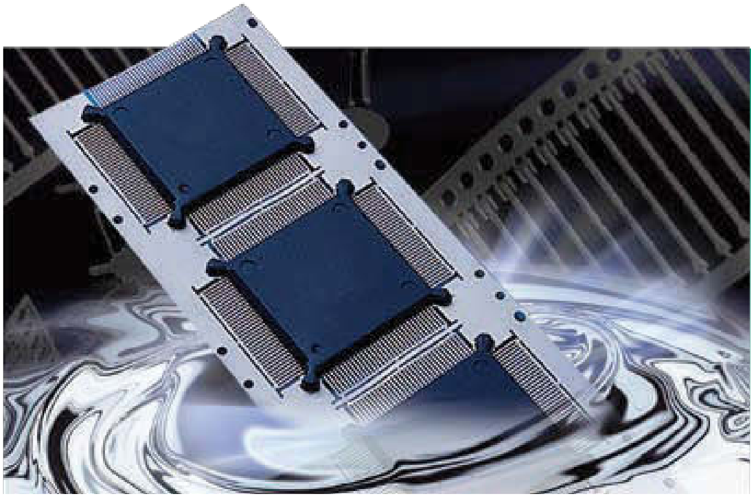
We will never restore our PCB manufacturing might by bringing back the old ways, and that includes HDI, however one defines that term. It will require new interconnection paradigms, something that will also be discussed in the next article.

In Part 2, I will suggest how Ms. Vardaman's insights at the beginning of the article, will take form, offering new opportunities to those ready to transform the way PCBs are now fabricated and assembled.

Suggested further reading

If Apple Onshored iPad Production it Would Create 67,000 American Manufacturing Jobs! **PCB**

For more than 30 years, Harvey Miller has been watching the printed circuit industry for as an economist at the University of Michigan, analyst, and database creator. He began his electronics career as a components engineer for computer and telecom OEMs, Burroughs and GTE among them. At present he's putting it all together, generating powerful marketing database tools for the global PCB industry www.FabfileOnline.com.

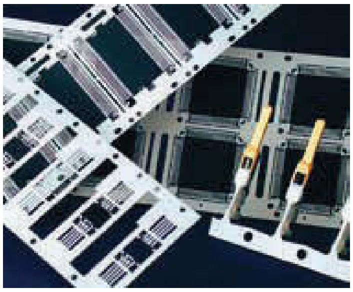


OMG Electronic Chemicals, LLC

Lead Free Plating Processes

Tin Plating Processes - Methane Sulfonate Pure Tin Technology

Product	Alloy	Finish	Application	Features
2544	100% Pure Tin	Matte / Satin	Reel-to-reel plating of semiconductor lead frames, connectors, contacts, wire, flat stock. Rack and barrel plating of connectors, contacts and miscellaneous parts.	Very wide current density range, great throwing power, excellent solderability, uniform deposit appearance, low carbon content.
2544 LF	100% Pure Tin	Matte / Satin	Reel-to-reel plating of semiconductor lead frames, connectors, contacts, wire, flat stock. Rack and barrel plating of connectors, contacts and miscellaneous parts.	Low foaming, unique controlled grain size and morphology, low stress, very low carbon content, additives are analyzable, excellent solderability.
2544 LF-20	100% Pure Tin	Matte / Satin	Reel-to-reel plating of semiconductor lead frames, connectors, contacts, wire, flat stock. Low ASF rack and barrel plating of connectors, contacts and miscellaneous parts.	Low foaming, large grain size, low stress, great tarnish resistance in high temperature applications, superior throwing power and thickness distribution, very low carbon content, excellent solderability.
2544 LF-P&S	100% Pure Tin	Matte / Satin	Reel-to-reel plating of semiconductor lead frames, connectors, contacts, wire, flat stock. Rack and barrel plating of connectors, contacts and miscellaneous parts.	Low foaming, two component additive system, maximum process control, control grain size from 1 to 10 microns, very wide current density range, excellent solderability.
SBT-100	100% Pure Tin	Bright	High speed reel-to-reel connector, flat stock, foil, and sheet steel applications. It can also be used in selective, connector, and stripe reel-to-reel plating applications.	High speed, low-foam electroplating system designed for the rapid deposition of low carbon content pure tin.



OMG Electronic Chemicals, LLC
400 Corporate Court, Ste. A, South Plainfield NJ 07080 • P: 800-536-1959 • F: 908-222-5885
5630 Pioneer Creek, Maple Plain, MN 55359 • P: 800-321-9050 • F: 763-479-3344
www.omgi.com

Start with Good Hole Wall Quality

by Michael Carano
OMG ELECTRONIC CHEMICALS

SUMMARY

A quality plated through-hole depends on a quality drilling operation. Pay close attention to hole wall quality after drilling — don't wait until the PCBs have been processed through metallization, only to discover plating defects that can be attributed to poor drilling practices.

It so often happens that a PCB fabricator suddenly experiences a rash of defects that unfortunately lead to lost revenue and potentially a loss of a long-term customer or two. When circuit boards are scrapped due to non-conforming defects, one looks to assign cause (read "blame") to wet processing typically. Before jumping to this conclusion and consuming countless hours looking in the wrong direction, one should first evaluate the mechanical drilling operation. While drilling equipment, tools and methods have been continually improved over the years, the fabricator must adhere to strict rules with respect to maintenance of the equipment, quality of the drill bits, the drilling operation parameters as well as the proper selection and use of entry and back-up materials.

Technology evolution also impacts the overall drilled hole quality. Increasing demand for higher density circuits is underscoring the requirement for smaller diameter plated through-holes. In general, smaller diameter vias are more difficult to drill. In this case, total drill hit counts must be closely monitored as hit count effect on quality is quite pronounced.

Since the formation of the through-hole in a PCB is fundamental to the PCB fabrication process, it only makes sense to ensure that the formed vias are of the highest quality. Simply put, this means:

- Clean hole wall, as free of debris as possible
- Smooth and uniform sidewalls of the via
- Minimal resin smear on the interconnect and through-hole walls
- Nailheading less than upper limits as described in Class 3 of the IPC 600
- No glass bundle tear-out or deep resin gouges

Certainly there will be some of these issues cropping up to some extent. However, a diligent drilling supervisor will ensure that these potentially harmful issues are minimized by observing good drilling practices.

Figure 1 below is a typical view of a mechanically drilled PTH that has yet to be desmeared. It is quite easy to observe the smear on the interconnect.

However, smear is only one issue. In the next set of cross-section photos, one can easily discern the poor quality of the drilled via. Figure 2 provides a view of a rough hole wall. This is usually caused by excessive infeed rates of the drill bit as it "punches its way" through the via rather than cutting through. One can imagine the chagrin felt by the folks in wet processing. With a hole quality as poor as this, it is very difficult to achieve a smooth and level (in addition to void-free) plated deposit. Note the severe nailheading as well.

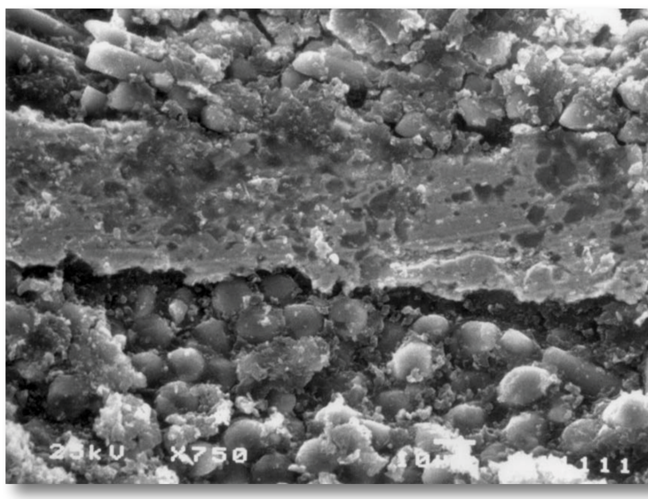
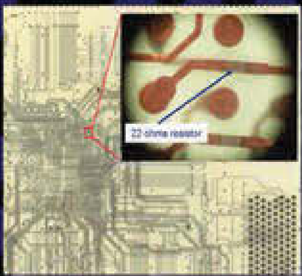


Figure 1: Example of drill smear.

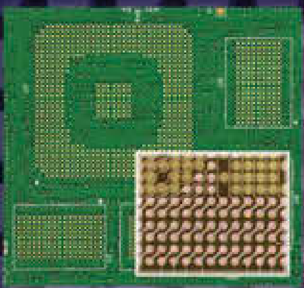
OhmegaPly® Embedded Resistors

Decades of Innovation



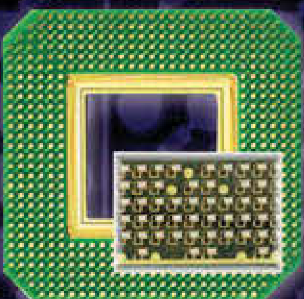
2010's

ORBIT ®
Built -In-Trace
HDI resistor



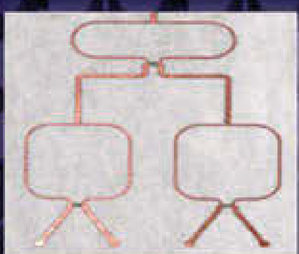
2000's

Series termination
under BGA device



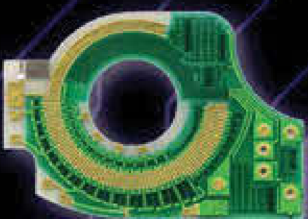
1990's

Parallel termination
of interposer board



1980's

Power divider
RF application



1970's

Potentiometer
surface resistor

Unsurpassed Reliability

Globalstar Satellite

Mars Express Lander

Phased Array Antenna

Ohmega
Ohmega Technologies, Inc.

4031 Elenda Street, Culver City, CA 90232
Phone: (310)559-4400 - *Fax:* (310)837-5268

www.ohmega.com

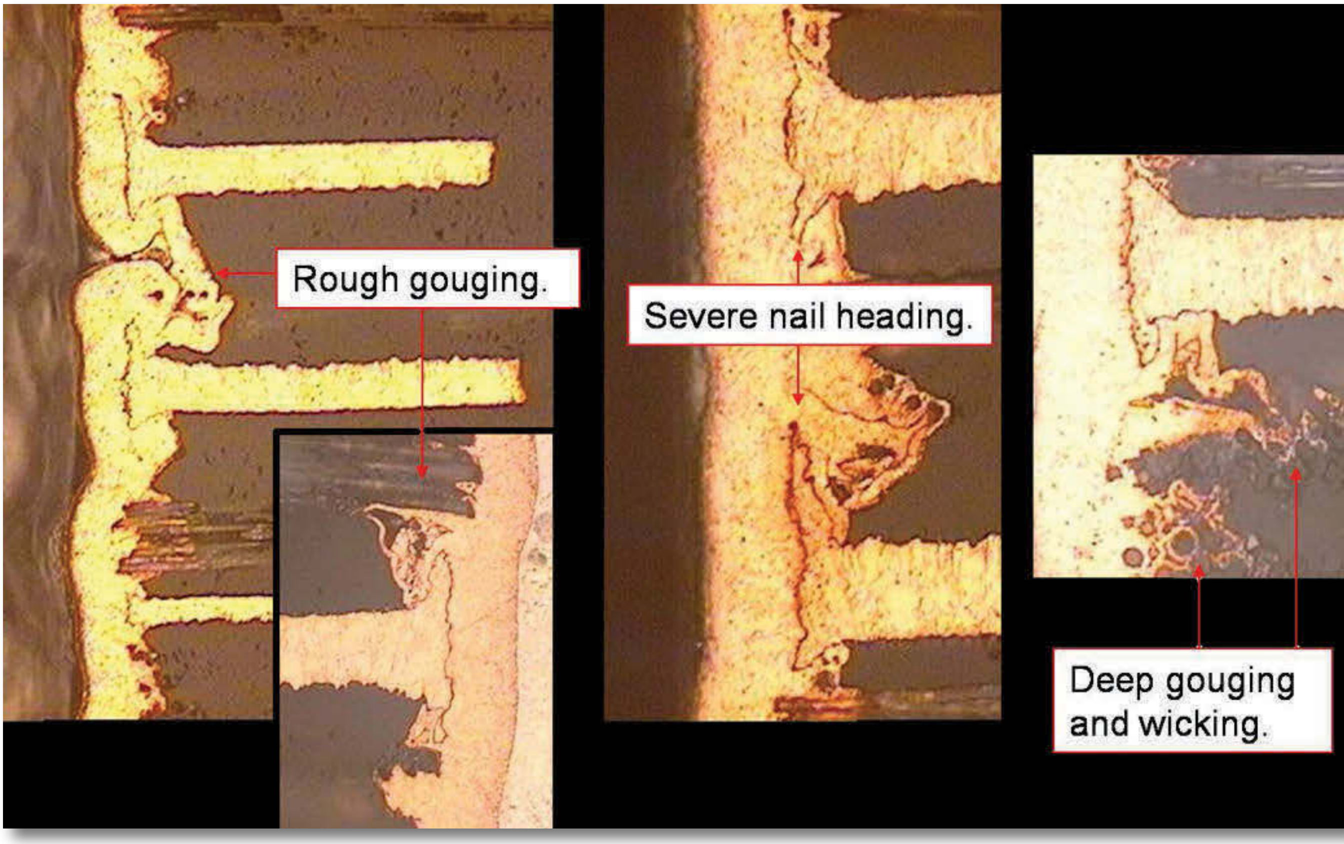


Figure 2: Rough hole walls, nailheading and deep gouging due to poor drilling practices.

Nailheading is the result of the drill bit (the cutting tool) dulling down. In addition, slow upfeeds (i.e., drill bit retract rate from the hole) will lead to nailheading and a less than ideal hole wall. As a rule of thumb, the retract rate (upfeed) should be 2X that of the infeed rate. Of course, monitor the number of hits the drill bill receives. As the hit count increases, so does the wear on the drill bit.

The longer the drill bit remains in the via, the greater the amount of heat and friction that will be generated. This will lead to more drill smear and increase the opportunity for torn out glass bundles and overall hole roughness. See Figure 3 below:

One remedy of the above situation is to adjust the feed and speeds of the drill. Reduce the in-feed rate and if necessary increase the rotational speed of the drill. Essentially one is adjusting the chip load. The chip load is defined as the in feed rate of the drill bit divided by the rotational rate of the drill. Too high of a chip load will cause the conditions shown in Figure 3 and excessively high chip loads will lead to drill bit breakage (Figure 4). However there are trade-offs as well. Increasing the rotational speed of the drill (in order to reduce chip load) will lead to higher friction and

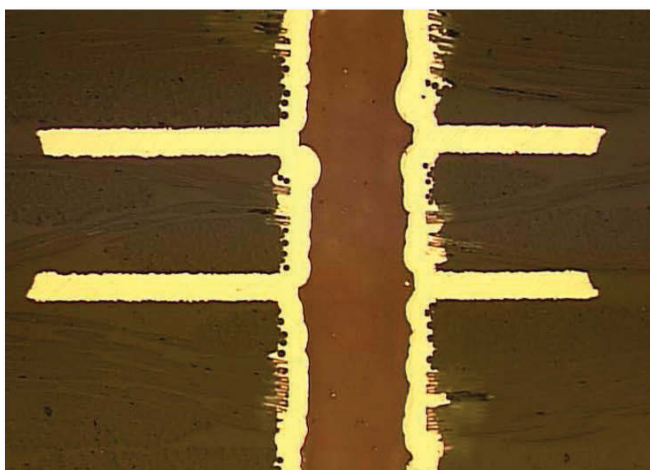


Figure 3: Rough hole wall with protruding glass fibers and torn out glass bundles.

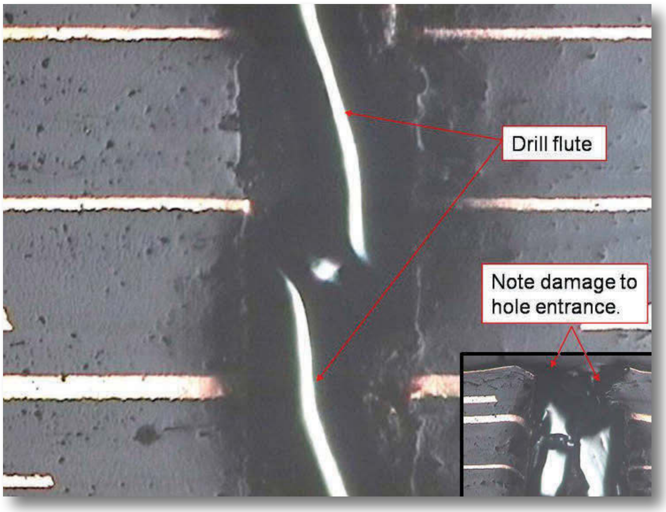


Figure 4: Broken drill bit remains in hole due to excessive infeed rates.

temperatures in the via. This action has a great deal to do with smear.

In summary, a quality plated through-hole is dependent on a quality drilling operation. Pay close attention to hole wall quality after drilling. Don't wait until the PCBs have been processed through metallization only to discover voids, nailheading and other plating defects that can be attributed to poor drilling practices.

A future column of Trouble in your Tank will present additional information on how to ensure quality drilled vias.

I would like to thank my colleague, Patrick Valentine, for his help in providing some of the figures shown in this column. **PCB**

Michael Carano is with OMG Electronic Chemicals (formerly Electrochemicals), a developer and provider of processes and materials for the electronics industry supply chain. He has been involved in the PWB, general metal finishing photovoltaic industries for over 29 years. Carano holds nine U.S. patents in topics including plating, metallization processes and PWB fabrication techniques.

MILAERO007 Highlights

Military Components Spending to Increase in China, APAC
With the reduced global expenses on new platforms in North America and Europe as troops continue to withdraw from Afghanistan and Iraq, there will be more spending on upgrades and retrofit programs. Sales in North America, Europe and Japan will be modest, while China will see significant increases.

DuPont Materials Enable Hypothermia Prevention Gear
DuPont Electronics & Communications and North American Rescue launched a line of hypothermia prevention thermal wraps, built upon DuPont's Kapton polyimide film.

Trilogy Circuits Achieves AS9100C Certification
Trilogy Circuits recently became one of the first Texas companies in the industry to achieve AS9100C Aerospace Quality Standard System Certification.

Career Technologies-USA Earns ITAR Renewal
Career Technologies-USA has received an approval letter from the United States Department of State, Office of Defense Trade Controls Compliance and remains an ITAR-registered company.

Military Radar Systems Market Hits $9.16 Billion in 2011
Developed economies are seeking to maintain dominance in new radar technological developments and, as new radar development will normally take between five and 10 years to reach production and deployment stage, countries are willing to inject large sums into their development.

It's Only Common Sense: That Ship has Sailed
With the capture of one of our intelligence drones by Iran recently there is renewed concern in Washington D.C. about our secret technology falling into enemy hands to be replicated by our enemies. But that ship has sailed.

Eltek Reports 33% Sales Increase in Q3
"The third quarter was another solid one for Eltek, as we managed to demonstrate continued strength in our core military and medical device markets," said Arieh Reichart, president and CEO.

Canadian Circuits Completes Upgrade, Increases Capacity
Canadian Circuits, a provider of boards for the aerospace industry, finished a complete upgrade of its facility in Surrey, British Columbia.

SCROOGE LEARNED A LESSON, BUT CAN WE?

by Gray McQuarrie
GRAYROCK & ASSOCIATES

SUMMARY

If you are happy just having any business at all, then you should be thinking seriously about a new business model this year. As a country, our manufacturing operations must be the best in the world to capture a second chance. A country that no longer knows how to build cannot innovate. The stakes of getting it right this time are high.

Do you hear it anymore? The whoosh? It's the sound of jobs leaving, equipment leaving, factories leaving. That was the horrible ghost of Christmas past. Today, we don't hear the whoosh. A small number of us have figured out how to survive the cold economic winter in the U.S. Not only that, we think we might see something optimistic coming our way in 2012 and beyond, something warm in the fire place: onshoring.

Onshoring is the ghost of Christmas Present. As we stare into this optimistic warmth, what does the ghost of Christmas Present yet to come say about it? Like the Dickens tale, if we don't seriously change and learn from our past, our second chance to do manufacturing right will disappear. Without changing, what we will see in our future is, "R.I.P. Innovation," along with "R.I.P. Workers, Engineers and Managers." Silicon Valley will return to orange groves. There will no longer be high-paying jobs. We will cut trees, grow crops and raise livestock. Wall Street will be tumbleweed. We have to change now. We must learn to love manufacturing and become extremely skilled at it.

I never thought I would see the day when the old moth-balled PCB plants would be started up again with millions of dollars in new capital and equipment, but it is happening. Onshoring is real. However, onshoring raises the serious question about why we offshored to begin with?

Offshoring in our PCB industry was based on several myths. The first myth was that what we did as PCB fabricators was just a commodity. Anyone can do it anywhere in the world. Making these complicated and complex PCBs was like making socks or flipping burgers or making shoes in a sweat shop. This thinking caused us to run our factories very badly. The second myth was that nothing much was going to change with regard to how we made our products. Everything was known, nothing was a surprise and nothing much will change about our fabrication processes, our material sets, our equipment sets, or our methods. We felt that the PCB industry will

> “With this brain drain, we are not only losing our ability to build stuff, but we are also losing our ability to innovate.”

be slow to advance so it made sense to let factories leave and move offshore. The third myth was it didn't matter about the quality of labor; what mattered was just the price of labor and that price was the differentiator. Anyone could learn how to do a job in our plants. Experience, training, attitude and behavior didn't matter. The fourth myth was the non-importance of the location of plants. Costs associated with transportation didn't amount to much. This lack of need of manufacturing proximity leads to the fifth myth, which was manufacturing knowledge and experience is irrelevant to innovation and technological growth of electronic devices. These myths created a one-dimensional decision. Move your plant to where labor was cheapest. This myth is proving to be wrong!

Once we let our PCB manufacturing go, we've seen the talent going, too. If what we do is such a commodity, then why do we need our best and brightest people to spend multi-year assignments in Asia and other emerging countries? With this brain drain, we are not only losing our ability to build stuff, but we are also losing our ability to innovate. We are becoming more and more dependent on foreign innovation to create the electronic devices we crave. We may have developed the graphical user interface with our friendly mouse, but who developed and designed the capacitative touch screen used on the iPhone? Who makes the iPhone touch screen? How important is manufacturing to innovation? How we build stuff, how we come up with new designs and how we come up with new ideas, is all linked. In order for this linkage to occur everything has to be in close physical proximity. By allowing our PCB factories to leave, we broke this linkage. As a result, our ability to innovate has weakened.

Too many of us today are just happy we have business, while others are feeling jubilant that we have, in fact, a thriving business. What we should be doing, instead of just being happy, is thinking seriously what the new business model needs to be going forward. What do we need to do differently to make our manufacturing operations the best in the world so that we can capture this second chance? The main challenge we face going forward isn't the technology, in terms of new materials and new equipment, it is in the technology and techniques and how fast we can reconfigure and optimize our businesses.

What am I talking about? We need to learn how to optimize our business because of the complex nature of the new, compact designs. We are forced to run very small lots, we are forced to go multiple times through the same operations, we are forced to mix together exotic materials that just weren't meant to be together, and this is just the start. Along with the challenges of trying to figure out how to make a product quickly and crashing entire lots of material at times, we don't know how to determine quickly the true cost of making a board. In my article, *Reject Cost Accounting's Answer and Build Jobs That Make the Most Money,* I explain how using linear programing (LP) allows you to focus on the true cost of different jobs and how to maximize your product mix. However, this is just part of the answer. We also need to become skilled at understanding variation.

By variation, I am not talking about a control chart or a Six Sigma type of variation, but a much deeper and sophisticated understanding of how variation impacts our operations. Let me give you an example by way of a question. Which is worse, having more variation in your upstream processes or having more variation in your downstream processes? The answer is that variation is your downstream processes is much worse for you than upstream variation.

Let me give you another example. In order for many of us to get ahead it will be tempting to accept low-yielding jobs. I explained in my article *Are We Nothing More than a Pair of Socks at Walmart?* that scrap consumes your plant's capacity. But it does something else that is extremely damaging. It makes your business exponentially unpredictable. In a future column, I am going to take the example in the aforementioned, Reject Cost Accounting's Answer… and show you something called a discrete event model. If you don't know what this is, you need to learn about it fast, because it will be one of the tools we are going to rely

Job	Profit	Scrap
1	$700	25%
2	$450	10%
3	$100	70%

Table 1.

on in the future to figure out how to optimize our manufacturing operations. Consider jobs 1, 2 and 3 described in Table 1.

Now, take a look at Figures 1, 2, and 3 below. Figure 1 shows the results of the discrete event simulation done with 20% Job 1 and 80% Job 2. Figure 2 shows the results of 10% Job 1, 80% Job 2 and 10% Job 3. What is surprising is just a little bit of this problem job had a huge effect on the variation of the profitability results. It made the business much more unpredictable. It would be easy on paper to reject this job. However, suppose this job went to an assembly plant your company owns and it was vital for their customers' next-generation device. The PCB shop doesn't make money on the part, but the company as a whole does. The problem is that deliveries are unpredictable, because the board is next to impossible to build. What can you do to produce a certain quantity of this problem job each month and still make your PCB board shop profitable? In other words, how can you take out the variation and its cause?

The jobs coming into this shop that produced Figure 2 were random. What happens when you produce in a specified scheduling sequence? You get Figure 3! Granted, the result isn't profitable, but there is far less variation. There is an optimum sequence of running problematic jobs with jobs that run through your factory well! Upon reflection this may seem obvious, but it isn't something many of us have considered, done and optimized. Understanding variation is a core competency that we must develop in order to achieve our future vision of manufacturing excellence.

We aren't going to be able to figure out all of this stuff and meet manufacturing challenges if we don't change our attitude about people. If we don't learn how to value the human asset, we will fail again. In the future we must build collaborative work teams. These teams must be devoid of bad behaviors so that they thrive in a productive culture. As Louis V. Gerstner said, "I came to see, in my

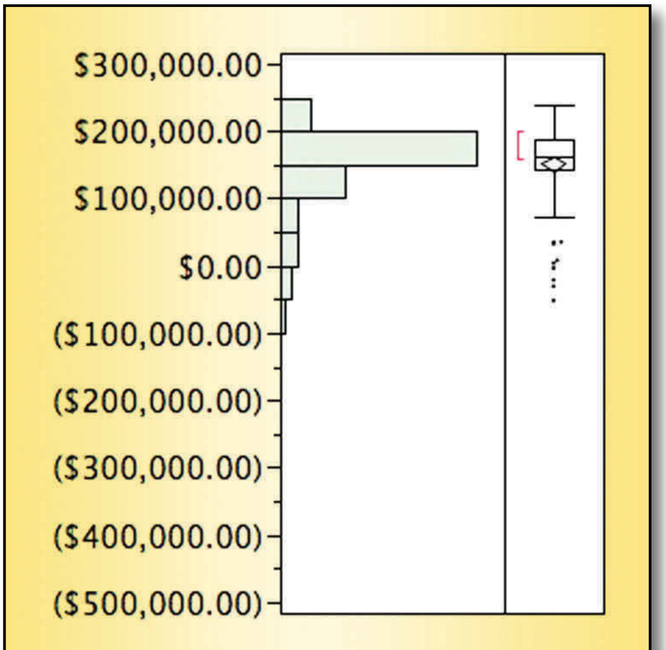


Figure 1.

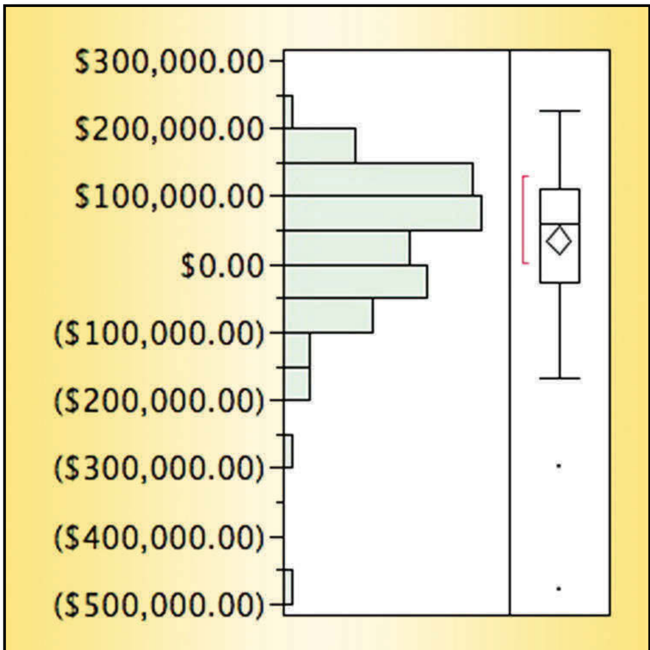
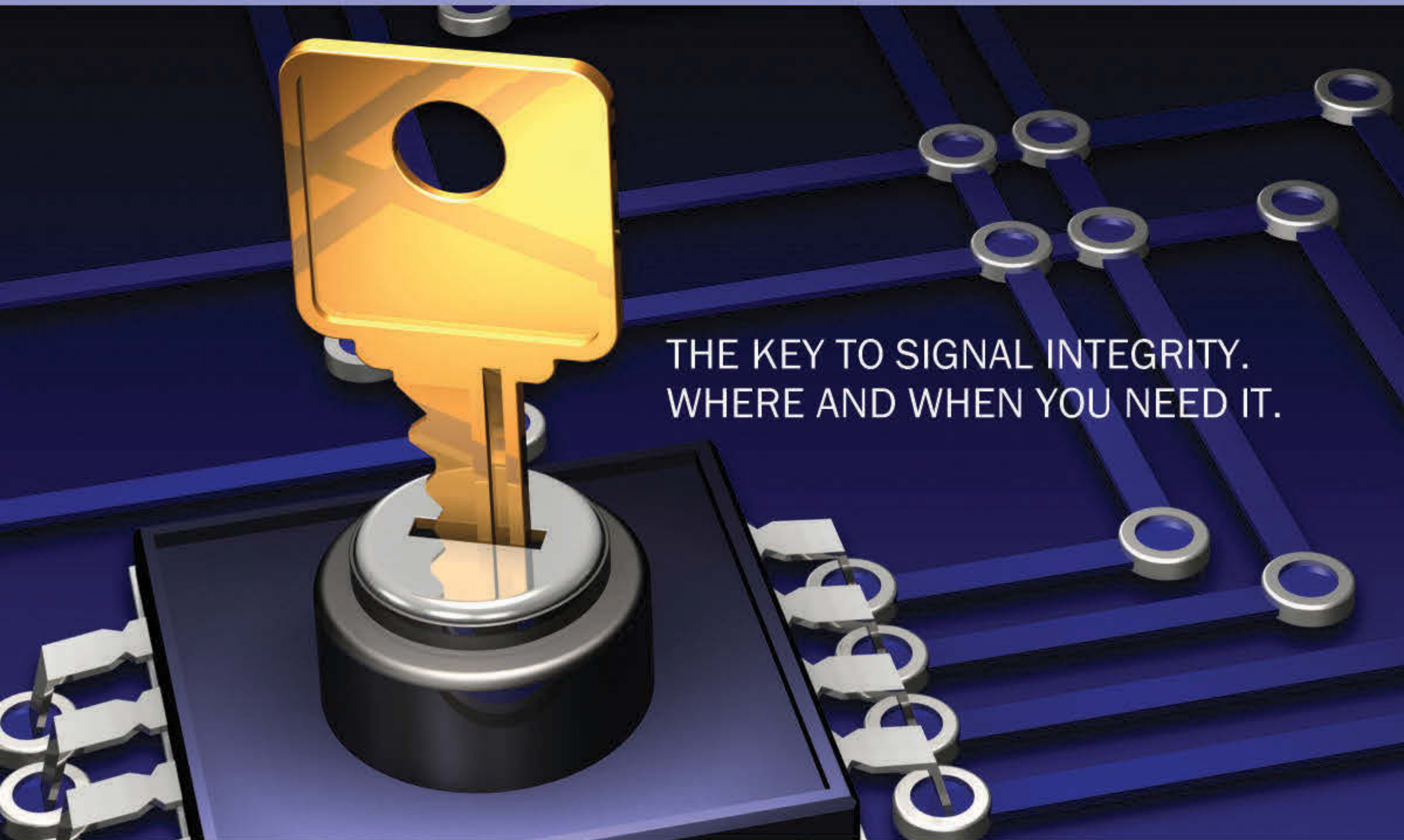


Figure 2.


THE KEY TO SIGNAL INTEGRITY.
WHERE AND WHEN YOU NEED IT.

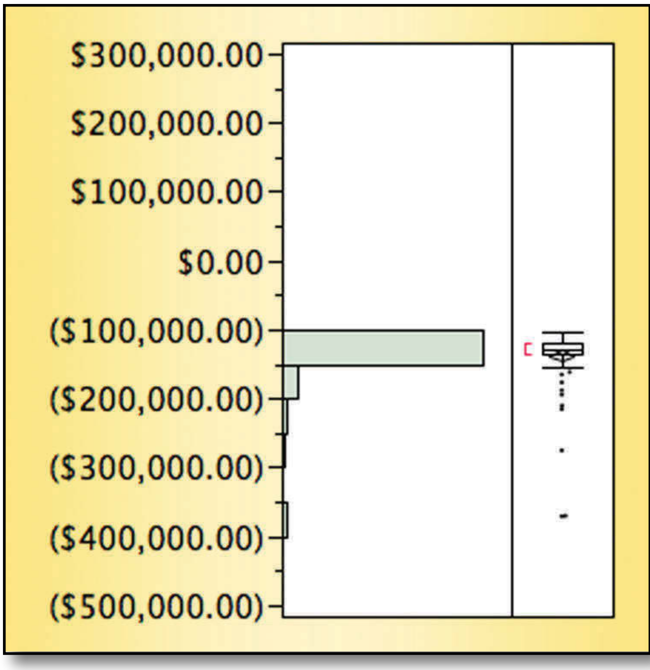


Figure 3.

time at IBM, that culture isn't just one aspect of the game, it is the game."

In order to have an effective team culture we have to be productive, we have to communicate, and we have to make discoveries. In my book, *Change Your DAM Thinking,* I talk about the five thinking DAMs, which block our progress and lead to an ineffective culture. These are listed in Table 2.

Going forward we must be able to observe, measure and hold our employees accountable for their behavior and we must observe, measure and provide feedback on behavior of those with whom we work. Our businesses in this industry must become serious about changing our DAM behavior, or we will fail. Without this DAM behavior we can work collaboratively to solve our complex problems quickly.

Along with being able to work with each other much more productively we also need to be able to take a different approach on how we think of problems. Our work is going to be so challenging going forward, that we must change our notion of problems. We must view problems as good and describe them as opportunities for discovery. If we encounter a quality problem we should ask, "What was the source of the problem?" Then, "When is this problem going to happen again?" And finally, "What are we doing to prevent it from happening?" What we need to do is have models that contain our understanding and assumptions, which then can be used to predict outcomes. At that point we can judge if the event was entirely unexpected or not. In both cases we need to figure out what we need to do to be better. This type of an approach goes far beyond the standard FMEA of Six Sigma. Are there companies doing this? Yes, but none that I know of in our industry. We not only need to become the best at building PCBs in the world, but our industry has to be known for its manufacturing excellence. We should be so good at manufacturing that Toyota wants to learn from us, because of our ability to perform, create and adapt. We need to understand and take lessons from the likes of Proctor & Gamble and other great companies that love and advance their manufacturing operations.

Finally, what we need to do in the future is become a real partner to our customers. Let me explain what this really means. It doesn't

Question	Dam	What it Blocks
Fail to ask for help?	Ego DAM	Productivity
Manage people's feelings?	Feelings DAM	The Business Purpose
Afraid to speak up?	Trust DAM	Communication
Won't let go?	My Precious DAM	Growth
Know it all?	Learning DAM	Discovery

Table 2.

mean the customer is king. It doesn't mean the customer is always right. It doesn't even mean you always have to be nice. It means you have a relationship. It means if you have a problematic job, you don't merely accept it, you work with the customer to make it better. And you do this with powerful ideas and flexibility. There are always options. There is never one perfect way. The future isn't going to be about individual achievement. The future is going to be about our ability to work collaboratively to find solutions to very complex problems. Our customer needs to be a collaborating partner in what we do and we need to be a collaborating partner in what they do. In so doing, we will both grow on many levels. We will set new standards of excellence, but not standards imposed upon us by our customers. They will be standards we set for ourselves because we want to be leaders. We want this second chance to work. We want to be the best manufacturers on the planet.

Finally, everything comes full circle — back to innovation. The manufacturing side is as important as the design side for an electronic device. In order for this to work there has to be localized manufacturing close to companies that are coming up with the ideas and doing the design. There are so many different ways to combine materials and components and so many different ways of packaging these components onto a board that the best combinations of things can't be understood when the manufacturing operation is thousands of miles away, with cultural and language differences. You can't innovate over Skype no matter how hard you try! The myth that offshoring has no impact on our ability to innovate is proving to be very wrong. We need to get our manufacturing operations to be the best in the world. We need to become collaborative partners with our customers. And we need to be able to figure out alternatives and solutions to complex problems, fast. If we do this while always operating our plants profitably then we will have a bright future. The easy days are over. Now it is time to show the world what we are made of. **PCB**

Gray McQuarrie is president of Grayrock & Associates, a team of experts dedicated to building collaborative team environments that make companies maximally effective. McQuarrie is the primary inventor of the patent, Compensation Model and Registration Simulation Apparatus for Manufacturing PCBs. McQuarrie is a monthly contributor to The PCB Magazine with his column, *Solving DAM Problems*. For more information, visit www.grayrock.net, or email McQuarrie at gray@grayrock.net.

PRINTED CIRCUITS, INC. EXPANDS CAPACITY, LOWERS COST

Printed Circuits, Inc. (PCI) has expanded their manufacturing facility. Through the acquisition of new equipment, PCI has increased inner layer capacity and improved manufacturing yields resulting in a cost reduction on their rigid-flex products, and the additional service offering of higher volume production to their customers.

Among the new capital equipment investments undertaken, PCI has installed and qualified new conveyorized wet processing equipment from Chemcut, USA, including a chemical clean line, cupric chloride etcher, and resist stripper. The added capacity and equipment has positioned PCI's cost structure for higher volume rigid-flex manufacturing, while improving repeatability, manufacturing yield, and process capability on trace technology below 75µ/75µ.

"In the last 12 to 18 months, we have been taking larger orders and higher volume, lower cost rigid flex" said Ken Tannehill, President and CEO of Printed Circuits. "This new equipment gives us the capacity to handle the higher volume projects at a lower cost, and a much higher yield. We have been pleased to see the improved yields since qualifying the process lines for production earlier this year."

To learn more, call or e-mail Bob Burns at (952) 886-9307.

2012: NO LACK OF CHALLENGES

by Agustin Araujo
PENTALOGIX, INC.

SUMMARY

Predicting what's to come for the world's PCB industry is like trying to predict the weather — it's a bit risky. While some areas may experience growth and success, others may remain flat or decline. There are challenges facing PCB industry this year, but companies can overcome them and thrive in the year ahead. Here's one view of how that may happen.

At PentaLogix, we divide the PCB market into the "Do It Yourselfers" (DIY), OEMs, EMS providers and suppliers to the industry. They all need software, bare boards, components, prototypes or golden boards. They also need stencils, solder, flux, reflow equipment, cleaning equipment, as well as formats for process control and test and measurement. Amid those needs, obviously supply and demand will occur, tempered by the economies in this country and of others around the globe.

China & ROTW

The good news is, anecdotally we hear of some growth. Many of the mergers and acquisitions have been completed and leaders have emerged in most sectors. The same is true of geography. Indeed, China has been on everyone's mind and it should be. According to IPC's recently released Roadmap, China accounts for about 80 – 85% of all circuit boards made worldwide. Over the next several years, China's PCB market share is expected to increase to 90%.

What does that leave for the rest of the world? Asian markets, such as Thailand and Malaysia, are starting to emerge, actually. Surprisingly or not, if we have been paying attention, South America has also been forging ahead. Brazil, part of the BRIC countries (Brazil, Russia, India and China) and long seen as a hot up-and-comer, gradually has been easing up on the burden imposed by taxation and other government restrictions on nations who do business there. Indeed, nudging from the likes of large industry players like Foxconn Electronics can do wonders.

India is another country positioning itself for a growth spurt. Another BRIC nation, India is working increasingly with manufacturers as an outsourcing partner in areas other than customer service functions. We think outsourcing of electronics to that country is growing in appeal to a great degree because of its well-educated, English-speaking population in the medical, electronics and other high-tech industries.

High-reliability: North America's Edge?

High-reliability (hi-rel) industries such as the medical electronics, military and aerospace, aviation, and increasingly, automotive, usually are of the smaller-volume, higher-mix nature, versus the high-volume, low-mix business on which China has built its strength.

Within all the hi-rel industries, failure has grave consequences in that lives could be lost; therefore, liability is extraordinarily high. Even without the threat of losing lives, failure could still mean huge recall costs. In those cases, at least for the foreseeable future, North America maintains an edge. Ask us how long we think that edge will remain, and we truthfully don't know. Things could change for a variety of reasons, not the least of which is erosion of experience.

> *"Even without the threat of losing lives, failure could still mean huge recall costs."*

One of the tragedies of the economic crisis is that in the resulting mass downsizing, practically the entire layer of middle management, with the engineering experience and "lore" that comes with it, evaporated suddenly and completely. It was almost back to square one for some companies. We have made up for that somewhat as those who remained have gained experience. But as we did so, companies in other countries quietly amassed the same type of experience we had lost, eroding one of our greatest advantages.

Until now, North America's edge also came from design innovation, manufacturing reliability and the ability to have a designer interact with the manufacturer within hours, rather than having to fly overseas to solve a problem. When a line is down, so are profits, and minutes count. But that, too, is changing. Interactive software now allows global sharing and analysis of data, often resulting in the ability to solve problems "remotely."

One way we've seen North American companies cope with the China threat, as well as with any China production problems, is to bring some of the business previously outsourced to China back to North America, locating in Mexico, either in maquilladoras (OEM factory subsidiaries, mostly staffed with and run by Mexican citizens employed by the OEM) or in EMS facilities of companies such as Foxconn or Celestica. Another reason for this is labor costs in China, which have been increasing steadily to the point of outweighing advantages for some companies. Mexico is obviously cheaper as far as shipping parts and equipment and much faster for an OEM designer to physically reach the EMS production floor.

We believe this migration of work from China will continue, but a threat to this positive trend is the escalating violence that has overtaken Mexico in many major manufacturing areas, with kidnappings and murder of executives increasing. We know of companies that currently do not risk sending engineers to Mexico and may again turn overseas, perhaps to Europe this time. The destination could be either Eastern or Western Europe, wherein we do see some specialization emerging, depending on the product needed.

Industry-Specific Challenges

There are definitely many other challenges for the industry this year. As we mentioned, North America is prominent for hi-rel military, medical and automotive. Defense budgets are being drastically cut, but will still provide opportunity for those who are ITAR

(International Trade Arms Regulation) certified and can meet the Government's requirements to provide risk management solutions as part of proposals for bids.

There are, however, increasingly expensive requirements on the table to be enacted. The IPC estimate that the conflict minerals requirement for traceability and accountability will cost the electronics industry even more than the RoHS has and be much harder to implement.[1] (By the way, if you want to have a voice in all this, we suggest you join the IPC Government Relations Committee. They're a great group and an excellent resource.)

Another ramification from the political arena for 2012 is the inability to foresee what will or will not come from an increasingly unstable group of governing bodies, whether it's the fractiousness of the United States, or the monetary woes of the Eurozone and its individual members. It's hard to plan for expansion or R&D, which keep our industry alive, when you don't know what you will be forced to pay in your own country, or how you will be affected by the monetary woes of others.

In addition to the economic worries, we will also face ever-evolving quality challenges in PCB manufacturing, such as the problems we encounter in the form of defects and field failure. As we increase the complexity and functionality of components, the boards on which they are mounted must accommodate those changes. In other words, we need to recognize that now it's also the components, not just the designers, that determine what kind of boards and what kinds of processes — soldering, cleaning, profiling and testing — we need.

For example, we must have bare boards prepared for the finest of pitches, as well as the largest of BGAs. We need to realize the existence of the largest thermal co-efficiency challenges perhaps ever seen, and exacerbated even further by the growing use of multilayer boards. Processes that were sometimes skipped, such as thermal profiling, must be done religiously. Additionally, because of increasing government regulations, more and more of what we do must be documented. Therefore, software must be robust and it must be transferable for conferencing around the world. The more design checks and test/inspection devices built into it, the better.

Heat, humidity (RH) and electrostatic discharge (ESD) will continue to rear their ugly heads, especially in the lead-free arena. If not controlled, for instance, humidity can reduce both storage and, eventually, field life. As in each of the last three or four years, we will most likely learn that more and more components fall under any of a number of "sensitive" categories. Education of our people needs to reflect this. We must send them to workshops, whether online or on site. Company sales reps must become educators and consultants, not just order takers. As in most things, we feel education will be the key. Things like tin whiskers still haven't been completely figured out and will no doubt continue to be a problem, even with the use of conformal coating to slow them down.

Security — From Inside and Out

Another question involves security. Where can we outsource what we are not prepared to do ourselves, and how do we select a good partner? Will they steal our designs or process? And what can we do to safeguard ourselves as industrial espionage reaches epidemic proportions?

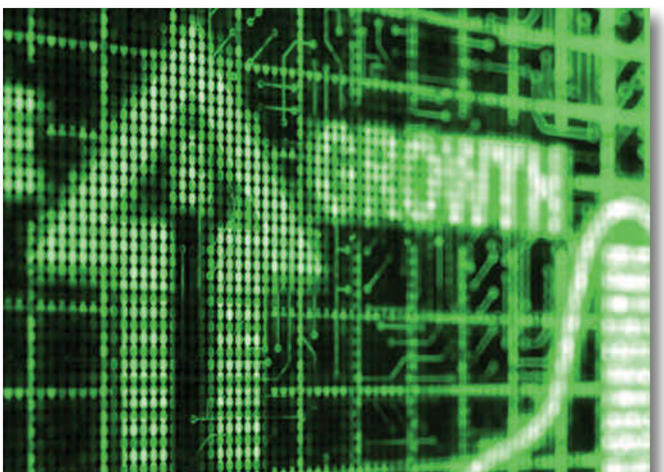
According to Retired Colonel Jill Morgenthaler of CJMI, who specializes in

Homeland Security measures, it begins with awareness that we are always vulnerable, both from the inside and the outside. Many companies still refuse to acknowledge this.

Once we acknowledge that basic fact, we need to survey, both visually and by asking questions, who has access to our designs? Are they required to have ID badges? Are they allowed to remove work or laptops from the building? Are entrances to buildings secured? Must visitors sign in and identify themselves with photo ID? Is there a strategy in place to protect the proprietary information should the building be invaded by hostile individuals or forces?

In our own headquarters, and in EMS facilities, is there a plan for how to continue work in the face of natural or man-made disasters? Do employees know exactly who is in charge if such an incident occurs? Are files backed up in a remote, secure location? Are there alternative sources for key materials? These are questions that must be asked of every OEM and EMS, no matter how stellar their reputations in other areas. We saw with the recent Japan natural disaster how disruptive it was for countries all over the world. Only companies with plans — and alternative sources — could continue their businesses uninterrupted.

“We saw with the recent Japan natural disaster how disruptive it was for countries all over the world.”



Integration, Electronics Style

Suppliers to the industry must think in terms of integration. Do the things they produce require dismantling and discarding of old equipment and products, or will new items be additional “modules” that seamlessly add value to what is already in place? Are the footprints smaller and do they conserve energy? If suppliers can produce efficient, scalable, integrated products, they will have the advantage.

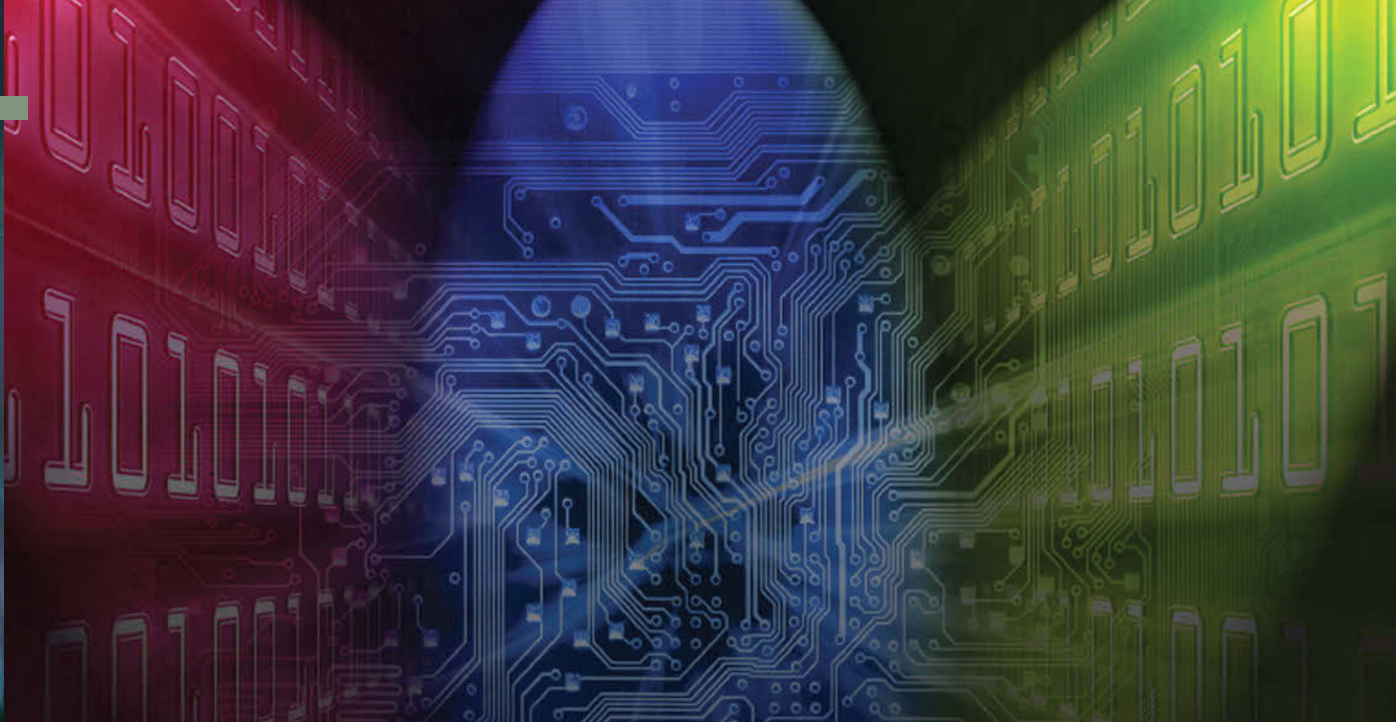
For actual numbers for equipment and dollar volume, there are many sources: the IPC at www.ipc.org, SMTA at www.smta.org, SEMI at www.semi.org and IMAPS at www.imaps.org, as well industry publications such as The PCB Magazine and its site, www.iconnect007.com.

You may find some contradictions from month to month, but that is to be expected in an industry dependent on and affected by so many outside factors. What we have tried to do in this article, is give you an overview of 2012 industry challenges and how you can potentially overcome them and thrive in the year ahead. May we all have much success in 2012! **PCB**

Reference

[1] IPC Government Relations Committee Meeting, APEX 2011.

Agustin Araujo is owner and president of PentaLogix, Inc. He has worked as a software engineer at companies including Mentor Graphics and US Bank. In the last several years, Araujo has developed PentaLogix into the only company to provide a single online site, www.pentalogix.com, that allows an engineer to both quote and order everything needed for an entire PCB project, from design through assembly.



Understanding the Rainbow Process: *"Get the jam on the bread and eaten before it hits the floor..."*

by Pete Starkey
I-CONNECT007

SUMMARY

From the showroom floor at productronica 2011, I-Connect007's Pete Starkey talks to CEO Jonathan Kennett, founder of Rainbow Technology Systems, Ltd., of its novel approach to primary imaging of PCBs — particularly multilayer inner layers.

A highlight of the 2011 productronica exhibition in Munich, Germany, was the launch of the Rainbow Process by Rainbow Technology Systems, Ltd., based in Glasgow, Scotland. The Rainbow Process takes a novel approach to the primary imaging of PCBs, particularly multilayer inner layers. Technical Editor Pete Starkey took the opportunity to discuss the concept and features of the process with Rainbow's founder and CEO Jonathan Kennett.

Starkey: *Jonathan, when we talked about primary imaging at productronica 2009, you hinted that you were working on something revolutionary. Two years later you launch Rainbow and it's attracting a lot of interest. In essence, what is it and what does it do?*

Kennett: Put simply, the Rainbow process is a three-in-one system: coating, imaging and developing in one automated unit, with the ability to print in extremely high resolution.

Starkey: *What sort of line and space resolution can you achieve?*

Kennett: The current industry norm seems to be around 75 microns. Rainbow can easily achieve 20-micron lines and spaces and 10-micron details are well within the capability of the system. Additionally, we have resolved 5-micron details in the lab.

Starkey: ***What particular characteristic of Rainbow makes this possible?***

Kennett: We have developed a unique wet resist that is solvent-free and does not need to be dried before imaging. We expose this wet resist with a phototool directly in contact with it, using an LED light source.

Starkey: ***What specific advantages does your resist offer over conventional wet resists?***

Kennett: The main issues with conventional wet resists are the maintenance and running costs of the ovens. In addition there are handling issues and the constant risk of airborne contamination. In the Rainbow system we don't need an oven so these issues just don't exist, and we don't have to deal with solvents.

Starkey: ***What sort of phototools do you use?***

Kennett: Conventional silver halide films. Because they are in direct contact with the resist, the optical path is only about 8 microns, light scattering is dramatically reduced and we don't need collimated light.

Starkey: ***Jonathan, I'm sure everyone wants to know why the phototools don't stick to the resist. What's the secret?***

Kennett: No secret — we have developed a protective coating which gives a low-energy, scratch-resistant, non-stick surface. This is applied as a liquid on a separate coating unit and UV cured to give a final thickness of about 4 microns.

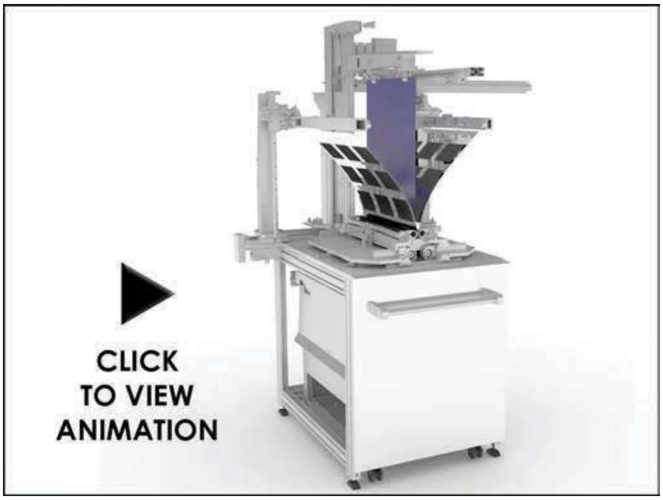
Starkey: ***What level of exposure energy does Rainbow resist need?***

Kennett: Only 2 milliwatts per square centimeter, whereas dry films typically require ten times as much. We use an LED light source with a beam spread of +/- 6 degrees. Because the optical path is so short we get negligible undercut and dust particles will not resolve.

Starkey: ***You have mentioned dust particles and airborne contamination. How do you keep them out of the system?***

Kennett: The Rainbow unit is fully automatic and operates within its own controlled environment. The operator is outside the unit and the process. Any particles present on a panel after surface prep are removed by a contact-cleaning machine as it enters the unit, which is maintained at a positive pressure with HEPA-filtered air. All process operations are carried out with the panel held vertical, and the panel is imaged within seconds of being coated so there is no time for particles to attach themselves; this is a major benefit of bringing the coating and imaging process together in a controlled environment and in a short timeframe. In effect this gets the jam on the bread and eaten before it can hit the floor! I would like to find a more technical analogy to explain this but it is a bit like an internal combustion engine where you arrange a combustible gas in a controlled environment and introduce a spark; then, you get an instantaneous change of state. This is the same way Rainbow brings liquids and light together in one moment of time in a controlled environment producing the desired end.

Starkey: ***Jonathan, I think the bread and jam analogy conveys the point very clearly! And I like the vertical processing concept — what other benefits does it offer?***



Rainbow Technology Systems' Rainbow Processing Unit, for coating and fine line imaging of panels.

Kennett: Apart from the avoidance of contamination, the main benefit is the ability to handle thin inner layers and substrates. Supporting the material by its corners and under slight tension gives us excellent panel control. And of course it allows us to keep the system extremely compact.

Starkey: ***Jonathan, we have talked about coating and exposing, but the system develops the image as well. Can you tell us about this part of the process?***

Kennett: During exposure, the image area of the resist is polymerised. Unexposed resist is still wet, and remains on the panel after the phototools are removed. This wet resist is easily removed using industry-standard sodium carbonate developer solution, followed by a rinsing and drying. The panel emerges from the Rainbow unit dry and ready for etching.

Starkey: ***I am sure that Rainbow offers environmental benefits; what are the main ones?***

Kennett: As I already pointed out, the resist is 100% polymerisable — there are no solvents to worry about, and there is no polyester protective film needing to be stripped and disposed. The LED exposure system does not create any ozone and uses minimal energy — indeed the whole system uses no more power than a domestic kettle.

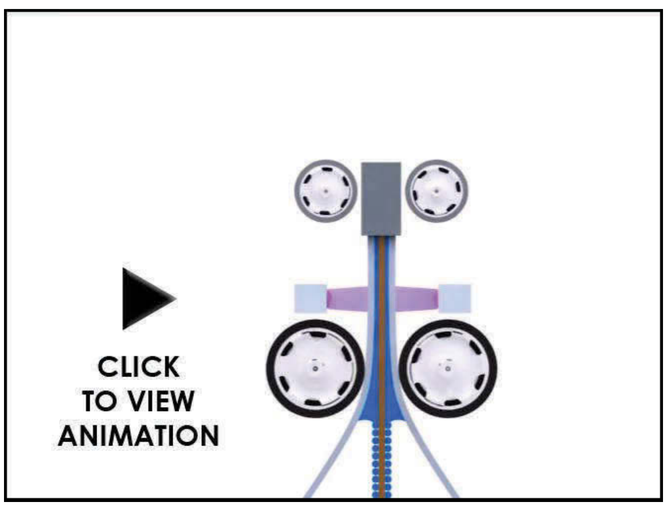
Starkey: ***Apart from energy, where are the main cost savings?***

Kennett: The cost savings are many — internal and external to the process. Internally, the base cost of the resist is very competitive with other resists. Externally, the system does not need a full clean room and the whole unit is only six metres long; it only occupies 12 square metres of floor space. The entire three-stage unit, which combines coating, imaging and developing, sells competitively against an image-only laser system, and of course there is no expensive laser replacement program either. One operator can monitor all three processes, which reduces labour; the automatic handling further cuts down rejects.

Starkey: ***Jonathan, I will ask the inevitable production manager's question: What's the throughput?***

Kennett: In its current configuration, the unit will process one double-sided panel every 15 seconds.

Starkey: ***And what's the yield?***



Kennett: At this stage we don't have actual production data — the system is just too new. However, we have addressed the major issues to which circuit manufacturers attribute poor yields. The main causes for rejects tend to be centred on handling damage, contamination from particulates and deviation from the process specification. The process is fully automatic so there are no handling issues. The closed environment in the Rainbow process is a major benefit as it excludes operators from the most sensitive parts of the process. The elapsed process time for a panel is around 60 seconds. From the moment a panel exits coating, it is being imaged within 10 seconds — there just isn't time for debris to get on the panel! The wet resist is constantly purging the phototool of any rogue particulates, so there is no likelihood of repeat image defects. Our expectations are therefore very high in this regard.

Starkey: *How easily can Rainbow be integrated into an existing PCB facility?*

Kennett: For any company already using silver halide phototools, it will be simple. The Rainbow process starts right after surface prep where a dry-film laminator would normally be placed, and finishes with a panel ready to etch. The input and outlet to Rainbow are both horizontal, even though the process itself is vertical.

Starkey: *What other equipment would be needed to run Rainbow?*

Kennett: We supply two further stand-alone units: one for coating the phototools and one for registering them on their carriers.

Starkey: *How about maintenance?*

Kennett: Any competent technician can maintain the unit, and it has an open architecture so access is very easy. Cleaning down is very simple and quick as there are no solvents involved. A major benefit of the resist is that it does not dry out and the whole machine can easily be left in standby mode for an hour or more.

Starkey: *Besides PCB imaging, what other applications do you see for the Rainbow Process?*

Kennett: The process is very scalable and we are in discussion with some touch panel producers who are interested in fine conductive lines. It is also applicable to chemical milling. We are developing other inks such as soldermasks and plating resists as well as seed layer technology for additive applications. We expect to release a reel-to-reel version of Rainbow in Q4 2012.

Starkey: *What practical assistance can you offer PCB manufacturers who want to explore the benefits of Rainbow?*

Kennett: We have a fully operational line at Rainbow Technology Systems. We will be delighted to arrange demonstrations for potential customers and to produce samples to their specifications.

Starkey: *Jonathan, thanks for giving us such a comprehensive insight into the Rainbow system. We wish you every success!* **PCB**

I-Connect007 Technical Editor Pete Starkey, based in the UK, has served as editor for both the Good PCB Guide (UK) and Printed Circuit News. With more than 30 years in the PCB industry and a background in process development, technical service and technical sales, Starkey is a fellow of the Institute of Technology, a member of the SMART Group Technical Committee, and an active supporter of the European Institute of Printed Circuits.

Rainbow Technology's New Primary Imaging System

by Real Time with...productronica 2011

Jonathan Kennett, CEO of Rainbow Technology Systems, explains the details of a revolutionary primary imaging technique.

realtimewith.com

CML GROUP RECEIVES SUCCESSFUL EVALUATION OF IPC PCQR2

Since April 2011 CML EurAsia, as deputy of the CML Group, is registered as member of the IPC Process Capability, Quality and Relative Reliability (PCQR2) Benchmark Database. IPC PCQR2 Database is an innovative supply chain management resource developed by IPC and CAT for designers, purchasers, assemblers and manufacturers of PCBs. It is based on statistical data collected from industry-developed test patterns that quantify the capability, quality and reliability demonstrated by global PCB manufacturers. For more information, visit www.cat-test.info.

CML Group submitted an IPC-6R-E test panel, produced by main production partners Gainbase, Onpress and Techwise, according to IPC specifications. The IPC PCQR2 test results provide detailed information about the process capability and reliability of PCBs. Furthermore, these results serve as benchmark within the IPC PCQR2 Database members. We have already received the first test results of the submitted test panels of our partners Gainbase and Onpress. The results of the evaluated criteria rank in the top performance level of the IPC PCQR2 benchmark.

The results illustrate the strengths and weaknesses of the CML Group. We are able to identify areas for improvement faster and together with our production partner we can work on the continuous process optimizations. The IPC PCQR2 membership with the statistical process capability and the PCB reliability testing is a further milestone of our zero-defect strategy.

Furthermore, we are very pleased that our employee, Catherine Dinnissen, is certified as IPC Trainer and is able to support our partner factories during the process optimization.

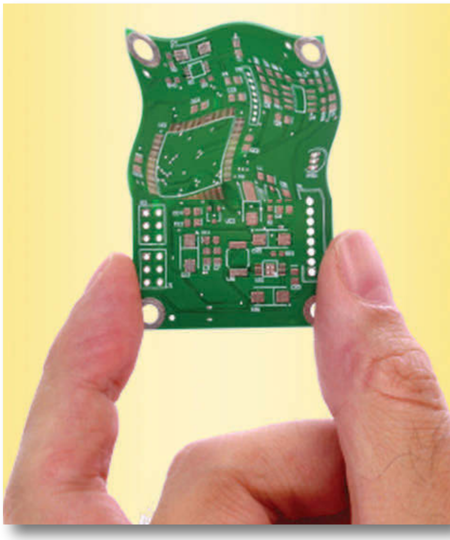
Rigid-flex Manufacturing Part I: Introduction

by Dale Smith
DAS FLEX CIRCUIT CONSULTANTS, LLC

SUMMARY

What are the requirements needed to meet certain challenges of rigid-flex design, and what are the preferred raw materials, manufacturing methods and techniques employed for achieving success in rigid-flex manufacturing?

Rigid-flex has been around now for more than 30 years, yet it still has a "black magic" stigma of being complex, extremely difficult to build and cost prohibitive. Many rigid board fabrication shops have tried to complement their rigid board business with rigid-flex offerings, but found the challenge quite daunting. In this new series, I discuss the requirements needed to meet certain challenges, as well as the preferred raw materials, manufacturing methods and techniques employed for achieving success in rigid-flex manufacturing. First, a short background on why rigid-flex has become such a popular interconnecting method for numerous electronic applications.



What is rigid-flex?

Rigid-flex is the marrying of rigid board materials with flexible materials to produce a multilayer printed wiring board with conductor paths that can be bent, twisted or folded. Designers of rigid-flex are typically referred to in IPC-6013, MIL-P-50884 and MIL-PRF-31032 as product "Type IV."

Rigid-flex is different from a flex circuit with an added stiffener. Rigid-flex incorporates plated through-holes that extend through the entire rigid section to create a robust plated through-hole interconnect similar to a rigid board. As with rigid boards, we can also incorporate microvias using HDI process technologies to take advantage of today's most advanced packaging components.

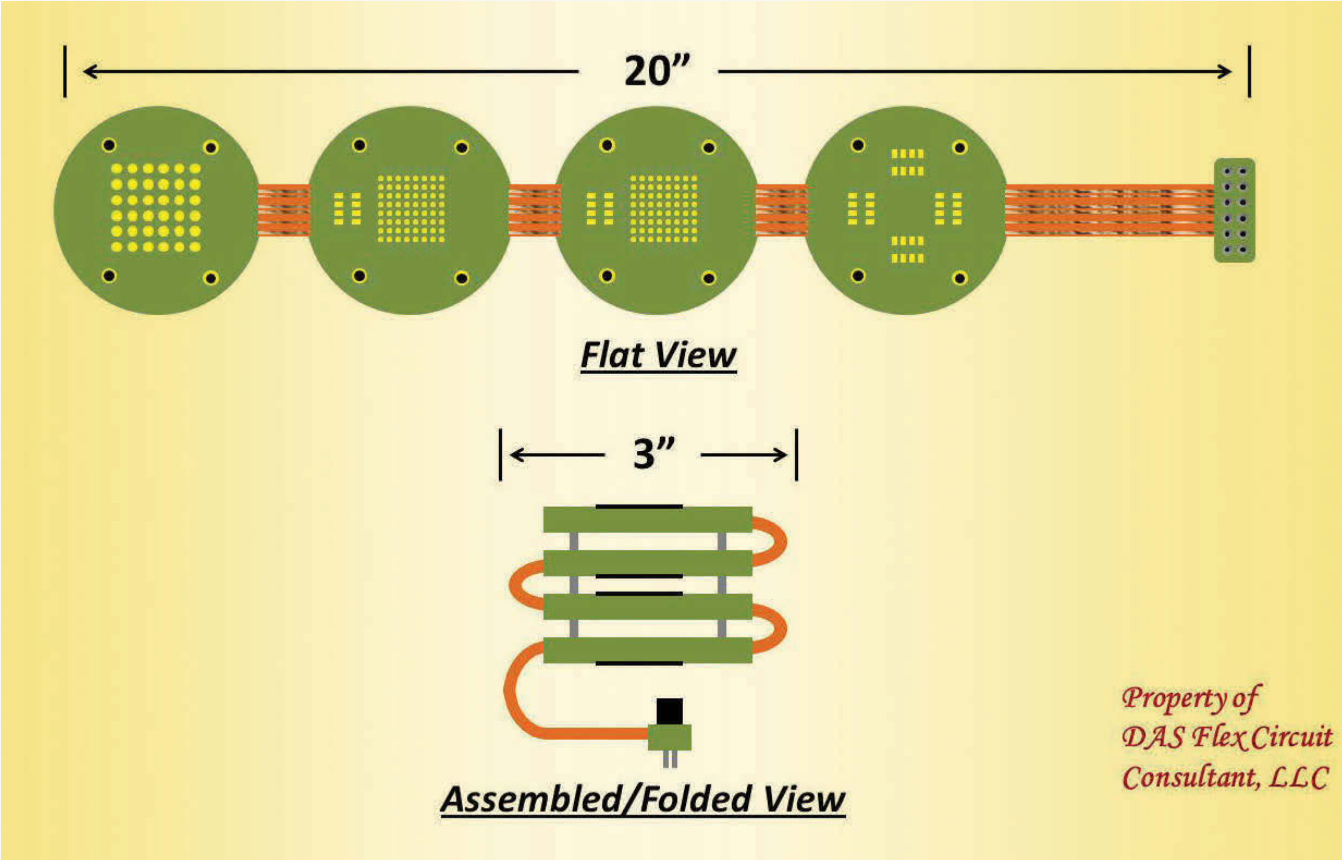
The flexible sections typically do not have plated through-holes to avoid reducing flexibility. However, some designs may require "sub-laminations" with plated through-holes or microvias. Another design may have inner layers that terminate with plated through-holes for receiving soldered connectors. Obviously, these requirements will add complexity to the design.

Why design rigid-flex?

The benefit of rigid-flex design is to provide a "three dimensional" interconnect solution for compact/limited-space applications that cannot be accomplished using conventional PCBs. Interconnects can be consummated in multi-directional planes even when they do not have "fixed" locations. Terminations can be flexible (no pun intended) with the addition of service loops to the flexible sections, simplifying the interconnect task without requiring precise dimensions or tolerances. They can also snake through or around obstacles inside an installation that would otherwise be impossible to interconnect (e.g., inside a cockpit).

The rigid section provides strength and stability for component placement and fixed hardware mounting. The flexible section enables conductor paths to be bent or folded (virtually into any shape) while maintaining electrical integrity. For example, some rigid-flex designs may have lengths of twenty inches or more, but when folded and installed, may only take up three square inches of area.

Rigid-flex offers many advantages when compared to conventional, rigid PCBs:



- Reduction in space/area consumed.
- Customized designs to fit almost any interconnect situation and an endless combination of materials and technologies that can provide solutions for the most challenging applications/installations.
- Thinner structures can be manufactured due to the availability of glass-free, thin copper-clad dielectric substrates, while maintaining excellent electrical and thermal properties. Some of the higher performance flexible materials actually have better electrical and thermal properties than rigid printed circuit boards.
- Lower weight (critical where weight is a concern; fuel cost, weight of equipment/hand-held devices, etc.)
- Elimination of connectors sometimes used to integrate rigid boards with ribbon cables, sometimes referred to as "jumper flex" or "poor man's" rigid-flex. Integration into one homogeneous unit improves signal integrity and eliminates these additional connections. Even EMI/RFI shielding, differential and twisted pairs can be incorporated into the design.

How has rigid-flex evolved?

Most rigid-flex was originally manufactured for military, aircraft (military and commercial) and aerospace applications due to space and weight limitations. Today, however, they can be found in industrial equipment, automotive, medical and commercial (even cell phones). There have been huge advancements in materials, equipment and processes that have dramatically improved reliability and driven down costs.

Today, rigid-flex of more than 40 layers has been successfully manufactured and is capable of meeting the rigorous environmental demands in military and aerospace environments. The rigid-flex market segment continues to grow year after year.

Rogers, Hitachi Partner on High-Speed Materials

by Real Time with...productronica 2011

Sean Mirshafiei, market development manager for Rogers Corporation, discusses Rogers' new alliance with Hitachi with I-Connect Editor Ray Rasmussen. The venture is designed to accelerate Rogers' expansion into materials for the high-speed digital market.

realtimewith.com

What are the disadvantages of rigid-flex?

The biggest disadvantage of rigid-flex remains higher cost when compared to conventional PCB's. This is due to:

- Higher material costs
- More complex design
- Reduced yields
- Increased number of fabrication steps
- Fewer fabricators with required manufacturing expertise
- Handling (even when the product is complete) and installation

These higher costs are usually justified by providing the interconnect solution needed where other methods are not possible, reliable or cost effective.

Market Segment

In this series, the rigid-flex market segment being discussed is for high-reliability applications (i.e., military, aerospace, industrial and high-end commercial). Here, the product is expected to perform long term in life support systems and/or harsh environmental conditions (temperature extremes, shock, vibration, multiple solderings, etc.). The most robust materials and processes are typically preferred to offer the highest manufacturing yields and superior end use performance. Rigid-flex for these types of applications is generally produced in batch-type manufacturing processes.

Next month, in Rigid-flex Manufacturing, Part II, the selection of materials for meeting the challenges of rigid-flex manufacturing and the performance demands of the end-use applications will be discussed. **PCB**

Dale A. Smith is President of DAS Flex Circuit Consultant and has more 30 years experience in the printed circuit industry. He specializes in rigid/flex high reliability manufacturing and Lean Manufacturing implementation into PCB shops. Reach Smith at 267-424-0690 or by e-mail at dale.smith68@verizon.net.

PCBDESIGN007 Highlights

Agilent Enhances RF Simulation Speed with EMPro
Agilent Technologies launched the latest release of its Electromagnetic Professional software, EMPro 2011.11.

Mentor Graphics FloEFD Targets New Applications
Mentor Graphics launched the next-generation of the FloEFD concurrent computational fluid dynamics (CFD) simulation tool.

Design-Focused Courses at IPC APEX EXPO 2012
IPC will offer a variety of design-related classes at IPC APEX EXPO 2012 in San Diego, during the Designers Forum and the professional development courses.

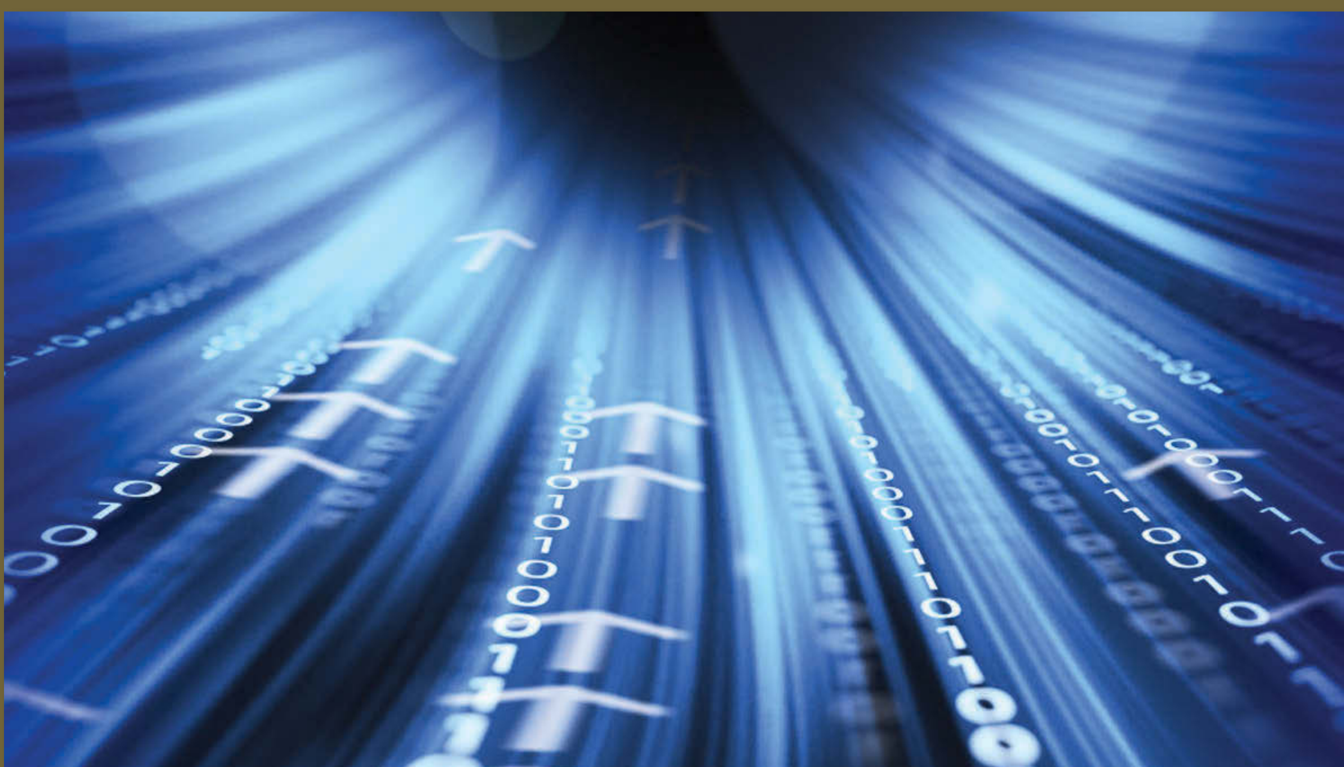
Zuken Launches Power Integrity Advance for CADSTAR 13.0
Zuken released Power Integrity Advance, a tool that conducts EMC analysis and AC and DC power integrity analysis.

Fujitsu Semi Standardizes on Mentor Graphics HyperLynx
Fujitsu Semiconductor standardized on the Mentor Graphics HyperLynx SI technology.

Screaming Circuits, element14 Partner to Enhance Offerings
The element14 online community for design engineers partnered with Screaming Circuits.

DfR Solutions, ISD Italia Partner on Sherlock ADA Software
ISD Italia partnered with DfR Solutions to offer DfR Solutions' Sherlock automated design analysis tool.

PentaLogix Now Offers Full Turnkey Service
PentaLogix began offering turnkey PCB services, from design through final PCB assembly.



What to Look Forward to in 2012

by Rick Almeida
DOWNSTREAM TECHNOLOGIES, INC.

SUMMARY

Financial experts will blog about the direction of the stock market and political pundits will tweet predictions of who is going to win the Presidential election, but 2012 also promises to be full of new and exciting products and initiatives for the PCB industry.

It's that time of year again, when people from all walks of life offer their opinion on what's going to happen next year. Hundreds of stories will be written about the hottest fashion trends, who will win the World Series and what drama will follow the Kardashian's around. Financial experts will blog about the direction of the stock market and political pundits will tweet predictions of who is going to win the presidential election.

So, with that in mind, here is one man's opinion on three things to look forward to in 2012 with regard to the PCB industry.

IPC-2581 Will See Continued Acceptance

In March 2004, an IPC committee developed and released IPC-2581: Generic Requirements for Printed Board Assembly Products Manufacturing Description Data and Transfer Methodology. This open, neutrally-maintained, global standard for data transfer, which some believe allows for improved efficiency, encourages innovation and helps reduce product development costs.

In June 2011, the IPC-2581 Consortium was founded by a variety of suppliers from across the PCB design/supply chain including Harris, Ericsson, Fujitsu, nVidia, Sanmina-SCI, Adiva, Cadence Design Systems, DownStream Technologies and Zuken. The mission of the new consortium is to bring companies together to enable, facilitate and drive the use of the IPC-2581 standard.

With dozens of companies joining the IPC-2581 Consortium and, supporting it with the development of products and services to import, export and consume IPC-2581, our industry has established a path towards the acceptance of an independent, neutrally-maintained data transfer format.

Design for Manufacturability Will Move into the Mainstream PCB Design Flow

As everyone would agree, it really doesn't make any sense to design a PCB that is

"As everyone would agree, it really doesn't make any sense to design a PCB that is problematic to manufacture."

problematic to manufacture. Additionally, with all of us doing more with less resources, and the pressure of saving money hanging over everyone's head, isn't it time we looked for a better answer?

One answer is design for manufacturability, or DFM. This general engineering art of designing products so that they are easy to manufacture, DFM is quickly becoming part of the everyday PCB design flow. In the PCB manufacturing environment, DFM is often being required as a first step because DFM issues detected too far along the manufacturing process create longer lead times, higher manufacturing costs and delayed product shipment.

PCB designs which pass electrical verification within the virtual PCB CAD system may unknowingly contain critical manufacturing flaws such as acid traps, soldermask slivers, copper slivers, starved thermals, soldermask coverage and more. These errors can create costly delays in manufacturing as the fabricator must address them in pre-production either by having the designer fix the problem or fixing the problem themselves. A design re-spin takes time; if the fabricator fixes the problem, the fixes they come up with are rarely fed back into the source CAD data, resulting in repeated DFM violations on later runs. In worst-case scenarios, design intent may also be unknowingly sacrificed when a third party manufacturer alters a PCB design. Thus, the greater acceptance of DFM into the mainstream design flow.

The Growing Importance of Stackup Design and Analysis Will be Recognized

Stackup refers to the types and arrangements of layers in a multilayer board. Today, there is a growing need to plan and design the stackup properly, as well as optimize it for better overall performance and communicate and document stackup information between engineering and fabrication. As with all other parts of the design flow, any errors found before release of the PCB design to fabrication can help reduce costs and increase output.

How important is the stackup design and configuration? Very important, because a well-built PCB substrate can effectively reduce electromagnetic emissions and crosstalk, while also improving signal integrity. Designed properly from the beginning, this can dramatically improve overall time-to-market by improving the manufacturability of the board.

Stackup design issues reside on both sides of the design-manufacturing chain. Engineers define stackup data based on electrical requirements. Fabricators need to replicate the stackup design with the materials available at the time of manufacture. In many cases, changes may be needed to build the PCH without delay. This creates a need for engineers and manufacturers to resolve stackup intent and stackup reality to ensure first pass success.

There are a few tools now on the market for stackup design including Speedstack® from Polar Instruments®, Hyperlynx® from Mentor Graphics® and Stackup Planner® from ICD®. In addition, the IPC-2581 consortium is looking at using the IPC-2581 standard as a metric to drive and share stackup information between design and fab. Expect to see more emphasis placed on stackup design and collaboration between engineering and manufacturing, including integration with these stackup tools and the core EDA CAD tool.

In the areas above, as well as many others, 2012 promises to be full of new and exciting products and initiatives. Here is looking forward to a successful and prosperous 2012 for our entire industry. **PCB**

Rick Almeida is a co-founder of DownStream Technologies, where he oversees company strategy and direction as well as corporate communications and product marketing. Previously, Almeida was VP of marketing for Innoveda's Product Realization Group and prior to that, VP of World Wide Marketing for PADS Software.

Is There a Future for "Made in America?"

A Q&A with Mass Design's Bill Gately

by Dick Pirozzolo

SUMMARY

What is the future of the PCB industry in North America, particularly in the low-volume, high-reliability sector? A leading U.S. PCB manufacturer weighs in on who is surviving and why.

As sales manager of Mass Design, Inc., a PCB manufacturer based in Nashua, New Hampshire, Bill Gately has more than two decades of experience in the PCB design, fabrication and assembly industry. Prior to joining Mass Design, Gately spent ten years with CGI Circuits. His position gives him a unique perspective from which to comment on the future of the PCB industry in North America, particularly in the low-volume, high-reliability sector.

Gately works with a diverse customer base made up of R&D, start-up and Fortune 500 Companies who produce military, medical, aerospace and industrial products, as well as PCB contract manufacturers. He has increased sales at Mass Design by attracting and retaining customers who see working with a U.S.-based company as a big benefit — especially when it comes to fulfilling orders that are as few as five boards, or as many 2,500.

Pirozzolo: ***We hear about outsourcing all the time. You're an American manufacturer with partners in Asia. What's the future of the "Made in the USA" label when it comes to PCBs?***

Gately: I'm optimistic. Coming out of the worse part of the recession, we saw an industry-wide recovery in 2010 sales for U.S. companies. According to IPC, rigid board sales were up 18% and flex board sales increased 16% from 2009-2010.

In 2011, Mass Design increased sales at the same rate, though industry-wide figures have not been as dramatic. Sales for 2011 are up about 4 percent for all electronic products including PCBs; however, the economy is growing at 2.5%. I see PCB sales outstripping general economic growth as a very positive sign.

"Sales for 2011 are up about 4 percent for all electronic products including PCBs; however, the economy is growing at 2.5 percent."

Year-to-year and month-to-month statistics only tell part of the story; long-term, the potential for growth in our industry is as immense as it is incomprehensible. Take a single simple item — hotel room keys. Keys have universally been replaced by card readers, all of which require electronic circuitry. That's just one change of a commonplace product that occurred 15 years ago. Kids today wouldn't know what to do if the front desk clerk handed them a key. These same kids are all walking around with iPhones that are more powerful than the laptop I bought last year.

The American auto industry is coming back, too. Compare your five-year old car to the 2012 models that routinely have rear video cameras, electronic braking systems and tire pressure monitoring gauges and, of course, built-in GPS, satellite radio and bluetooth — with many of these new features mandated by law.

Likewise, capital investment in U.S. infrastructure isn't all about bulldozers and concrete. It's about improved railway signaling, automated toll collection, video surveillance and improved fuel efficiency — again, increasing demand for PCBs.

Pirozzolo: ***Industry-wide, that's terrific. But with the bulk of PCB boards outsourced to Asia, what's left for U.S. and Canadian manufacturers?***

Gately: There isn't much future on the mass-produced, consumer-products side of the equation — toasters, video games and cell phones. That business has gone offshore and margins are too slim for North American companies to compete.

Where there once were about 4,000 PCB manufacturers in North America, now there are only about 200, with a smaller number of PCB manufacturers doing really well.

The real success stories are the U.S. companies that have developed a hybrid-manufacturing strategy by offering complete capability in the U.S. combined with the skill needed to negotiate with, and manage, reliable manufacturing partners overseas — particularly China and Taiwan. U.S. manufacturers outperform Asian manufacturing reps with a phone and desk, because they offer complete preproduction capability in the States and can quickly turn around prototypes. It's something customers see as a huge benefit even if their manufacturing is eventually outsourced.

Pirozzolo: ***So, no matter how you slice it, U.S. production costs are going to be higher?***

Gately: I was just getting to that. For small quantities, in our high-reliability niche, the marginal savings from manufacturing in a foreign country is outweighed by quicker turnaround times and delivery costs.

For longer runs, we've had times when a delivery from Asia is delayed. We'll turn around the same product in-house in the USA and ship it within 24 to 48 hours to meet a customer's deadline. We've also had times when 3,000 boards are due to come in from Asia in a month or so, but we'll produce 500 immediately in Nashua so the customer can start production.

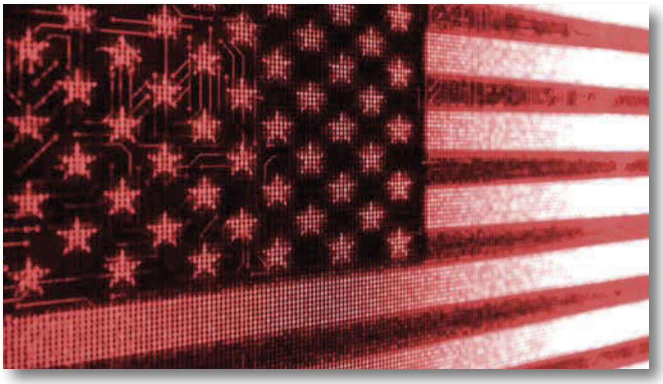
That kind of assurance and comfort level is what the successful U.S. manufacturers are offering. It's a customer-focused way of thinking and it pays off.

Pirozzolo: ***What about ITAR-compliant manufacturing that has to take place in the U.S.? Do cutbacks in defense spending worry you?***

Gately: No. Just look at how electronics are replacing active-duty people and military infrastructure. Compare the manpower cost of maintaining a fighter squadron in a warzone to piloting the Predator Drone from an Air Force Base in Nevada. Less manpower simply means more technology.

The same goes for missiles, satellite tracking, weapons controllers and command and control systems — it's all low-volume, high-reliability work and the five-year outlook is bright.

Pirozzolo: ***ITAR keeps that work in the U.S. Can't high-reliability be outsourced when it comes to avionics and medical applications?***



Gately: Medical devices are produced in small to medium volumes with solid markups. It's an industry where manufacturers want their boards released to them monthly or quarterly, so overseas transit times, cost and complexity become a factor. These manufacturers won't risk a production delay by going overseas.

Medical devices present a big growth area for the PCB industry. Physicians are using more handheld diagnostic tools and hospitals are sending patients home earlier with electronic monitoring equipment. For some patients, dialysis can now be performed at home. We are also seeing potential as the age-at-home population increases and relies on electronic pill monitoring and other systems. All of this means smaller, lighter, more rugged and foolproof equipment.

It's good news for our industry, especially in flex where you have to cram much more capability into smaller spaces and the equipment has to be drop-proof. We saw 18 percent growth in flex sales at Mass Design last year and I'm sure we're not alone.

Pirozzolo: ***Any other indicators?***

Gately: Sure. Boeing's new 777 is being produced in the U.S. and the volumes, in terms of avionic equipment, are so low and the reliability requirements so high that it just doesn't make sense to outsource that kind of work.

Going green is also becoming an incentive. We're hearing from customers who feel more comfortable buying in North America, because of our high wastewater- treatment and worker safety standards — something they can't always count on overseas.

Pirozzolo: ***Is there anything else a U.S. manufacturer can do to keep an edge?***

Gately: Customers want to keep as much labor out of their shops as they possibly can. Value-added products and fulfillment, such as offering box-builds to streamline the costumer's operation, is a winner. One of our customer's does box-builds for a producer of handheld electronic trouble-shooting devices for auto service centers and ships about 50 orders per day.

Pirozzolo: ***Is there a four-minute mile barrier to thin and the number of layers possible in a PCB?***

Gately: Smartphones have led the way to thinner boards and are increasing the number of layers — a minimum of 10-12 layers and an overall thickness .015. Copper is a quarter-ounce with thicknesses from .5 mil to .3 mil. Standard part placement of 0201 is now 01005. Additionally, interlayer leaves are much thinner.

Those standards on the consumer side are simply driving expectations in the low-volume, high-reliability market.

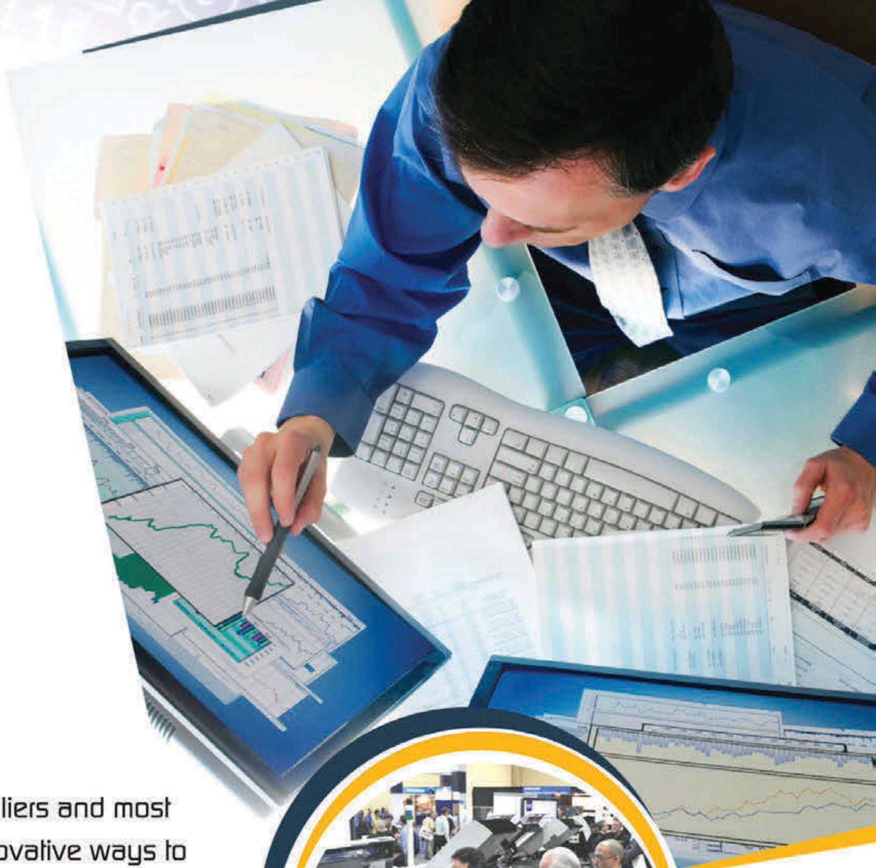
Pirozzolo: ***You seem optimistic. In a nutshell, who will be the U.S. PCB success stories five years from now?***

Gately: The successful American companies of the future are the ones that continually invest in well-educated, talented people. They also invest in capital equipment that automates their manufacturing processes, as well as advanced software to assure reliability. Above all else, they will offer great customer service. **PCB**

Dick Pirozzolo is a member of the Society of Professional Journalists and the Foreign Press Association, and writes about global technology and energy.

Innovation drives results

APEX EXPO® IPC



In this economy, innovation is everything and IPC APEX EXPO 2012 is where you'll find the information to inspire it!

We are gathering the brightest minds, top suppliers and most exciting technologies to help you discover innovative ways to manufacture, manage your supply chain and succeed in 2012. Learn at the world's premier technical conference with leading experts and top-tier educational courses. With an exhibit hall overflowing with advanced and emerging technologies in printed board design, manufacturing, printed electronics and electronics assembly, this event is second to none.

Invest just a day or more and take home the tools that will drive your results for 2012 and beyond. Register early for FREE exhibit hall admission and extra savings at

www.IPCAPEXEXPO.org

"The conference and exhibition gave me a lot of information on BTCs and flex circuits to address our challenges in working with these technologies and to drive positive results. It was time will spent!"

- Larry Davis, Production Engineer, Hach

CONFERENCE & EXHIBITION
February 28–March 1
San Diego Convention Center

2012

• DESIGN • PRINTED BOARDS • ELECTRONICS ASSEMBLY • TEST

with a new focus on printed electronics

Use your smart phone reader! Enter for a chance to win a free All-Access Registration or a free hotel stay!

INFORMATION that INSPIRES INNOVATION

2012: Slight Growth Following a Sluggish Start

by Matt Scherer
DATABEANS, INC.

SUMMARY

Consumer spending trends indicate that those product categories that became popular this year will further pick up steam in 2012, benefiting from new technologies, new operating systems and the return of consumer electronic demand.

The year 2011 was characterized by major ups and downs and several reversals of fortune across the semiconductor industry as a whole. On the one hand, the exploding popularity of tablets and smart phones has caused great demand for certain semiconductor products, notably NAND Flash memory. On the other hand, thanks to a combination of excess inventory, manufacturing overcapacity, natural disasters and slowing consumer demand across most regions, the year 2011 will end on a sour note for IC suppliers. In fact, starting in the third quarter, most suppliers' revenue fell well below seasonal averages, leading to a weaker-than-expected fourth quarter. This was despite a positive holiday season for portable electronics sales. As a result, Databeans expects that the first quarter 2012 will continue this trend, but the market should begin to improve towards the middle part of 2012 as consumer confidence returns in Europe and the U.S. while regional currencies stabilize (Figure 1).

Intel, the world's largest semiconductor firm by sales, re-exerted its market dominance in 2011 after its leadership role has been slowly eroded over the last few years by Samsung Electronics. This is because Intel gets much of its revenue from the sale of microprocessors (MPUs) and NAND flash memory produced in partnership with Micron Technology, which happened to be two of the hottest segments in the entire chip industry for 2011. Particularly, Intel's Sandy Bridge second-generation core processors which were first introduced in January have become the high end processor of choice in numerous laptop and tablet designs. In November 2011, the company further demonstrated its dedication to the Sandy Bridge line with its latest version, the powerful 6-core Sandy Bridge-E (Extreme) processors. Heading into 2012, Intel's primary plans is to focus on pushing its new, powerful Ultrabook laptops with the brunt of its marketing efforts. Currently, the few devices on the market from OEMs like Toshiba, Acer and Lenovo feature Sandy Bridge processors,

> "Heading into 2012, Intel's primary plans is to focus on pushing its new, powerful Ultrabook laptops with the brunt of its marketing efforts."

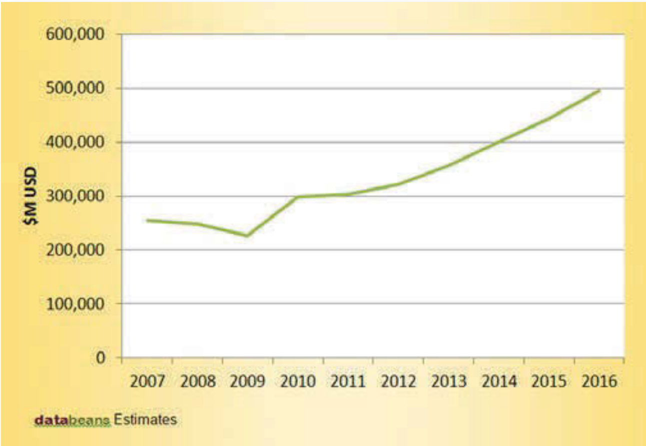


Figure 1: Worldwide Semiconductor Revenue Forecast.

but starting in 2012 several firms will release new touch-screen Ultrabooks that incorporate Intel's more power-efficient Ivy Bridge processors. The Ivy Bridge platform will begin to close the computing power gap between laptop PC's and tablet interfaces.

Samsung, the world's second-largest semiconductor player by sales and also the largest memory supplier, had a somewhat mixed year. On one hand Samsung benefitted from its market leadership in NAND and rose to become the world's largest top smartphone seller. However, the company is also the world's top supplier of DRAM, which has reached a critical inventory overstock situation causing prices to plummet, negatively impacting Samsung's profit margins. Ultimately, this situation could benefit the company in the future, as the field of competitors may soon shrink and allow them to raise prices. The competitors worst affected have been those in Taiwan's once prominent DRAM industry, including Inotera, Powerchip Semiconductor, Nany, and Rexchip. These companies have faced a devastating 2011 thanks to the collapse of DRAM pricing. In fact, all together, the four top Taiwanese DRAM players posted a total loss of about $928 million for the third quarter period between July and September 2011, which forced them to cut expectations for the fourth quarter and slash total capital expenses that were previously planned for 2011 expansions. As a result, the situation for these smaller DRAM players will remain grim in 2012 and could force many to seek diversification or consolidation just to survive.

For Texas Instruments, the overall leader in analog chips and the third-largest semiconductor supplier, 2011 was eventful as the company purchased rival analog chipmaker National Semiconductor for $6.5 billion, giving the new combined firm an even greater grip over the profitable analog chip industry. Despite poor results across most other segments, analog and mixed-signal semiconductors have managed to maintain growth for the year as a whole. In particular, products in the consumer, wireless and

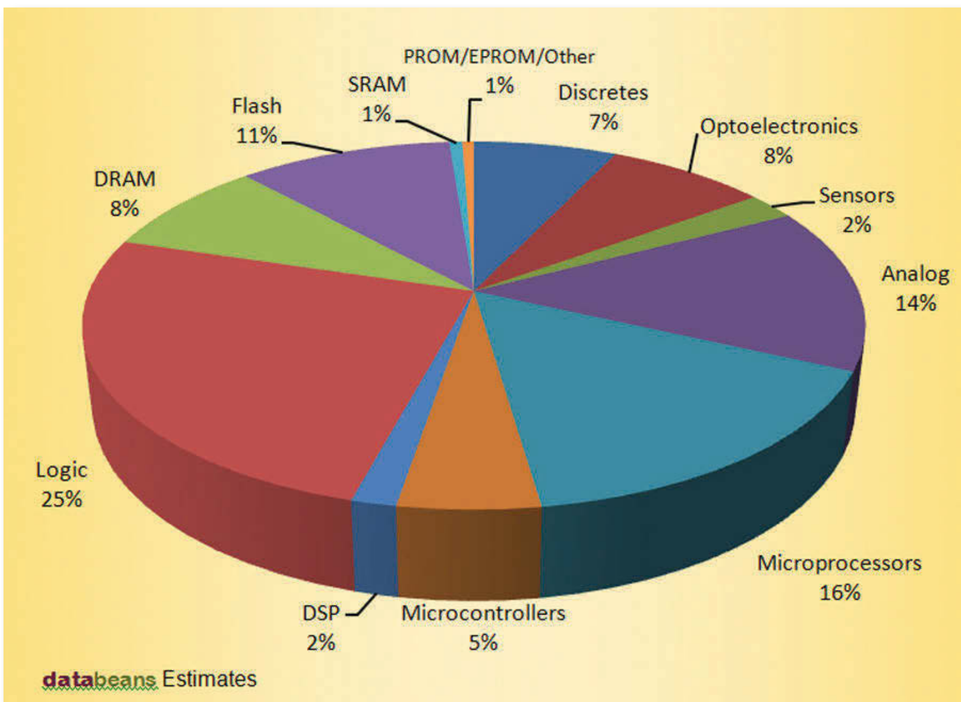


Figure 2: 2012 worldwide semiconductor revenue shares by market type.

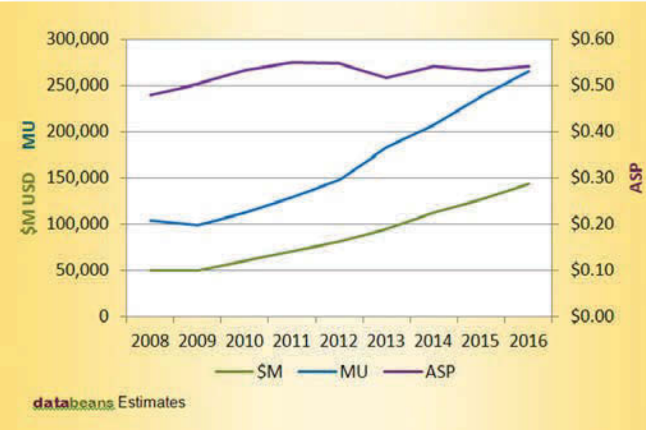


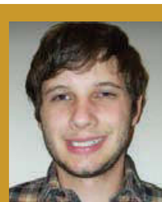
Figure3: Worldwide wireless semiconductor market forecast.

tablet computer markets are performing the best, which will continue into the next year. Heading into 2012, Texas Instruments' plans to focus on its popular dual-core ARM application processors for mobile devices like smartphones and tablets. The company will continue to focus on developing dual-core processors in the short term, while TI's competitors such as Nvidia and Qualcomm will continue their push for more powerful and more power-hungry quad-core designs in mobile devices. You can expect to see these devices at CES in January and in consumers' hands by mid-2012.

The traditional PC market, once the darling of the semiconductor industry, will continue its slip from 2011 into 2012 as consumers rapidly make the shift to the mobile computing offered by tablets and smartphones. Intel hopes to reverse this trend somewhat with the push towards the Ultrabook class of computing, which the company projects will account for 40% of all laptop shipments by the end of next year. While it is unclear whether Intel will reach its target or not, Ultrabooks are already proving to be highly popular in the enterprise segment and with high-end computing enthusiasts.

As a whole, the year 2012 should be improved over 2011 for both suppliers and OEMs. Consumer spending trends indicate that those product categories that became popular this year will further pick up steam in 2012, benefiting from new technologies, new operating systems and the return of consumer electronic demand. For example, in the smart phone market, Google's Android mobile OS has continued to dominate, surpassing half of the global smartphone operating system share in the third quarter of 2011. Thanks to the high volume of popular phones from players such as Samsung and HTC, as well as the fourth-quarter release of its fourth-generation version, Ice Cream Sandwich, Android should remain the top mobile OS in 2011. Apple's iOS, the second-largest operating system, continues to gain popularity, particularly in the tablet PC space, thanks to the incredible popularity of the iPad 2. This will continue in 2012, as the firm will most likely announce a third-generation version of the device sometime in the next year.

In conclusion, the 2012 year will consist of a slowly growing overall semiconductor market. This effect is achieved by flourishing markets such as NAND flash and automotive counteracting the diminishing effects of DRAM and other memory such as PROM and EPROM. Most large gains will be made with consumer interest in mobile devices and inventory replacements for the industrial and computer segments. Suppliers and OEMs will be competing for sockets in the low power market and expanding their reach to the strong growth markets. Companies that capitalize on the wireless market and the infrastructure needed to support its steadily increasing speeds will fair best over the next year. This month's CES will focus on devices that rely on many types of sensors and optoelectronics, which will make them two of the more prominent growth segments for the year. **PCB**



Matt Scherer is a research analyst for Databeans, Inc., a market research firm focused on the semiconductor and electronics industry. Since its inception, Databeans has grown to become both a source of information and advisor on all semiconductor product categories and markets. For more information, contact Brice Esplin (brice@databeans.net) or Matt Scherer (matt@databeans.net).

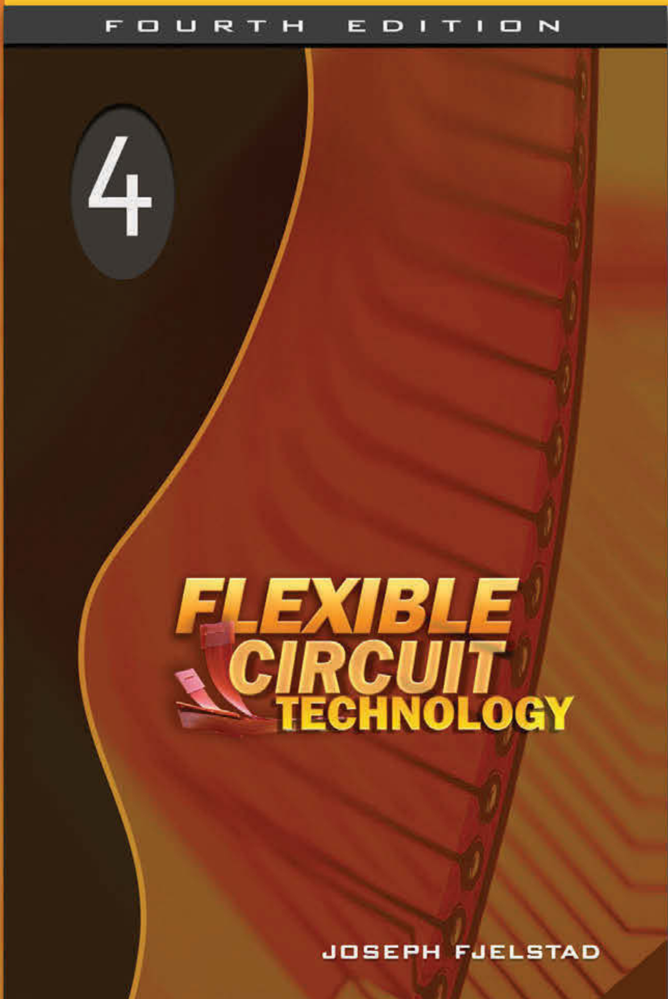
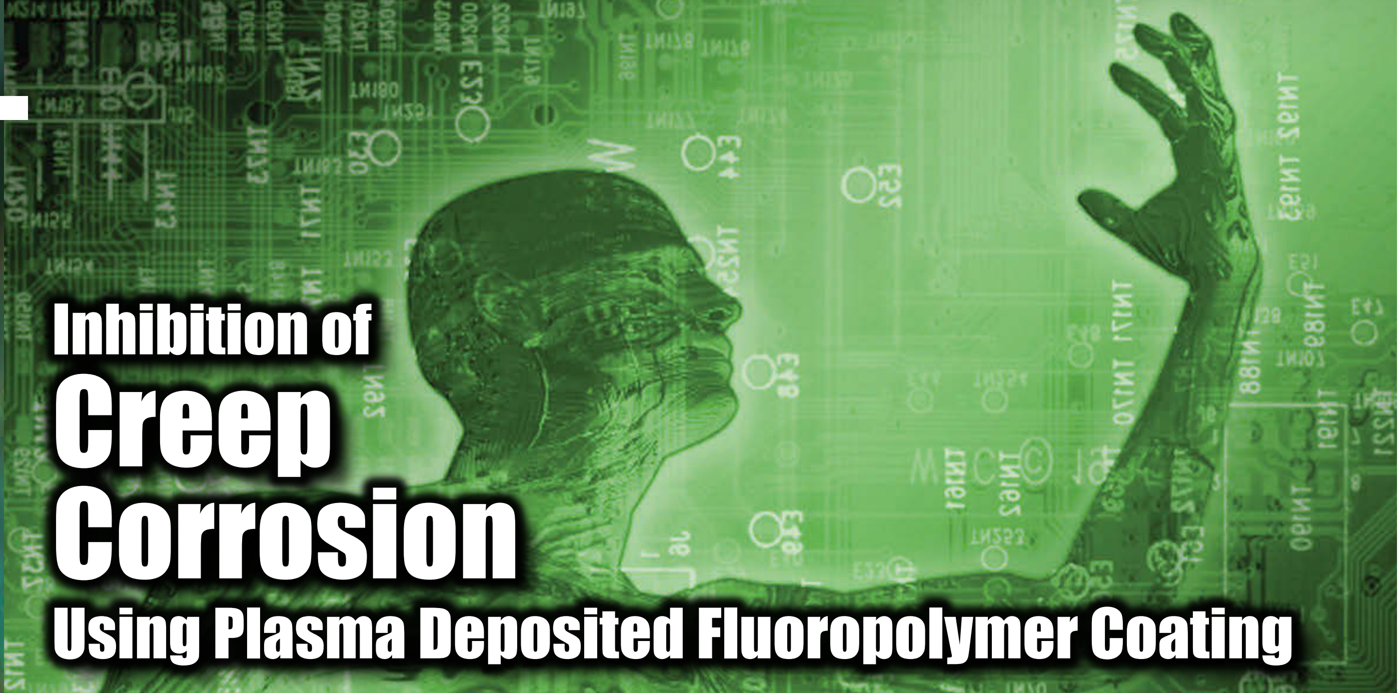


Table of Contents



Inhibition of Creep Corrosion Using Plasma Deposited Fluoropolymer Coating

by Tim Von Werne
SEMBLANT LTD.

SUMMARY

Creep corrosion is a growing problem for the electronics industry, where a copper sulfide corrosion product migrates across the surface of a PCB in a high-sulfur, high-humidity environment. A new coating used as a final surface finish offers the ability to prevent creep corrosion in aggressive environments.

Introduction

Throughout the past decade the electronics industry has faced challenges to existing manufacturing processes as it responds to changes in legislation and guidelines that impact electronic products. Additionally, technology continues to evolve and demands products that are smaller, faster and cheaper to produce. The environment in which electronic products are now expected to regularly operate is growing ever harsher due to increased pollution levels and the increasing use of these products in high-heat, high-humidity areas. Failure of electronic products due to corrosion caused by environmental exposure is becoming increasingly common.

One particular type of corrosion that has begun to manifest itself is creep corrosion. This failure mechanism is of particular concern as it can lead to catastrophic failure in products either by causing electrical shorts between adjacent tracks or electrical opens in conductive tracks. Creep corrosion has been well studied over the last five to seven years, and the failure mechanism is now fairly well understood. Creep corrosion manifests as blooms of conductive material that grow outward, across the surface of the PCB from exposed metal contacts. The composition of the growing corrosion product has been shown to be primarily copper sulphide (Cu_2S) [1]. The failure is caused by excessive levels of sulphur in the atmosphere, combined with moisture to create a highly corrosive environment at the surface of the electronic assembly. Creep corrosion differs from electrochemical migration in that no applied bias is required in the case of creep corrosion, and the corrosion product tends to grow out in all directions from the metal contact whereas in electrochemical migration, dendritic growth occurs between two biased electrodes. In some cases, product failure was occurring within weeks, although two to four months of service life was common.

While nearly every final surface finish has been shown to exhibit creep corrosion in the most aggressive atmospheres, immersion silver is generally understood to be the most susceptible surface finish to this failure mechanism. The explanation for this particular susceptibility is that silver is highly sensitive

> *"Creep corrosion has been well studied over the last five to seven years, and the failure mechanism is now fairly well understood."*

to sulphur, combined with the fact that the immersion silver coating is typically very thin and can be prone to small defects. Defects in the PCB manufacture, such as solder mask attacks, can leave small areas of exposed copper, which is also highly sensitive to attack by sulphur. Because of this, solder mask-defined pads tend to be especially vulnerable to creep corrosion. The combination of exposed copper in the presence of silver and sulphur can even then accelerate the growth of the copper sulphide corrosion product by setting up a galvanic cell, as shown in Figure 1, reproduced with permission from R. Schueller.

There have been several test methods developed to accelerate creep corrosion. The first widely reported method was designed to reproduce conditions in the field in a particular location and was based on exposing samples to an atmosphere created by modelling clay [1]. The Chavant Clay method involves placing the test samples in a closed container containing sulphur-bearing modelling clay which has been heated and a moderate level of humidity. A second method, also based on actual field conditions is the Tyre Factory Test published by Toscano et al [5]. This test also uses a high sulphur content atmosphere, generated from sulphur powder and mercaptobenzotriazole. Both tests have been used to generate creep corrosion on a wide range of samples, but these methods can be so aggressive that they lack the ability to differentiate between different surface finishes and may not accurately indicate which samples are truly likely to suffer from creep corrosion in the field. Alcatel

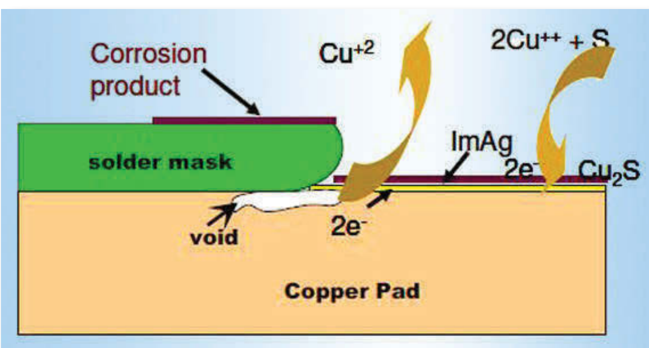


Figure 1. Schematic showing creep corrosion mechanism.

Lucent has developed a modified mixed flowing gas (MFG) test based which possibly offers a more controlled and repeatable test method [4], but this test requires specialized equipment and can be expensive to run. This MFG method achieves creep corrosion by greatly increasing the amount of H2S in the test, up to 2000ppm. This paper will mainly present results from the Chavant Clay method, but these findings have been confirmed using the Tyre Factory test; mixed flowing gas testing is currently ongoing.

Semblant SPF Plasma Deposited Fluoropolymer Coating

The use of a new plasma deposited fluoropolymer coating as a solderable final surface finish for PCBs from Semblant was first presented in October 2009 [2]. The plasma finish was shown to exhibit exceptional corrosion resistance while maintaining good solderability for SMT and through-hole soldering. This surface finish has the potential to change the PCB manufacturing process, shrinking environmental impact by reducing water consumption and heavy metals usage.

The use of gas plasma as a processing technique can be a very efficient and powerful tool. Plasma processing is used extensively in industries as varied as semiconductor manufacturing, automotive component manufacturing and medical device fabrication. The use of plasma as a deposition technique represents a novel approach for coating PCB panels. In a conventional plasma cleaning process, a gas such as oxygen or argon can be used to chemically or physically etch a surface. In the plasma deposition process, a reactive precursor gas is used which can polymerize in the conditions generated in the plasma chamber. The growth of the polymer film can be controlled by adjusting input parameters, and the continuity and quality of films deposited using this method has been well documented [2]. In the case of the plasma coating for PCBs, a fluorocarbon precursor gas is introduced to a plasma chamber that has been specifically designed for the treatment of large format PCB panels. Striking the plasma in the chamber activates the precursor gas and a thin coating of a dense, highly cross-linked fluoropolymer film is deposited across the entire surface of the PCB panel.

The coating provides protection to the PCB against corrosion and surface oxidation and has been previously shown to impart high levels of resistance to corrosion in high sulphur environments. Printed circuit boards coated with the plasma deposited fluoropolymer were subjected to a Class III environment mixed flowing gas test, consisting of 100ppb of H_2S, 200ppb of SO_2, 200ppb of NO_2 and 20ppb of Cl_2, for a period of 20 days and were compared against unprotected PCBs finished with immersion silver (ImAg), organic solderability preservative (OSP), and electroless nickel immersion gold (ENIG). The fluoropolymer coated samples produced virtually no corrosion (<5% of the surface area), while the unprotected PCBs were all substantially corroded.

Experimental Method

The fluoropolymer coated samples were prepared by cleaning test samples to remove any contamination or oxidation using a dilute sulphuric acid wash, then placed into the plasma deposition chamber. The chamber was evacuated, the process gas was introduced and the plasma was struck and the deposition process was allowed to run for seven minutes. The coated samples were removed from the chamber and the coating thickness measured using an FTIR spectrometer as previously described [2]. The average thickness of the plasma deposited fluoropolymer coating was 40nm.

The Chavant Clay creep corrosion test was performed by heating 50g of Chavant J-525 sulphur bearing clay to the point of bubbling in a microwave oven. The clay is then immediately placed into a glass desiccator with the test samples and 5ml of deionized water and the chamber is sealed. The clay is re-heated twice daily and replaced with new clay every 48 hours. The samples are inspected regularly under a microscope for creep corrosion and the test is run until the uncoated reference samples exhibit severe corrosion.

Creep Corrosion Testing Results

Chavant Clay creep corrosion testing has been performed on a series of samples in order to determine the effectiveness of the fluoropolymer coating at inhibiting creep corrosion. Several reference samples were included in the test to confirm that the conditions would generate creep corrosion and to serve as comparisons. The test set consisted of bare copper and the Semblant finish over copper PCBs. The bare copper boards were included because exposed copper has been shown to be susceptible to creep corrosion. The samples were tested after one lead free reflow, to ensure that the fluoropolymer would survive the harsh thermal excursion. Images taken of specific features on the PCBs are shown in Figures 2 and 3. The low (left) and high (right) magnification images in Figure 2, show the surface of the bare copper PCB after the seven-day exposure to the creep corrosion test. The bare copper sample shows significant amounts of creep corrosion occurring after seven days exposure to the test environment, with the tell-tale copper sulphide blooms appearing. The surface of the bare copper is

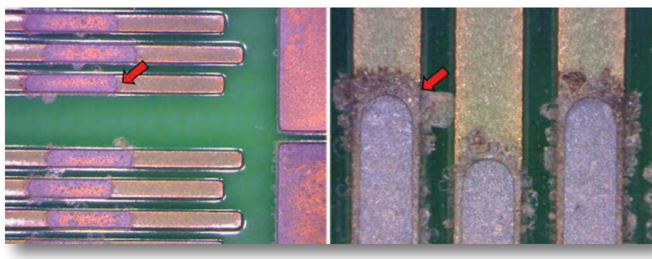


Figure 2. Micrographs showing bare copper PCBs subjected to seven-day exposure to the Chavant Clay creep corrosion test conditions.

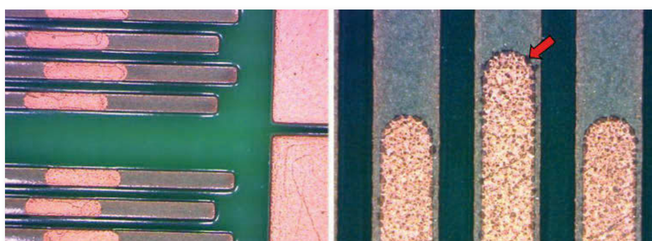


Figure 3. Micrographs showing Semblant SPF™ coated copper PCBs subjected to seven-day exposure to the Chavant Clay creep corrosion test conditions.

heavily corroded. The creep corrosion in these samples is most prevalent in areas where the solder mask over laps the underlying metal trace, and confirms that solder mask defined features are particularly susceptible to this issue. Figure 3 shows low (left) and high (right) magnification images of the surface of the Semblant SPF™ coated copper PCBs after the seven-day exposure to the test environment. The fluoropolymer coated copper PCB show a significant improvement over the bare copper sample. The copper does not appear to have been extensively corroded, and very little, if any, creep of the corrosion product is evident. Even at the copper / solder mask interface almost no creep corrosion was observed.

Creep Corrosion Inhibition Immersion Silver

In order to test the effectiveness of the fluoropolymer coating at inhibiting creep corrosion, the coating was tested as an over coat on immersion silver finished PCBs. fluoropolymer coating was applied either before or after a lead-free reflow cycle using the same deposition recipe that was used to coat the copper samples, with an average coating thickness of approximately 40nm. The PCBs were then placed into the corrosion chamber, along with several bare ImAg samples for reference, following the test method detailed above.

Representative images of the test samples are shown in Figures 4 and 5. The low (left) and high (right) magnification images in Figure 4 show an uncoated ImAg finished PCB at seven days exposure to the sulphur clay environment. As would be expected in a high sulfur content environment, tarnish and corrosion are apparent very quickly, and by the end of day seven, the corrosion product has begun to creep across the solder mask, as highlighted by the arrow. The higher magnification image from the same sample shows an area of solder mask defined pads which are particularly affected by the creep corrosion.

Figure 5 shows tht one observation from this test is that the ImAg finish under the fluoropolymer does not tarnish in the same way as the uncoated sample, and even at the end of the seven-day test the silver finish does not appear to have been compromised. There is a deposit that seems to form on the surface of the fluoropolymer, as highlighted by the arrow, but on closer inspection this appears to be condensed clay rather than a corrosion product. The higher magnification of a solder mask defined area highlights again the lack of creep corrosion across the surface of the solder mask.

Discussion of Prevention Mechanism

There are several factors which need to be combined for creep corrosion to occur: an exposed metal surface that is susceptible to sulphur induced corrosion, a positive interaction between the corrosion product and the surrounding surface which enables it to creep, and ideally an overlap between the metal and a solder mask. Careful design of the PCB can reduce the risk of creep corrosion in many instances by addressing the latter factor. Use of solder mask to cover as much of the board as possible, along with minimizing the use of solder mask defined features can reduce the exposed metal and solder mask interfaces

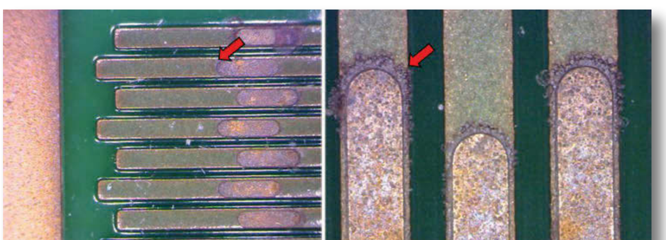


Figure 4. Micrographs showing bare immersion silver PCBs subjected to seven-day exposure to the Chavant Clay creep corrosion test conditions.

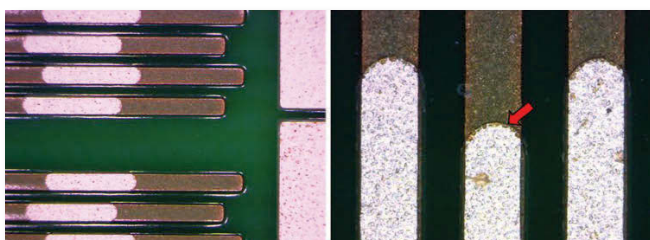


Figure 5. Micrographs showing Semblant SPF™ coated immersion silver PCBs subjected to seven-day exposure to the Chavant Clay creep corrosion test conditions.

which are the sources of the creeping corrosion products. The choice of solder paste and flux has been shown to be important in determining if creep corrosion occurs, and the careful selection of these materials may be able to minimize creep corrosion in some instances [4]. Removal of flux residue contamination from the surface of the assembled PCB can also help as these residues can encourage the creep of any corrosion products that have been formed by eliminating the positive interaction between the corrosion product and the surrounding surface. Recently, several organic top coats have been developed to be used in conjunction with the ImAg surface finish to prevent creep corrosion [3,6]. These top coats are self-assembled monolayers applied as post-dips to the plated silver, and they work by inhibiting the corrosion of the silver and any exposed copper. Each of these methods can independently address one of the three factors influencing creep corrosion.

The fluoropolymer coating is thought to act against creep corrosion by addressing multiple factors: First, it is an effective corrosion inhibitor. This coating has been proven to prevent sulphur driven corrosion in a number of different corrosion tests. However, as with the self-assembled monolayer methods mentioned previously, no corrosion barrier is perfect and eventually all samples can be expected to show some signs of corrosion. Simply adding a corrosion barrier will extend the lifetime of a product susceptible to creep corrosion, but cannot be guaranteed to prevent it from occurring in the long term. The fluoropolymer coating also addresses a second factor influencing creep corrosion: it changes the surface energy of the surrounding solder mask. In order for the corrosion products to creep across the solder mask surface, the interfacial energy between the copper sulphide and the solder mask must be lower than the surface energy of the solder mask material. The addition of the fluoropolymer coating drastically reduces the surface energy of the solder mask, thus making the creep of the corrosion product unfavourable.

Other possibilities include the integration of this material into the manufacturing process flow. The coating can be applied as a top coat by the PCB fabricator, prior to assembly. Alternatively, the coating can be applied by the electronics manufacturer after the board has been assembled. In this case the coating would act more as a conformal coat. **PCB**

References

[1] Schueller, R., "Creep Corrosion of LF PCBs in High Sulfur Environments", SMTA Int'l Proceedings, 2007.

[2] von Werne, T. Ph.D., Brooks, A., Woollard, S., "Latest Developments in Surface Finishing of PCBs Using Plasma Deposition", SMTA Int'l Proceedings, 2010.

[3] Long, E., Ph.D, Swanson, J., and Toscano, L, "An Immersion Silver PWB Surface Finish to Combat Harsh Environments", Journal of the HKPCA, No 38, 2010.

[4] Chen, Xu et al, "Creep Corrosion of PWB Final Finishes: Its Cause and Prevention", IPC/APEX Proceedings 2009.

[5] Toscano, L., Long, E., "Creeping Corrosion of PWB Surfaces in Harsh Sulfur Containing Environments", SMTA Int'l Proceedings, 2007.

[6] Kenny, J., Antonellis, T., Sun, S., Wang, C. Ph.D., Kudrak, E., Abys, J. Ph.D., "PWB Creeping Corrosion Mechanism and Mitigation Strategy", SMTA Int'l Proceedings, 2008.

A version of this article was originally published in the Proceedings of the SMTA International Conference, Fort Worth, Texas, October 16 - 20, 2011.

Tim Von Werne is CTO of Semblant Ltd. and manages a team of engineers in the company's R&D facility near Cambridge, UK. Prior to joining Semblant, Von Werne spent seven years as director of research at Plastic Logic, where he was responsible for the development of a new process for the fabrication of flexible displays based on polymer thin-film technology.

SUPPLIER/NEW PRODUCT Highlights

Lee Ritchey, Isola Team for Differential Signaling Class

The course, entitled, "PCB Stackup Design — Optimizing Signal Integrity, Manufacturability and Reliability of a Printed Circuit Board," focuses on high-speed differential signaling issues.

DuPont MCM Releases New Silver Conductor for PE

DuPont Microcircuit Materials (MCM), a business unit of DuPont Electronics & Communications, has introduced its latest screen printed silver conductor material for the printed electronics market.

Schmid Group Unveils Vertical Etcher

Precision machine manufacturer Schmid Group earns 50% of its revenue in the Chinese market. But, the company is a significant supplier to the PCB industry worldwide, serving the U.S., Taiwan, Japan and Europe. Laurent Nicolet, VP of the PCB Business Unit, discusses the company's new vertical etcher and wet process solutions.

HAM, Union Tool Enter Sales Cooperation Agreement

The technology company Hartmetallwerkzeugfabrik Andreas Maier GmbH (HAM) and Union Tool SA, the world's leading maker of carbide tools for manufacturing PCBs, have entered into a sales cooperation agreement for Europe starting in January 2012.

Park Electrochemical Introduces New Digital Products

Park Electrochemical Corporation announced the introduction of N4800-20 and N4800-20 SI®, the company's new high-speed digital electronic material products. N4800-20 and N4800-20 SI are available globally in both prepreg and laminate forms.

Eltek Reports 33% Sales Increase in Q3

"The third quarter was another solid one for Eltek, as we managed to demonstrate continued strength in our core military and medical device markets," said Arieh Reichart, President and CEO.

Dow, Rohm and Haas Recognized as Global Innovators

Dow, Rohm and Haas, acquired in 2009, was honored in Thomson Reuters' first-ever ranking for Global Leadership in Innovation Performance.

Atotech Addresses Challenges of Greener Plating

Atotech's Dr. Gerd Linka carries responsibility for sustainable development, risk management and compliance. He gives Editor Pete Starkey a brief insight into the challenges to be overcome in the development of greener plating technologies.

Technic Launches New Gold Process Chemistries

Technic Inc. has announced entering into the final phase of development on enabling technologies designed to reduce the use of gold in electronic packaging and connector applications. The new line of products will be marketed under the name Goldeneye and consists of equipment, electroplating chemistry and ancillary process chemistries.

Camtek Unveils Next-Gen AOI for PCB, IC Substrates

Camtek has launched Phoenix, the next generation of automatic optical inspection (AOI) systems for the PCB and IC substrates industry.

Networking…
an Important Southern California Event






APEX EXPO
IPC

2012 Golf Outing

Sunday February 26, 2012
San Diego, California

Networking on a golf course just doesn't get any better than this! Coronado Municipal Golf Course boasts immaculately manicured greens, wide inviting fairways, with dramatic views of the Coronado Bay Bridge, the famous Hotel Del Coronado, the San Diego Bay and the stunning San Diego skyline.

This golf outing is being co-sponsored by **IPC** and the LA/OC Chapter of the **SMTA.** It will be hosted at the picturesque Coronado Golf Club located just minutes away from the San Diego Convention Center on Coronado Island on Sunday, February 26th. More information can be found on IPC's website.

Sign up today! Click here for registration form.

TOP TEN

Most-Read News Highlights from PCB007 this Month

1 EIPC's John Ling Breaks Down productronica 2011

In 2009, it seemed that our old friend productronica was in a bad way. In 2011, it seems that the patient has recovered somewhat, but major surgery has changed him. His sense of direction is not the same and his personality is less cheerful; he has become quite thrifty and is worried about his future. The visiting hours are the same, but the number of visitors, other than devoted family members, remains small.

2 Embedded Actives Next High-Density Design Technique

Resistors and capacitors have been buried into circuit boards for years, but that doesn't provide enough space savings for some designers. They're beginning to squeeze active components into the substrates.

3 EPTE Newsletter from Japan: PWB Industry Q3 Performance

The numbers are much better for flexible circuit shipments during September. Revenue shrunk by 4.3%, but volume increased 10.5% compared to the same month last year. You don't have to be an economist to figure out that manufacturers have lowered prices to maintain volume.

4 N.A. PCB Shipments Down 9.4% in October

For rigid PCBs and flexible circuits combined, shipments in October decreased 9.6% from October 2010, as orders booked decreased 0.1%. "North American PCB sales are down from the previous month, but this is a normal seasonal pattern," said Sharon Starr, IPC Market Research Director.

5 Canadian Circuits Completes Upgrade, Increases Capacity

Praveen Arya, president and owner, said, "We have all worked very hard the last year and a half making sure that Canadian Circuits will be there, now and long into the future, for our customers. We have not only improved the company, but we have also increased our sales to the point where we are now hiring key people to help us meet the demands of our growing customer base."

6 TTM Wins Stoke's Supplier Excellence Award

"TTM Technologies has helped Stoke create breakthrough solutions through a close and collaborative manufacturing partnership," said Paul Cianci, vice president of operations.

7 Ibiden Releases CSR Report 2011

The company is putting into action the Global IBI-TECHNO 100 Plan, the medium-term management plan that will run through fiscal 2012 with the aim of instilling the dynamism and corporate culture that will enable the company to achieve sustainable growth and stable profits for the next 100 years.

8 IPC China Aims for Solid Expansion

China's growth as an electronics manufacturing powerhouse has been solid, with no signs of slowing down. If IPC China's general manager gets her wish, Chinese manufacturers will see equally solid growth. "I have a five-year plan. By 2016, I'd like China to account for a third of IPC's membership," said Leesha Peng, General Manager of IPC China.

9 Omni PCB Launches PCB Coffee Talk

"This series is intended to be an entertaining, yet informative session to provide basic PCB information to people involved in the procurement of PC boards," said Elizabeth Foradori, President.

10 Schweizer Begins Diversification Strategy

Schweizer Electronic AG, a leading manufacturer of premium PCBs, as well as innovative solutions and services for automotive, solar and industry electronics, is taking the first concrete steps regarding the diversification of their business strategy.

THE SALES CYCLE

The Team Makes All the Difference

by Barry Matties
I-CONNECT007

SUMMARY

With the New Year here, it's time to make our 2012 resolutions. Perhaps the best one you can make for yourself and your business is to become a great organization. Most of us think we are great, but there is always room to improve. The first step is to make sure you have the right team in place. The team makes all the difference.

As a leader, you are continually faced with choices that help ensure your company is on a clear path to success. One key area of focus with regard to choices is making sure you have the right team in the right place within your company. I read a fantastic book many years ago by Jim Collins, titled "Good to Great," which described the importance of, as he put it, having the right people on the bus, and even more importantly, making sure they are in the right seat. When you have achieved that, your company can stay focused on being great. Without it, you will find less time is spent on being great and much more time is spent working through the shortcomings.

You can always tell when the right people are in the right roles. We can see examples of this concept in the world of sports, where winning teams know they are a team and there is one clear objective: To win. Not like Charlie Sheen's "winning," but a true victory based on incredible teamwork, with a team consisting of aligned individual contributions. Why would you have a great first baseman playing right field, a star quarterback on the defense or, even worse, an amateur playing in a professional league? If you want to win, you wouldn't. If you were the manager or head coach of that team you would not have a job very long if you did such things.

So how do we do it in business? I'm sure many of you don't think you have the wrong people and in some cases, you're right. However, when the wrong people are in place, what keeps them there? Is it that you are just satisfied with mediocrity, or less? Do you have so many other fires in your organization you have no time to deal with the people issues? Maybe it is because you don't like confrontation or you don't want to negatively impact another person's life. I can certainly understand that; after all we are humans dealing with human emotions, but as a business leader you still have to deal with it. Or, is it you who are in the wrong seat?

How do you know when you have the right players on your team? One way to tell immediately is by how engaged they are in their activities and how they interact with co-workers. Do you have people asking, "So exactly what does so-and-so do?" If so, you probably have a person who is misplaced. Other signs include attending meetings but not actively participating. In fact, it may come to the point where they feel they don't even need to attend the meetings. These same people will often make mistakes that should not be made. They do not always follow through on action items. They could be poor communicators, not because they don't speak well, but because they are just not connected

"Having the right people on the bus and in the right seat is clearly one of the most important things you have to do for your business."

with the others in the company and therefore do not have much to contribute. They feel like outsiders, but occupy space on the inside. These are just a few of the signs.

The idea is to build a game-winning team, across the board. There are some natural lines of separation within organizations. Just as you have an offense and defense in football, these lines also exist in businesses. Of course one of the classic lines of separation is between sales and manufacturing. The sales team is on the front lines dealing with and learning what is relevant and important to the customers; the guys running the manufacturing processes have little contact with customers. How many times do you see a salesperson in the back running a process to gain an understanding of what his co-workers do every day? It's one thing to walk through a factory and watch others working for 20 minutes, but it's an entirely different perspective to sit for 40 hours a week and load plating tanks or solder boards.

And for the manufacturing guys, how often do they hear directly from the angry customer whose order has been delayed or completely scrapped? Do they even know how difficult it really can be just to make a sale in the first place? At least in competitive sports you can see firsthand the challenges that each of your teammates is faced with. You are there during practices and games and for the wins and losses. You feel it with your team members. When a salesperson loses an order, they feel it, too, and oftentimes they feel it alone.

Having the right people on the bus and in the right seat is clearly one of the most important things you have to do for your business. Don't just settle; really make sure you get it right. If you don't have the right people in the right seat, switch seats. If people can't find the right seat, stop the bus and open the doors. In the end, you are doing them, the other team members, the customers and yourself a favor. It may not always feel like it, but once you have the right people in the right seats, you'll ask yourself why the hell you waited so long.

One thing about business is that you have to learn — and sometimes relearn — how important a solid hiring process is. If you find that you have the wrong people in your company, you may want to re-examine your hiring process, because that is most likely the system that is failing. If you are satisfied that the process is reliable, then rely on it and bring great people onto your bus and enjoy the ride! **PCB**

Barry Matties started in PCB manufacturing in the early 1980s. In 1987, he co-founded *CircuiTree Magazine.* Nearly 13 years later, *CircuiTree* was sold as the leading publication in the industry. In the early 2000s Barry and his former *CircuiTree* partner, Ray Rasmussen, joined forces again and acquired PCB007. Over the years, PCB007 has grown and continues to thrive. In July of last year, Barry and Ray acquired *SMT Magazine.* With his many years of business leadership skills, Barry now produces this column for anyone who has a desire for success. The column relates 25 years of successful business leadership, including marketing and selling strategies that really work. Read a few and decide for yourself.

EVENTS

- IPC Calendar of Events
- SMTA Calendar of Events
- iNEMI Calendar
- PCB007 Online Events

Editor's note: In light of the holidays, our Events Calendar this month will include two full months' of content — January and February. We will resume our usual one-month Events Calendar beginning with the February 2012 issue.

2012 INTERNATIONAL CES
January 10 - 13, 2012
Las Vegas, Nevada, USA

41st INTERNEPCON JAPAN
January 18 - 20, 2012
Tokyo Big Sight, Japan

29th ELECTROTEST JAPAN
January 18 - 20, 2012
Tokyo Big Sight, Japan

13th IC PACKAGING TECHNOLOGY EXPO
January 18 - 20, 2012
Tokyo Big Sight, Japan

PWB EXPO
January 18 - 20, 2012
Tokyo Big Sight, Japan

ELE TRADE
January 18 - 20, 2012
Tokyo Big Sight, Japan

CAR-ELE JAPAN
January 18 - 20, 2012
Tokyo Big Sight, Japan

MATERIAL JAPAN
January 18 - 20, 2012
Tokyo Big Sight, Japan

LIGHTING JAPAN
January 18 - 20, 2012
Tokyo Big Sight, Japan

Soldier Technology 2012
January 23 - 26, 2012
Marriot Gateway, Arlington, Virginia

11th annual Flexible Electronics and Displays Conference and Exhibition
February 6 - 9, 2012
Phoenix, Arizona

Dallas Expo & Tech Forum
February 7, 2012
Richardson (Dallas), Texas

Houston Expo & Tech Forum
February 9, 2012
Stafford, Texas

Southern Manufacturing 2012
February 15 - 16, 2012
FIVE, Farnborough, Hants

Electronics For You Expo 2012
February 16 - 18, 2012
Pragati Maidan, New Delhi, India

IPC APEX EXPO
February 28 - March 1, 2012
San Diego Convention Center, San Diego
California

PV SYSTEM EXPO 2012
February 29 - March 2, 2012
Tokyo Big Sight, Japan

MASTHEAD

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

PUBLISHER: **RAY RASMUSSEN**
(916) 294-7147; ray@iconnect007.com

SALES: **ANGELA ALEXANDER**
(408) 915-2198; angela@iconnect007.com

EDITORIAL:
GROUP EDITORIAL DIRECTOR: **RAY RASMUSSEN**
(916) 294-7147; ray@iconnect007.com

MANAGING EDITOR: **LISA LUCKE**
(209) 304-4011; lisa@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: **MIKE RADOGNA, BRYSON MATTIES, SHELLY STEIN**
INNOVATIVE TECHNOLOGY: **BRYSON MATTIES**
COVER: **MIKE RADOGNA**

PCB007 Presents

the pcb magazine

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

January 2012, Volume 2, Number 1 • The PCB Magazine© is published monthly, by BR Publishing, Inc.

AD INDEX

It's a Material World — and the February Issue of The PCB Magazine Proves It!

Printed circuit materials have to satisfy many physical, mechanical, thermal and electrical requirements. From laminates options and the bewildering catalogue of material properties and characteristics, how are the right materials chosen? What are the latest advances in technology and how can they help the designer achieve required performance levels? Our February issue answers these questions and then some in feature articles from Mentor Graphics, Saturn Electronics, DuPont and more!

Don't miss our columns department, fresh video from productronica 2011 and more! Not a subscriber? Click here to get The PCB Magazine delivered to your inbox every month.