I-Connect007 Presents

SPECIAL ANNOUNCEMENT

I-Connect007 GOOD FOR THE INDUSTRY AWARD

The very first recipient of this award is revealed on page 14.

AUGUST 2016

the pcb magazine

voices of the industry

Click here to search for your listing and upgrade NOW!

THE PCB List works ...and it can work for you!

"When I joined Cirexx last year I shared with my colleagues the positive experiences I've had with The PCB List.

We listed Cirexx with a detailed Showcase on The PCB List and it has resulted in quotes and orders!

The PCB List is not only brand and image must for all PCB manufacturers, but also another set of boots on the ground directly generating business."

- **Al Wasserzug**
  Sr. Business Development Manager
  at **Cirexx International**

Over 3000 members are already registered and using *THE* PCB List.

thepcblist

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

www.thepcblist.com

The PCB industry's go-to resource for transitioning to ISO 9001-2015

ISO 9001:2015

THE RIGHT APPROACH CONSULTING LLC

Quality systems.
Process improvements.
Business coaching.

Do you want to increase your business ...and your profits?

I can help.

FREE DOWNLOAD: ISO 9001:2015 READINESS CHECKLIST

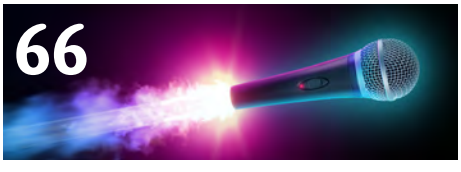
CLICK TO DOWNLOAD

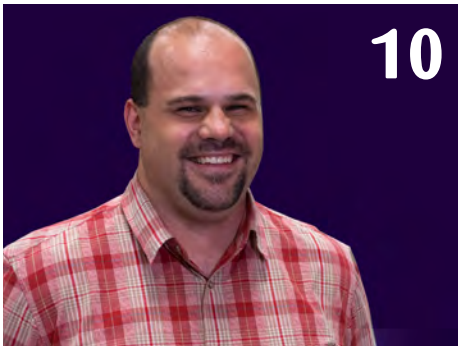
TheRightApproachConsulting.com

I Connect007 Presents
the pcb magazine

August 2016

Featured Content





Voices of the Industry

Welcome to this month's topic—Voices of the Industry—otherwise known as YOU! We've surveyed readers, including engineers, fabricators, technologists, CEOs, and others, just about everyone we could reach, to find out exactly what's on their mind when it comes to today's electronics manufacturing industry.

Six reasons why outsourcing CAM Engineering can be a great idea for you:

- On-Demand Capacity
- Improved Automation
- Fast Turnaround
- Reduced Costs
- Improve Quality
- Build Redundancy

...and here is the delivered value:

	In-House	With Entelechy
Availability	48 weeks	52 weeks, 24/7
Overhead Cost	>30%	0%
Scalability	Limited	Unlimited
Redundancy	Variable	100% Bulletproof
Operational Efficiency	Status Quo	Continual Improvement
Total Cost	Fixed	Significantly lower

> "In our company, the CAM department was the bottleneck. As a result of our working with Entelechy, we can now accept orders that we had to refuse in the past."
>
> *-Thomas Hofmann, CEO/Owner*

HOFMANN LEITERPLATTEN

Click here to check out our risk-free Quick Start program»

We deliver the highest quality PCB CAM and Mechanical CAD engineering services to customers around the world. We can help you beat your competition by teaming up with us.

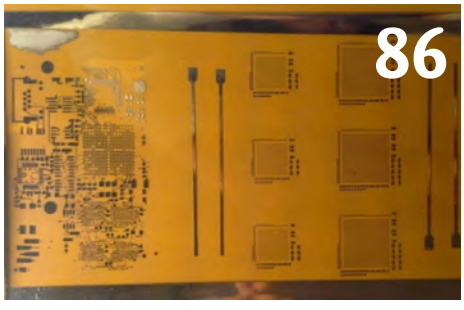
ENTELECHY GLOBAL

www.entelechyglobal.com

I Connect007 Presents

the pcb magazine

August 2016
Vol. 6, No. 8
thepcbmagazine.com



More Content

185HR

High-Tg, Low-CTE Materials with High Thermal Reliability

FEATURES

- **High Thermal Performance**
  - Tg: 180°C (DSC) (Base Laminate)
  - Td: 340°C (TGA @ 5% wt loss)
  - Low CTE for reliability
- **T260: 60 minutes**
- **T288: >15 minutes**
- **Lead-free Compatible and RoHS Compliant**
- **UV Blocking and AOI Fluorescence**
  - High throughput and accuracy during PCB fabrication and assembly
- **Superior Processing**
  - Closest to conventional FR-4 processing
- **Industry Approvals**
  - IPC-4101D-WAM1 /98 /99/ 101 /126
  - UL – File Number E41625
  - Qualified to UL's MCIL Program
- **Core Material Standard Availability**
  - Thickness: 0.002″ (0.05 mm) to 0.060″/0.062″ (1.5 mm)
  - Available in full size sheet or panel form
- **Prepreg Standard Availability**
  - Roll or panel form
  - Tooling of prepreg panels available
- **Copper Foil Type Availability**
  - Standard HTE Grade 3
  - RTF (Reverse Treat Foil)
- **Copper Weights**
  - ½, 1 and 2 oz (18, 35 and 70 µm) available
  - Heavier copper available upon request
  - Thinner copper foil available upon request
- **Glass Fabric Availability**
  - Standard E-glass
  - Square weave glass fabric available

KEY APPLICATIONS

- **Heavy copper, 6 & 12 oz automotive applications**
- **Military**
- **Medical**
- **Industrial**

ORDERING INFORMATION:

Contact your local sales representative or visit **www.isola-group.com** for further information.

Isola Group
3100 West Ray Road
Suite 301
Chandler, AZ 85226
Phone: 480-893-6527
info@isola-group.com

Isola Asia Pacific (Hong Kong) Ltd.
Unit 3512 - 3522, 35/F
No. 1 Hung To Road, Kwun Tong,
Kowloon, Hong Kong
Phone: 852-2418-1318
info.hkg@isola-group.com

Isola GmbH
Isola Strasse 2
D-52348 Düren, Germany
Phone: 49-2421-8080
info-dur@isola-group.com

Speak Up!

by Patty Goldman
I-CONNECT007

If you have paid any attention at all to our newsletters and other publications, you should have been aware of this month's topic long before opening this magazine. We tried to reach… everybody, so we could get their thoughts on the industry for this issue.

We got quite a few responses, but I wish there had been many more. Apparently the PCB industry has a huge silent majority. So, what is it? Why is everyone so quiet? What about *you?* No opinion, no thoughts, no ideas to share—you just keep your head down and do your job? Do you wish you were in a different industry? Trust me, you are sucked in and will always have PCBs in your blood, so you might as well just start making the most of it.

When I was a newbie chairman at IPC I sat in my first CCC (committee chairman's council) meeting and was totally awed by a huge table lined with other chairmen (almost all men, of course). As usual, the meeting started with everyone introducing themselves around the room. It took all my courage to muster up my name, company and subcommittee I was chairing because I feared stumbling over the words. Many people nowadays find that awfully hard to believe of me; yes, I can be pretty outspoken at some of those meetings now! What changed? Well, I developed some confidence in myself but what really happened was that I became involved in our industry. I cared about the standards and specs that were being developed, as well as the companies and people that would be using them, and I cared (and still do) very much about where the PCB industry was headed. And if any of you had been at one of those meetings and seen the solder joint guys arguing, or the vehement discussions on adopting metric terms in documents, you would know that I was and am not alone!

So I'm just saying, don't sit back and wait out your time. Put a little thought in it, a little emotion, and you will be surprised with the results.

And now on to this month's issue, which isn't strictly "Voices;" we do have some great technical content for you too.

We start off the magazine with a newly created award, the I-Connect007 *Good for the Industry Award,* introduced by our publisher, Barry Matties. Once in a while someone or something in the industry impresses you beyond all measure

and stirs you to action. That has happened here at I-Connect007 and we felt we had to let the world know about it. Alex Stepinski of Whelen Engineering has made such a significant contribution that we cannot ignore it.

We follow this with the voices of your colleagues, which I have found most interesting to read and I am confident you will too. Maybe you would like to reply to one or more of them or maybe you have some thoughts of your own that you want to share. Let me know—my email is at the end of this column—and keep in mind that we never publish without permission (you may remain anonymous if you feel a need to be).

As you must know by now, we interview a lot of people in the industry…a lot. We ask them about their companies, the industry, their plans for the future and whatever else they want to talk about. We've included a few in this issue, and the first "voice" is that of Steve Driver, managing director at Spirit Circuits. Steve discusses the new plant being built in Romania and the influence Whelan's Alex Stepinski has had on the project.

And no doubt you are well aware that the seventh edition of the *Printed Circuits Handbook* came out this year and it is certainly the most read and most venerated book on printed circuits and electronics in our industry. So we asked the many authors to tell us their thoughts and we gathered them into a single section and put it in all three magazines. It starts off with a wonderful history of the handbook written by Clyde Coombs and continues with a dozen more authors' "voices." Get the inside scoop from the man who started it 50 years ago!

Our next interview is with Eltek USA's Jim Barry, who shares his thoughts on devices that are getting tinier, the industry driving it (maybe not what you think) and the cross-overs between the end markets. He also discusses the challenges to the PCB business, where he thinks things are headed, and throws in a little of his own philosophy as well.

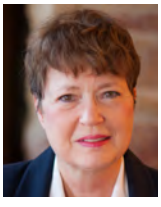
In our third interview, Stan Hetzel of the European Space Agency discusses the paper he presented at the EIPC Summer Conference on qualifying one's PCBs for the space industry. He mentioned that they were raising the bar in terms of levels of testing and inspection that are required. It's nice to hear from some European voices.

As I mentioned earlier, we have interviewed a lot of people over the past year, both video and audio. So this seemed a great time to pull them together into an index, if you will. And it's quite a long index, organized alphabetically by the interviewees' last names. Just an FYI, you can always search our website for an interview by using the search box in the upper right and putting your search term in quotes.

Lest you techies feel put out by all the "touchy-feely" voice stuff, our regular columnist, RBP Chemical's Mike Carano, continues his discussion on PTH reliability and Keith Sellers of NTS talks about reliability testing in the new product development cycle, covering both mechanical and environmental testing. And we have an interesting article by David Ciufo of Intrinsiq that may be of especial interest to the flex folks, a discussion on the adhesion of very thin film copper for ultra-fine line circuits.

Our final entry for this month is Barry Lee Cohen's *Launch Letters* column on your brand—and he is not just speaking to marketing and management folks. Whether it be your company brand or your personal brand, Cohen calls upon you to fully embrace it and believe in it. As always I find his columns to be inspiring and I'm thinking you will, too.

Next month, we turn to the military/aerospace electronics industry, examining and exploring the challenges, technology and market trends. How about you? Does your company supply the military and/or aerospace electronics industry? This is your chance to tell our readers what it is like, what is different from other industries and what you see for the future. Drop me a note by clicking here. **PCB**



Patricia Goldman is a 30+ year veteran of the PCB industry, with experience in a variety of areas, including R&D of imaging technologies, wet process engineering, and sales and marketing of PWB chemistry. Active with IPC since 1981, Goldman has chaired numerous committees and served as TAEC chairman, and is also the co-author of numerous technical papers. To contact Goldman, click here.

Alex Stepinski, Our First Recipient of the Good for the Industry Award

I-Connect007 is proud to introduce the *I-Connect007 Good for the Industry* award. This new and prestigious award is bestowed upon individuals and companies that are good for the industry.

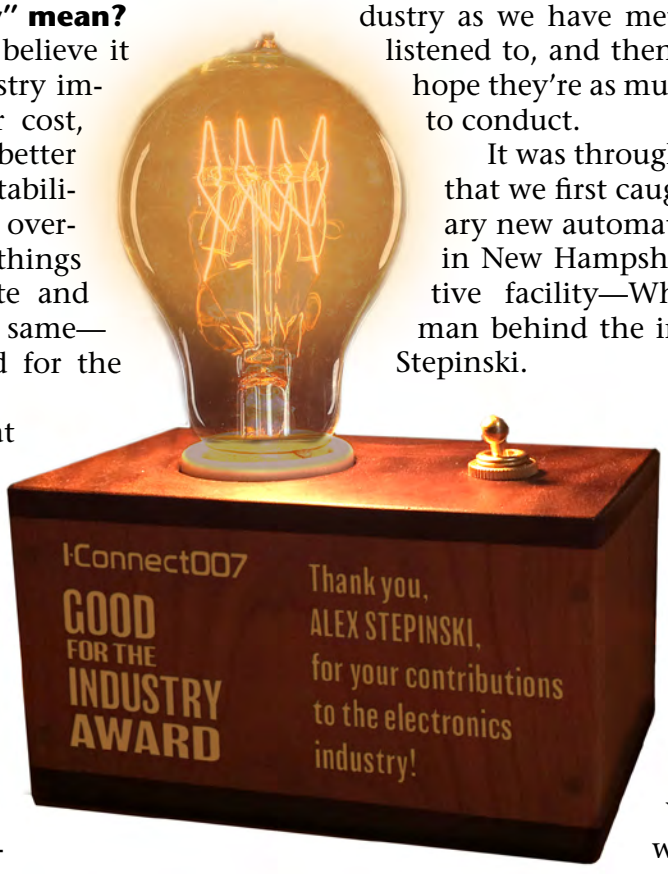
What exactly does "good for the industry" mean?

At I-Connect007 we believe it means helping the industry improve cycle time, lower cost, increase yields, build better products, increase profitability, reduce waste, become overall more efficient, do things differently, and motivate and inspire others to do the same—all things that are good for the industry.

This is what we at I-Connect007 do every day…we believe in this, and we know others do, too. We want to recognize those people that are good for the industry and share their stories.

Our goal is to consistently bring our readers fresh new content related to those characteristics listed above. Throughout the past year, the I-Connect007 team has done this by conducting interviews all over the world with hundreds of people from the industry, from congressmen to electrical engineering teenage prodigies to some of the top PCB executives. These interviews have given us a unique perspective into the industry as we have met all sorts of people and listened to, and then shared their stories. We hope they're as much fun to read as they are to conduct.

It was through one of these interviews that we first caught wind of a revolutionary new automated PCB fabrication shop in New Hampshire, being built as a captive facility—Whelen Engineering. The man behind the innovative design is Alex Stepinski.



Why is Alex Stepinski good for the industry?

Alex Stepinski is good for the industry because of his first-of-a-kind, proof-of-concept creation—a fully automated PCB factory with zero waste water discharge—and his and Whelen Engineering's willingness to open the

doors and share their new captive manufacturing facility for others to learn from.

When designing and building the factory, Alex did not rely on the status quo. Instead, with a clear mission and a new way of thinking, he sought out and adopted leading-edge automated technology that would substantially cut manufacturing cycle time and the need for costly labor. Alex's innovative factory design has fewer than 20 people operating it on all shifts and has caught the attention of and gained recognition from some of the industry's biggest names. When I-Connect007 sat down with Gene Weiner at the most recent HKPCA show, he had this to say of the Whelen factory:

"The idea of the containment of the operating equipment, the maintenance, the effluent, and the central control overlooking the whole thing that forms a circle, from which you can side-step to do ENIG or some other special process or finish, is sheer genius in the way it was designed and built. Several pieces of equipment were created especially for that line as well as the waste treatment, such as digitizing the use of a plasma system for desmearing that uses oxygen instead of an organic. There are a lot of innovative things at Whelen."

The Whelen factory represents the first fully automated PCB facility to open up on United States soil in decades, which is fitting because Whelen is a very pro-American company that manufactures sirens and indicator lights. Rather than continuing to spend $7 million a year on product in China, for a little under $12 million they were able to set up Alex's manufacturing line and start building their boards in-house. Taking chances on unproven technology and automation paid off in a big way and allowed Alex to significantly cut costs and reduce ROI. The Whelen factory offers American manufacturers evidence that adopting advanced technologies can pay off with even more opportunities in the future.

Of course, it's risky to take chances on expensive new technology when you're struggling to get by, which makes Whelen's openness even more admirable. They've offered to let anyone come in and look under the hood to see what they're doing. They want to share this technology and their unique approach with American companies.

The benefits and scope of Alex's design go beyond just North America, however. During our short visit at Whelen, a gentleman from the UK was examining the factory with hopes of setting up a similar facility in Europe. And these techniques and solutions aren't just beneficial to the West; it's expected many Asian manufacturers will be interested in Whelen's waste treatment system as environmental regulations continue

to get ramped up in China. Veteran PCB expert Happy Holden summed it up:

"Every couple of decades a "perfect storm" of opportunity presents itself to our industry. What Alex Stepinski has done at Whelen is that perfect storm! I like to call it, LEAN PLUS GREEN. Putting together a number of new innovations like inkjet printing, horizontal plating and conductive polymer metalization, along with his own innovations in water and chemical regeneration/recycling, he has cut the labor, environmental and chemical costs out of the PCB equation. This allows PCB merchants and OEMs alike to relook at the concept of re-shoring and captive production here in the USA, since waste emissions and labor costs are no longer relevant.

I congratulate Alex and his team for earning this first I-Connect007 Good for the Industry award."

Alex designed a factory that accomplished a number of firsts, and it looks like it could be adding more soon related to plating on plastics. It is for these ongoing accomplishments and continued innovation, as well as the willingness to share this information to further advance the industry as a whole, that we present Alex Stepinski with our *Good for the Industry* award. We've spent some time with Alex, and we've found that he is a genuinely nice guy.

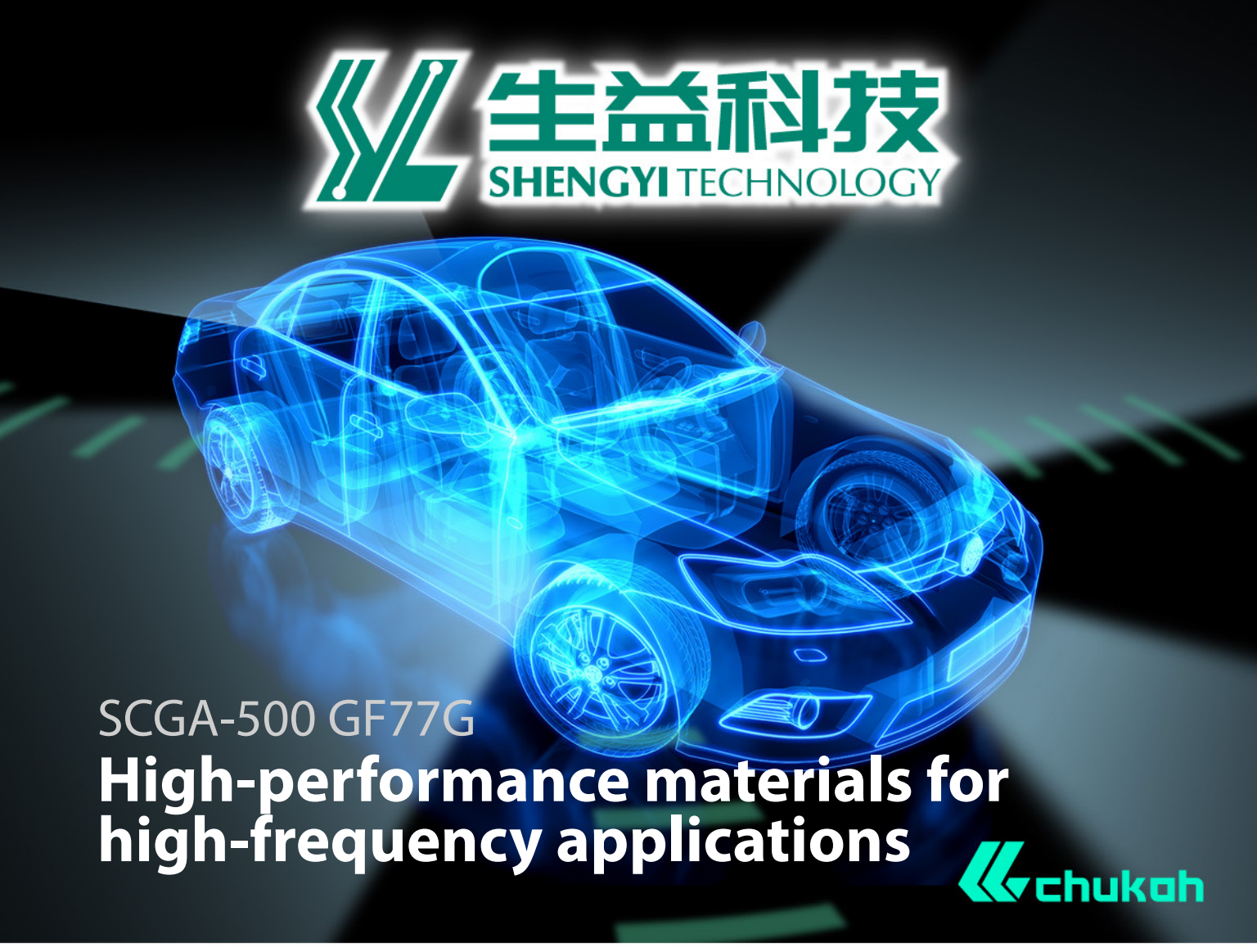
If you get a chance, tour Whelen Engineering's new PCB manufacturing facility in New Hampshire. Once you meet him, you will agree that Alex Stepinski is good for the industry.

To read the full story on Whelen Engineering and Alex Stepinski, click here.

The I-Connect007 team is on a constant search for people or companies that can be considered good for the industry. If you know of someone like Alex who is *good for the industry,* let us know why, and perhaps one day it will be them being recognized in one of our magazines. **PCB**

Drone footage capturing Whelan Engineering's fully automated PCB factory.

生益科技
SHENGYI TECHNOLOGY



SCGA-500 GF77G

High-performance materials for high-frequency applications

chukoh

Key product features:

- Consistent permittivity in high frequency range
- Low Df at high frequency
- Steady properties in high frequency range
- Very low dielectric loss tangent, high performance in the high frequency band

Learn more about SCGA-500 GF77G

Designed to work with our next-generation FR-4 materials

Autolad0

Mid Tg Applications

Autolad3

High Tg Applications

Photo Chemical Systems

www.photochemicalsystems.com

PARAMOUNT

www.paramount-usa.com

voices of the industry

Editor's Note: In this issue of our magazine, we decided to focus on you, our readers, and let you give a voice to your thoughts on the industry, your company or job, and/or a favorite tip or trick to share with others. As we always say, read and learn… and enjoy!

Dave Becker
VP Sales and Marketing
ALL FLEX FLEXIBLE CIRCUITS, USA

My favorite tip or trick to share is...

…a handful of things I have learned during my career:

• A quick response followed by a course correction is often a better path than waiting for all the information. Sometimes the fastest distance between two points is not a straight line.

• There are no risk-free decisions, including making no decision.

• Doing the basics day after day, the blocking and tackling, provides better results than most management fads.

• Mistakes will occur. The best relationships are developed from reactions after something goes awry.

• Culture can be crushed if management actions don't map with the message.

Barry Cohen
President & Managing Director
LAUNCH COMMUNICATIONS, USA

My favorite tip or trick to share is...

…programs, not projects.

Launching a new service should never be confined to one tactic. Introducing a capability such as a new product, corporate initiative or participation at an upcoming technical conference can and should be deployed using a mix

A new set of rules for PCB production

Less process steps. CAM to etch in minutes instead of hours. Maximum yield. Minimal environmental impact.These are just a few of the PCB production rules that Lunaris has redefined.

We've put together a complete set of all the new rules made possible by Lunaris ...get yours today!

Download all the new rules:
The Official Lunaris Rulebook

CLICK HERE TO DOWNLOAD

mutracx
THE BETTER WAY

www.mutracx.com

of both digital and traditional media to increase frequencies and enable different exposures. There does not need to be a "Sophie's Choice" made regarding what tactics to use, regardless of budget or nature of the offering—whether revolutionary or evolutionary. An impactful and powerful market launch looks beyond the obvious news release and employs an integrated program relevant to the targeted customers and markets, where it will create engaging and differentiated value. The sales team is critical to providing a voice for the customer and should be instrumental in communicating the differentiated value that can be achieved.

Tara Dunn
President
OMNI PCB, USA

If they would only listen...

…to each other! The advances in electronics over the past 10 years are truly inspiring. I often find myself using the phrase, "We are really only limited by our imagination," which is true, but we also have to acknowledge the fact that it takes all aspects of the electronics supply chain working together to make these advances. Raw materials suppliers, designers, fabricators, and contract manufacturing ALL play an equally important role in supporting the OEM. If we could just step back and LISTEN to each other, and the OEM, new product development would be so much more productive and exciting.

Product development rarely follows a linear path; there are always obstacles, lessons learned and restarts. All too often, when troubleshooting why something didn't progress as planned, fingers start pointing. PCB fabricators blame raw materials, contract manufacturing blames fabricators and everyone blames the design. Just imagine the progress, and the fun, of stepping outside that vicious cycle, bringing everyone's expertise to the table, and having honest discussions and solving problems. Sounds like fun to me!

Michael Gasch
Semi-retired consultant
DATA4PCB, GERMANY

I was based in Asia, came back to Europe, worked with a large PCB manufacturer, and now (as a hobby?) I work as consultant to the PCB industry as a "one man show", establish statistics, and speak at conferences.

Whether in the USA or in Europe, PCB manufacturers (or anyone in the electronics industry) seem to be complaining about Asia—in particular China. I have the impression that "if we do it, it is free market; if they do it, it is dumping." But it isn't that easy.

The political decision to open China came in the 1970s and coincided with unprofitable companies in the West trying to improve their bottom lines and having the vision to enter an enormous market. However, it was overlooked that China had its own agenda and the way to allow foreigners to produce against a majority shareholding by Chinese nationals was the first step. Other steps followed (new technologies, transfer of know-how, establishing facilities in inner China), and always granting favours, but with a subtle hook to catch companies.

China-as-copy-cat was known, but ignored. The side effects are felt today. Successful companies are invited, their products and sales strategies are studied and parallel to this, their own products are developed. As soon as these products are ready for the market the same government makes it more and more difficult for the foreigner to work in the country. So far, the global market grew, and this strategy was discarded as pinpricks, but with declining economy it is a question of survival.

And who could condemn a nation that thinks this way? In America there is a similar sentiment—one has just to listen to the slogans during elections. We have to keep in mind that for more than 2000 years (!) China was the largest economy in the world (Andus Maddison University of Groningen[1]). Only after the 1850s did other regions became larger (opium war, industrialization, crude oil).

What is the cure? Establish a reliable base of suppliers (at least a second source in your own country). The cheap price could be the most expensive error when looking at total cost. A good relationship to a good, close-by supplier is more important because technical advice and help in design matters is easily available. This could help to save a lot of money—more than the cheap price. If you only use Asian suppliers, you throw your design over the wall and you get what you designed. A local supplier will give a helping hand in miniaturization and a more reliable and longer lasting product. You will have fewer problems because of quality issues. (One of the largest problems in China is fluctuation. Labour turnover in the best case is 6–8%—per month! In the worst case it could happen to be 30% per month! Not very attractive to maintain a high quality level—or is it?)

We should use more common sense—whether in the USA or in Europe, and we should have a better view on cost, rather than solely on price.

Connie Herring
Sales
CARTEL ELECTRONICS, USA

What I really like about the industry is...

...that when it is slow, it gives greater opportunity to sales people who work hard. Everyone is a "good salesperson" when money is easy to grab. But in a down market, the "good sales people" are home complaining...

Poor attitude = Poor actions = Poor results = Poor lifestyle.

A slow market gives you time to develop relationships and learn the ever-changing personality of the market and grab the lost opportunities of the "good salespeople" who spend their time complaining instead of servicing and selling.

Emma Hudson
PCB Industry Lead
EMEA+LA, UL, UK

If they would just do this...

If the laminate and coating manufacturers would just start UL recognizing their products using solder limits representative of real soldering processes, the PCB manufacturers would be much more likely to do the same (re: reduced and no-test programmes for adding materials to PCBs need solder limits to be at least the same or more severe) and then the SAFETY testing done on the PCBs would be much more realistic of the real life situation, and in turn we in the industry would be doing our bit to make the world a safer place.

Most companies still use solder limits that equate to the old solder float test, but we know that assemblers are typically using multiple SMT reflow operations when soldering the board; everyone accepts that this influences the reliability of the PCB but very few seem to think about the influence on safety.

What I really like about the industry is...

This industry is full of wonderful people who are always willing to help you and do what they can to support you. I am very pleased to be a part of it and hopefully will be for many years to come.

James Latham
Chief Marketing Officer
ESI, USA

What I really like about the industry/ my company is...

What I really like is the dynamic nature of the industry. PCB processors, contract manu-

facturers, and the brands driving the innovation in personal technology are all part of an incredibly fast-moving industrial supply chain that stretches from San Francisco to Singapore to Shanghai. That dynamism brings constant challenges for us to up our game in providing evolving capabilities to the supply chain to do better, faster—at an affordable price. In the end, using that long tenure of 70 years of technology expertise, innovation, and insight, we provide solutions that empower our customers to make products that transform lives in fundamental and exciting ways—every day.

Andy MacGregor
Sales Manager
P.W. CIRCUITS LTD, UK

If I were in charge...

…I would strictly control the either implied or stated status of being a "British manufacturer." Buying boards from offshore, packaging and labeling them with a "Made in Britain" sticker does not wash with me.

The main cost element of the finished product is the populated PCB assembly and not the housing/labeling.

Likewise, importing Chinese boards when your UK manufacturing facility has been mothballed does not justify marketing your company as British.

Other thoughts...

One other issue, somewhat linked to my previous statement, is our concern at the number of so-called PCB manufacturers who do nothing of the sort—relying 100% on offshore PCB production. They hide behind their registered office address.

As one of the last remaining PCB manufacturers, we of course feel the pain of cheap Chinese imports but resent the falsehoods prevailing in the industry. As a privately owned family company, we have been employing members of the local community since 1968, paying rates and UK taxes, but feel our hands are being tied.

Companies don't even seem to bother any more as to where their products are sourced from and will not hesitate to ping off an enquiry to Shenzhen.

We are also deeply concerned about the exploitation of the Chinese labour force, which seems to be ignored by so much of the British electronics industry. The transparency of the offshore market is almost nonexistent, with even Apple manufacturing companies conspiring to hide the truth of the desperate working conditions.

Ken Moffat
Business Development Manager
AMERICAN STANDARD CIRCUITS, USA

What I really like about the industry/ my company is...

…everything is constantly changing. People are moving from one company to the next, larger corporations buy up or spin-off smaller companies and then there are new start-ups that seem to pop up out of nowhere. Change is a good thing and in order to be successful one needs to stay on top of the changes in one's territory or suffer the consequences.

The constant change is definitely challenging—not to mention tedious at times—but nonetheless it is something that only seems to happen quicker as I get older.

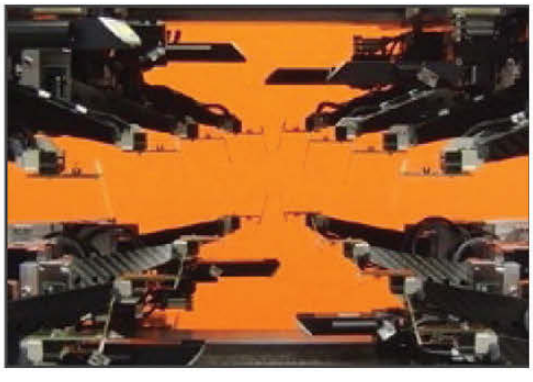
My favorite tip or trick to share is...

…trying different approaches to contact prospects. I do not rely solely on any one method to prospect new business. Social media is a great tool but it's not the only one. Don't forget to use the telephone and try to exhibit or attend various trade shows.

Enjoy your profession or change it and try something different.

Unrivaled Speed with Full Automation

Introducing the atg A8-16a with 16 test probes at an unrivaled test speed of up to 250 measurements per second and full “lights out” Automation.



Test area:	610 mm x 620 mm (24.0” x 24.4”)
Number of test heads:	16 (8 top + 8 bottom side)
Smallest pad / pitch :	35 µm (1.4 mil) / 80 µm (3.2mil)
Test voltage:	Up to 1000V
Max. measurements / minute:	Up to 15000 measurements /min. *
Loading capacity:	390 mm , max. 80 kg weight
Max board weight:	20 lbs
Marking option:	Barcode label

Get more info

atg Luther & Maelzer GmbH
Zum Schlag 3 • 97877 Wertheim • Germany
Phone +49-9342-291-0 • Fax +49-9342-395 10
sales@atg-lm.com • www.atg-lm.com

atg®
LUTHER & MAELZER®
an Xcerra company

Dale Paulin
Marketing Manager
ESI, USA

What I really like about the industry is...

…the excitement. The industry is never static, it is always moving, and it presents a new challenge every day. From the initial spark of insight, to the continual drive for innovation and the creative problem solving that gets applied at every step in the value chain, this industry keeps moving forward and delivering breakthrough products to market in shorter time frames. It's exciting to see customers meet new production challenges head on and continue to deliver products that add value to our lives.

Kirsten Smit-Westenberg
Executive Director
EIPC, EUROPE

What I really like about the industry/ my company is...

…working in an international environment on a daily basis. We from the European Institute of Printed Circuits try to create networking platforms all over Europe. All different countries have different values and norms, but we try to work together with all of them.

My favorite tip or trick to share is...

…to finish what you start; if you don't, your success rate will always be zero.

Pete Starkey
Technical Editor
I-CONNECT007, UK

Reflections of a Superannuated PCB Technologist

At the age of 70, several years having passed since I had any significant hands-on involvement in PCB manufacturing, I can only reflect upon my time in an industry that gave me a good living, made me a lot of friends and allowed me the scope to make a few innovations. The battle scars fade with time, but memories of happy days remain.

I grew up in an age when the industry was still feeling its way—PCB manufacture was migrating from captive OEM shops to the free market and, besides a handful of large and professionally organised factories, hundreds of small entrepreneurs saw circuit boards as their route to a fortune. Very few of them really knew what they were doing, and they relied heavily on the technical support of their suppliers, who priced their products accordingly! But those PCB manufacturers could afford to pay—they charged what the market would stand, and there was so much potential profit to be made that a job could be scrapped three times, delivered weeks late, and still not at a loss. At the most basic level, manufacture was not capital-intensive—a functional set-up could cost as little as £20,000 if second-hand kit was used. And, with a bit of ingenuity, quite complex product could be realised—even if yields were not spectacular, as already mentioned. We had great fun pushing the technology along and achieved a lot as the industry grew up around us. The strongest survived; more than a few fell by the wayside.

How times have changed! Huge investment is required to enter the game (would it be better spent elsewhere, brewing craft beer or baking artisan bread, for example?), the technology is challenging, materials and consum-

ables are priced as commodities, margins are narrow and high yields are mandatory. And automation is rapidly soaking up the routine and repetitive job functions in the fabrication process. I admire, but do not envy, the technical managers and process engineers of today's industry and the burdens of responsibility they have to carry.

It's nice to be on the outside looking in and having privileged access to conferences and exhibitions, with the opportunity to observe and understand how the technology continues to advance, and to have the experience to be able to write reviews that deliver the essence of an event to those not able to attend, while comfortable in the knowledge that the fortunes of a company and the livelihoods and welfare of its employees do not hinge upon my day-to-day technical and managerial decisions. Instead, I can devote my attentions to matters no more serious than choosing which brand of cat food to purchase.

John Talbot
President
TRAMONTO CIRCUITS, USA

What I really like about the industry/ my company is...

...that customers are continually challenging us, in the industry and in my specific company! They come to us with tougher design/ assembly issues every day. We love that type of customer. Their job isn't easy and they are grateful to work with a supplier who doesn't mind them making our job tough as well.

I've been in this industry for nearly four decades and it's currently the hardest it has ever been to compete in. But it is also the most gratifying time to succeed in.

My favorite tip or trick to share is...

...to listen. To your customers. To your employees. To your vendors. To the FedEx guy. Listen to everybody that is willing to talk with you about your company.

Irene Tso
Technical Service Engineer
TAIWAN UNION TECHNOLOGY CORPORATION, TAIWAN

What I really like about my company is...

...its resilience and flexibility to be able to cater to the ever-evolving needs of our PCB market. Being a PCB laminate material supplier in Asia (Taiwan), I have seen our product mix evolve from the typical cookie-cutter FR-4 standard to current high-performance materials with low Dk/Df properties, improved thermal properties, and reliability in a span of 10 years. With many material suppliers out there who are almost trusted institutions, to be able to be almost at par with these international brand names in terms of PCB fabs and OEMs serviced is indeed a giant leap. It feels like a David and Goliath thing—big fish and small fish. The company does not skimp on training nor hiring the right people to enrich its technical aspects. Also, improvements in laboratory equipment and production equipment is always a never-ending task. With a customer-oriented attitude and the passion to move forward, I think our company has a bright future ahead.

Voices from the China PCB World

A PCB equipment manufacturing marketing manager:

In Dr. Hayao Nakahara's 2015 NTI-100 list of the world's largest PCB fabricators, domestic Chinese companies' numbers—already 33% of the list—rose from 29 to 34. The total operating income of these domestic Chinese companies has grown by 7.29%, making them a global leader. But none of the mainland companies has a turnover of more than US$1 billon, and that revenue only accounts for 16.2% of the NTI-100. The domestic Chinese PCB fabricators accounted for more than 70% of the sin-

gle/double-sided and generic multilayer PCBs, which are not less advanced, low value-added products.

But domestic enterprises reduced their production costs by optimizing production processes, such as large array production, LDI exposure machine, automated production, to enhance competitiveness and gained rapid market growth. As a PCB production equipment manufacturer, we will strive to provide equipment and production solutions that matches or exceeds imported products and enhance the strong competitive position of domestic PCB enterprises in the low-end PCB products, while gradually improving technical capacity to expand into high value-added PCB board production field.

A PCB equipment marketing and research engineer:

All rapid growing large-scale domestic PCB fabricators are concentrated in the low value-added, low technical requirement area such as single/double-sided, generic multilayer PCB production, by adopting 25" x 28" and larger area production, two-pin mechanical drilling machines, LDI to replace conventional exposure equipment, and other means to improve production efficiency and reduce costs. But due to the limitation and easy replicability of these techniques, the headroom for fabricators' technical capabilities is seriously suppressed, causing difficulty for companies entering into the high-end field. And if this continues, it will result in even more tragic competition in the low-end PCB manufacturing field, a rapid decline in corporate profits, and the domestic PCB industry could suffer irreversible damage.

CPCA South China Office staffer:

What I like the most about the PCB industry is working with the veterans of this industry, like Wang Longji, Lin Jindu, Wang Hengyi, and Gong Yonglin. They look dozens of years younger than they actually are. Overall, the people who work in the PCB industry are not only making money, but they are also very healthy and happy. I wish more new people would join China's PCB industry, to make a contribution for the future of the market.

PCB fabrication process engineer:

As a process engineer in PCB fab, I miss the good old days of stepping into a trade show, when every piece of technology and equipment drew my attention. Back then, orders and profits were great. Our factory is now doing more HDI and profitable products. **PCB**

References

1. GGDC.net

Delivering the Next Generation of Connected Clothing

Broadsword® Spine® has been designed specifically as an alternative to the heavy portable data and power supplies currently carried by military and emergency service personnel. BAE Systems has agreed an exclusive new partnership with world-leading e-textiles developer, Intelligent Textiles Limited (ITL), to deliver the ground-breaking wearable product in high volumes at low cost.

Broadsword Spine is an e-textile based layer that when added to a user's clothing creates an invisible electronic network and power supply, by using conductive fabrics instead of wires and cables. With the innovative network, users can plug vital electronic devices straight into their vest, jacket or belt and have them instantly hooked into power and data via USB—all delivering an estimated 40% weight saving per user versus alternative solutions.

Panasonic

Circuit board materials for further High-speed ICT equipment

MEGTRON Series

We create the value with our customer.

Contact us

More information

http://industrial.panasonic.com/ww/electronic-materials

Partnering to go beyond.

Electronic Materials
Panasonic Corporation

Steve Driver

Spirit Circuits:

Building a New Factory in Romania...

with an Alex Stepinski Influence

by Barry Matties

Recently I sat down with Managing Director Steve Driver of Spirit Circuits. We discussed the company's future, and the interesting decision to build a new facility in Craiova, Romania, and how Whelen's Alex Stepinski influenced the design.

***Barry Matties:** Steve, we're sitting at your facility in what appears to be a sunny day here in England.*

Steve Driver: Absolutely. Every day is sunny in England.

***Matties:** That's why I carry my umbrella.*

Driver: It's a myth. The rain's a myth.

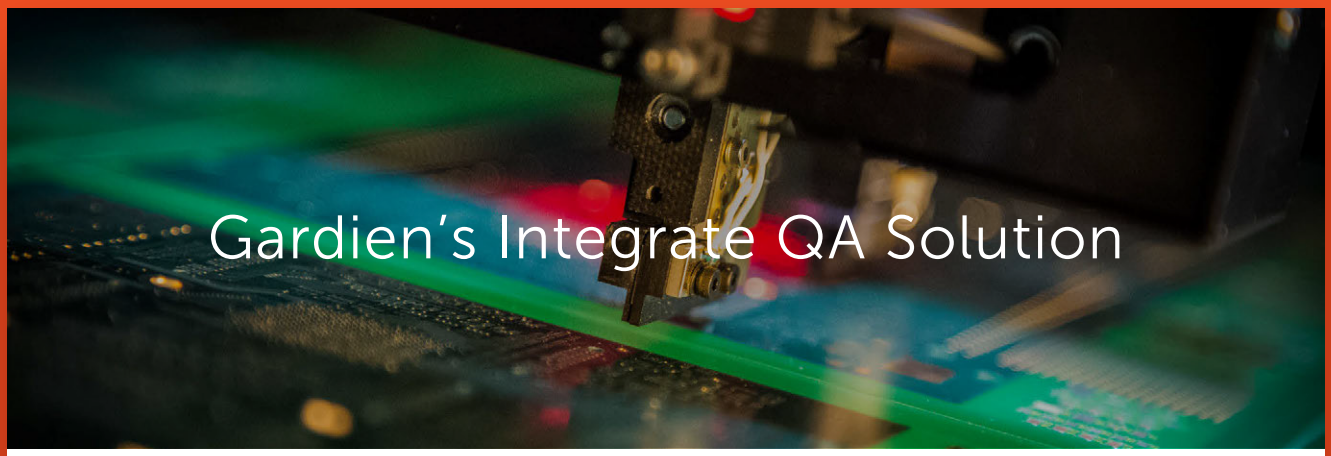
***Matties:** I just went on a nice tour of your facility, and it looks like you guys are doing some interesting things with metal backed boards here, but you're also setting up a new facility in Romania at the moment. Let's just jump right into that, since I've already spoken with your managing director, Martin Randall, about what you're doing here. Tell me about what you are planning to do in the future, in Romania.*

Driver: The Romanian project is customer-driven. We have a dominance in the LED PCB sector. More than 30% of our business revenues here are on LED boards. We have a high demand for metal backed PCBs, single-sided PCBs using CEM-1 or FR-4 laminates. Nobody wants to make it any more. It's expensive to come from China because the freight is killing it. The freight is 20–25% of the landed cost. So we're building our first plant in Romania and it's going to be for single-sided PCB manufacturing. We have the building ready. The containers are being loaded with the equipment now. Pilot production will be around the September/October time frame. We have 12 employees here from Romania being trained, with another four arriving on the 30th.

***Matties:** What sort of manufacturing is this? You were talking about a fully automated process, is that the idea?*

Driver: For Phase 1, no. Phase 1 is using all the equipment we've got. We have made several acquisitions over the last few years, so we've got a whole bunch of PCB equipment around us. That is going to be installed in the next few months. It will be a standard shop using standard technology—screen printing, photo mechanical, etching, drilling, routing—with some new equipment from Viking and Technic. For Phase 2 we plan to install the Lunaris machine from Mutracx and have a digital section of the factory. This will be Q1 2017. Then we have a plan for Phase 3 to build a turnkey digital PTH shop, but that's about 18 months down the road.

Gardien



Gardien's Integrate QA Solution

WHAT IS INTEGRATE?

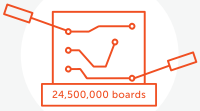
Integrate is a new approach to bare board PCB quality assurance. We address and are responsible for 100% of your tail end QA needs, whether it's AOI, testing, final inspection, or a combination of all three. We can provide the service in one of our service centers or fully integrate within your facility.

WHAT MAKES IT DIFFERENT?

Partnership pricing. Payment is based on product that actually passes quality assurance, not on panels produced or tested. Our success is built entirely on how successful we can help you become.

3900 jobs tooled per month

18100 orders processed per month



24,500,000 boards processed per month

Find out if our Integrate solution is right for you!

www.gardien.com/why-integrate

***Matties:** That's still pretty close. That's a pretty aggressive timeline from Phase 1 to Phase 3.*

Driver: It is, and we are 14–15 months into this project. The due diligence and the bureaucracy and getting everything organized has been a lot slower than we'd perhaps have liked. We are building a bespoke single-side PCB shop. There is already a lot of competition in the PTH and multilayer market. We feel that Spirit Circuits has had a good model here for the last 15–16 years on 1–8 layer quick-turn. Over eight layers we've got strong competition in the UK anyway so we've concentrated on the lower end, but do it well, do it quick, and with a specialism in thermal management. We have very good thermal engineers working for us and we give customers an educated choice on what materials they choose to use. We don't ally ourselves with any one particular manufacturer of materials, but we do extensive testing and we do design and offer some assembly as well.

***Matties:** Why did you choose that location over this one?*

Driver: The footprint in Romania is the attraction. Most people want to say it's the labor cost. Currently the labor cost is of course an attraction, but it won't stay that way, as it didn't in Poland or the Czech Republic. It's the actual footprint. The building, the land and the utilities are all low-cost. It's a low-cost central Eastern European location. To put it in context, the electricity is one-third of the cost per kilowatt-hour to that of the UK, water is one-third of the cost of the UK, and the local government taxes are one-tenth of the cost. Our total cost of operation is so much less expensive than being here.

Why Romania? Also in Romania, location is pretty important. A lot of our business is dealing with contract manufacturers in the Czech Republic, Germany, Hungary, and France. Probably 40% of our business now is in Europe and we see more of our customers moving over into the central and Eastern European locations. Romania itself wasn't our first choice. We looked at other countries like Bulgaria and Macedonia, but we've had very good encouragement from the educated workforce that's available in Romania. There is also the opportunity for European funding, which we haven't had any yet but that we're hoping to get.

We're coming down into the Southwest of Romania, whereas in the Northwest at the moment there is quite good development there. We've got big companies like Flextronics and Kimball that are already very well established up in Timisoara. We're down in Dolj County at a town called Craiova, where there is still some government assistance available and where there's still around 10% unemployment. We have our first 16 employees here. None of them have PCB experience. They are here at Spirit for two to three months for training. There isn't any significant PCB manufacturing in Romania so we are completely starting from zero with our skill base and knowledge base.

***Matties:** That could be a big plus actually.*

Driver: I'd like to think so. We want to be a little bit disruptive with some of our processes and

not have any preconceived ideas. We are also very keen to work with the universities. We've aligned ourselves with a couple of universities where there are some very intelligent young people coming out of university with not very good prospects in the towns that they are living in. There are some very large companies in Craiova that have shed lots of jobs, so we are able to offer some encouragement to these people.

***Matties:** Aside from perhaps economics, why wouldn't you start with the Phase 3 model? I understand you have the equipment in hand.*

Driver: We don't have the demand and the risk is mitigated in what we're doing because everything that we're going to be making in Romania we currently buy in China, so we're going there with an order book. If we were to start with something different to what we currently have as orders, we'd then have to go and fill the plant, whereas we are building this now where one customer already satisfies one shift. Our whole objective of this is to repatriate some of the work that we're doing in China, where we have a very small margin, and give our customers better agility. If we bring product by sea, you're typically looking at 10–12 week total lead times, and customers are changing their minds during that period. Bring it by air and we lose the margin on the freight, so we're doing this with a known order book. That's what is mitigating the risk and that's the attraction.

***Matties:** It's a mirror of what you're doing.*

Driver: It's literally repatriating the work and using our existing order book. Yes, I would love to build a factory like Alex Stepinski and have a digital plated-through-hole factory. We don't have that as an order book at the moment but that will be our Phase 3. We will go for digital manufacturing of plated-through multilayer once we've got ourselves established. There also is the risk factor of not having any skilled labor available in the country. We're starting from zero with the labor. Once we've got some basic skills there, we can then build on those skills.

***Matties:** That would almost make the case for jumping to Phase 3 first, because the reduction of skill in Phase 3.*

Driver: It would, but then you need the money to build the plant. We don't have the money.

***Matties:** Right, that would be finances aside.*

Driver: Finances aside, yes, I'd probably go straight into phase three. If I had $10 million to go and build a factory I'd probably go for it. We could make it a digital, hands-off operation, although it's never really hands-off.

***Matties:** No, I think Alex has 17 people where normally it would be 70 or 80. How about waste treatment over there? How do you handle that? Are you going for a zero-discharge setup?*

Driver: We are engaged with Alex and we're looking for zero discharge. Alex was here at the weekend and he is going to be engaged with us with some of his innovations on waste treatment. I'm impressed with his thought process.

***Matties:** The nice thing about what Alex is doing is he's repurposing equipment that already exists to achieve the goal. It's not a lot of special new technology that you have to chase.*

Driver: He's being disruptive and he's challenging the norm. I have 15 years on Alex and have been making boards a lot longer and it's annoying, some of the obvious things that he's chal-

lenging at the moment, but at the same time refreshing and encouraging to see a new generation with a passion for our industry.

Matties: *Fresh perspective is always refreshing.*

Driver: I'm very impressed with Alex's disruptive mind. We're going to be working with Alex and he's going to have some input into how we design the factory.

Matties: *That's great news. Anything else you would like to discuss?*

Driver: I don't think so, but it's very nice to see you here, and speaking on behalf of the EIPC it was very nice to see you guys at the EIPC Summer Conference. The conference in Edinburgh was fantastic as well as the global presence there. You had some of your colleagues over from the U.S. and it was very nice to see you guys here. I'm very pleased to welcome you actually.

Matties: *Thank you. It's been great to be here.*

Driver: We're a very proud PCB company here, and it's nice to have global visitors and global coverage.

Matties: *It's our pleasure, absolutely.*

Driver: I'm sorry that you had to suffer Jake [president of Viking Test] today. [Laughs]

Matties: *He's going to teach me how to play golf while I'm here. I'm sure I'm about to get schooled.*

Driver: As you're very aware Barry, we are neighbors to Jake, we are neighbors to Cemco, and we are in a very fortunate position. We've got some good support around us.

Matties: *In fact, it was an unannounced meeting we had with Peter Lymn of Cemco this morning. It was just fascinating. I've known Peter by name for many years but this was my first time to actually get to know him a little bit better.*

Driver: He's a fantastic guy, a guru.

Matties: *His mind works a little differently too, doesn't it? In a positive way.*

Driver: It does. He's a good neighbor for us. I like innovative people and he's certainly one of those, and he is a character for our industry.

Matties: *Then I see Jake's equipment from Viking throughout your facility, so that's great.*

Driver: Jake's done well for us.

Matties: *I saw some Chinese equipment here too. Maybe some routing equipment you have down there?*

Driver: Not routing, but we've got scoring that Jake supplied and XY measuring machines.

Matties: *Martin said you're really happy with it and that it's just great equipment.*

Driver: It's perfect. We wouldn't buy it without support of a distributor or a service team. There are opportunities to buy Chinese equipment yourself, but we wouldn't do it without Jake or another. It hasn't been without its problems. For example the scoring machine arrived with a Chinese operating system on it and Jake's brother Sam was here and fixed it. It's not without problems. It's not without risk, but you need a good partner. We're very happy with Viking.

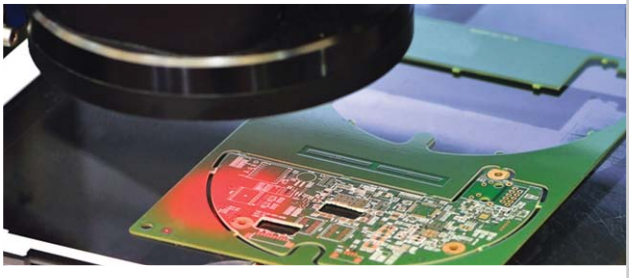
Matties: *Thank you very much, Steve. Appreciate it.*

Driver: Cheers. **PCB**

In North America

Uyemura is

for ENIG, ENEPIG,
Via Fill Coppers,
Electroless Golds
. . .and now for ***EPIG!***



UIC is proud to be the Final Finish Leader,
with more new ways to help our industry innovate and compete.

New from the Uyemura Labs:

EPIG, a process for fine line boards that plates palladium, then gold, directly on copper. Ideal for RF/microwave applications, this nickel-free process significantly increases signal speed. EPIG is the future for shops that require maximum process flexibility.

TWX-40, a hybrid gold bath for heavy deposition. Where competitor products congratulate themselves on plating 2 to 2.5 µin, TWX-40 plates 7-8 µin of gold on palladium. TWX-40 is ideal for wire bonding and other applications with heavy gold requirements.

UYEMURA
USA

As we have since 1985, Uyemura-USA is committed to providing its customers with game-changing advantages in performance and cost, and to supporting each program with the industry's finest technical support.

Corporate Headquarters: Ontario, CA • ph: (909) 466-5635 • (800) 969-4842 • www.uyemura.com **Tech Center:** Southington, CT • ph: (860) 793-4011 • (800) 243-3564

Supply Lines
Highlights

Next-Generation PCB Drilling
During the recent NEPCON China event in Shanghai, I spoke with Andreas Schneider, sales director for cutting and structuring laser at LPKF Laser & Electronics, about the latest developments in laser drilling, how customers can justify investments in laser drilling machines, and their outlook for the industry in Asia.

Ventec International Group Earns ISO 9001:2015 Certification for German Facility
Ventec International Group, a world leader in the production of polyimide and high-reliability epoxy laminates and prepregs, is proud to announce that its Central European facility located in Kirchheimbolanden, Germany has received certification under the ISO 9001:2015 quality management standard.

Viking Test Services: Much More than Test
Recently, 007 Technical Editor Pete Starkey and I toured Viking Test's facility in Hants, UK where we met Managing Director Jake Kelly to learn about the history of the company, their recent involvement in the Indian PCB market, and what he thought of the EIPC Summer Conference.

Rogers Corporation Launches UL 94 V-0 Circuit Material
Rogers Corporation introduces RO4730G3 UL 94 V-0 antenna-grade laminates to meet present and future performance requirements in active antenna arrays and small cells, notably in 4G base transceiver stations (BTS) and Internet of Things (IoT) applications as well as emerging 5G wireless systems.

Faster, More Accurate AOI is More Important than Ever
Among PCB manufacturers, there is a common perception that AOI has not changed much since it was introduced decades ago. Over the last few years, however, new advanced technologies have made AOI solutions smarter, faster and much more accurate. AOI system data have even been integrated with newly developed automatic 3D shaping tools.

Candor Improves Positive-Acting Liquid Photoresist Process; Installs ENIPIG Line
Toronto-based high technology circuit board manufacturer Candor Industries has recently taken steps to improve certain aspects of their fabrication processes.

Laser Pointers: Special Edition—Those Five Words I Didn't Expect to Hear
I'm going to hijack the column this month for kind of a public service announcement regarding a topic that hit very close to home for me, and it may well hit home for others. On February 18, 2016, I heard the words, "Mike, you have bladder cancer."

Nano Dimension Achieves Important Milestone in 3D Printed Circuit Boards
Nano Dimension Ltd., a leader in the field of 3D printed electronics, announced today that it has completed the development of the initial version of its software package, which will be integrated in the company's DragonFly 2020 3D printer.

Zuken Launches Latest Version of E3.series Software
Zuken's latest version of E3.series offers engineers a live-feel, collaborative environment, along with a range of individual and multi-user usability and productivity features.

Digital Imaging Update
Through the years, I have repeatedly covered and updated digital imaging in this column, from as far back as 1997 in CircuiTree, and through a column in this magazine in November 2015. Several reasons for this extended coverage include the fact that technology had a slow, long incubation time that eventually led to accelerated improvements and acceptance for mass production.

esi®

PRECISION IN MOTION

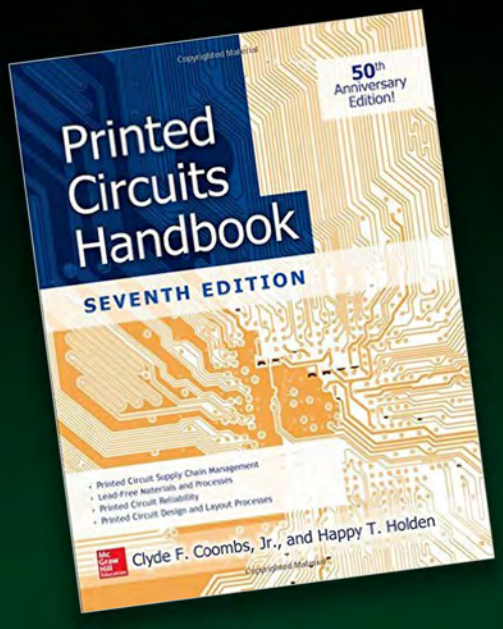
For over 40 years, ESI has been providing the laser technology and expertise needed to keep up with an industry that is in constant motion. By offering reliable platforms teamed with world-class service and support, we help the world's top PCB processors meet their customers' needs for quality and accuracy.

And you can be assured that this experience and expertise has been applied to nViant, an HDI microvia drilling solution engineered for affordability, reliability and accuracy at high volume.

Choose ESI as your full-service partner to stay ahead of the HDI manufacturing game.

Learn more at esi.com

The Authors of the Printed Circuits Handbook "Speak"



Editor's Note: The seventh edition of the Printed Circuits Handbook *was published this spring, which was also the 50th anniversary of the first edition. For this issue—"Voices"—we asked the many authors of the Handbook for their thoughts—their voices. We asked a few questions to get them started; though not everyone spoke strictly about the Handbook, we found their comments interesting and thought-provoking, and we hope you do as well. We begin with a wonderful history of the Handbook by the main man himself, Clyde Coombs.*

Clyde Coombs
Editor-in-Chief
(Chapter 1)

Background of the *Printed Circuits Handbook*

The *Printed Circuits Handbook* is now in its seventh edition, and we are observing the 50th anniversary of the publication of the first edition. This long-term level of importance in an industry is remarkable, but the need for this book seems obvious today. This is to put the concept of the book into the context of the industry when the first edition was published, and try to explain why there was a book in the first place, and what led to this long string of successful subsequent editions.

Touring a modern, technology- and capital-intensive, highly-automated printed circuit factory of today, supported by a staff of trained specialists, many with advanced degrees in science, engineering and systems, would be a totally different experience than touring a printed circuit shop of 1959. For the most part, those shops were the creation of entrepreneur artisan platers or silk screeners, and the facilities were called "bucket shops" for good reasons. With the exception of IBM, Collins Radio, RCA, and a few others, along with the founding members of IPC, the estimated several thousand shops in the United States (numbers at the time ranged from 4,000 to 7,000) were operated by rules of thumb, years of experience in related trades, and generally considered an art, not a technology.

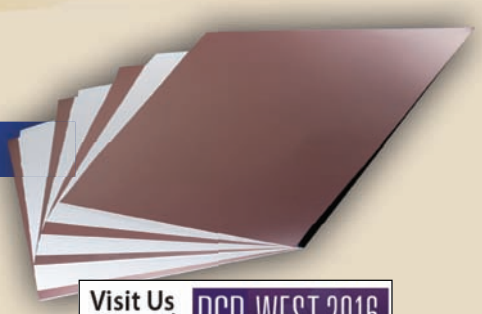
Shops were divided into two categories: captives, which made boards as a part of a vertically integrated OEM, and independents, which made and sold boards to OEMs that did not make their own. Both categories of shops could be justified since it was generally accepted that it did not take significant technical skill, or a large capital expenditure to start a shop. However, in 1959, the printed circuit world was on the brink of a major revolution that few shops were prepared to cope with, and most shop managers did not understand. The spark for this was the sudden introduction, and swift adoption, of the transistor into electronic devices. As vacuum tubes disappeared, and more functionality was designed onto much smaller boards, there was a sudden need to be able to connect circuits on both sides of a board reliably.

Entire OEM futures were being staked on whether printed circuits would even serve as an interconnection system. National advertis-

Don't Let Your Temperatures Rise

Let ROGERS' Thermal Management Solutions *Keep You Cool*

COOLSPAN® TECA Film • 92ML™ StaCool™ Laminates • ML Series™ Laminates & Prepregs



Heat can be damaging, especially when it is not managed. That's why Rogers Corporation invested so much time and energy into creating an array of material-based thermal management solutions to keep heat rise to a minimum in printed circuits. From automotive circuits to LED modules to power supplies, ML Series laminates and prepregs effectively conduct heat away from the source, while COOLSPAN thermally & electrically conductive adhesive (TECA) materials enhance the thermal management of new and existing designs. And for that extra cooling edge, 92ML StaCool laminates feature a thermally conductive metal bottom plate to enhance the heat dissipation.

Product	Thickness	Thermal Conductivity (Z-Axis), W/mK	Thermal Impedance, C-cm2/W	Tg, C	CTE (Z-Axis), ppm/C		Dk, 1MHz	Df, 1MHz	Breakdown Voltage, kVAC	Flammability
					<Tg	>Tg				
92ML	8mils	2.0	0.52	160	22	175	5.2	0.013	>50	HF V-0

Don't let your temperatures rise. Use Rogers' thermal management solutions.

Visit Us @ Booth #201 PCB WEST 2016 Conference & Exhibition

ROGERS CORPORATION

Advanced Connectivity Solutions

www.rogerscorp.com/acs

USA - AZ, tel. +1 480-961-1382
EUROPE - BELGIUM, tel. +32 9 235 3611

ing campaigns were focused on the fact that the product did not use printed circuits. Hewlett-Packard's first push into this process world was a disaster, which brought into question whether a superbly engineered product could be left to the questionable quality that would result if printed circuits were the foundation of its manufacturing. At the same time, it was also obvious that, with transistors being designed into all new products, hand soldering was not a viable option, even for the short term, let alone the entire new product line.

> "The first try at printed circuit assembly—crimping component leads onto board pads—was slow and unreliable."

The first try at printed circuit assembly—crimping component leads onto board pads—was slow and unreliable. The next step was the use of eyelets, which had some success on single-sided boards, and was proposed as the basis for two-side connection. It's almost impossible to describe the level of controversy that existed in the electronics industry of the time over whether eyelets or plated-through-holes were the best alternative. Today, that seems laughable, but the discussion was deadly serious at the time. The general betting seemed to be on eyelets. The reasoning went as follows: It was absurd on the face of it, that a process could be developed that would plate copper on a nonconductive surface, and do it in hundreds of small holes in a board—without missing even one—while eyelets were positively put in place and missing one would be obvious.

This environment of uncertainty spawned a furious search for an alternate circuit packaging technology and resulted in some very creative proposed solutions. Most were developed for specific products, or types of products, but were often put forward as general purpose packaging solutions. Most are now lost even to history.

As a recent graduate in electrical engineering and a new HP hire, with three years of experience in the United States Navy, my first assignment was to sort this out, propose, and develop a process that would be reliable and cost-effective, and could be integrated into the manufacturing capabilities of the company. I soon realized that they had assigned the wrong person. Instead of an EE, who could communicate with product developers, and board designers, what they really needed was a chemical engineer who could understand the chemical issues of plating and board materials. However, I had the job and needed to do a lot of learning.

First, however, I quickly eliminated eyelets, which HP was using successfully on single-sided, low volume boards, because 1) the difference in the coefficient of thermal expansion between the eyelet and the board material meant that after soldering, the two no longer could be counted on for an intimate contact, leading to potential intermittent, or open, circuits and required considerable touch-up with soldering irons; 2) eyelets had to be installed one at a time; and 3) the smaller eyelets needed for transistor interconnection were more expensive and even harder to install. That left plated through-holes, but the question remained: How could they be done reliably and consistently?

I went to the corporate library and found two books that claimed to be on printed circuits. One was really about silk screening, and the other was a high-level discussion with no real, or useful, information. The only actions left were to evaluate commercially available chemicals, and to see how others were doing it.

Giving some support to the concerns about the possibility of developing a reliable through-hole plating process was the problem that early chemical processes for reducing copper ions were very unstable. It was quite common for the solution to "go critical" and all the copper ions in solution suddenly reduce to a lump of copper at the bottom of the tank. In addition, there was a distinct interface between the copper that was deposited and the copper laminated to the base material. This required a separate sanding process to remove the copper deposited on the surface, leaving only the holes with new copper.

The result was no standard and low confidence.

Rising to meet these issues, however, a series of patents filed in the early 1960s introduced a chemistry that had a time-controlled reduction and resulted in copper deposited on the surface that did not require a sanding process. I've always felt that this was one of the most important technical advances of the twentieth century, and also the least appreciated. It became a commercial product from Shipley Chemical. We made two instruments with boards plated with this chemistry, and after full stress testing and evaluation, all discussion of eyelets stopped, and within six months all HP active boards were being made this way. The rest of the electronics industry took a little longer, but the plated through-hole has been at the heart of almost all electronic devices ever since.

At this time, the San Francisco Bay area was dominated by relatively large captive shops, with a number of small independents. I took this opportunity to leverage the Hewlett-Packard name and get invitations to visit as many of these shops as possible. The captives were just proud to show what they could do, while the independents saw us as a possible customer, but, in any case, not competitors. As a result, people were quite candid in telling me what they were doing. After a few visits, it became obvious that everybody was doing it the same way, thinking they had something unique. But it also became clear that what we were talking about was just chemistry, and of course it was basically the same. Since the people I had met did not feel competitive, I suggested that printed circuit association for our area might offer an opportunity for suppliers and users to get together to discuss technology issues confronting each. The result was the California Circuits Association (CCA). I had no idea how this would play out, but it grew from the five charter members in the Bay Area to three chapters, including two in Southern California, and more than 150 members at its peak. The biggest thing for me, however, was that I got to know who was doing what, and, more important, who really knew what.

One of the early efforts by the CCA was to hold a one-day seminar on all aspects of through-hole plating, featuring a PhD from Stanford Research Institute, Ed Duffek, and his technical assistant, Ernie Armstrong. It included the chemistry of getting a reliable coating down in the holes. The event sold out immediately, and we could have used a much bigger auditorium and charged much more. That's how hungry people were for real information. As part of the cost of admission, we provided a summary of all the information Ed and Ernie presented (this later became the basis for the chapter on plating in the first edition).

With the success of the Plating Seminar, we asked if there were other parts of the printed circuit process that would yield similar interest and we decided that just about all parts of the process would get a big reception. However, being a volunteer organization, we did not have the resources to repeat this effort very often, and we decided to concentrate on monthly meetings.

> "Every book that was available was done by a single author and tended to concentrate on that person's field."

The need for real information was still there, however, and those with expertise in each process step were clear, so I decided that it was time to rectify the problem that there was no authoritative book on printed circuits in print. Every book that was available was done by a single author and tended to concentrate on that person's field. I felt that a contributed handbook, which is a McGraw-Hill specialty, could allow the use of a leader in each area. I made a formal proposal to McGraw-Hill based on this approach, and they sent it to a board of review for comment. One reviewer said he thought it would be OK, another said it represented a discussion of an unimportant issue and would be a huge mistake, a third said it was a great idea and McGraw-Hill would probably find they couldn't print them fast enough. The third reviewer was right.

When the book was published it went through printings as fast as McGraw-Hill could order them. Obviously, printed circuits were a very important issue, with a big, unsatisfied need for information to help technologists understand it. Large companies were buying the books by the hundreds and giving them to their entire technical teams. Small companies were using them as cookbooks. An authorized Japanese translation was quickly published, and an unauthorized reprinting appeared in Taiwan. I started getting mail and invitations to visit facilities around the world. That was 50 years ago, and the book has maintained its position as the industry's best source of information on all things printed circuits ever since. Also, as the industry has changed, so has the book, with new editions bringing new developments into focus and continuing to answer real questions. It's still a useful book about an important subject.

Looking back, I think that the key to the success of the book, from the first edition to the seventh, is the convergence of a series of points: 1) the printed circuit is the basic building block of electronic devices, and rather than receding in importance as components have seen more and higher levels of integration, it has become even more important as an application specific interconnection system; 2) it covers the entire process; 3) each chapter contains real information on how to perform processes and how things work; 4) as the need for information has developed in many parts of a company beyond manufacturing, we have put the same level of information on design and engineering issues.

Will there be an eighth edition? I can't say, but the challenges in the electronics industry haven't abated.

Happy Holden

Editor

(Chapters 1, 5, 16, 25, 26, 27, 43, 58, 65, 66)

Q: What would you do differently for this Handbook (if you could do it over again)?

A: There is nothing I would do differently, but I do have regrets. It is sad that we had to remove chapters and content in order to meet the publisher's size requirement and add all the new content that required a new edition. That is always a trade-off with a hardbound, published book. That's one reason I like e-books: they can be longer, in color, with hypertext connections and electronic searches. I have totaled the pages from the 1st edition of the handbook through the 7th. More than 1,200 pages have been removed. Most of that content was not obsolete, but used less often, or not in current practice or interest or covered in other publications that are readily available. Newcomers to our industry may never know these technologies or techniques.

The handbook also focuses on practice, not theory. So you may not understand the WHY when an author talks about the best practice. My regret is that I have nearly 35 boxes in my basement full of PCB information collected over my 45 years in the industry and very little of that found its way into the handbook. There was just too much information—most of it not available on the Internet.

Finally, we don't have a chapter called "Neat Stuff." I would love to have added a chapter on the neat stuff created over the last 45 years by printed circuit innovators. The AT&T/Western Electric metal-core additive of early rotary phone days; Pete Peligrino's flow-motion plating that would deposit a mil of copper in 15 minutes; plated-post technology that created microvias twenty years before laser drilling; Kollmorgen's Multiwire® and Microwire® boards; the unique properties of tin/nickel plating instead of nickel plating as a barrier metal; and landless vias!

Q: Is there going to be an eighth edition?

A: Well somebody will write it, and maybe I will edit it—if I'm still around. Most assuredly, it will be an e-Book. This industry is always changing. There will always be a need for a Handbook. Future Printed Circuit Handbooks will probably include printed electronics; metal inserts/wires and cavities for power dissipation; 3D laser-shaped circuitry on molded plastics, embedded components and new methods of optical wiring.

Do you want the best via-to-pad alignment?

Use Tongtai's Single Station Drill Equipped with Nano System's Optical Registration System

Tongtai

SD-116

Cognex™ Hi-Res Camera

- *Machine vision seamlessly integrated with Sieb & Meyer™ control*
- *Expose up to 21 layers with a single percussion countersink*
- *Enhanced illumination for skived layer targets*
- *Automatic and manual modes*
- *Vision data log files generated for each layer*
- *Target data can automatically integrate with Xact PCB™ software*
- *Machine vision calibration & fiducial selection training included*
- *Optional precision depth control to ± 20µm*
- *160K rpm Westwind air bearing spindle*
- *Dependable North American service centers*

NANOSYSTEM

nanosystem-usa.com

contact us

Nano System, Inc.
Milpitas, CA (408) 935-2270
Manchester, NH (603) 645-1500
sam.sekine@nanosystem-usa.com

My concern: who is going to write the chapters? Clyde and I found it very difficult to find the new authors for the 7th Edition. Those with the most expertise were also the busiest, and maybe didn't have time to write a chapter. Fortunately, we found those experts and they made time to help us. We are eternally grateful that they took time to help the industry. Next time, the selection may be much smaller—or for me—not English speaking.

Q: Is there going to be enough new technology to justify an eighth edition?

A: There is probably enough new technology now to start an eighth edition. The growth of digital imaging alone would need its own chapter. Not just laser direct imaging, but the use of powerful LEDs as light sources and TI's DMD micro-mirrors have started to proliferate. Inkjet printing of etch, plating, solder mask and legends is also growing. Given the use of these techniques at the new Whelen Engineering PCB fab facility, we also should bring back green technology (as it saved them a lot of money and "no permits required"). I like to call this new captive facility a "Lean plus Green" example. Lean is emphasized and results in a panel cost half that of China but with a two-day turnaround time instead of six weeks. These Lean concepts have simplified the entire PCB manufacturing process. They used horizontal pulse electroplating technology to reuse the copper from their etching process as well as other novel techniques to totally recycle all their rinse waters. It is also significant that all the technology came out of the U.S. or Europe, not Asia. But now that the cat is out of the bag, we may see more of it implemented in Asia. One key attribute is the less than three-year return on investment. Hopefully, this will now interest banks and other loaning institutions in financing more PCB factory growth in North America.

Q: How will the Handbook be used by newcomers to the PCB profession?

A: We now have a generation in college, and more coming up through K–12 schooling, that have grown up with digital devices, video gaming, mobile phones and social networking. The effect of all of this has changed the nature of how they learn. To continue their education in electronics manufacturing—and specifically printed circuit fabrication and surface mounted assembly—we will have to adjust our training and education to this new generation of learners. For someone as old as I am, the challenge is to adapt my style of teaching to this new digital learner. I would like to work with the IPC to create a comprehensive, online, self-paced two-year and four-year degree in engineering (or at least a certificate of completion) for manufacturing in electronics that has the 7th edition of the Printed Circuits Handbook as one of its texts. PCB007 has other very good free e-books that can also be tapped in making up this new course. Given the work to create this, an eighth edition of the Handbook would have to wait until the coursework is finished.

Tim Rodgers

Adjunct Professor, University of Colorado
(Chapters 3, 4, 6, 7, 8)

One area that I would have liked to explore in more detail in the latest edition of the *Printed Circuits Handbook* is the topic of supplier performance. A lot of people seem to think that the most effective way to ensure high performance is to threaten suppliers with legal action or the loss of future business. A supplier who works to avoid negative consequences may achieve a minimum level of performance, but probably not much more than that. If you expect your supplier to represent your interests when you're not actively observing their performance, you have to provide a reason for them to do so. What's in it for them?

A supplier is more likely to behave as a partner if they get something more out of the relationship than money for services rendered. What do suppliers want? The answer varies, but here are some examples:

• Large, well-known customers that they can use in their advertising to attract new customers. This is especially valuable for smaller suppliers that are looking for revenue growth.

• Technical capabilities that can be leveraged to other customers. If the customer's requirements drive the supplier to develop new technology, then the supplier will be able to attract other customers.

• Entry into new markets. Suppliers that focus on specific markets (e.g., consumer electronics, semiconductors, automotive, aerospace) are at risk due to economic and demand cycles. A diversified portfolio of customers and markets provides more stability.

• Predictable demand for better asset utilization. Suppliers are just like any other business—they like being able to confidently plan into the future. This is so important that some suppliers are willing to give a discount if the customer is willing to commit to use a fixed level of their capacity over a period of time.

Most suppliers operate with very small profit margins, and if they are in a position to choose their customers, they have to consider the cost to service each customer. If you can't give them a reason to value your business, then you shouldn't be surprised or disappointed if they don't go the extra mile.

Andy Shaughnessy

Managing Editor, I-Connect007
(Chapter 18)

When Happy Holden asked me to contribute to the 7th edition of the *Printed Circuits Handbook,* I jumped at the chance. I still have an older edition of the book that I was given when I first started covering this industry, and just being associated with the movers and shakers of the PCB world is a real honor. I've edited Happy's articles for 17 years, so it was a little bit of a role reversal to have him editing my content. But Happy and Clyde couldn't have been easier to work with, and they really have this process down to a science.

They were great about dealing with my schedule. I still don't know how I found the time to put this chapter together. I remember working on weekends, and on Thanksgiving before and after our turkey dinner. But it was a lot of fun. Have you ever been on a video conference with Clyde and Happy?

Much of this chapter focuses on tools by the "big guys" of EDA, since they do have the lion's share of the market. But I was struck by the number of inexpensive and free PCB design software tools available. And these are solid tools that actually work well. Some of these free and budget tools are robust enough that the big EDA companies may be getting a little worried!

I'm glad to see the handbook expanding to include EDA tools and supply chain management. I think Clyde Coombs and Happy Holden deserve all the credit for updating this "Bible of the industry." If they ask for my help on the next edition, count me in.

Bill Hargin

President, Z-zero—PCB Signal & Power Solutions
(Chapter 20)

Q: What would you have done differently in this Handbook?

A: I do believe that the chapter Mark Montrose and I created adequately captures introductory educational material that's well-suited for people just getting into signal integrity, power integrity, and EMC. I've been doing that since the mid-'90s, and Mark, an EMC consultant, has been doing it even longer than that. I do wish there was more time to revamp the mechanical design section. That may be the focus for the next spin.

With the aging of baby boomers, like myself, I see educational materials like this book, which I use myself sometimes, as filling a critical need in the PCB design and fabrication world. In fact, my own company, which is an electronic-design software startup, is targeted, in part, to bridge that gap—taking about 20

Geek-a-Palooza

We're ***stronger*** in numbers!

Geek-a-Palooza is a social networking event for all of us electronics geeks that focuses on expanding our resources, building relationships and bringing every aspect of the local electronic industry together.

Our next event:
Geek-a-Palooza Boston | September 21

Register Now!

www.geek-a-palooza.com

Get your Geek on!

Boston
Wednesday, September 21

Register Now!

Minneapolis
Thursday, October 13

Register Now!

The Geeks Speak!

"Geek-A-Palooza combines the best aspects of technology and events in a fun and interactive environment. Having worked in this market for over 15 years, I think this innovative and fresh take on networking is going to shape our industry's future engagements!"

-Amanda Scacchitti, AAES Consulting, LLC

Love to one of our sponsors!

Omni PWB

Omni PCB- The Printed Circuit Board Specialists

With over 40 years combined experience in the PCB industry, Omni PCB helps its customers make informed decisions regarding their PCB designs and purchases.

Visit their website

Geek-a-Palooza

We're ***stronger*** in numbers!

years of lessons learned, and putting it in a software tool.

I will say that it was tough to carve out the time to create something meaningful. I felt like I was in college again, cranking out pages and graphics into the evenings and on weekends as the deadline approached. But, it's done, and I think it'll be a helpful introduction to signal integrity.

***Q:** Is there going to be an eighth edition?*

A: Heck if I know, but if there is, I'll probably double down on updating and improving the mechanical section.

Suzy Webb

Sr. PCB Designer, Design Science
(Chapter 21)

As PCB designers, we have the responsibility to bring order and inclusion to the chaotic needs of all the groups we work with. Those groups include the circuit design, mechanical design, fabrication, assembly, test, reliability, EMI issues, data sheet and model requirements, industry standards, industry specifications, and so on. In short, we have much to understand, and many hats to wear! All that, and it is our responsibility to prevent problems as much as possible, so that the product will perform as expected, will not need re-design, and will reach the market in a timely manner.

Many of us understand these principals because we have been around the business a very long time. Unfortunately, that body of knowledge may be fading away from the newer designers. The recent recession thinned the designer ranks considerably, and other long-time designers are getting near retirement age. There are no schools or engineering classes that teach the nuances of PCB design. There are no books or classes that can teach all of the ramifications of choosing this issue over that (although The Printed Circuits Handbook and books on specific topics are a very good start). Some of us try to share our knowledge with newer designers by leading classes, writing articles, speaking at conferences, and consulting at businesses. We encourage others to share their knowledge through user forums, software group meetings, and Designers Council meetings. Whether a designer is relatively new to the business, or an engineer is designing his own boards, we encourage him to read those articles or forums. Attend those classes, workshops, and conferences even if you must do it on your own time. (Many of us did it too.) Now is the time to be pro-active about your career! You will need a good foundation because there are many new challenges coming.

Michael Carano

VP Technology and Business Development, RBP Chemical Technology
(Chapter 32)

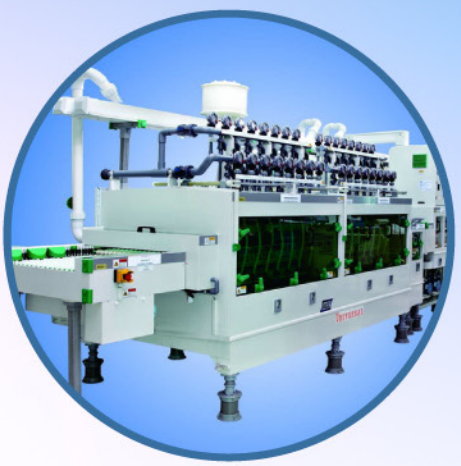
As an industry veteran with 36-plus years of experience, I have come full circle in my belief as to how the North American electronics industry can compete in today's global economy. Many years ago I felt strongly that the North American electronics industry in general and the printed circuit board industry in particular were second to none. Then, as OEMs shifted to an outsourcing model, much of the expertise and know-how went with it. As I traveled globally I experienced this trend first hand.

First the migration of cellphones then smartphones, television sets, even semiconductors were finding a home outside of the United States. What would be the future? Well, a few data points recently allowed my belief to complete the circle. As an example, last October (2015), IPC hosted the HDI/Flex Conference in Minneapolis. The two-day event was well attended by over 115 participants. The latest information on via filling, reliability, materials and solderable finishes flowed from the experts to the audience. The PCB fabricators in attendance were some of the biggest and the best: Multek, TTM, and FTG. And all North American headquartered. Very impressive. All have a large manufacturing footprint on the continent.

UCE WET PROCESS LINE

Etchers

Scrubbers



Developers

PTH

Oxide Replacement

THIN CORE HANDLING • PC CONTROL • RELIABLE • LOW COST

CLICK TO VIEW OUR UCE WET PROCESS LINE

The leading European PCB Equipment and Parts Supplier

www.vikingtest.com
sales@vikingtest.com

Viking

Then fast forward to the recent publication of the *Printed Circuits Handbook!* Just take a look and see the vast quality of information presented in this latest volume. You can't help but recognize that the reader would be getting a priceless education from the authors of the various chapters in the book. And just take a look and note where all these authors and editors reside. Yes, the North American electronics and printed circuit board industry is in good hands.

George Milad

National Accounts Manager for Technology, Uyemura USA
(Chapters 33, 35)

I think that OEMs should discontinue specific process qualification and concentrate on specifying the attributes that they desire in the finished product.

The OEM may run a battery of tests on a specific vendor product and it may pass and then they specify the qualified vendor. Among the drawbacks of this method is that, defective parts may still occur as a result from variations in the processing conditions, that the manufacturer or the OEM, may not be aware of. In addition, new processes with a wider operating window may be available that the manufacturer may want to try but feels trapped in the qualified process.

Another advantage of specifying the attributes that they desire in the finished product is that the responsibility for the delivered parts remains with the manufacturer. In this case the manufacturer chooses the process that fits his shop conditions and may choose a vendor that he feels is knowledgeable and offers him the best service and problem resolution as the need arises. Some of the qualification procedures may be so extensive and cumbersome that they impede the progress of the industry as it comes up with better more robust processes that have not undergone qualification.

Jason Keeping

Corporate Engineering Analyst, Celestica, Inc.
(Chapter 44)

***Q:** How was the experience of putting together your chapter of the Handbook?*

A: As a technology leader for our organization, I have written documents for industry in the past; however, this was my first opportunity to have information put into hard copy book not as a document but as a book, and it was a great pleasure. As the content of this chapter had not changed much in the past decade based on the legacy of information that was already available, my real goal was to find content that was new that could enhance this section for readers. With recent work that I had completed for the IPC-HDBK-830 Conformal Coating Handbook, I had this fairly easily available and just needed to confirm all aspects and have them incorporated into this handbook. After this was done the overall flow and images just needed to be amended and put more into the current century. With current industry progress on nano materials and new developments, the next time this section may have new material types as well as content to empower its readers both in hardcopy and digital form.

With the speed that our industry is developing at the moment, the amount of changes that were encountered within this edition will most likely be exponential and not linear, just to maintain a parallel path to these market innovations and changes.

***Q:** Do you see any challenges in PCBs in general or PCB education in particular?*

A: Within my role I work with all market segments between military/aerospace, industrial, telecommunication and even consumer covering all aspects of ruggedization from assembly cleaning, staking, edge/corner/complete underfill, and selective/atomized/vapour/dip deposited conformal coating and even component encapsulation and full assembly potting tech-

nologies. However, these technology sectors are continuously in development due to newer material research and cost pressures to compensate for newer harsh environments than previous designs and/or even legal requirements such as RoHS and/or even local requirements connected to emission either into the air or down the drain.

With this all stated, one aspect that is a hot topic—depending on whom you discuss with—is no-clean. From the information that I have obtained over the years, this was a great marketing term; however, is not true if you have the perception that no-clean means that assembly cleaning is not required. For the bare PCB this may be true; however after you run a PCB via the SMT process, add manually and/or via automated equipment components for a wave soldering process, then perform various manual touch points such as handling for test and/or inspection, the resulting ionic levels at this point usually do not mean the assembly is clean—yet all materials are "no clean." At this point the challenge is what the next step should be? This would be a challenge that I see coming that the industry will need to understand and what test methods and guidelines should be used and followed.

Laura Turbini

International Reliability Consultant
(Chapters 45, 59)

Q: What do you really enjoy about the industry?

A: I have had the privilege of working in the electronics industry for almost 40 years, starting in 1977 when I joined Western Electric's Engineering Research Center. As one of the early women in the industry, I was treated with respect and support by my many male associates. One thing that always impressed me in my colleagues is the way they worked together and helped each other. Over the years I had a chance to learn from the industry's leading thinkers. The insights I received from them led me into the area of research related to failure modes in printed circuit boards—particularly to the study of CAF (conductive anodic filament) formation. "Standing on the shoulders of giants" enabled me and my students to identify the chemical nature of CAF and how it is formed.

I was first asked by Clyde Coombs to contribute to the 4th edition, which published in 1996, and subsequently to the 5th, 6th and 7th editions. The present 7th edition was very important as a central resource for lead-free soldering. Writing a technical book chapter took a great deal of time to compose it in a clear manner and to make sure there were no errors—but I had learned to do that early in my career when I was the editor of the Western Electric Engineer. I am pleased to have been a participant in this book and I hope my former students will pick up where I left off.

Reza Ghaffarian

Principal Engineer, Jet Propulsion Laboratory, California Institute of Technology (or JPL/NASA, CIT)
(Chapters 60, 61)

***Q: Why did you decide to write two chapters on PCB reliability for the* Printed Circuits Handbook?**

A: Well, when I look back, I find it to be interesting, even maybe amusing, as how a simple request by Happy, a dear long-time colleague friend and a co-editor, caused me months of hard work followed by a final proud moment of accomplishment. I am not joking since this for me is another extra activity, only work during weekend since I have a day job to do and that also put additional burden of as what I can publish. Well, I said OK to a simple request of review and republish with minor modification of a previously written chapter on PCB reliability. I thought a few weekends would be OK, especially knowing the Christmas holiday of 2014 was on the way. A few weeks became more than six months of nonstop weekend work and now

a feeling of relief and accomplishment and I am concentrating on other activities. Rather just minor modification, I made a complete face-lift and one chapter becomes two, one on PTH reliability and the other on a new topic of micro reliability. My intention always has been to pay back to the industry, but this became a hefty one even though I am proud of it. I am proud to be able to add the most updated research on PCB reliability. Again, just a contribution to technical community with no financial gain.

Q: It has only been a few months since the Handbook was published, but have you received any input from the industry?

A: Just this month I received two emails, one asking for a good reference on PCBs and the other one had comments after review of the chapter on microvia reliability, especially on drawback of current coupon testing. Even

“Just this month I received two emails, one asking for a good reference on PCBs and the other one had comments after review of the chapter on microvia reliability, especially on drawback of current coupon testing.”

though I have heard similar comments during IPC Thermal Stress Test Methodology Subcommittee meetings, but this is on the stack microvia which is an interesting one and I like to share since it is timely too. Jerry Magera wrote that for stacked microvias, they have observed that opens may occur at temperature above the PCB Tg (glass transition temperature), more typically around 220°C, it stays open through the peak temperature, followed by reconnect/closure during cooling. They have tested these types of failures to thermal cycling, -40°C to 125°C, and found that such latent defects were undetectable. Solder float tests also does not detect these failures. They also found that current induced testing is not appropriate for acceptance test and also unable to detect microvia issues. He recommends that “the current induced test coupons should be used by PCB manufacturers to monitor their process for PTHs.” So the two chapters on PCB reliability need to be revisited to include new technologies and requirements when the co-editors decided to publish the 8th edition. I want to send my sincere appreciation to the two well-known coeditors of the 7th *Printed Circuits Handbook* for their tireless effort to put together such an invaluable new edition.

Joe Fjelstad
Verdant Electronics
(Chapters 67, 68, 69, 71)

Electronic Assembly… Is it time to remove the “training wheels”?

Many children learn how to ride a bicycle by using training wheels—two extra wheels attached to the rear axle to allow them to keep upright as they learn how to ride. The down side of training wheels is that they slow the rider down and impede changes in direction. In many ways, solder is the training wheels of electronic assembly. It slows down manufacturing and impedes the industry’s ability change direction when it comes to making products that are better, more reliable and less expensive. However, in order to gain those benefits, as with riding a bike, the would-be user has to exercise balance, discipline and good judgement. To illustrate, consider the following example.

Below are images of two electronic assemblies, they are identical in function and were designed by the same designer, Darren Smith[1], using the same general design rules. Yet as can be seen, one is substantially larger than the other and has twice the number of circuit layers. What makes that possible? Three

We would like to publicly thank Dan Beaulieu for his continued marketing assistance and guidance during our relentless mission to provide

YOUR ULTIMATE SOLUTION.

A valuable consulting asset to Taiyo or any company…

A colleague with knowledge that spans several industries…

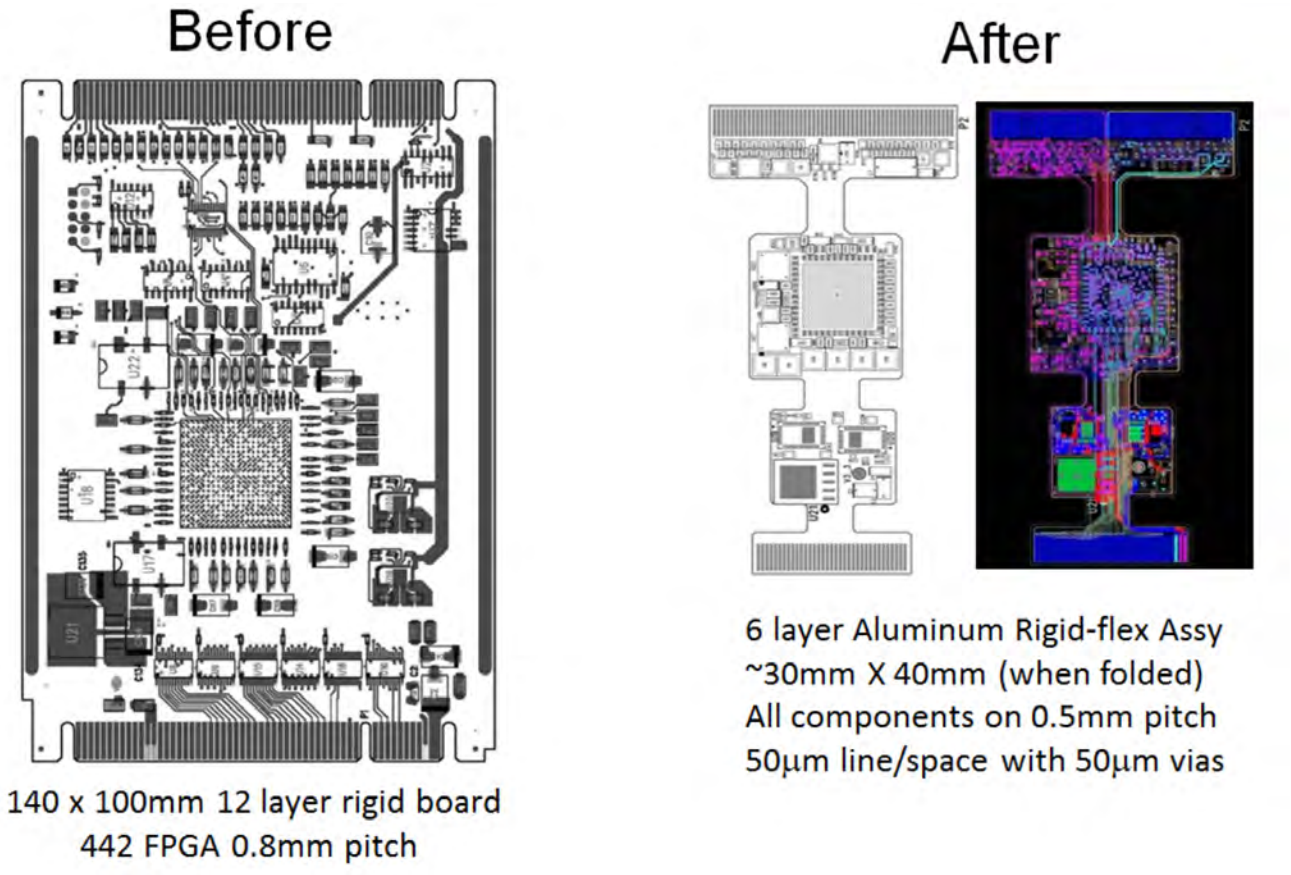
An intelligent, well-connected and kind gentleman…

THANK YOU DAN!

TAIYO AMERICA

Your Ultimate Solution

Phone [775] 885-9959 • www.taiyo-america.com • info@taiyo-america.com



Elimination of solder makes possible significant improvements in design efficiency.

things: the first is the elimination of solder; the second is the placement of all components (fully tested and burned in) beneath the surface with terminations facing up; and the third is selecting only components that have terminations on a common grid pitch. The combination of these three simple principles makes it all possible.

Consider first the elimination (or at least the minimization or possible replacement) of the use of solder. Solder pads take up a significant amount of space on the outer layers of a circuit assembly leaving little room for routing circuits. The fact that hundreds of millions of dollars are arguably wasted annually, inspecting, testing of PCBs and assemblies, x-raying and repairing defective solder joints seems, if one reflects without prejudice, a reasonable thing to try and eliminate. However, there is also the cleaning process which is getting ever more difficult as lead pitches shrink. Eliminate (or replace) solder and you eliminate all of these wasted steps.

The elimination of solder is made possible (with the least number of steps) by the second item from above. That is the building of the components into the board and then applying the circuits. More succinctly, rather than building a circuit board and soldering components to the surface, build a "component board" and then build up the circuits on them. Some might consider this impractical but it can be done and fortunately while using nearly all of the current infrastructure. Circling back to the analogy that bikes work better without training wheels, the manufacturing process can as well.

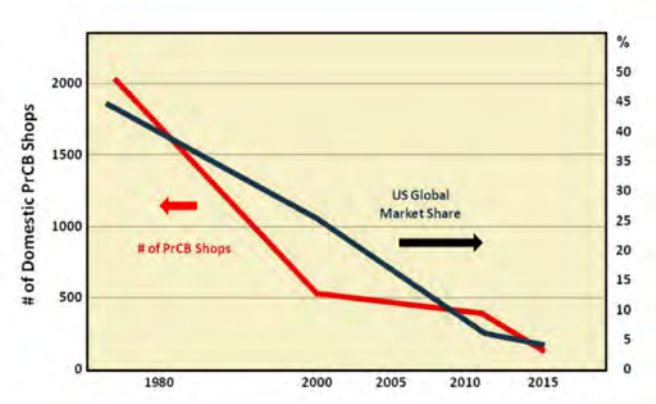
The last item is perhaps the most challenging because components come with leads on many different sizes, shapes, pitches and lead forms. J-leads, gull wing leads, though-hole leads, radial leads, pins, balls and even no leads for

QFNs and LGAs. It is these latter two that are of greatest interest and value because they are well-suited to getting to a standard grid (e.g., 0.5 mm). Moreover, in a solderless assembly they require no solder on their terminations. Less manufacturing steps and the elimination of high-temperature excursions which can impact long term reliability are both good things. The common grid pitch simplifies design by making routing more predictable. When all three of the principles suggested are combined, well, the results are self-evident above. It needs to be conceded here that some components cannot be easily adapted to the concept in their present configurations, but they are what they are because there were no real physical constraints when they were introduced.

In summary, it is possible to eliminate solder from numerous applications and minimize its use in countless others, and by doing so, enjoy significant benefits in design and manufacture. Not every company will be comfortable without the training wheels. They will worry about falling and it can be argued they will be less nimble as a result. Yes, eliminating the "training wheels" from electronic assembly requires some risk-taking but it is through taking risks that progress is made. It is the combined discipline of design, procurement and manufacturing that will carry the day. From experience it can be found that a key guiding principle of excellence in manufacturing is to first do the right things and then do those things right.

P. Marc Carter
Printed Circuit Engineer
(Appendix)

Those of us who deal with other aspects of electronic production realize how fortunate the printed circuit world is to have long-recognized, centralized resources like the *Printed Circuits Handbook*. The handbook (and a few other consensus resources, like IPC) allow us to speak in a common language. Often as not, we ARGUE, but at least we argue from a common starting point. Historically, assembly and microelectronics haven't enjoyed that same common tongue.



It is now a truly international business, and the shift of much of the fabrication base out of North America has resulted in (for an English-language resource of finite length at least) a necessary de-emphasis on fabrication details, and an increased emphasis on assembly, and securing printed circuit boards from remote sources. Another reviewer commented that an e-book that allowed the reader to focus on those areas that were of particular interest in their case might be preferable. The only other way I can think of to deal with that issue would be a two-or three-volume *Printed Circuits Handbook*, each volume addressing the needs of a particular segment. **PCB**

To read a book review by Karl Dietz, click here.

To view the Table of Contents on Amazon, click here.

References

1. Darren Smith may be reached by clicking here.

ELTEK
Looking for Strong Growth in the USA

by Barry Matties

At the recent Nadcap meeting in London, I spoke with Eltek USA President Jim Barry about the state of the company, further investment from their new owner, and their focus on the U.S. market.


Jim Barry

***Barry Matties:** Jim, why don't we start with a little bit about Eltek and what you do.*

Jim Barry: Eltek is a manufacturer of high-density, high-reliability PCBs. Our market is primarily medical, military and aerospace. The majority of our technology mix is rigid-flex boards. We focus on the high-reliability, high-density, "can't-fail" type of technology.

***Matties:** There was some recent news about ownership buying additional shares of stock in the last couple of weeks, correct?*

Barry: Correct. Our controlling shareholder and chairman and board, Yitzhak Nissan, bought control of the company about two and a half years ago and has made some changes within management. We've downsized management level a bit, and we've become leaner and meaner, I guess you could say. 2015 ended with net profit, and we are shipping quality product on time. So to show his confidence in our growth, he recently went out and purchased about 620,000 shares in our company, increasing his holding in Eltek from 50.5% to 56.6%. I think it was around $583,000 worth of his cash he's investing in the company. His confidence factor in the growth of the company shows, and we've been growing in the United States significantly. As the Israeli market is tightening and getting some offshore competition, the U.S. market is actually growing with our military and aerospace customers.

***Matties:** Just for clarification, you have two manufacturing locations?*

Barry: Yes, Eltek's parent company is Nistek, but Eltek itself has two locations. Eltek LTD is our main factory located in Petah Tikva in Israel. It's about 100,000 square feet, maybe 350+ employees. It has a small facility in Germany called Kubatronik. They're small, maybe 40 people, and do smaller batch sizes. They service the German market, or the European market. Then we have a smaller division in the United States called Eltek USA, which is our U.S. subsidiary. It is a

Productive Fabrication Equipment

MDI-DT

WorkCell²

Hybrid CombiDrill UV/CO_2

schmoll optronics

BÜRKLE PROCESS TECHNOLOGIES

schmoll optronics

MDI Micromirror-Digital-Imaging

- *Cost-efficient & high-quality DDI*
- *High-power UV-LEDs & DMD-technology*
- *Operator-friendly concept w/compact footprint*
- *Modular machine concept & field upgradable*
- *Flexible number of heads and head distances for shortest exposure times*
- *Multiple wavelength concept*

Vacuum Lamination System

- *Even pressure across the platen surface*
- *State-of-the-art heating system*
- *All-steel construction*
- *Modular machine concept & field upgradable*
- *User-friendly visualization system*
- *Affordable initial investment*

Laser Drilling/Cutting System

- *Microvia drilling 50-300µm diameter*
- *Space saving compact design*
- *Best power/size ratio in market*
- *Pulse Stabilizer*
- *European technology center and USA support center*

Bürkle North America supplies a full complement of productive equipment for fabricating printed wiring boards. From imaging through metrology inspection, Bürkle North America's equipment offering sets the standard in the industry for Imaging, Registration, Lamination, Mechanical or Laser Drill/Routing and Feature Metrology Inspection.

BÜRKLE NORTH AMERICA, INC.

11105 Knott Ave., Cypress, CA 90630

Contact Dave Howard for more details.

BURKLE NORTH AMERICA 25 year ANNIVERSARY

www.burkleamerica.com

Phone: 714-379-5090

small company with sales, customer service, engineering, and applications. You could say it's the front door or the store window for Eltek LTD in the U.S. It allows us to bring in military and aerospace work into Eltek USA and bring it to Eltek Israel through our ITAR export license to build for U.S. customers.

***Matties:** And you're the president of the Eltek USA?*

Barry: That is correct.

***Matties:** What is your main area of responsibility?*

Barry: I wear many different hats. My responsibility is for the sales in the United States, engineering, customer service, and basically bringing the work into the factory, filling the factory, making sure our customers are happy. We have a strong technical sense here. I'm a veteran in the industry for 38 years, have run many companies, and have been in executive management as well as engineering management.

***Matties:** With the investment coming back into the organization, there must be a lot of confidence and growth.*

Barry: Yes, absolutely.

***Matties:** What sort of growth are you guys looking at?*

Barry: In 2015, our sales in the North American market increased by 27% and amounted to $7.5 million. This represents approximately 18% of the company's overall revenue, but the long-term plan is to grow it further. Since we are a public company, I cannot get into projected numbers. My personal goal in the U.S. is to get us to much higher figures in the next number of years. I don't want to put a number on it, but we have an aggressive growth plan.

***Matties:** Let's talk a little bit about the military market, ITAR, and how you're able to manufacture offshore. That's a tricky business, I would think.*

Barry: Yes, it is. Eltek USA is a USA company, so we're registered with the Department of State. We have our DTC registration. The Department of State has approved our technical control plan, and we actually have a license, it's called an MLA, a manufacturing licensing agreement, which is a 10-year license. The license is actually between Eltek USA and Eltek Israel. So you as a U.S. customer would buy from Eltek USA; we are the name of record on the export, so we control the data. It comes to us, we sanitize as necessary, we upload using an encrypted file system to Israel, they manufacture and we ship back to the United States customer.

In the past 13 years, we've shipped probably close to $67 million into the United States and at least 40% of that has been military/aerospace work. We deal with a number of Tier 1 companies. I don't want to state names of my customers without getting their prior permission, but we deal with a lot of the major Tier 1 companies and some of our products are missile system-related, night vision goggles-related, aerospace controls, etc. We also do a tremendous amount in the medical field for implantable medical devices. The reason being, Eltek is known to be one of the highest reliability companies out there in the industry today. We tend to take on the challenges that nobody else either wants or can figure out.

***Matties:** When the ownership took over a couple years ago, you were talking about underperforming, no profitability, and losses. Cycle time or delivery time must have been a factor in that.*



Barry: Correct. Our delivery performance was suffering.

Matties: ***What shifts did you make? I know you talked about changing management; culturally, and from a process point of view, what were they able to do to turn it around so quickly?***

Barry: Well, I think our new controlling shareholder and CEO, Yitzhak Nissan, is a shrewd and smart businessman. He's tough and holds people accountable. I think accountability became a big thing; you must perform. We've also thinned down the management team to become much more efficient. We cut out multiple levels of management so that the answers come quickly, the decisions come quickly, and we're very responsive. Our market's lead time decreased somewhat. On time delivery is nearly 100% these days to the U.S. market, and technology growth is going up. In the three years ended December 31, 2015, Eltek invested $6.7 million in fixed assets. We actually have a piece of equipment that's brand new in the industry, serial number one, from Orbotech. Being they're an Israeli company, we often get to be their beta site.

We're now using direct imaging for solder mask as well as direct imaging for nomenclature so no more screening. We're plating our vias closed now. I'd say a major part of our technology these days is blind and buried. It's very common for us. We don't see a lot of through-hole all any longer as the devices are getting smaller and smaller and density is just ramping up.

Matties: ***Printing the solder mask makes a lot of sense. I'm surprised it's taken so long to really become something that the mainstream can adopt.***

Barry: It needs to be done as devices become tinier. We have many medical customers and I really think the medical market are the ones who push the bleeding-edge of technology because impantables are getting small and getting tiny; the military is starting to pick up on that. Night vision goggles are a good example. Old night vision goggles might have been two inches by two inches, but they're now down to the size of a half inch by a half inch square, they can be put into an eyeball-sized device. With medical implantables as well, there are implantables now for sleep apnea, neurostim and Parkinson's disease, and it's amazing how dense and how small and tiny they are. Without having that blind and buried vias technology, you wouldn't be able to get to the density or the reliability. The solder mask obviously has to get down that small, which is now going to start pushing the assembly people to get smaller and smaller with their dies and their chips.

We see a lot of wire bond on the rigid-flexes as well now. You see wire bond actually jumping from the old ceramics now to chips right onto the board or chips onto the flex. It's just amazing where the technology is going.

Matties: ***Do you also serve the manufacture of flex boards?***

Barry: We don't do a lot of single- and double-sided flex, and the majority of what we do is rigid-flex. Flex is a huge market. There are a lot of players in it. We don't do it because, I don't want to say it's too simple, but I could build an 18-layer or a 12-layer rigid-flex with blind and buried versus building a double-sided flex utilizing my capacity and my factory. It's a much higher profit product to do the higher tech stuff. We're a little bit like Einstein; we can build a 28-layer blind and buried without an issue but struggle with the simple stuff, as our mindset

is focused on the high technology end of the business.

***Matties:** From a buyer's point of view, what is it that they need to be aware of when they're dealing with a company and are looking to buy high-reliability military, medical, and that sort of thing?*

Barry: I think buyers get bombarded every day with all these people trying to sell them their technology. Granted, there are a lot fewer companies now than there were before, but people are still calling them every day. It's difficult because everyone says they can do everything, and that's just not true. Companies have to look for specifics. Again, I mentioned we're not good at doing single- and double-sided flex, so we turn it away. There are companies who specialize in that. There are companies who specialize just in prototypes who can build maybe a 10-layer blind and buried rigid-flex, but can't take it from that 100-piece quantity to 1,000–10,000. Eltek is more of a high-mix, low- to medium-volume so we can build you the 10, but we can also go to the 100 and 10,000. When it comes to high-volume rigid boards, there are companies who specialize in that.

To the buyers out there, you can't have one company that does it all. I think you have to have a matrix of maybe three or four companies that fit your needs—one for prototype, one for production, one for maybe rigid-flex. You have to be very careful.

The other thing is reliability. Anybody can build one of anything. It's being able to build over and over and over with high reliability. Our products have been out there for over 40 years. We're an old company. We've been around a long, long time even though some people haven't heard of us. You've got to look at reliability. How many products do they have out there in the field? We've shipped 14,000 rigid-flexes towards night vision goggles. We've shipped thousands and thousands of products that are in missile systems, space and medical. You have to look at the company, its history and what it has done.

Also, I run into a lot of comments like, "You're in Israel. You're in an unsafe part of the world. You're in the Middle East." It's kind of a false statement. People fear that part of the world, but it's actually safer there than it is here in the U.S. We (Eltek) were born from a military company so we know military reliability and urgency. People say, "Well, it's unsafe." I highlight to people some of the recent incidents that have happened in the States and around the world. It's not safe anywhere in the world, plus there are companies that are affected by snowfall, tornados, hurricanes, tsunamis, earthquakes, etc. We heard of a well-known U.S. facility that was shut down two years ago because of snowfall. It is a global manufacture space now…

***Matties:** It's always something.*

Barry: There's always something. We might have some bad neighbors, but we've never missed a day of work. With all of our armed conflicts in Israel, we've never missed a shipment to the United States. Again, I've been involved for 13 years. We have sold probably close to $10 million a year and are growing.

***Matties:** Is it common for buyers to go to your factories for tours?*

Barry: Yes. We've had a number of the Tier 1 facilities visit; sometimes they'll send an engineering team out there or sometimes they'll hire

Ucamco
Former Barco ETS

SOFTWARE | PHOTOPLOTTERS | DIRECT IMAGERS | NEWS | SERVICES | COMPANY | DOWNLOADS | CONTACT

Software

UcamX

New ground-breaking CAM Software

- For Rigid, Flex and HDI PCBs
- New Graphical User Interface with Workspaces
- Optimal performance using Parallel Processing
- Multi Job Processing via load balanced Sessions
- Support for Gerber version 2 (X2) format

Read more

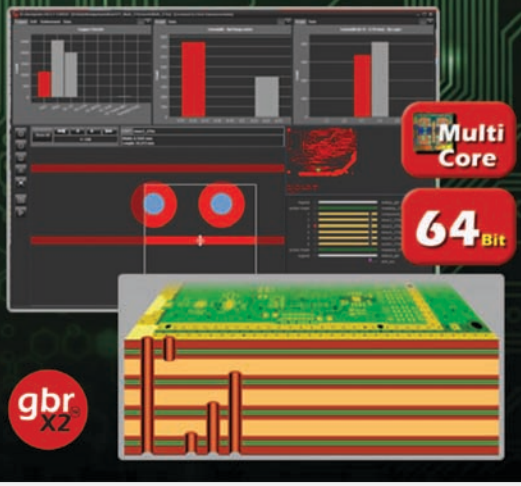
gbr X2 | 64 Bit | Multi Core

Software

Integr8tor

Automated data entry and design analysis

- Ucamco's 9th Generation Sales & Engineering Tool
- Field proven with more than 250 installations in daily production
- Seamless integration into any environment



Multi Core | 64 Bit | gbr X2

Read more

Ucamco
Former Barco ETS

SOFTWARE | PHOTOPLOTTERS | DIRECT IMAGERS | NEWS | SERVICES | COMPANY | DOWNLOADS | CONTACT

PHOTOPLOTTERS

Calibr8tor & SilverWriter

- Ucamco's Photo Plotters, the world-wide benchmark for accuracy, throughput and reliability
- Field proven with more than 450 installations in daily production
- Seamless integration into any CAM System

New

Field upgradable
with new high resolution
Nano AF100 Opto Electronics

Read more

For more information please contact info@ucamco.us,
call (415) 508-5826 or check out our new website at www.ucamco.com

somebody in Europe or Israel to go. This week, we're here in London at the Nadcap meeting. We support Nadcap. We're an aerospace supplier. We received our Nadcap accreditation a number of years ago. I actually sit as a supplier voting member on the Nadcap ETG task group. We work closely developing the specifications with many of the Tier 1 primes.

***Matties:** Is the goal of the meeting this week to come up with some of these specifications?*

Barry: Yes, Nadcap is always looking at its documentation and improving for its auditors and for companies like ours. As a supplier sitting in on these committee meetings, we have the opportunity to add our comments into it. It's a great forum; I think it's really important to come for the sake of meeting with your Tier 1 customers, but also you have some influence on the specifications that you're audited to.

***Matties:** You're part of the conversation.*

Barry: Exactly, so it's critical.

***Matties:** Part of being a business is being part of the conversation, because if you're not part of the conversation you're just overlooked.*

Barry: Correct, exactly. You have to live with what they give you.

***Matties:** Just some final thoughts. What do you think the greatest challenge is today of being a board fabricator?*

Barry: Oh my. Well, price is a big thing with erosion of the U.S. market and a lot of it going offshore. Some of that's coming back, so price is a big thing. You have to be able to have a factory that's lean so that you can be price sensitive to a customer. You have to be a factory that has a global mindset and can deal with customers all around the world. Not only do we do business in the U.S., but we're in Europe, we're in India. We're starting to deal all around the world.

***Matties:** When you say you're in those locations, do you mean with sales offices?*

Barry: No, not sales offices, but an actual customer base. We do a great deal of business around the world. You have to be a global supplier and think globally. Technology is critical, but I think even more than technology it's reliability. If I'm a buyer and I'm going to look at a supplier, I'm going to go to the supplier that may not necessarily be the cheapest, because cost isn't everything, but I'm going to go to the supplier who I can count on and who I can depend on. Where I know when I get the product, it's going to be reliable and that he does what he says he's going to do. I think those are the critical things. The PCB industry has had a bad rap for poor performance or poor quality, and then on the other hand everybody tries to buy for just low cost.

I've had customers actually go away from me for a few dollars, but then they have huge fallout from possibly an offshore supplier for yield.

***Matties:** That's not an uncommon statement. We hear people shopping for price over cost and value and they wind up just losing their ass.*

Barry: We've had several customers go offshore and then come back to us because they couldn't build the technology. There's also a lot going

on in the industry right now. You have the major merger of TTM and Viasystems. It's probably one of the biggest companies out there in the world now. There are a number of mil/aero customers who are afraid of this; mainly because now they have to seek out additional sources. We're seeing some residual business from this merger. You've got the one giant, you've got a handful of what I'll call mid-sized companies, the $50–100 million guys, and then you've got a whole slew of smaller ones. It's going to be interesting in the next number of years to see where this goes and how it's going to go.

***Matties:** Also it's going to be interesting in the next number of years to see how you meet your sales goal.*

Barry: I think we're going to be very successful.

***Matties:** It sounds like you have a good foundation to build on.*

Barry: Yeah, we have a strong customer base in the U.S. and we're growing. We just brought on another additional direct sales guy. We utilize direct sales people around the country as well as sales reps. Our mindset is not one of let's bring on a 1,000 new customers. Ours is one of let's bring on a manageable number of Tier 1 and Tier 2 faithful customers that we can help grow.

***Matties:** I think the big advantage you also have is an ownership structure that invests in the organization, empowers their people, lets you do your job, but holds you accountable.*

Barry: Absolutely.

***Matties:** I think that is key. We go from company to company; I've interviewed hundreds of people by now, and it's amazing how lacking accountability is in the business world.*

Barry: Yeah, I think it's key and I have a little thing on my email that says "accountability breeds responsibility." It's a statement from Steven Covey, who as you know, wrote The Seven Habits.

***Matties:** I did notice that on your email and thought, "Well, here's a man who's enlightened," because it's the truth.*

Barry: Yes, it is.

***Matties:** Are there any other thoughts that you have or topics you would like to cover?*

Barry: No, I appreciate your meeting me here in London and giving me the time to speak. I look forward to future growth with us and future conversations and being involved in the industry. That's all I think I have at this point.

***Matties:** Thank you so much. I appreciate it.*

Barry: All right, thanks. **PCB**

Electronics Industry News
Market Highlights

Artificial Muscle for Soft Robotics: Low Voltage, High Hopes

Researchers at the Harvard John A. Paulson School of Engineering and Applied Sciences (SEAS) have developed a dielectric elastomer with a broad range of motion that requires relatively low voltage and no rigid components.

Graphene Could Revolutionize the Internet of Things

EPFL researchers have produced a tunable, graphene-based device that could significantly increase the speed and efficiency of wireless communication systems. Their system works at very high frequencies, delivering unprecedented results.

Asia Pacific Dominates Global Consumer Electronics Market

The global consumer electronics market features several players dominating the market for a particular type of electronic device.

Engineers Integrate Nano-scale Sensors, Electronics and Microfluidics into Threads

For the first time, researchers led by Tufts University engineers have integrated nano-scale sensors, electronics and microfluidics into threads, ranging from simple cotton to sophisticated synthetics, which can be sutured through multiple layers of tissue to gather diagnostic data wirelessly in real time.

A New Generation of Electron Devices Makes Waves

No one has managed to push radiofrequency technology into the trillions of cycles per second, or Terahertz (THz), range. With the Terahertz (THz) Electronics Program, however, DARPA has begun to make it possible.

Military Airborne Collision Avoidance System Market to Grow 4.5% by 2020

Research and Markets has announced the addition of the "Global Military Airborne Collision Avoidance System Market 2016–2020" report to their offering. The report forecasts the global military ACAS market to grow at a CAGR of 4.5% during the period 2016–2020.

IoT Communication Protocol Market Worth $15.8B by 2022

The market size, in terms of value, is expected to grow from USD 11.44 billion in 2015 to USD 15.80 billion by 2022, at a CAGR of 4.7% between 2016 and 2022.

World Record: 248,000 Industrial Robots Revolutionizing the Global Economy

The worldwide sales of industrial robots achieved a new record number of 248,000 units in 2015, according to the International Federation of Robotics (IFR).

New Detector Overcomes Key Challenge in Using Light for Wireless Communications

Today's high-speed wired communication networks use lasers to carry information through optical fibers, but wireless networks are currently based on radio frequencies or microwaves.

Automotive LED Market has a Bright Future with the Rise of Replacement Products

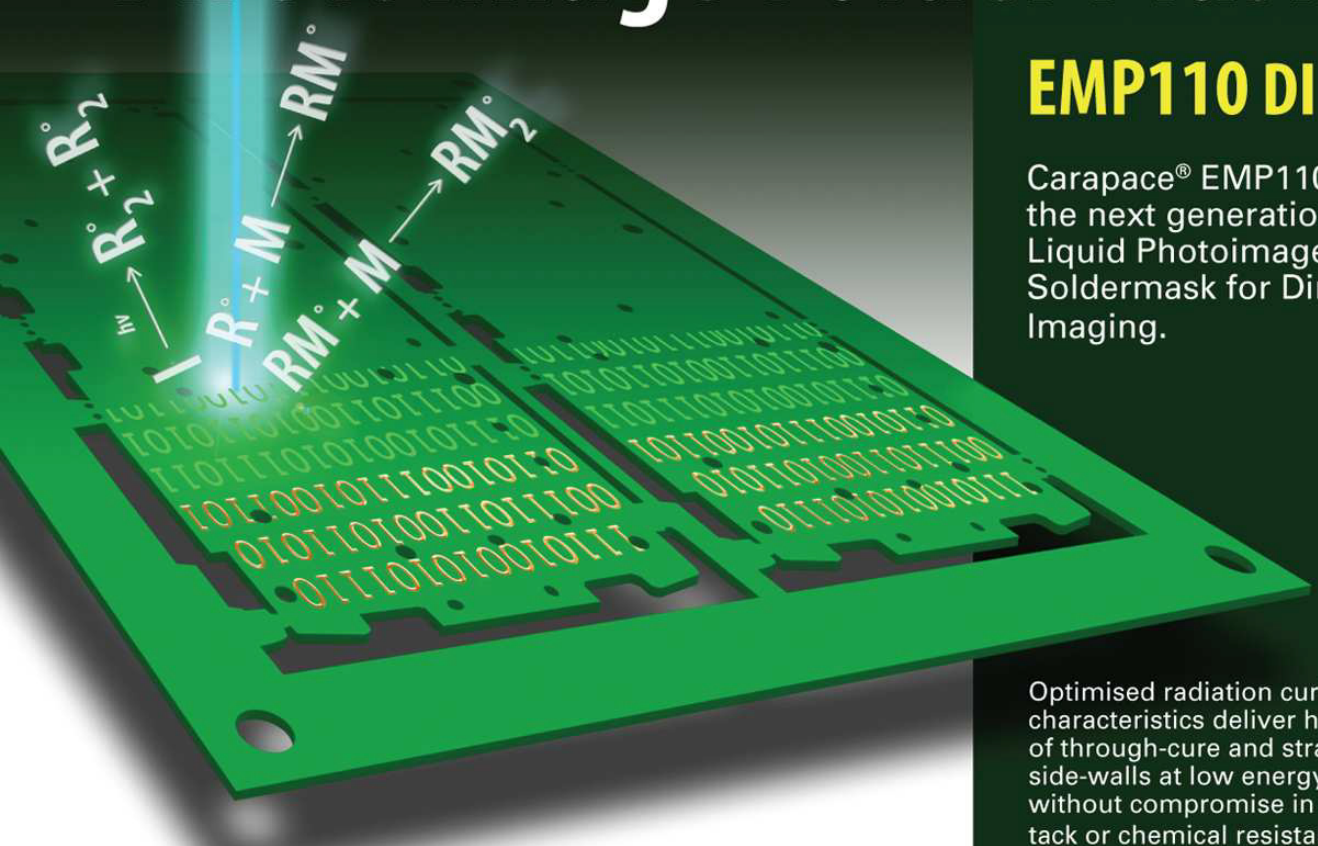
The value of the global LED market has been growing at a crawling pace as demand for backlight LEDs contracts. Automotive LEDs, however, are one of the few sizable application markets that are developing rapidly along with other smaller markets for niche industrial LED products.

ELECTRA

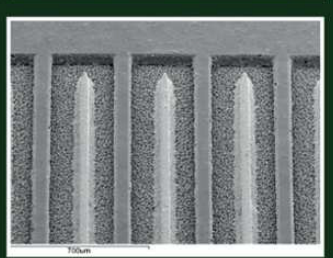
Direct Image Solder Mask

EMP110 DI

Carapace® EMP110 DI is the next generation in Liquid Photoimageable Soldermask for Direct Imaging.



Optimised radiation curing characteristics deliver high levels of through-cure and straight side-walls at low energy levels without compromise in surface tack or chemical resistance.



- **Suitable for Laser and LED Direct Image Systems**
- **Versions available for single, dual and multiple wavelengths**
- **Low Exposure Energy**
- **Fine Solder-dam Resolution (50 micron, 2mil)**
- **High Chemical Resistance to ENIG & Sn Processes**
- **Screen-print and Spray**
- **Halogen-free, RoHS and fully REACH compliant with no TGIC or other SVHC content.**

www.electrapolymers.com

The European Space Agency on Reliability

by Barry Matties

Stan Heltzel is a materials engineer working for the European Space Agency, and he is tasked with the job of procuring and qualifying PCBs that end up in satellites. I met with Stan at EIPC's Summer Conference to discuss his presentation on qualifying a fabricator, his role at the ESA, and updating of space standards.

***Barry Matties:** Stan, please start by explaining a little about what you do.*

Stan Heltzel: I'm a materials engineer at ESA and my responsibility is the qualification of printed circuit boards and the development of technology and qualifying the technology for space projects.

***Matties:** Everything you do is all about high reliability?*

Heltzel: Yes.

***Matties:** You gave a presentation today on the qualification process, and talked a lot about the various approaches, the testing, and so on. What did you want the audience to take away from your presentation?*

Heltzel: First of all, it's the interaction between ESA, the national space agencies, contractors for satellite integration, and PCB manufacturers. They are working together in various organizational bodies and trying to address problems that no one organization could ever solve, so we're doing that jointly. For instance, the nicest example I can think of is the cleanliness of the raw materials where PCB manufacturers were not in a position to ask for different quality standards from laminate suppliers until we got together as a group and addressed it all together.

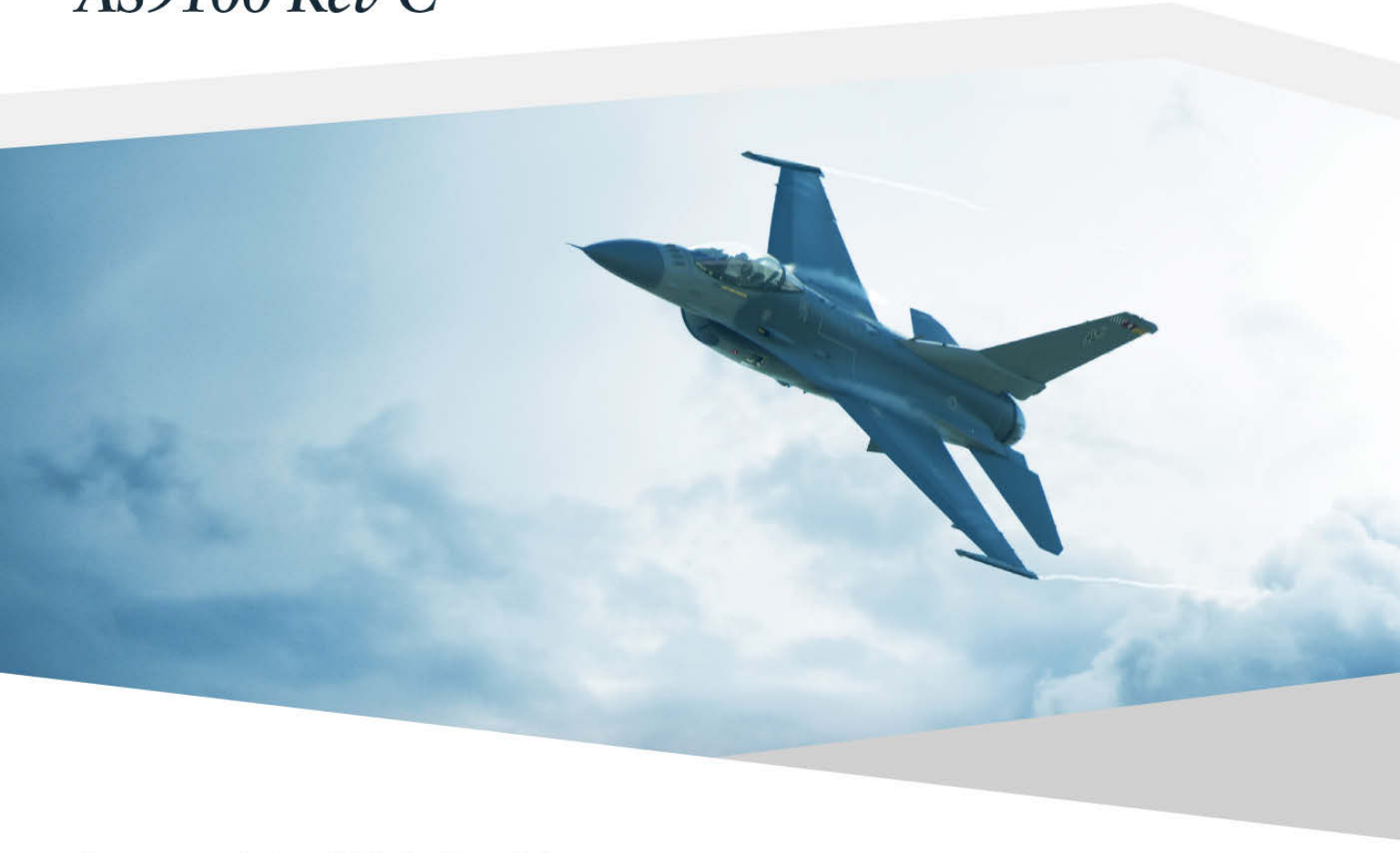
***Matties:** Today you were showing some images of where you had peeled back the copper and found foreign debris in the mix. Is that something that's commonly found, or is that an anomaly?*

Heltzel: I think we should consider it an anomaly. We're used to good quality materials that we get from laminate suppliers, but what we want is to be able to set the bar so we are able to distinguish between bad batches, if they oc-

Delivering the highest quality standard for aerospace & defense

Fully certified AS9100 Rev C

ventec
INTERNATIONAL GROUP
騰輝電子



Do you manufacture PCBs for the aviation, space and defense industries? With Ventec's AS9100 Rev C certification, you can rely on our fully accredited supply chain for high reliability laminates and pre-pregs.

From manufacture through fabrication and delivery, Ventec's high quality product portfolio of polyimides, high reliability FR4 and our latest 'tec-speed' range of high speed/low loss materials are all covered by the accreditation.

Ventec - your strategic partner for safety critical supply chains

Ventec International Group
179 Ward Hill Ave,
Ward Hill, MA 01835,
United States
T: 978-521-9700
E: ordersCA@ventec-usa.com
Follow @VentecLaminates

www.**venteclaminates**.com


Stan Heltzel

casionally happen, and good ones. I don't think it makes any sense to have inspection that does not discriminate between good clean materials and not-so-clean materials.

***Matties:** How do you even go about qualifying a fabricator to be a producer? What is it that you're really looking for?*

Heltzel: What's important is that the qualified PCB manufacturer is committed to supply to the space industry for the long term. Then there are all kinds of technical requirements that are now written down in our standards, and for us to meet those technical requirements. Of course, some of these requirements are technically challenging to meet them 100% of the time.

When we face problems, the really important thing is that we don't give up. We should always have the mindset of continuous improvement and commitment to the space project to sort out any problems if they occur. I think that is one of the most important points to ESA.

***Matties:** Do you embed engineers from your agency into fabricator shops?*

Heltzel: No, we do not.

***Matties:** You rely totally on their engineering and support to solve problems?*

Heltzel: I audit PCB manufacturers. Also, the whole industry chain is involved with the quality assurance from PCB manufacturers, but inside those we do not have embedded engineers.

***Matties:** What is the greatest challenge in this whole process, from design to a completed board?*

Heltzel: Mostly pricing, I'm afraid.

***Matties:** Really? I thought you had all the money you needed. It's a government agency, right?*

Heltzel: Let's say the space project has a high budget, and it is an expensive business. But still, ESA does not buy the PCBs. It's the industry buying the PCBs and the industry is pushing budget and they push back on PCB fabricators to provide a reliable PCB at a low cost. That is a conflict.

***Matties:** That sure is. The other aspect, of course, is that inspection is a standard step in almost every process for your application, I would imagine. That's expensive, but necessary. A lot of people tell me, "If you have a process that's in stable circumstances and you're in control, you don't need inspection." It seems like that would be a mistake in this case.*

Heltzel: It would be. We inspect a lot with regards to PCB procurement and PCB qualification. We are in the process of raising the bar in terms of the level of testing that we're requiring and the number of inspections that we're requiring. That work costs money but we justify that and we can quite easily convince industry that it's just a fact that those inspections will then prevent failures later in the chain in manufacturing the product. Really, the expensive thing is when things go wrong and 20 people have to sit around the table once a week in a physical meeting to sort out the problems on schedule.

***Matties:** When things go wrong in the finished product, it's hundreds of thousands of miles away perhaps, with no chance of repair.*

THE MOST IMPORTANT POINTS IN MULTILAYER ALIGNMENT ARE

ACCURACY ACCURACY ACCURACY

DIRECT OPTICAL REGISTRATION

The only system on the market that will cut the layer to layer alignment tolerance in half of the best pin process available—and we have the customer data to prove it!

34 layer panel with thin core and subs in the final stack-up

DIS inc

DIS, Inc.
85 Hoffman Lane, Suite Q
Islandia, NY 11749

Designed, Patented and Made in U.S.A.

For more information email:
sales@distechnology.com
(631) 851-0102
www.distechnology.com

Heltzel: When I speak about when things go wrong, it's still ground-based.

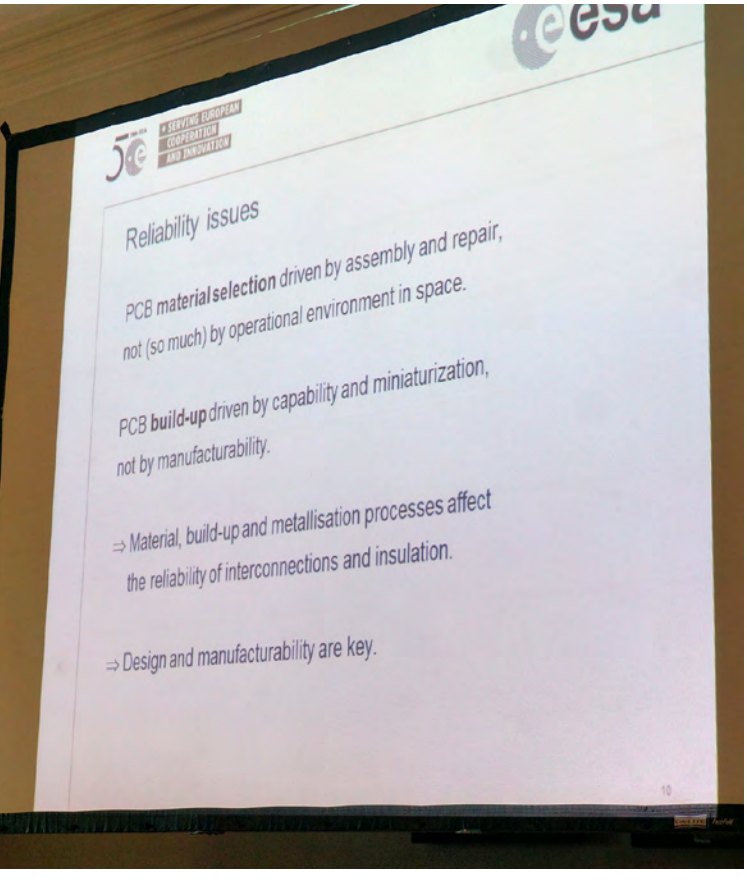
Matties: ***That's where you want to catch it.***

Heltzel: You want to catch it before that. At the PCB level, you want to catch mistakes before they're being assembled. It's very late if we are catching things when we are performing ground-based environmental testing on the unit or even worse on the satellite. Keep in mind that these tests are also pretty severe tests of any electronic equipment that is passing, like a thermal vacuum campaign on the ground is expected to do well up in space.

Matties: ***Who's doing your circuit design? Is that an internal function or is that farmed out?***

Heltzel: That's the industry's responsibility—the OEMs.

Matties: ***So you basically go in and say, "We need something that gives us this functionality."***



Heltzel: Yes.

Matties: ***They design it and they engineer it and you just buy it? Is that how it works?***

Heltzel: It's a bit more complicated than that. There are space projects, some are done by ESA, and ESA issues contracts for those space projects. Those contracts include somewhere in the supply chain that some supplier will manufacture the equipment that will eventually be built into a satellite. That equipment manufacturer is responsible for procuring the PCBs. My job is independent from those space projects. I have the general support function just for the PCB technology.

Matties: ***I see. What's the most surprising thing that you've seen in your career? The thing that made you say, "I can't believe they did this," or "We should've done it that way."***

Heltzel: I think the lesson that has been learned is the fact that the standard we've been using in space was allowed to be outdated. That causes some problems. We had to provide maintenance in the form of memoranda. It's a European industry standard, but that standard was allowed to be slightly outdated because when PCB technology increased in terms of ability, the standard was not in sync with that capability and of course with the new capabilities, you get different failure modes. The fact that this is now being revised will be a major milestone.

Matties: ***How long does it take you to revise the standard and make it become live?***

Heltzel: This one has now been running for two years.

Matties: ***So it's quite a process. It seems rather long, doesn't it? If it's something outdated to begin with...***

Heltzel: Yes, but then after that, I expect it will be up-to-date for many years. The industry in space isn't changing that quickly—it is still quick but not as quick as an iPhone.

***Matties:** Which is more powerful than our first rocket ship that went to space, I think.*

Heltzel: In terms of computing power, yes.

***Matties:** There's something odd about that. Is there anything that circuit board fabricators should know?*

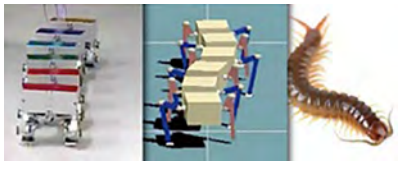
Heltzel: We rely on them to do their own quality control and we will check and jump in where needed. Also, if they need us or need a bigger expertise, if they don't have resources, we are there to support them. When I say 'we' I am talking about the full supply chain, but in the end it is their responsibility to make sure that the reliability of their product is good and they should have the resources available.

***Matties:** Thinking about the laminate, are you specifying material or do you leave that entirely up to your contractor as well?*

Heltzel: The original design for the qualified stackup is up to the industry to select, but once it's selected it's cast in stone. It gets written up in the process identification document and that cannot be changed without performing the qualification work.

***Matties:** Thanks for spending time for this interview. I appreciate it.* **PCB**

Hey Robot, Shimmy Like a Centipede



Centipedes move quickly. When one is coming directly at you, you might not care to spend a moment pondering its agility, so fear has likely been affecting our understanding of why centipedes move with such dexterity. Researchers at Kyoto University have turned to computer simulations and ultimately robotics to find an answer.

What they have uncovered is a surprising insight into the mechanics of locomotion itself, namely that taming instability—a factor that might be a disadvantage—is a key to the centipede's success.

"During locomotion, many legs are in contact with the ground to support the body against gravity and produce propulsive and decelerating forces," explains lead scientist Shinya Aoi. "These many legs are physically constrained on the ground, and this constraint can impede their maneuverability."

Centipedes overcome these constraints by harnessing instability, producing the creature's characteristic undulating movement.

"Our group developed a mathematical model of centipedes and found that the straight walk becomes unstable and body undulations appear through a supercritical Hopf bifurcation by changing the locomotion speed and body axis flexibility," continues Aoi, referring to a mathematical description of the walking system's tipping point from stable to unstable.

First with computer models and then with segmented, multi-legged robots, the team was able to replicate the centipede's movement, including the wave-like body motion, as described in a paper in the online journal Scientific Reports. Down the line, such knowledge could lead to better motion for robots.

The Many Voices Over the Past Year

The PCB007 Interview Index

Editor's Note: Here at I-Connect007, we are constantly interviewing people in the industry, from CEOs to new employees right out of college. In this issue, we present a collection of the interviews that we conducted over the past year. Alphabetized by interviewee's last name, this is a great chance to meet some of the many people in our industry who have talked with us and shared their insights, ideas and visions.

Catching up with Cosmotronic's Gary Abel
Gary Abel, Cosmotronic
November 4, 2015

IPC's Fern Abrams: Keeping up with Regulatory Matters
Fern Abrams, IPC
The PCB Magazine December 2015

Striving for One-Stop Solutions, Manz Eyes U.S. Market
Frank Baron, Manz AG
February 18, 2016

No Substitute for Experience: Dan Beaulieu on EXPO, the Industry and I-Connect007
Dan Beaulieu, DB Management
March 22, 2015

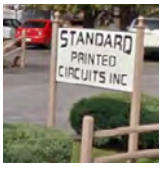
Williams and Beaulieu: Board Shops and CMs Must Communicate Better
Dan Beaulieu, DB Management Group, and Steve Williams, The Right Approach Consulting
April 5, 2016

What a Difference a Year Makes: Voxel8 and the 3D Printing Pioneering Spirit
Michael Bell, Voxel8
February 15, 2016

Catching up with Frank Bevans—the PCB Industry's Premier Photographer
Frank Bevans, Photographer
June 23, 2016

How North American Fabricators Benefit from Attending HKPCA
Peter Bigelow, IMI Inc., and Alex Stepinski, Whelen Engineering
February 4, 2016

The Opportunities for Plasma Processing
Andre Bodegom, Adeon Technologies B.V.
December 4, 2015



Catching Up with Standard Printed Circuits' Bob Bormann
Bob Bormann,
Standard Printed Circuits
March 24, 2015

ECI on the Many Benefits of Automated Process Lines
Peter Bratin, ECI Technology
February 11, 2016

Catching up with… Author David A. Brock
David A. Brock, Author
July 7, 2016

INSULECTRO

INSULECTRO now is the exclusive distributor for Oak-Mitsui Copper and ABC!

WHAT'S NEXT INSULECTRO?

Exclusive Distributor for CAC & Oak-Mitsui

- High Temperature elongation (HTE) foils
- Very low profile (VLP) foils
- Double-treated foils
- High ductility foils

The Ultimate Copper

the technological leader in the manufacture of high-quality electrodeposited copper foils

OAK-MITSUI

insulectro.com

Insulectro • 20362 Windrow Drive, Lake Forest, CA 92630 • 949.587.3200

The Many Voices Over the Past Year

IPC's Phil Carmichael on Asia, New Standards, and the Future
Phil Carmichael, IPC
April 15, 2016

HKPCA Executive Director Daniel Chan on the Upcoming Show and More
Daniel Chan, Hong Kong Printed Circuit Association (HKPCA)
November 25, 2015

HKPCA & IPC Show 2015 Kicks Off in Shenzhen China
Daniel Chan, Hong Kong Printed Circuit Association (HKPCA)
December 2, 2015

The Value of IPC's Validation Services
Randy Cherry, IPC
The PCB Magazine
December 2015

Jason Chung Offers the Asian Perspective and Explains Ventec's Aim to Differentiate
Jason Chung, Ventec
May 11, 2016

Mutracx: First Install Achieves CAM to Etch in 5 Minutes
Peter Coakley, Mutracx
The PCB Magazine October 2015

Martin Cotton Explains Ventec's Plan for Growth
Martin Cotton, Ventec
April 8, 2016

Walt Custer Elaborates on his Annual IPC APEX EXPO Forecast Presentation
Walt Custer, Custer Consulting Group
March 31, 2016

A Conversation with Walt Custer: Market Report
Walt Custer, Custer Consulting Group
May 16, 2016

ESI: Drawing on a Deep History to Create a Vision for the Future
Michael Darwin, Electro Scientific Industries Inc.
April 4, 2016

New Leadership at MuTracx
Arnoud De Geus, Sioux Group
October 8, 2015

For Pioneer MuTracx, the Future Looks Bright
Arnoud de Geus, Sioux Group
December 7, 2015

Catching up with Nujay Technologies: A Conversation with Urvi and Nesh Dholakia
Urvi and Nesh Dholakia, Nujay Technologies
October 16, 2015

Carmakers Now Embracing Plasma Treatment
Jonathan Doan, Nordson MARCH
The PCB Magazine September 2015

An Examination of the China Market with Laminate Supplier Shengyi
Jack Dong, Shengyi Technology
December 18, 2015

Catching up with Winonics' Mark Eazell
Mark Eazell, Winonics
December 11, 2015

WKK's Hamed El-Abd on the Current State of China
Hamed El-Abd, WKK
January 18, 2016

The Supplier View: An Interview with Hamed El-Abd
Hamed El-Abd, WKK
March 30, 2016

Are There Advantages to Changing Your Registration System?
Tony Faraci, DIS Inc.
March 22, 2015

A Conversation (and Day) with Joe Fjelstad
Joe Fjelstad,
Verdant Electronics

Part 1: May 3, 2015
Part 2: May 5, 2015
Part 3: May 6, 2015
Part 4: May 6, 2015
Part 5: May 8, 2015

ATG's Swen Fleischer on China: The Market, the Future and CPCA
Swen Fleischer, ATG Luther & Maelzer
April 14, 2015

An Interview with Gardien Group's Jason Fraser
Jason Fraser, Gardien Group
May 24, 2015

Fighting the War on Failure with Supplier Analytics: An Interview with Fane Friberg
Fane Friberg, Cephas
The PCB Magazine August 2015

Printing PCBs... In Your Office!
Simon Fried, Nano Dimension
The PCB Magazine February 2016

Japan's MicroCraft on the Japanese Market, Technology, and More
Soichiro Fujita, MicroCraft
February 22, 2016

Optima Discusses Advantages of In-line AOI
Gillad Galor, AOI maker Optima Ltd
January 15, 2016

The Pinless Registration System, Explained by DIS' Bernd Gennat
Bernd Gennat, DIS Inc.
February 17, 2016

Catching up with David Gerber
David Gerber, Author
January 26, 2016

Specialist Coatings for Specialist Applications
Josh Goldberg, Taiyo America
December 8, 2015

Patricia (Patty) Goldman Inducted into IPC Hall of Fame
Patty Goldman, I-Connect007
March 18, 2016

Navigating the Global Materials Supply Chain: A Roundtable Discussion
Mark Goodwin and Jack Pattie, Ventec; Michael Keuthen, Schoeller Electronics; and Bob Willis, National Physics Laboratory
December 3, 2015

What You Probably Don't Know About NASA
George Gorospe, Ames Research Center, NASA
June 13, 2016

Catching up with... Fastrak Manufacturing's Phil Guzman
Phil Guzman, Fastrak Manufacturing
April 1, 2016

Atotech's Uwe Hauf's View of the Global Electronics Manufacturing Market
Uwe Hauf, Atotech
April 12, 2015

Isola Launches Low-loss Laminate for 100 GB Ethernet Apps
Fred Hickman, Isola
March 30, 2015

Koen Hollevoet Explains Rogers' New PEEK-based Material for Extreme Temps
Koen Hollevoet, Rogers Corporation
April 28, 2016

In Their Words: Hong Kong Printed Circuit Association (HKPCA)
Hong Kong Printed Circuit Association (HKPCA)
The PCB Magazine December 2015

The Many Voices Over the Past Year

An Optical Update with TTM
Marika Immonen,
TTM Technologies
April 16, 2015

ESI's New Gemstone Changing the Rules for Laser
Mike Jennings, ESI
April 7, 2015

Interview with Dean Kamen, Segway Inventor and Founder of FIRST
Dean Kamen, FIRST (For Inspiration and Recognition of Science and Technology)
April 25, 2016

Catching up with...Tom Kastner
Tom Kastner, GP Ventures
February 11, 2016

The Importance of Harmonized Standards that Benefit All
Sean Keating, Amphenol
The PCB Magazine May 2015

Bernie Kessler: Pioneering Spirit Then and Now
Bernie Kessler of Bernard Kessler & Associates
April 7, 2015

Schmoll Keeping an Eye on the Future—and on LDI
Thomas Kunz, Schmoll Maschinen GmbH
March 16, 2015

A PCB Manufacturers' Representative Point of View
David Lanagan, Mandalay Technologies
July 7, 2016

How to Make Mass Lam Outsourcing Successful
Steve Law, Option Technologies
June 15, 2015

Catching Up with Circuit Connect
Bob Lazzara and Paul LaLiberte, Circuit Connect
November 19, 2015

The Newest Flex Shop in the U.S.
Mark Lencioni, Lenthor Engineering
July 7, 2016

Manz: A Total Process Solution
Alex Liu, Manz AG
April 1, 2015

Catching Up with Vector Fabrication's Quang Luong: Here Comes Vietnam!
Quang Luong, Vector Fabrication
June 24, 2015

The Future of Computer-Integrated Manufacturing in the PCB Industry
Alex Liu, Manz China Souzhou Ltd
May 9, 2016

Rogers: 'To PIM-test Our Material, You Have to Build a Circuit'
Tony Mattingly, Rogers Corp.
April 21, 2016

Interim CEO Jeff McCreary Discusses Recent Changes at Isola
Jeff McCreary, Isola
October 21, 2015

Catching up with Gardien's Rick Meraw
Rick Meraw, Gardien
December 17, 2015

Working for the Industry: Renee Michalkiewicz, NTS
Renee Michalkiewicz,
NTS Baltimore
The PCB Magazine December 2015

Uyemura USA's George Milad on Updating Final Finish Specs
George Milad, Uyemura USA
The PCB Magazine December 2015

A Conversation with IPC President and CEO John Mitchell
John Mitchell, IPC
The PCB Magazine December 2015

DELIVERING COMPETITIVE ADVANTAGE FOR
PCB SYSTEMS DESIGN

FIND OUT MORE ON
www.mentor.com/pcb

Mentor Graphics®

The Many Voices Over the Past Year

A Conversation with IPC's President John Mitchell
John Mitchell, IPC
January 20, 2016

IPC President John Mitchell on the Past, Present, and Future, Pt. 1
John Mitchell, IPC
April 7, 2016

IPC President John Mitchell on the Past, Present, and Future, Pt. 2
John Mitchell, IPC
April 8, 2016

Ken Moffat of American Standard Circuits Talks Aerospace Accreditation, DfM, and More
Ken Moffat, American Standard Circuits
June 17, 2016

Morgan and Starkey on CPCA 2016, Automation, and the Upcoming ECWC14
Alun Morgan, European Institute of Printed Circuits (EIPC)
April 12, 2016

Alun Morgan on the EIPC Summer Conference 2016
Alun Morgan, EIPC
June 17, 2016

EIPC: Furthering the Reach of Electronics Industries in Europe
Alun Morgan, European Institute of Printed Circuits (EIPC)
The PCB Magazine December 2015

Multilayer's Viny Mulani on what it Takes to Survive Offshoring
Viny Mulani, Multilayer Technology
June 3, 2016

MacDermid's Research Team Talks New Cyanide-free Immersion Gold at SMTAI
Jun Nable and Cherry Santos, MacDermid
December 14, 2015

Bob Neves Receives Award, Talks Bergman Tribute
Bob Neves, Microtek Laboratories China
March 5, 2015

Growing Their Portfolio: Camtek's One-Stop-Shop in Functional Inkjet Technology
Dr. Boaz Nitzan, Camtek Ltd
April 20, 2015

A Conversation with Joe O'Neil, the New Chairman of the IPC Board of Directors
Joe O'Neil, IPC
April 11, 2016

DuPont, Taconic and PFC Team Up For High-Speed Flex
Glenn Oliver, DuPont; Tom McCarthy, Taconic; and Steve Kelly, PFC Flexible Circuits
March 1, 2016

PE on 3D Objects
Mike O'Reilly, Optomec
May 19, 2016

Insulectro Alliance with Freedom CAD
Ken Parent, Insulectro
April 28, 2015

Accomplish Change Together (ACT)
Ken Parent and Jason Marsh, Insulectro
July 31, 2015

Catching up with Sunrise Electronics' Ashok and Jigar Patel
Ashok and Jigar Patel, Sunrise Electronics
July 12, 2016

Big Strategies for Success
Pratish Patel of Electronic Interconnect
The PCB Magazine September 2015

Recruiting for Growth – An Interview with Jack Pattie, President & CEO of Ventec USA
Jack Pattie, Ventec USA
November 3, 2015

Ventec International Group Expands North American Focus
Jack Pattie and Chris Alessio, Ventec USA
April 27, 2016

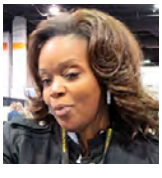
An Update on the Rogers Material Supply Line
John Pavlak, Rogers Corporation
The PCB Magazine July 2015

Orbotech's Latest Technology at IPC APEX EXPO
Micha Perlman, Orbotech
May 2, 2016

Catching Up with HSIO's James Rathburn
James Rathburn, HSIO
January 11, 2016

The ESI Solution: High-Throughput Roll-to-Roll UV Laser Drilling of Thin, Flex Materials
Patrick Riechel, Electro Scientific Industries Inc.
April 13, 2016

New Management and Strategies at eSurface
Alex Richardson and Rick McCann, eSurface Technologies
June 3, 2016



Behind the Scenes with IPC Technical Conference Coordinator Toya Richardson
Toya Richardson, IPC
December 9, 2015

LPKF Launches New Laser Drilling System for Flex Printed Circuits
Florian Roick, LPKF
April 14, 2016

Bürkle Discusses Cycle Time in PCB Lamination Systems
Stefan Roller, Bürkle Process Technologies
January 15, 2016

Rex Rozario: Investing in Graphic PLC Through Technology and Capacity
Rex Rozario, Graphic PLC
April 7, 2015

Rex Rozario: The PCB Industry's True Renaissance Man
Rex Rozario, Graphic PLC
March 14, 2016

Rex Rozario, Part 2: The Beat Goes on: New Developments at Exeter, the Music Scene, and China
Rex Rozario, Graphic PLC
March 15, 2016

Rex Rozario, Part 3: The Future Beckons
Rex Rozario, Graphic PLC
March 16, 2016

Rex Rozario, Part 4: A 10,000-ft. view of his Business Ventures, the Industry, and Life
Rex Rozario, Graphic PLC
March 17, 2016

Rex Rozario Brings IPC Board Duties to a Close
Rex Rozario, Graphic plc
April 26, 2016

Graphic PLC... Are the Rumors True?
Rex Rozario, Graphic PLC
June 22, 2016

ESI Taking Laser-Based Tooling to New Levels
Chris Ryder, ESI
December 28, 2015

Catching up with... PNC: Open House Planned for May
Sam Sangani, PNC
April 6, 2016

Next-Generation PCB Drilling
Andreas Schneider, LPKF Laser & Electronics
July 5, 2016

Sun Chemical Expands Range and Market Share in China
Tony Searle, Sun Chemical
April 15, 2016

A New Source for Laser Drills
Sam Sekine, Nano System Inc.
June 10, 2015

Nano System, Two Years On
Sam Sekine, Nano System Inc.
April 22, 2016

21 Years of Growth: A-Gas Electronic Materials Has 2020 Vision
Jonathan Sellars, A-Gas Electronic Materials
April 9, 2015

Global Machine and Management and First EIE Strike Important Relationship
Carl Spitko, Global Machine and Maintenance, and Jean Paul Birraux, First EIE
April 6, 2015

First EIE and CCI Eurolam on Direct Imaging, Printers and Partnership
Gregory Stoeckli, First EIE, and Lawson Lightfoot, CCI Eurolam
January 25, 2016

IEC: Celebrating 50 Years in Business
Shawn Stone, International Electronic Components Inc.
April 27, 2016

Catching up with PNC
Calvin Switzer, PNC
November 11, 2015

Catching up with KT Technical Sales: Looking at Things from the Rep's Point of View
Ken Thompson and Tim Miranda, KT Technical Sales
April 20, 2016

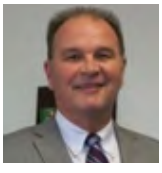
World Wide PCB on the "Purification" of the China Market, Industry Changes and More
Francis Tsoi, World Wide PCB Equipment
March 11, 2016

Francis Tsoi on the Future: What's in Store for Technology, Automation in China's PCB Industry?
Francis Tsoi, World Wide Group
April 1, 2016

Camtek Takes Inkjet Technology into the Future
Amir Tzhori, Camtek Ltd
The PCB Magazine February 2016

Integrating with Gardien Group's Roland Valentini
Roland Valentini, Gardien Group
August 26, 2015

American Standard Circuits Taking Business to New Heights
Anaya Vardya, American Standard Circuits
November 3, 2015



Zentech's John Vaughan on the Mil/Aero Sector: "It's Headed Up"
John Vaughan, Zentech Manufacturing Inc.
April 9, 2015

Multek CTO Excited About the Challenges of the Fast-Moving Wearables Market
Dr. Joan Vrtis, Multek Corp.
March 26, 2015

Orbotech's Gaby Waisman: 'The Future is Digital'
Gaby Waisman, Orbotech
April 18, 2016

New CEO Jeff Waters Outlines his Roadmap for Isola
Jeff Waters, Isola
February 8, 2016

An Interview with Gene Weiner at HKPCA
Gene Weiner, Weiner International Associates
January 19, 2016

Reliability and Harmonization of Global Standards at Forefront of EIPC Efforts
Michael Weinhold and Alun Morgan, European Institute of Printed Circuits (EIPC)
March 18, 2015

The Importance of Choosing Equipment Suppliers Carefully and Investing in the Future
Jianming Wu, Burkle China
April 7, 2016

In Their Words: The China Printed Circuit Association (CPCA)
Jin Zhang, China Printed Circuit Association (CPCA)
The PCB Magazine December 2015

The Launch of a Lifetime: Catching Up With Barry Lee Cohen
Barry Lee Cohen, Launch Communications, May 2, 2016

What a Long, Strange Trip It's Been—and It's a Long Way from Being Over
Harvey Miller
February 6, 2016

RealTimewith... Video Interviews
Check out the many, many video interviews conducted over the past year by clicking here or on the individual links below:

IPC APEX EXPO 2016
CPCA 2016
Productronica 2015
SMTAI 2015
CPCA 2015

Nagji Sutariya, Industry Veteran and Beloved Figure, Passes Away

The electronics manufacturing industry is saddened to learn of the recent passing of Nagji Sutariya, founder and president of Saturn Electronics Corporation. As many were aware, Nagji had been battling brain cancer for the past seven years, beating it four times.

Nagji was born in India on March 11, 1945; in 1969, he moved to the United States to attend the Michigan Technological University, which he graduated from with a Master's degree in engineering in 1971.

Among many other things, professional and personal, Nagji was a dedicated business owner, loved by everyone, customers and competitors alike. He financed several other PCB shops throughout the United States. He was also a generous philanthropist. From the Saturn Electronics website:

"Nagji Sutariya spearheaded the efforts to build a world-class hospital in a rural area of India serving 450 villages and millions of travelers on the adjacent highway with no previous immediate access to medical or dental care. The hospital was completed in March 2006 and now caters to anyone free of charge inclusive of medications and regardless of the ailment. This effort was funded by Mr. Sutariya and his generous donors."

Funeral services were held on Thursday, July 28 at the Ford Community and Performing Arts Center, in Detroit, Michigan. Many speakers touched on his many accomplishments in business along with his honesty and generosity.

Nagji's two sons, Yash and Raj, remain as working owners of Saturn and plan to continue embracing the commitment to customer satisfaction their father instilled in them.

Building Reliability into the PCB, Part 2

by Michael Carano
RBP CHEMICAL TECHNOLOGY

In Part 1 of this column on reliability, I presented the common PTH failures encountered when reliability is less than robust. PTH reliability is influenced by several factors including the quality of the PTH after drilling, plating thickness and plating distribution in the PTH. In this column, I will present additional factors, including the Coffin-Manson model in the context of understanding reliability failures.

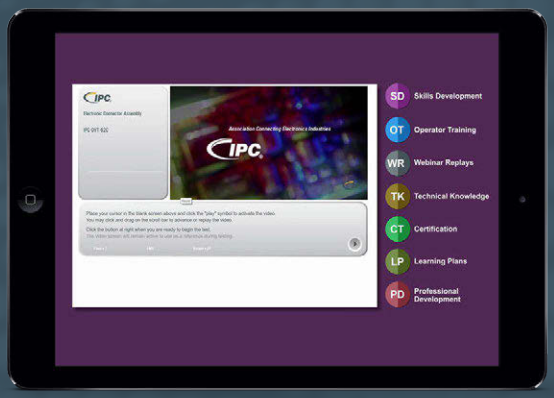
Explaining the Failures

Many engineers have been in this situation: Process audits are completed. Personnel who have direct influence are properly trained. Analytical controls are in place. Great! Yet there are boards that are not meeting specification (IPC 6010 Series, IPC 600, IPC 610). And now the team has to deal with the rejects and provide corrective actions.

In the case of the PTH failures presented previously, reliability experts have long sought to frame and predict these failures with different models. One popular model is the Coffin-Manson model. This model functions on the premise that cycles to failure in a PTH is predicated on the resistance of the plated copper to fatigue and the overall strain imposed on the copper during the thermal cycles[1]. For ductile materials such as copper, the majority of the strain is plastic strain. While the thermal expansion of individual materials is reversible, the majority

Get a Whole New EDGE

Introducing **IPC EDGE** Online Learning from IPC



IPC, the market leader for standards and certification in the electronics industry, is excited to introduce our new online learning platform—***IPC EDGE***. From the executive office to the factory floor, ***IPC EDGE*** provides the critical education it takes to excel in the electronics industry. ***IPC EDGE*** will support your learning goals and provide knowledge you can apply to your work today.

With *IPC EDGE* you can also:

- Prepare for Certified IPC Specialist (CIS) certification, the most recognized certification from IPC in the electronics industry
- Get highlights of the latest revisions and updates to IPC's most popular standards
- Learn about many other electronics industry-focused topics

IPC EDGE has you covered 24/7 from your desktop, laptop or tablet!

Get your competitive *EDGE* ... visit www.ipc.org/edge today and start learning!

SEE IPC EDGE IN ACTION—WATCH OUR VIDEO ON OUR YOUTUBE CHANNEL: IPCASSOCIATION.

IPC EDGE

IPC EDGE Online Education and Training for the Electronics Industry

of the plastic strain that is found in the copper PTH barrel is not reversible.

The PTH acts like a rivet, which resists this expansion, but the copper barrel is stressed and may crack, causing electrical failure. There is increasing strain on the barrel associated with a high-temperature excursion. Failure may occur in a single cycle or may take place by initiation and growth of a fatigue crack over the course of a number of cycles. For high-aspect-ratio through-holes subject to repeated thermal shocks from room temperature to solder reflow temperatures (220–250°C) during board fabrication (e.g., hot-air solder leveling) and assembly (reflow, wave soldering, rework), it is not uanheard of to encounter failures after 10 or fewer of these thermal cycles. While the Coffin-Manson model works fairly well in explaining failures, it does not take into account several other issues that may affect PTH reliability. These are:

- Poor copper plating distribution in PTH
- Poor leveling leading to thin areas of copper plating (Figure 1)

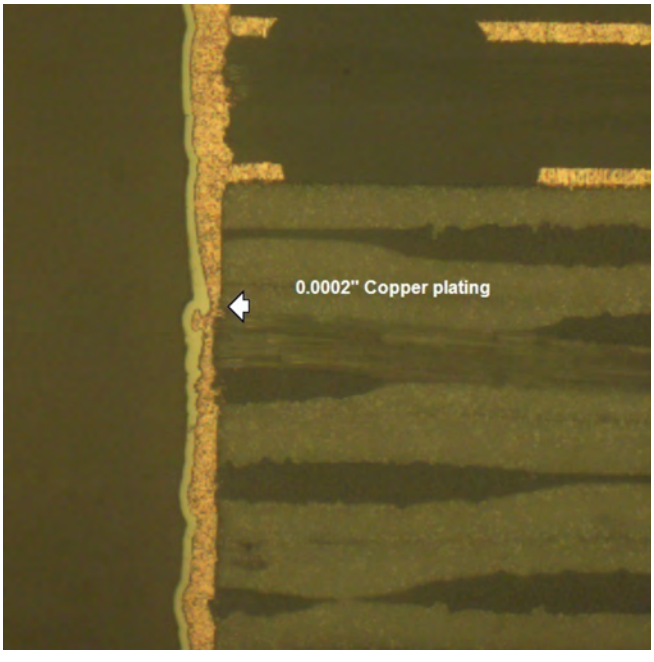


Figure 1: Inadequate leveling leading to thin plating in the PTH (Source: IPC, Bannockburn, Illinois).

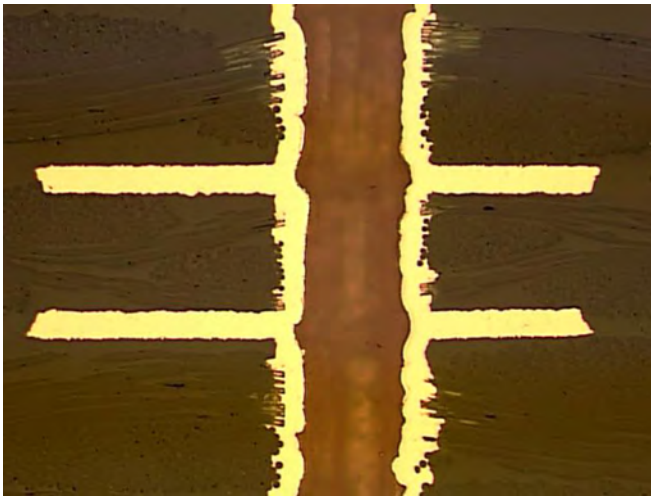


Figure 2: Rough and/or poorly drilled PTH. Glass fiber protrusions are evident.

- Glass fibers protruding from poor drilling (Figure 2)
- Resin interactions with glass weave
- Changes in copper grain structure from lot to lot
- Rough plated copper
- Copper ductility variations from lot to lot, board to board

The majority of the issues listed above can be controlled while others depend on other factors such as circuit board design, resin material properties and copper foil thickness, to mention a few. Process design, robust chemical processes and equipment will also influence the reliability of the PTH through various interactions.

Protruding glass fibers (Figure 2) affect PTH reliability in several ways. As an example, poor drilling practices cause the glass fiber bundles to be ripped out from the hole wall. These fibers then can act as a high spot leading to poor plating distribution within the hole wall. These areas are stress points that can lead to PTH failure during thermal loading.

Additionally, a critical concern with respect to PTH reliability is the wedge void. Obviously, there are a number of possibilities leading to this type of defect. First, one should note (although it is not exclusive) that the wedge originates at the B-stage side. The mechanism of the "wedge void" formation discussed below will illustrate

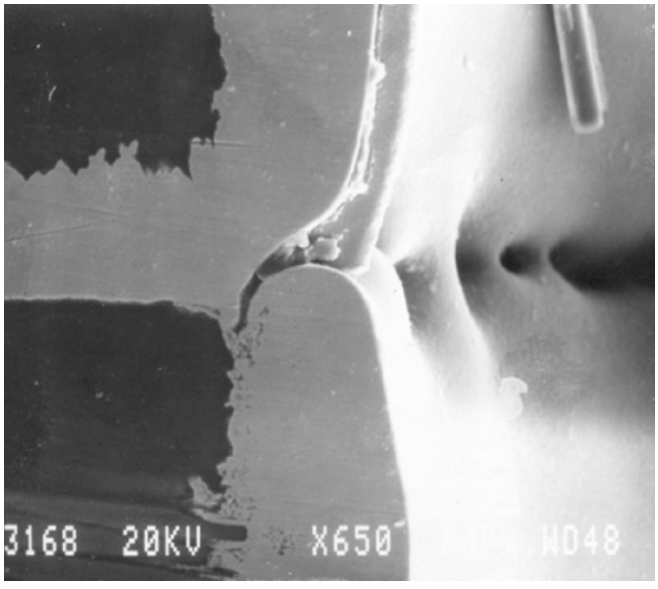


Figure 3: Wedge void (Photo courtesy of Dr. Karl Dietz, formerly with DuPont).

the significance of the defect and the genesis of a plating void.

One often overlooked cause of wedge voids is the drilling operation. During the drilling operation, vibration of the drill bit can create microfractures in the copper/dielectric interface. Of course, dull drill bits, excessive chip loads, extremely high heat generation during drilling and poor quality entry and back-up material will increase the likelihood of a wedge void. Figure 3 shows an extreme example of a wedge void caused mainly by poor drilling practice. (Note evidence of gouging in the hole wall and turn-out glass bundles.) This type of drilling will generate significant heat leading to the opening of the interface between the B-stage and the copper. Copper plating subsequently is unable to "level" out the wedge leading to copper plating folds. These folds typically are very thin in the plated copper and thus more susceptible to cracking from thermal stress. Keep in mind, thickness of the plated copper in the PTH does matter (Figure 4).

Summary

The integrated nature of printed circuit fabrication requires that engineers recognize the influence of up and down stream processes.

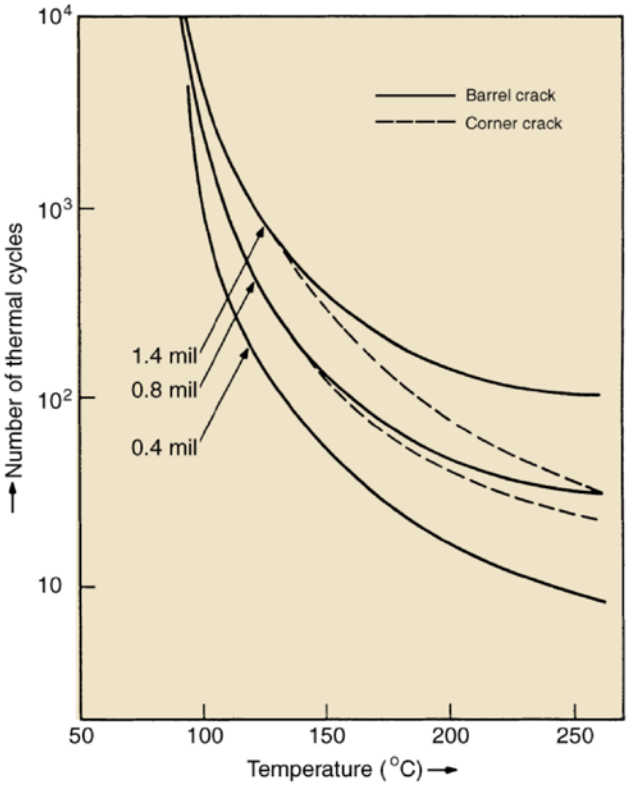


Figure 4: Effect of PTH plating thickness on number of thermal cycles to failure for thermal cycles with the indicated peak temperatures. For acid-sulfate copper and FR-4 170 Tg boards. (Source: Printed Circuits Handbook, Clyde Coombs, 2006.)

Information presented in this month's column underscores the fact. When poor drilling or lack of control of plating processes causes low copper plating thickness, long-term reliability is compromised. **PCB**

References

1. Lalanne, C. (2009). Fatigue Damage. ISTE—Wiley (ISBN 978-1-84821-125-4).

Michael Carano is VP of technology and business development for RBP Chemical Technology. He can be reached by clicking here.

MilAero007 Highlights

TTM Technologies Recognized by Raytheon Integrated Defense Systems

TTM Technologies Inc. announced that three facilities, Santa Clara (SC), Sterling (STE), and San Diego (SD) under its Aerospace and Defense/Specialty Business Unit were recognized by Raytheon Integrated Defense Systems on June 7, 2016 in two award categories for outstanding service, partnership, and excellence in engineering collaboration.

Today's MilAero Options: Outsourcing—'Everybody's Doing it' Not so True Today

There was a time, not so many decades ago, when that most commonly-stated mantra ("lower labor costs") behind offshoring printed circuit fab (and some assembly) operations, still had some case-by-case validity.

FTG Secures Licensing Agreement With EarthOne Circuit Technologies Corporation

Firan Technology Group Corporation announces a licensing agreement between FTG and EarthOne Circuit Technologies Corporation, whereby FTG will license the eSurface technology owned by EarthOne for use in the manufacture of printed circuit boards.

Teledyne Completes Sale of Printed Circuit Technology Business

Teledyne Technologies Incorporated announced today the successful completion of the sale of assets of Teledyne's printed circuit technology business (Teledyne PCT) to FTG Circuits Inc., a California corporation and subsidiary of Firan Technology Group Corporation (FTG), for US$9.3 million in cash.

Consultants to EU Publish Recommendations on RoHS Exemptions

The Oeko-Institut e.V., Institute for Applied Ecology and Fraunhofer-Institut IZM for Environmental and Reliability Engineering, consultants to the European Union (EU) Commission (Commission) recently published their recommendations on 29 requested renewals of RoHS exemptions.

Designing with Tighter Tolerances

David Ledger-Thomas is a PCB design engineer with Honeywell Aerospace. He's spent decades designing PCBs for a variety of applications, including defense, aerospace, computers, and high-performance audio. I asked David to share some of his thoughts on designing high-tech boards with increasingly finer spaces, traces and pitch.

Setting a Satellite to Catch a Satellite

The target is set: a large derelict satellite currently silently tumbling its way through low orbit. If all goes to plan, in 2023 it will vanish—and efforts against space debris will have made a giant leap forward.

OKI and Avio Ink Agreement on Transfer of PCB Business

The two companies will start negotiations on technologies and facilities transfer and a range of certifications from October 1, 2016 with a view to completing the business transfer by March 31, 2018.

Zentech's Matt Turpin on IMPACT Washington, D.C.'s Benefits

At the recent IMPACT Washington, D.C. 2016 event in Capitol Hill, Matt Turpin, CEO of EMS firm Zentech discusses with I-Connect007's Patty Goldman his expectations on the event, its importance, and how, so far, it has helped the electronics manufacturing industry in the United States.

OrbitOutlook Integrates Largest and Most Diverse Network of Space Sensors Ever to Help Avoid Collisions in Space

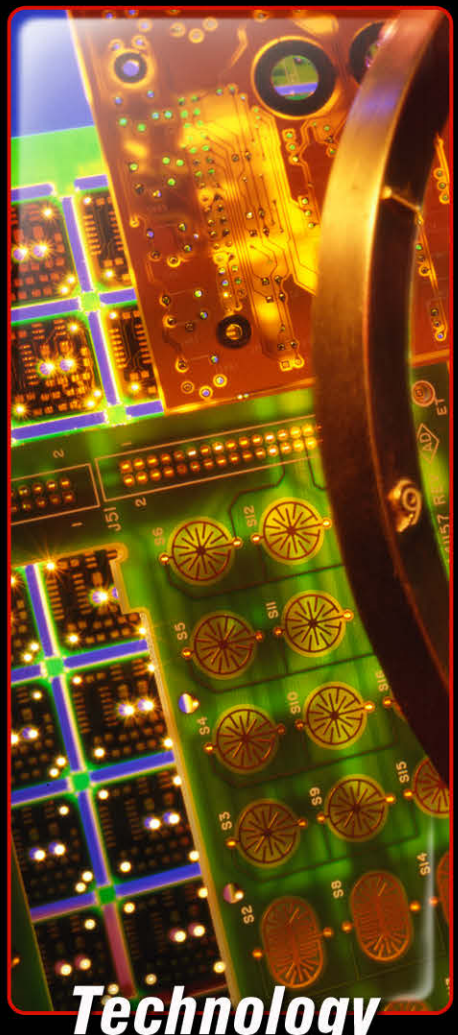
New data-gathering network aims to vastly improve the capability of the U.S. military and the global space community to make decisions about potentially hazardous space objects in near real time.

Proud Sponsor of Geek-a-Palooza
We're stronger in numbers!

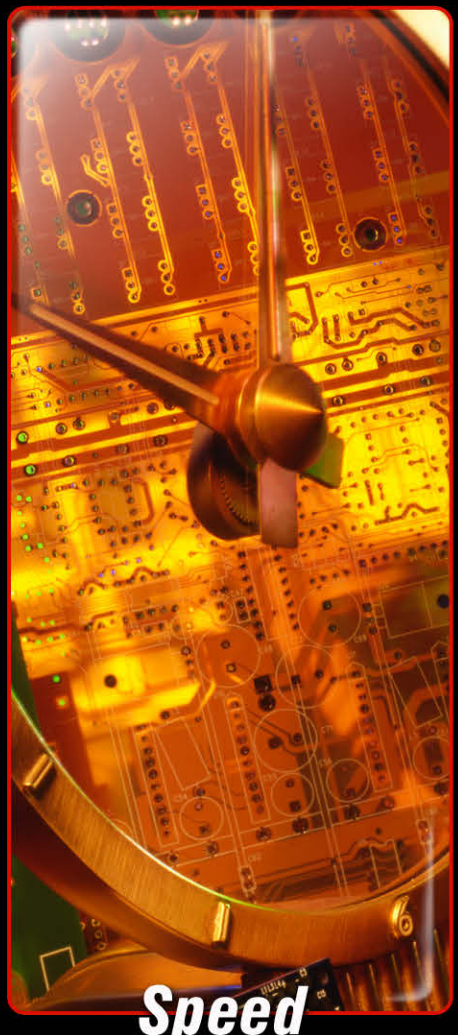
prototron.com

We Take the Time to do it Right

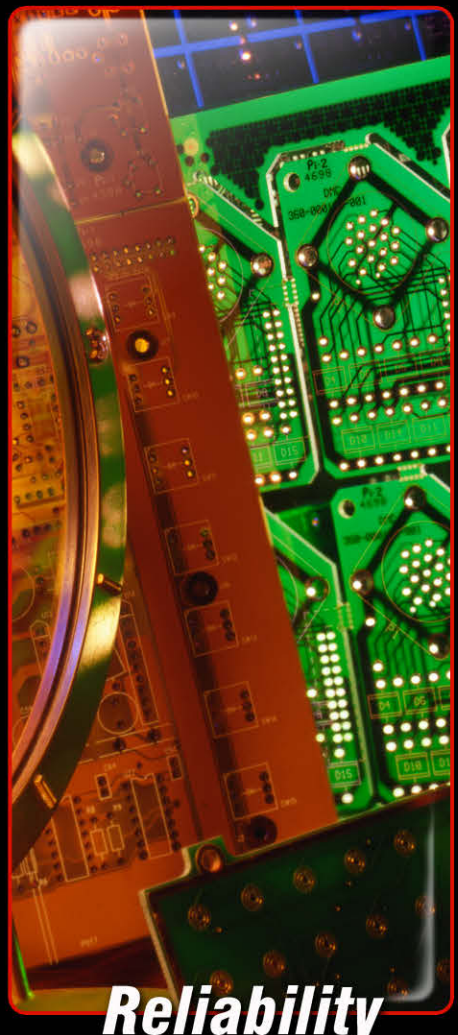
because you don't have the time to do it over.



Technology



Speed



Reliability

Prototron Circuits

High quality quick turn manufacturing, just another reason we are

America's Board Shop

Redmond Facility
15225 NE 95th Street, Redmond, WA 98052
Toll: 888.847.7686 Phone: 425.823.7000

Tucson Facility
3760 E. 43rd Place, Tucson, AZ 85713
Toll: 800.279.5572 Phone: 520.745.8515

Welcome Your Product to the Real World!

by Keith M. Sellers
NTS-BALTIMORE

The days, weeks, and (sometimes) years that go into a product's development usually are incomprehensible to the lay person. Like laws and sausages, no one wants to really understand what has gone on behind the scenes to make your "thing" a reality. They just care that your widget makes their life easier and/or more enjoyable!

With that, design and development kick off the process which eventually takes your widget from a "light bulb" idea all the way to actual production, but ultimately the testing that has been performed along the way shapes your widget into something that is reliable and thus more attractive to your customer.

At the core of this reliability-based testing is exposure of your widget to various potentially harmful environments. These potentially devastating environments help designers and engineers identify weaknesses in the product and, hopefully, identify ways to make your widget better! Generally speaking, there are two main categories of reliability exposures—mechanical and climatic. This is not to say that there are not other types of harmful environments; like an exposure to hazardous vapors or subjecting your widget to electromagnetic interference, but mechanical and climatic are the more traditional types and we'll explore those in a bit more detail in this column.

Mechanical Exposures

Generally speaking, mechanical test exposures involve hitting your widget with a large singular force or with a smaller repeated force at a high frequency. The large singular force exposure is traditionally referred to as a mechanical shock, whereas the repeated force at high frequency is traditionally referred to as vibration.

For mechanical shock, the widget is typically dropped from a specific height, onto a hard surface, with the widget checked for functionality pre- and post-exposure. Further, these types of shocks can be repeated or increased / decreased

Get the Job Done Without Breaking the Bank AND Reduce Cycle Time!

OS-253 Solder Stripper is a great way to increase productivity and minimize down-time. Long-lasting bath reduces tank shut down for cleaning / makeup. Easy waste treatment minimizes time and a built-in antioxidant allows indoor storage without having to deal with oxidized copper!

Call or email us today to place your order, you will be glad you did.

OS-253 Solder Stripper

- Economical
- High Capacity
- Fast
- Leaves Pristine Copper Substrate
- Fume Control
- Minimal-to-no Sludge

For over 45 years, our customers have come to rely on us for products that work extremely well. This, coupled with our commitment to personal service, is what makes us really outshine our competitors. We look forward to meeting you.

insidesales@os-tech.com

OS TECH

A company with chemistry

CALL: 1-888 OSTECH1 • 1-888 678 3241

in magnitude, dependent on the application or ultimate end use. From a simulation standpoint, mechanical shock testing would be the equivalent of knocking your widget off your desk, or possibly dropping your widget down the stairs.

For vibration, the widget is mounted to a table that has the ability to move a relatively small distance laterally at a high frequency. This high-speed movement exposes the widget to a vibrational stress. In terms of simulation, this type of exposure is designed to determine how the widget might perform when bolted to something like a jet engine or a clothes dryer. The widget is typically operated during the exposure, so its performance can be monitored under the real world conditions.

Climatic Exposures

This category actually has some sub-categories, as a spectrum of exposures are possible when delving into this realm. Of typical interest in the world of climatic exposure would be the following categories: temperature, humidity, and weathering.

Temperature extreme testing is always of interest in respect to reliability. This testing can be done at a single extreme temperature (elevated or sub-ambient) or with both extremes, the latter of which being commonly referred to as thermal shock testing. For thermal shock, the testing exposes your widget to temperatures well above and well below normal atmospheric conditions in a cyclic process. The extremes in these temperatures can commonly vary from 150°C all the way down to -75°C with many other options in between and above and beyond. As for the exposure itself, the cyclic nature of the testing—physically transferring the widget from hot to cold—results in a thermal stress being applied to the device under test.

During the exposure, the widget can be monitored for functionality or it can be simply tested pre- and post-exposure. In respect to simulation, cyclic thermal excursions are common for many things in the real-world and could show how your widget survives a hike up a mountain, where the temperature at elevated altitude could be very different than on the ground where you started, or how it might survive in the engine compartment of an automobile, where the temperature will cycle from ambient conditions up to the temperature of the engine itself.

Humidity testing is usually performed in conjunction with elevated temperature testing and is thus commonly referred to as temperature / humidity testing. Humidity levels of 60% to 95% RH are common for this type of testing, along with temperature levels from 30°C to 85°C. The environment is very harsh—hot and wet!—and is commonly used to explore the reliability of electronic-based widgets, such as what might happen if you took your GPS deep into the jungles of Brazil! Common tests like surface insulation resistance (SIR) and electrochemical migration (ECM) are performed in an attempt to promote an electrical issue for the widget that will cause it to malfunction. In terms of evaluation, pre- and post-exposure testing and/or monitoring while at conditions are common ways to check reliability.

Weathering type exposures are commonly centered around light effects. Commonly referred to as UV or Xenon Arc exposures, these types of tests expose your widget to a variety of light spectra…as if your widget had been left out in the sun too long! The main variables in this type of testing are the irradiance level (which can be thought of as the light intensity) and the test duration. Pre- and post-exposure testing is very common to determine what effect, if any, the light exposure has had on your widget. These tests could be related to visual characteristics, such as color and gloss, or mechanical properties, such as flexibility and tensile strength.

Ultimately, exposure of your widget to environmental stress testing is a key step in the development process of any product and the establishment of a reliability strategy to access the potential weaknesses, as a function of environment, for your widget is critical to its long-term success. **PCB**

Keith M. Sellers is operations manager with NTS in Baltimore, Maryland.

DIBBLE 4 LEADERS

THE FOUR NEW AGREEMENTS FOR SUSTAINABLE SUCCESS

Profitability is more about leadership and management than any other factor.

Any business can be transformed quickly, easily and sustainably with the proven methods we use. Contact us today for an initial consultation to learn how to get started on the path of doing the right things to allow profitablity to happen.


BOOM!

The Four New Agreements to EXPLODE PROFITABILITY

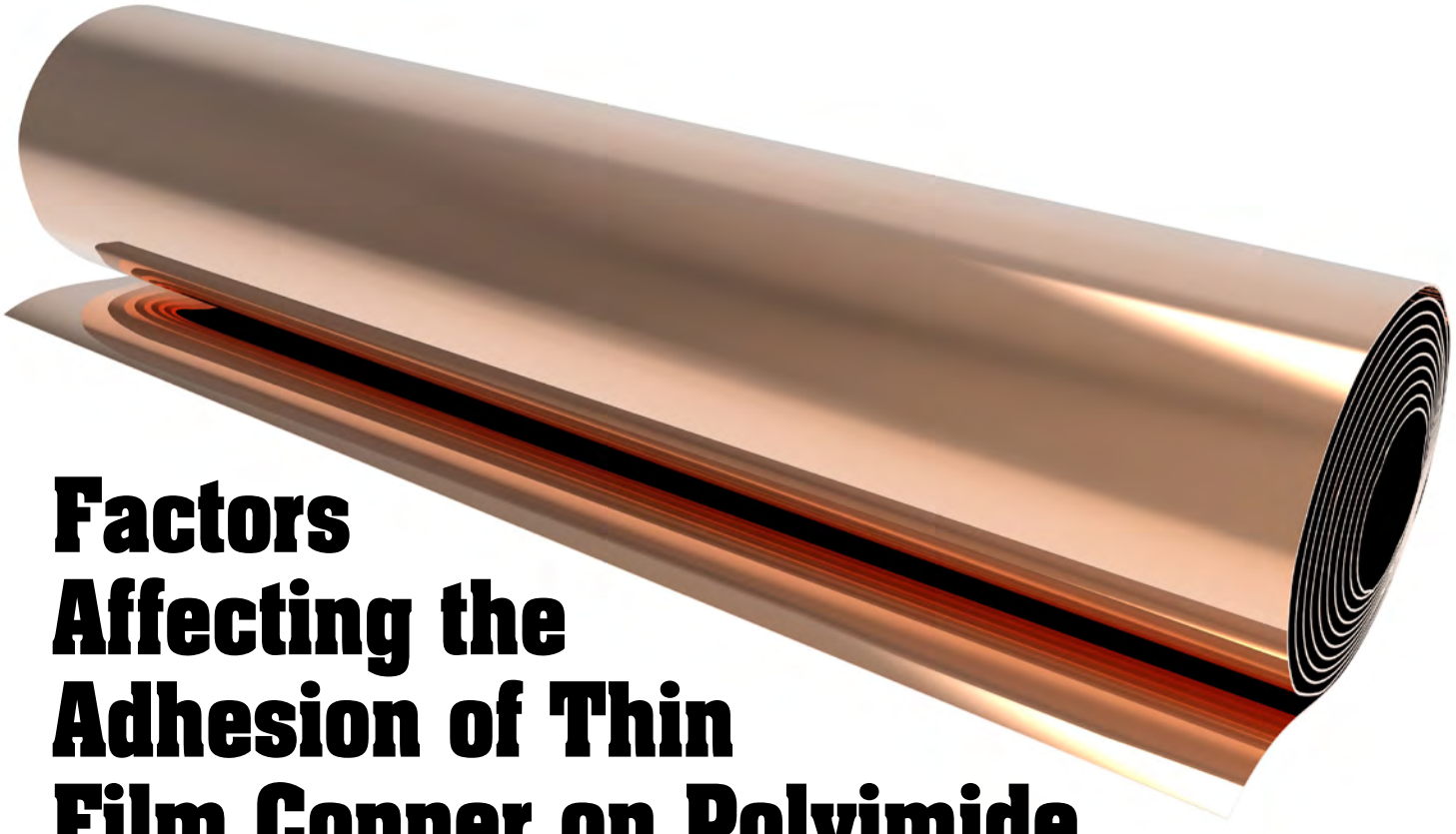
click to read

Learn more about the roadmap used to build great companies with a high level of profitabiity in this article from the March 2016 issue of **The PCB Magazine.**

For 25 years we have been doing Four New Agreements consulting and training, significantly improving businesses. This stuff really works!
—David Dibble

4newagreements4leaders.com

858-775-0722 Pacific



Factors Affecting the Adhesion of Thin Film Copper on Polyimide

by David Ciufo, Hsin-Yi Tsai and Michael J. Carmody
INTRINSIQ MATERIALS

Abstract

The use of copper foils laminated to polyimide (PI) as flexible printed circuit board precursor is a standard practice in the PCB industry. We have previously described[1] an approach to very thin copper laminates of coating uniform layers of nano copper inks and converting them into conductive foils via photonic sintering with a multi-bulb conveyor system, which is consistent with roll-to-roll manufacturing. The copper thickness of these foils can be augmented by electroplating. Very thin copper layers enable etching fine lines in the flexible circuit. These films must adhere tenaciously to the polyimide substrate. In this paper, we investigate the factors which improve and inhibit adhesion. It was found that the ink composition, photonic sintering conditions, substrate pretreatment, and the inclusion of layers (metal and organic) intermediate between the copper and the polyimide are important.

Ink factors include the intensity of photonic sintering. Better sintering leads to better cohesive strength of the nano copper layer. The ink solvent and the dispersant used to suspend the nanoparticles are significant both for adhesion and the colloidal stability of the dispersion. Pretreatment of the substrate by plasma roughening did not improve adhesion. We describe the effects of chromium and nickel interlayers which are typically used in standard foil laminates. Finally, we describe the types of peel strength testing used to assess adhesion.

Introduction

The goal of our ultra-thin film project is to create a thin (1–2 micron) film of uniform copper on flexible polyimide for application to flex circuits. Ultra-thin films allow very narrow copper lines on patterns created by photolithography, plating, and etching. Figure 1 depicts the invented process.

iNEMI MEPTEC THE NEXT GENERATION SMTA Surface Mount Technology Association

Medical Electronics Symposium 2016

September 14 & 15

Marylhurst University • Portland, Oregon

SMTA, INEMI, and MEPTEC have joined forces to again host this international conference, focusing on advances in electronic technologies and advanced manufacturing, specifically targeting medical and bioscience applications. Last year's conference attracted about 200 attendees and more than 30 exhibitors. In prior years, MEPTEC's and SMTA's conferences were held in Phoenix, Arizona and Milpitas, CA, respectively, drawing technology experts, entrepreneurs and service providers that work in this niche technology space. Typical applications within this space involve implantable defibrillators, neurostimulators and drug delivery, interventional catheters, pillcams, ultrasound transducers, hearing aids, biosensors, microfluidics, wireless communications, as well as future diagnostic and treatment solutions that may use stretchable electronics, microelectromechanical systems (MEMS) or nanoelectromechanical systems (NEMS). ◆

TECHNICAL COMMITTEE

General Chair:
Chuck Richardson, iNEMI

Technical Chair:
Peter Tortorici, Medtronic Microelectronics Center

Don Banks, St. Jude Medical
Bette Cooper, MEPTEC
Jenny Ng, SMTA
Bob Rogers, Intel Corporation

REGISTRATION SPECIAL
Four Registrations for the Price of Three!

Marylhurst University, founded in 1893, is Oregon's oldest Catholic university, and the first liberal arts college for women established in the Northwest.

Multiple Track Topics Include:

■ **TRACK 1:**

Designs, Components and Assembly for High-Density Medical Electronics Solutions

Chair: John McNulty, Exponent Failure Analysis Associates
Co-Chair: Dale Lee, Plexus Corp.

■ **TRACK 2:**

Next Generation Microelectronics for Changing Healthcare Markets

Chair: Celeste Fralick, Intel Corporation
Co-Chair: Erik Jung, Fraunhofer IZM

Sponsorship Opportunities and Exhibit Spaces Available

REGISTER TODAY AT SMTA.ORG/MEDICAL

Figure 1: A process for preparing very thin copper foil on polyimide.

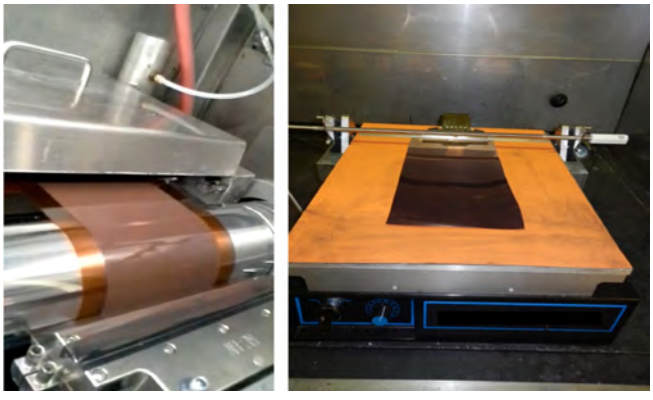


Figure 2: Two ways to coat uniform layers of nano copper ink: a) roll-to-roll coating; b) automatic Mayer bar coater.

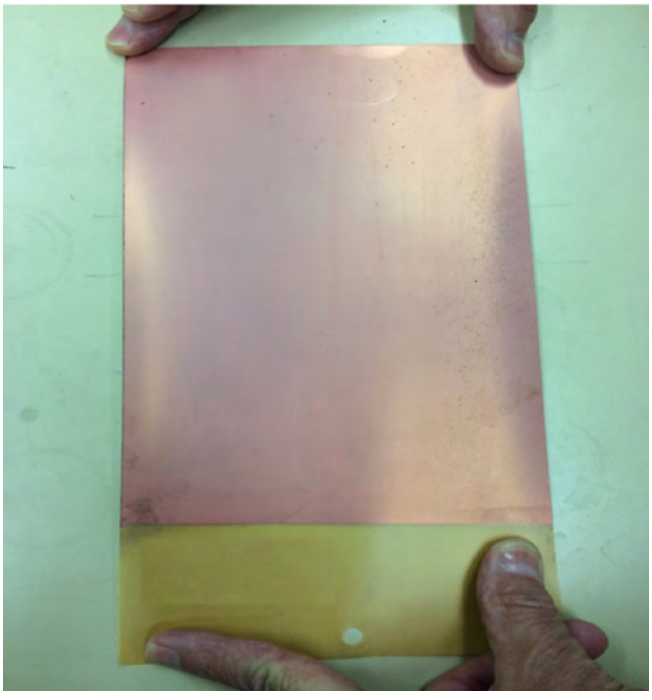


Figure 3: A uniformly sintered Mayer bar coating of nano copper ink on an epoxy BT rigid substrate.

A very thin layer of nano copper ink is created on polyimide by coating techniques. A picture of a coating made in a roll-to-roll (R2R) manner on a slot die coating machine is shown at left in Figure 2. After drying the ink (done by air impingement here), a film of copper nanoparticles in dispersant/binder of about 0.5–1.0 microns thick is produced. This film can be turned into a continuous film of conductive copper metal by photonic sintering with a flash lamp system. There have been developed special R2R machines with moving conveyors, multiple flash bulbs, and algorithms to control and interleave flashes to give uniform sintering of the coating[2]. On the other hand, the lab scale coatings are made by using an automatic Mayer bar coater (Figure 2, right), dried in a vacuum oven, and sintered by a single flash lamp unit it the lab. The resulting copper films are less uniform than the ones generated by the R2R process, and banding where the flashes overlap is usually observed. Alternatively, a uniform conductive copper film can be generated by sintering by immersing in a formic acid/argon reducing atmosphere at 190°C for an hour (Figure 3).

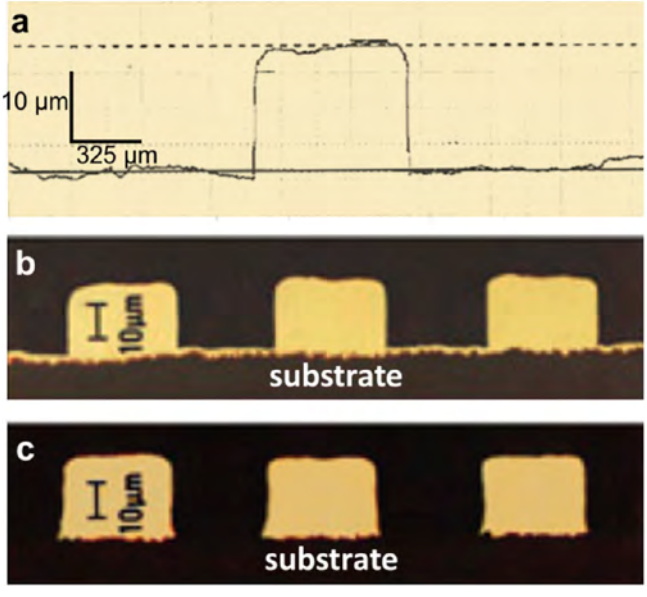


Figure 4: A profilometry graph of an etched trace (a), and cross section photographs of plated (b) and etched (c) traces.

This conductive layer can be plated in an electroplating bath to give copper anywhere from 3–50 microns of copper. As stated earlier,

1–2 microns thick would be best for masking off the sintered layer, electroplating a pattern, removing the mask, and etching off the unwanted copper. Since etching is isotropic, the sides of the plated traces will etch as well at the tops, and thinner traces give less under-cutting and straighter side walls. Using the ultra-thin film process, straight side walls of features on the etched film can be achieved, as shown in Figure 4. Present methods of creating these 1–2 micron films involve sputtering or etching thick foil laminated layers and are very expensive.

In the present paper, we discuss the adhesion of the thin copper films on flexible polyimide substrates created by ultra-thin film process, and the possible factors to improve the adhesion.

Methodology

The adhesion of ultra-thin films was tested by the ASTM 3359 cross-hatched tape test in which a pattern of horizontal and vertical cuts spaced 1 mm apart are made in the copper layer by a blade (a razor blade was used). A special tape is adhered to the cross-hatched area and peeled off at about 180°.

To test the adhesion of the patterned, plated and etched copper layers, a peel test following IPC-TM-650-2.4.8 is routinely used in the printed circuit industry. Here special coupons >30 microns tall are created. They are 1 mm wide by up to 96 mm long, with a wider tab at the end for attaching by a clamp to a force gauge. The trace is peeled in a controlled manner at 90 degrees to the surface of the polyimide and the force measured as a function of peel length. A photo of such a sample is shown in Figure 5.

Data and Results

One of the attractive features of this ultra-thin film process has been that the coated, dried and sintered 0.5-micron copper layer adheres very well to polyimide. Due to the fine thickness of the films, the ASTM 3359 cross-hatched tape test has been applied to test the adhesion of the coatings. In the present study, the tested thin film samples were prepared by a lab scale preparation, i.e., Mayer bar coating, vacuum dry and single flash lamp sintering, and it was found that none of the sintered copper is removed by the tape.

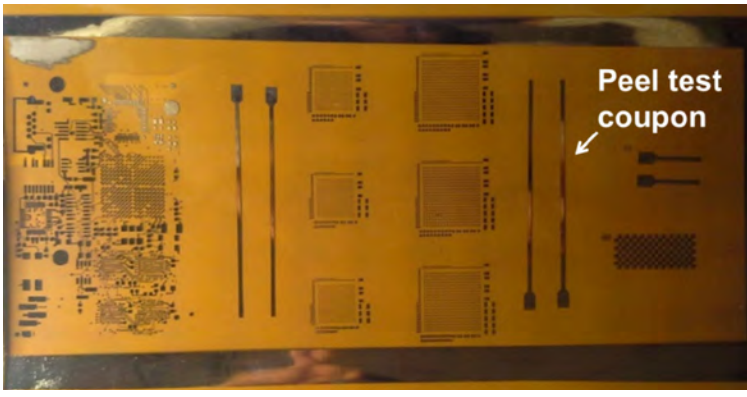


Figure 5: A plated, etched sample of nano copper ink on polyimide showing the peel strength coupon.

However, for the patterned, plated and etched copper layers, it was found that these 1 mm wide traces have highly variable adhesion. A peel force of about 10 N/cm is acceptable in the industry, and there have been samples that ranged from 0 N/cm (came off in the etching bath) to 8 N/m. So, a sintered copper layer was used that adheres very well to polyimide, but when that layer is masked, plated, and etched, the resulting (>30 micron) layer sometimes does not adhere very well. It is instructive to point out that the plated etched area is >30 microns thick because the peel coupons must be that thick to avoid tearing when connected to the peel test apparatus and pulled.

In order to understand the possible sources for this variable adhesion that are seen for plated and etched samples, we have investigated a number of factors which might affect the adhesion in our system. First, we investigated the flash sintering conditions. Well-sintered layers usually conduct the best and have the best adhesion. Additionally, we have mentioned already that sintering large areas of coated ink by overlapping repetitive flashes from a single bulb shows a darker band in the overlap area, and it has been found that the overlap area does not exhibit as good adhesion as the more copper colored area from a single flash (data not shown). The SEM images from the copper colored area and darker band (Figure 6) clearly showed the

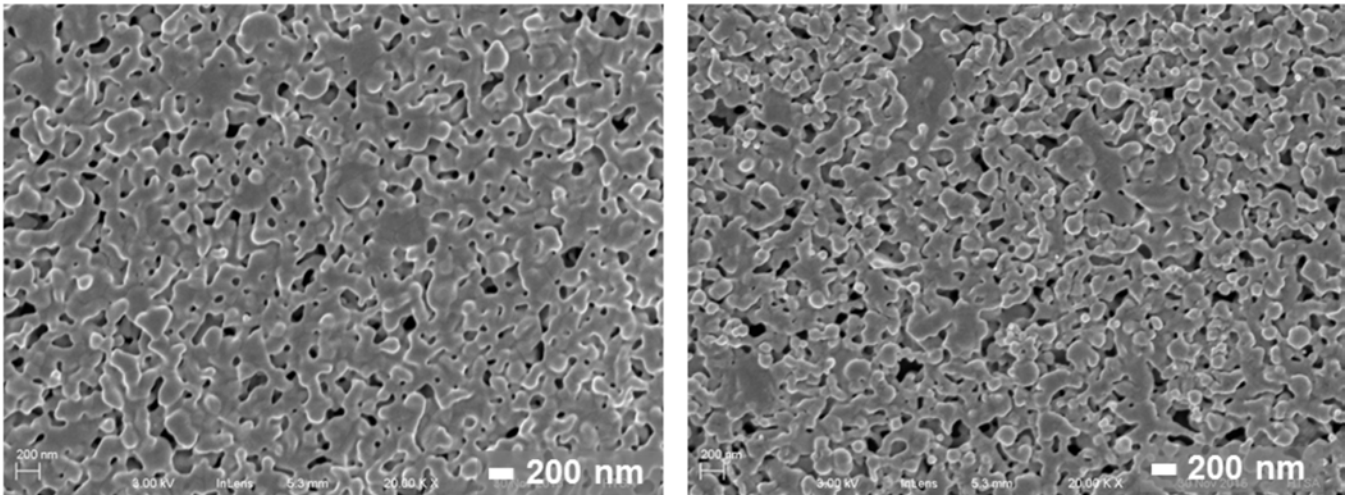


Figure 6: SEM images of the copper nanoparticles on copper colored area (left) and darker band (right) of a coated ink sample sintered by repetitive flashes.

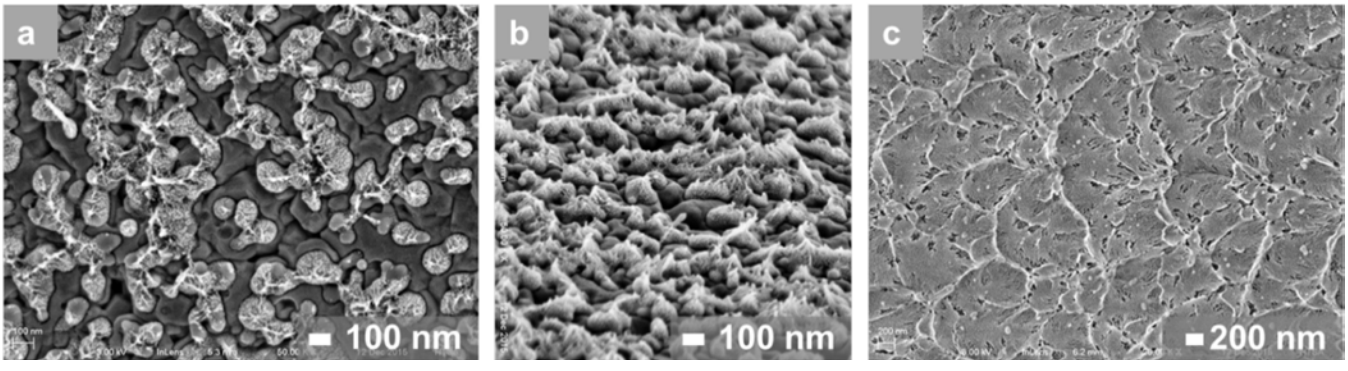


Figure 7: SEM images of gold sputtered samples of a sintered and plated copper film, peeled from the substrate: (a) top view of the peeled film; (b) tilted view of the peeled film; (c) top view of the substrate underneath.

difference in particle morphology, i.e., less particle necking and densification in the dark area. The result indicates that the sintering condition of the nano copper ink is important to the adhesion. This also points out the criticality of using a flash apparatus specifically designed to minimize such banding.

When the coated and dried ink is treated with the flash lamp system, the binder/dispersant decomposes and flashes off allowing the copper nanoparticles to sinter via necking. Vapors from this process can be observed on sintering. However, electron micrographic evidence suggests that some binder is left at the copper/substrate interface of a nano copper film plated up to ~5 µm thick. As the plated film was peeled off, the deformed polymer fibers (by the peel force) were left on both nano copper (Figures 7a, 7b) and the substrate (Figure 7c). Furthermore, the polymer layer left on the substrate (visible by the naked eye) can be wiped off by good solvents but not by bad solvents of the binder. The results clearly indicate the presence of binder/dispersant at the end of the sintering and plating process (data not shown).

The results suggest that the dispersant/binder is playing an important role as an adhesive layer at the interface, therefore affecting the adhesion. To investigate the binder effect, we examined a number of binders of different chemical structures and different solvent systems for their ability to disperse the nano copper and to

Dispersant	Solvent	Stable Ink
Polyvinylpyrrolidone	DEGBE	Yes
Polyoxyethylene octyl phenyl ether	EG/1M2P	No
Polyvinylpyrrolidone-polyvinylacetate copolymer	EG/1M2P	No
Polyvinylpyrrolidone-polyvinylacetate copolymer	DEGBE	No
Polypropylene carbonate	propylene carbonate	No
Polypropylene carbonate	N-methyl-2-pyrrolidone	No
Polypropylene carbonate	DEGMEEA	No
Hyperdispersant A	DEGMEEA	No
Hyperdispersant A	DEGBE	Yes
Polyvinyl butyral	DEGBE/MMB	Yes
Dispersant B	DEGBE	No
Polyether polyamide	DEGBE	No
Polyalkyleneoxy terminated polyamide	DEGBE	No
Polymethacrylate copolymer	DEGBE	Yes
Polymethacrylate copolymer	DEGBE/butanone	Yes

Table 1: A list of tested dispersants and solvents.

provide subsequent adhesion. A list of tested dispersant/solvent combinations is shown in Table 1. Five of these provide sufficient dispersion stability to allow further coating and adhesion testing. These inks were Mayer bar coated, vacuum dried, and flash sintered into 6x9 inch samples in the lab and the resulting uniform copper samples were sent to a local printed circuit house for preparation of peel strength coupons by masking, plating and etching. An assessment of the adhesion of these materials is shown in Table 2.

The maximum peel strengths of 2–3 peel test coupons on the plated (to ~30 µm) and etched films are recorded and listed in Table 2. Two samples showed very poor adhesion of the final film; zero peel strength is assigned as the traces fell off. For the other three samples, the numbers are quite variable, indicating non-uniform adhesion on the 6x9 inch coatings. The non-uniformity can be attributed to the sintering by overlapping repetitive flashes from a single bulb, resulting in less particle necking in the overlap area, as mentioned previously.

Another possible reason is the contamination on the polyimide surface (dust, grease, etc.) which largely affects the surface energy and therefore the thin film formation during ink

Dispersant	Solvent	Max. peel strength (N/cm)		
		1	2	3
Polyvinylpyrrolidone	DEGBE	4.4	4.4	---
Hyperdispersant A	DEGBE	6.8	0	4.5
Polyvinyl butyral	DEGBE/MMB	0	0	---
Polymethacrylate copolymer	DEGBE	8.0	3.8	3.4
Polymethacrylate copolymer	DEGBE/butanone	0	0	---

Table 2: Maximum peel strength of 1 mm wide peel test coupons on the plated and etched copper nano ink coatings (N=2-3).

drying. Interestingly, the last two inks on the table, that both employed polymethacrylate copolymer as dispersant/binder, showed very different adhesion results as different solvents were used. This suggests that the configuration of the polymeric dispersant/binder in the ink is possibly important to the adhesion. Overall, it is believed that we can further improve the adhesion by testing more binders that have similar chemical structures and properties with polyvinylpyrrolidone, hyperdispersant A and polymethacrylate copolymer.

It has been reported in the literature that the adhesion between copper and polyimide can be improved by applying a thin layer of chromium[3,4]. It has been attempted to sputter a thin (<100 nm) layer of metal like Cr, Ti, or Cr/Cu as a tie layer on plasma-treated polyimide substrates (20% O_2/Ar, 3 min at 15 mtorr), upon which to coat the ink. We found that there was a great deal of difficulty sintering these samples with a metal layer underneath, while no significant difference was observed on the plasma-treated (control) sample, as shown in Figure 8. The coated copper developed holes on sintering showing the sputtered metal underneath. The unexpected loss of adhesion might come from the nature of the deposited copper on the metal interlayer. In the literature, the metal layer promotes the adhesion of electron-beam evaporated[4] or sputtered[3] pure copper, but the copper nano ink includes dispersants/binders that may interfere with the interface. Since these sputtered samples had very poor adhesion at this point, none were masked, plated, and etched.

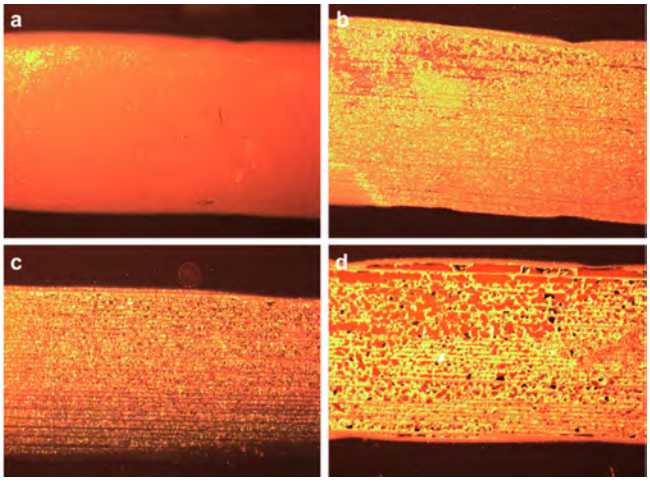


Figure 8: Micrographs of coated nano copper films on (a) plasma treated PI; (b) PI with 5–7.5 nm of chromium coating; (c) PI with 7.5 nm of titanium coating and (d) PI with 7.5 nm of chromium/100 nm of copper coating.

Polyimide	Max. peel strength (N/cm)		
	1	2	3
Regular	4.4	4.4	---
Modified	4.8	5.6	5.1

Table 3: Maximum peel strength of 1 mm wide peel test coupons on the plated and etched copper nano ink coatings on different PI substrates (N=2-3).

Surface cleaning and modification by physical and chemical means is often used to promote adhesion. A commercially available polyimide substrate, which was invented to promote the adhesion to copper, has been tested. However, we could not find significant improvement in adhesion comparing to regular PI substrate (Table 3). It is argued that the stronger adhesion to copper can be achieved through chemical functionalization and physical roughening of polyimide surfaces, by plasma or alkaline treatment[5-7]. However, the nano copper film on the plasma treated PI (Figure 8a) did not show stronger adhesion after plating and etching, compared to the film on untreated PI (data not shown). Further investigation on the effect of potassium hydroxide (KOH) treatment is in progress.

We have found so far that sintering condition and ink formulation (binder/solvent selection) have more impact on the adhesion of plated and etched films, while the surface modification of PI substrate does not seem to make much difference. We believe that in the system we use, the binder layer remaining at the interface after sintering is acting as an adhesive layer, and the adhesion failure happens between the nano copper layer and the binder layer. The surface modification of PI may promote the adhesion of the binder layer contacting with PI, but does not prevent the adhesion failure between nano copper layer and the binder layer. Sintering condition plays a complex role here, since it not only affects the connection between copper nanoparticles, but also the decomposition and therefore the remaining binder in the final film.

The fact that the sintered nano copper layer adheres well on PI but not after plating and etching makes us consider if the plating/etching chemistry is causing the adhesion failure. The sintered copper layer is not entirely homogeneous. As shown by the SEM image of a sintered trace (Figure 9a), there are voids in the sintered copper film. The sintered copper film which densifies as copper nanoparticles are connected by necking. At the same time, the voids between the nanoparticles join to form larger voids. This explains why the conductivity of this sintered copper by this technique is about 10% that of pure copper wire. When plating, those voids are filled in by much more uniform pure copper as shown in Figure 9b. Since this image is from the substrate facing side of a trace peeled from the PI, the presence of plated copper clearly indicates that the plating solution can get into the voids and all the way down to the PI substrate. If there is a negative effect of plating solution on the interface, the solution certainly has access to it. Strategies to minimize the impact of the plating/etching chemistry, such as mixing binary nanoparticles to create denser packing, will be further investigated in the near future.

Get the spark you need to tune up your sales team.

When an engine doesn't fire on all cylinders, performance is limited. The same goes for your sales team. We provide the spark to get your sales to the next level ...and beyond.

We've helped dozens of companies big and small improve their bottom line.

db MANAGEMENT GROUP

Learn how we can help YOU

207-649-0879 danbbeaulie@aol.com

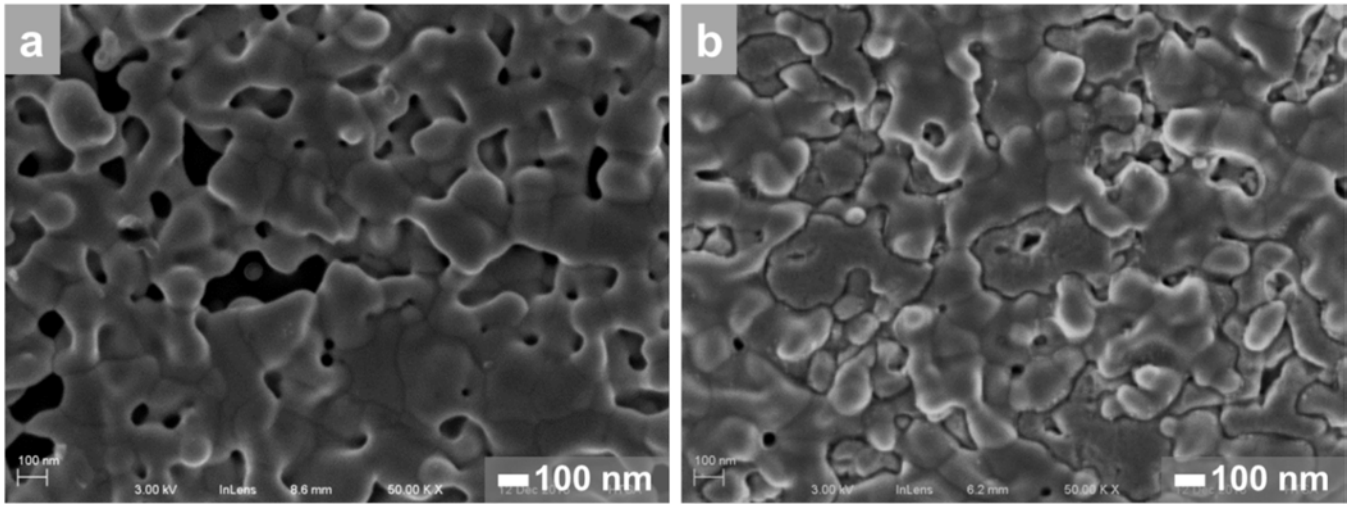


Figure 9: SEM images of (a) a sintered nano copper coating and (b) the substrate facing side of the same coating after plating to ~5 µm.

Conclusions

The ultra-thin film process is designed to put thinner copper cladding on both flexible and rigid printed circuit laminates than is currently available, by coating a 0.5~1 µm nano copper ink layer. There has been demonstrated excellent quality of etching on the patterns created by standard masking, plating and etching techniques. The adhesion is significantly affected by sintering condition and ink formulation, while not observed by the surface modification of PI substrate in the system. A few binders/dispersants have been found to give good ink stability and reasonable adhesion. It is concluded that the binder/dispersant layer between polyimide and nano copper is responsible for the adhesion, and the accessibility of the plating/etching chemistry toward the interface may determine the adhesion if it has any negative effect on the bonding at the interfaces. Overall, the technique provides the ability of the industry to greatly reduce minimum feature size and to also improve the performance of high speed electronics.

Acknowledgements

The authors would like to thank Brett Austin and staff at Nationwide Circuits, Inc as well as Professor Mark Poliks and his students at Binghamton University. **PCB**

References

1. Ciufo, D. In Nano Copper Materials in PCB Manufacturing, IPC Flexible Circuits-HDI Forum, Minneapolis, Minnesota October 28-30, 2015; Minneapolis, Minnesota.

2. Xenon Sinteron 5000 system.

3. Noh, B.-I.; Yoon, J.-W.; Choi, J.-H.; Jung, S.-B. Materials transactions 2010, 51, (1), 85-89.

4. Shih, D. Y.; Klymko, N.; Flitsch, R.; Paraszczak, J.; Nunes, S. Journal of Vacuum Science & Technology A 1991, 9, (6), 2963-2974.

5. Inagaki, N.; Tasaka, S.; Hibi, K. Journal of Adhesion Science and Technology 1994, 8, (4), 395-410.

6. Lee, S.; Park, S. S.; Lee, H. K. Macromolecular Symposia 2007, 249-250, (1), 586-590.

7. Wang, Z.; Furuya, A.; Yasuda, K.; Ikeda, H.; Baba, T.; Hagiwara, M.; Toki, S.; Shingubara, S.; Kubota, H.; Ohmi, T. Journal of Adhesion Science and Technology 2002, 16, (8), 1027-1040.

This paper was originally presented at IPC APEX EXPO 2016 and published in the proceedings.

David Ciufo is Program Manager PCB Technologies, Hsin-Yi (Sherry) Tsai is Research Scientist and Michael J. Carmody is Chief Scientist, all with Intrinsiq Materials in Rochester, NY.

thepcblist

THE best way to find a PCB fabricator, anywhere.

PROTOTRON CIRCUITS SOUTHWEST

Tucson, Arizona, U.S.A

Website | Contact Me | RFQ

Prototron Circuits
Quality Printed Circuit Boards

Overview | Contact | Specs | About | Videos | Photos | Brochures | News

Prototron Circuits is one of the industry's best and most reliable high technology quick turn companies in the country. With facilities in Redmond, WA and Tucson, AZ, Prototron is the quick turn printed circuit board manufacturer for people who value quality and on time delivery. Whether your need is a prototype PCB or medium volume production, our goal is to always deliver high technology "production quality" circuit boards on time.

With Prototron's strong engineering support, we can provide assistance with impedance calculations, as well as unique solutions to complex design issues.

Markets: Military/Aerospace

Board Types: Multilayer

Mfg Volumes: Prototype, Small, Medium

Other Services: Quick turn-around

Sr Blind/buried vias, Controlled Im
Sequential lamination

D 9001, ITAR registered, MIL-PRF-5

Click to see a partial list of registered OEMs!

Click here to see a demo

Why YOU should Showcase:

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

www.thepcblist.com

Embrace Your Brand

by Barry Lee Cohen
LAUNCH COMMUNICATIONS

Embrace your brand. It's your promise. It's your word. It's the trust, knowledge and confidence you communicate to your customers, the community, and the industry you interact with *and react to* every nanosecond. Break your brand, and you'll be breaking your heart and your aspirations.

Quite simply, your brand is what others believe to be your value, and is expressed more often than not upon having your name come up during a third-party conversation. Keep in mind, brands are built by individuals, not organizations. The person's view may not be true, in fact, it may be completely off the mark. Regardless, *it's their truth.*

When you let others define your brand it's like an alien force that invades your mind and body. It wants to immediately mold your reputation and worse yet, potentially denigrate your self-confidence and perceived worth. Don't cede the power of you.

Speaking from my own experience, you can launch—or relaunch—your brand by being true to what you inherently know to be your strengths and has proven to make you successful and happy. Blastoff:

Be your own version of extraordinary—ordinary is boring. Subscribing to the status quo can often be debilitating, as well as regressive in your personal pursuits and professional aspirations. Making that extra effort—no matter how small or insignificant you may believe it to be at the time—is deemed to be significant and at times monumental to others. You may not even realize your dedication and principled approach for a given project has taken a

www.ipc.org/events

UPCOMING Programs

August 31, 2016

IPC Education Online — Wisdom Wednesday — Webinar
for IPC Members ONLY
2017: A New President, A New Congress and a New Fervor for Tax Reform

September 21, 2016

IPC Education Online — Wisdom Wednesday — Webinar
for IPC Members ONLY

September 21–23, 2016

IPC India 2016 — Conference & Exhibition
Bengaluru, India

September 24–30, 2016

IPC Fall Committee Meetings — Meeting
Co-located with SMTA International
Rosemont, IL USA

September 26, 2016

EMS Management Meeting — Meeting
Rosemont, IL USA

October 19, 2016

IPC Education Online — Wisdom Wednesday — Webinar
for IPC Members ONLY

October 25–27, 2016

IPC-SMTA Cleaning and Conformal Coating Conference — Conference
Rosemont, IL USA

November 2, 2016

IPC Technical Education — Workshop
PCB Trouble Shooting
In conjunction with PCB Carolina
Raleigh, NC, USA

November 16, 2016

PC Education Online — Wisdom Wednesday — Webinar
for IPC Members ONLY
A Vision for the Industry
Raleigh, NC USA

November 29, 2016

IMPACT Europe 2016 — Conference
Brussels, Berlgium

December 7–9, 2016

HKPCA International Printed Circuit & IPC APEX South China Fair — Conference and Exhibition
Shenzhen, China

December 14, 2016

IPC Education Online — Wisdom Wednesday — Webinar
for IPC Members ONLY
Be a Resource for your Customers, Suppliers, and Team Members: Making the Most of IPC Services

February 12–16, 2017

IPC APEX EXPO 2017 — Conference and Exhibition
San Diego, CA USA

April 26–27, 2017

IPC Manufacturability Forum — Conference
Rosemont, IL, USA

May 2017

IPC IMPACT Washington, D.C. 2017 — Meeting
Washington, D.C., USA

June 27–28, 2017

IPC Reliability Forum — Conference
Dusseldorf, Germany

October 17–19, 2017

IPC APEX EXPO 2017 — Conference
Minneapolis, MN, USA

Wisdom Wednesday Webinar Series

This monthly webinar series educates IPC members on topics relevant to the industry. Each FREE 30-minute webinar is offered only to IPC members, with replays available online. These sessions are led by industry experts who share their technical knowledge and insights.

August 31, 2016

2017: A New President, A New Congress, and A New Fervor for Tax Reform presented by Keith Smith

Register Now

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

major—and yes, extraordinary—burden off the backs of your customers, colleagues, and management.

Revel in reliability, relevance and believability. Ever feel uneasy when you need to depend on others with unknown, inconsistent or questionable track records to get the job done? There's nothing worse than when you begrudgingly need to rely on these folks to deliver performance that reflects directly on your reputation, your character and in summary—your brand. I believe that there's nothing considered sexier in the C-suite than a colleague you can depend upon to deliver ROI results without sacrificing performance. The colleague becomes a trusted and relied upon resource and partner. The aura of subservience and the constructs of a traditional supplier relationship gradually melt away.

Decide to make a positive difference, even under difficult circumstances. This is easy to state, but tough to realize. It can, and arguably should be uncomfortable to offer different perspectives other than the norm, or for that matter knowing you must agree with a group for the short term, until your own position with grounded recommendations are fully formed and vetted. Indeed, silence is golden, but it should not be mistaken for indifference or worse yet, ignorance. The silence should be thought-provoking and absorbing until you can offer value that will benefit the organization and advance your brand.

Determination: never give up. Revel in your failures as you will learn from them, much more so than the successes you will achieve along the way. Determination can be both mentally and physically draining if you allow the pressure of a project or circumstance to divert your focus. Dare to be fully committed to reach the apex. Make sure to eat healthy and rejuvenate along the way. A few reps at the gym will replenish your energy level. It can be a steep and daunting climb, but well worth the journey if you are committed to bolstering your brand.

Be humble, but not timid. There's a big difference. Over the years, I've experienced some of the brightest minds vanquished due to lacking the courage of their convictions to establish or maintain their brands. Although in most cases you should let the work speak for itself, thereby enabling your brand to shine, there are instances when you need to unplug a few ears from all the elevator music, that others are happy to endorse and contribute. Don't be embarrassed. It is mutually beneficial to gently, yet directly remind key stakeholders the value you created as part of a collaborative team or independently. There's no shame in the truth.

Most importantly, don't let any individual or organization define you. Own your brand and you will own your destiny.

Barry Cohen is president and managing director of Launch Communications. He can be reached by clicking here.

Electronic Nose Smells Pesticides and Nerve Gas

A team of researchers led by Ivo Stassen and Rob Ameloot from KU Leuven, Belgium, have made it possible to detect pesticides and nerve gas in very low concentrations.

Researchers have built a very sensitive electronic nose with metal-organic frameworks (MOFs). MOFs are like microscopic sponges that can absorb quite a lot of gas into their minuscule pores.

The sensor can be integrated into electronic devices by applying the MOF as a thin film over the surface of an electric circuit, making it fairly easy to equip a smartphone with a gas sensor for pesticides and nerve gas.

Why choose Fein-Line?

Because there is a Fine Line between winning ...and the alternative.

After more than 44 years in the industry, just about everyone knows the name Dan (Baer) Feinberg.

There are few in our industry that have his experience and expertise in just about every aspect of the business. Dan knows what works, and perhaps more importantly, what doesn't.

When additional expertise is needed, or when senior management just needs an independent opinion, companies large and small alike get the guidance of Fein-Line Associates, especially in the areas of New Product Market Entry and M&A Due Diligence.

In our business, there's a lot on the line, not just sometimes, but every time. The line between success and the alternative can be razor thin. That's why winning companies choose Fein-Line time after time.

Dan (Baer) Feinberg
Fein-Line Associates

Sales Training General Marketing Assistance · New Product Market Entry
Merger and Acquisition Due Diligence · Printed Circuit Technology assessment
Market Research (It may work but will it sell) · Expert Witness assignments
Technology Transfer

Fein-Line Associates, Inc.
P.O. Box 73355, San Clemente, CA 92673-0112
(949) 498-7866
baer@feinline.com
www.feinline.com

TOP TEN

Recent Highlights from PCB007

1 The Newest Flex Shop in the U.S.

I-Connect007 sales team member Angela Alexander and I recently got a tour of Lenthor's new Silicon Valley flex board shop and then sat down with President and CEO Mark Lencioni to discuss the new flex facility, the markets, management, and the future.

2 The 21st Century PCB Factory–Designed to Eliminate Offshore Cost Advantages

More than 15 years have passed since North America and Europe ceased being the center of worldwide PCB fabrication and were supplanted by a Chinese market with low-cost labor, lax environmental requirements, and strong government support.

3 Happy's Essential Skills: Design for Manufacturing and Assembly, Part 1 and Happy's Essential Skills: Design for Manufacturing and Assembly, Part 2

Advances in interconnection technologies have occurred in response to the evolution of component packages, electronic technology and increasing complex functions. Therefore, it comes as no surprise that various forms of printed wiring remains the most popular and cost effective method of interconnections.

4 All About Flex: Imaging Methods for Etch Resist, Part 3: LDI

When LDI technology was first introduced around 20 years ago, throughput was an issue. LDI was often restricted to low volume or prototype runs. Subsequent advances in equipment as well as faster acting photoresist have made it practical for high volume circuit fabrication.

5 Against the Density Wall: Landless Vias Might be the Answer

I saw my first landless via multilayer while visiting NEC in Japan in 1985. You may not know much about landless vias. This has been a well-kept secret for the last 30 years, possibly because it is not permitted on military boards, and therefore, discouraged in all IPC standards.

6 Standard of Excellence: The Future is in Fine lines

The age of much finer lines and spaces is upon us. After years of slowly moving towards this technology our customers are now demanding that all of us provide them with fine lines and spaces. Our new trend in electronics is for denser and denser circuitry on smaller and smaller real estate.

7 Weiner's World


Gene Weiner

The way that I see the problem relates to overcapacity for fabricators in the consumer segment of devices with flat panel displays—mobile and otherwise—as well as a decline in notebook PCs, TVs and other household and portable electronic devices.

8 Happy's Essential Skills: Understanding the Concept of Managing Management Time

You can get the most value out of monkey management and one-minute management by using these principles together. Who knows? With these techniques, maybe you'll have a chance to take care of your own work instead of everyone else's!

9 Catching up with Sunrise Electronics' Ashok and Jigar Patel


Ashok Patel


Jigar Patel

I had been hearing about Sunrise Electronics for many years, so when Ashok asked me to come and see them for myself, I jumped at the chance. It was one of the most amazing plant tours that I've been on in a long time. I wanted to know more, so we sat down and had a chat.

10 EPTE Newsletter: Transparent Circuits Gaining in Popularity


Dominique Numakura

Transparent printed circuits were a popular item showcased at the JPCA Show 2016. Polyethylene terephthalate (PET) use to be the dominant material used for transparent substrates in optical circuits and devices and generated a huge amount of sales from touch screen panels.

Events

For IPC Calendar of Events, click here.

For the SMTA Calendar of Events, click here.

For the iNEMI Calendar of Events, click here.

For the complete PCB007 Calendar of Events, click here.

IPCA EXPO 2016
August 18–20, 2016
Delhi, India

NEPCON South China
Auguts 30-Spetember 1, 2016
Shenzhen, China

PCB West Conference and Show
September 13–15, 2016
Santa Clara, California, USA

Medical Electronics Symposium
September 14–15, 2016
Marylhurst, Oregon, USA

24th FED Conference
September 15–16, 2016
Bonn, Germany

ICT Evening Seminar
September 20, 2016
Hayling Island, UK

EDI CON
September 20–22, 2016
Boston, Massachusetts, USA

Medical Design & Manufacturing
September 21–22
Minneapolis, Minnesota, USA

EIPC Workshop on Reliability
September 22, 2016
Tamworth, UK

IPC India/electronica India 2016/ productronica India 2016
September 21–23, 2016
Bengaluru, India

IPC Fall Committee Meetings
September 24–30, 2016
Rosemont, Illinois, USA

SMTA International 2016
September 25–29, 2016
Rosemont, Illinois, USA

Manufacturing Day
October 7, 2016
An annual celebration of modern manufacturing meant to inspire the next generation.

electronicAsia
October 13–16, 2016
Hong Kong

TPCA Show 2016
October 26–28, 2016
Taipei, Taiwan

Electronica
November 8–11, 2016
Munich, Germany

FUTURECAR: New Era of Automotive Electronics Workshop
November 9–10, 2016
Atlanta, Georgia, USA

Printed Electronics USA
November 16–17, 2016
Santa Clara, California, USA

International Printed Circuit & Apex South China Fair (HKPCA)
December 7–9, 2016
Shenzhen, China

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

SALES: **ANGELA ALEXANDER**
(408) 489-8389; angela@iconnect007.com

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

EDITORIAL:
MANAGING EDITOR: **PATRICIA GOLDMAN**
(724) 299-8633; patty@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, SHELLY STEIN, TOBEY MARSICOVETERE**

INNOVATIVE TECHNOLOGY: **BRYSON MATTIES**

COVER DESIGN: **SHELLY STEIN**

I-Connect007 Presents
the pcb magazine

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

August 2016, Volume 6, Number 8 • The PCB Magazine© is published monthly, by BR Publishing, Inc.

ADVERTISER INDEX

Coming Soon to *The PCB Magazine:*

SEPTEMBER:
Military/Aerospace
Military/Aerospace electronics technology trends, manufacturing challenges and market outlook

OCTOBER:
The Leadership Issue
What makes a great manager and leader and the changing roles of leaders in our industry

I-Connect007

GOOD FOR THE INDUSTRY

FREE SUBSCRIPTION

Subscribe

007 Media Kit

subscribe.iconnect007.com

mediakit.iconnect007.com

EDITORIAL CONTACT

Patty Goldman
patty@iconnect007.com
+1 724.299.8633 GMT-4

SALES CONTACT

Barb Hockaday
barb@iconnect007.com
+1 916 365-1727 GMT-7

pcb design 007 | pcb 007 | SMT 007 | mil aero 007 | flex 007 | ein 007 | RTW

www.iconnect007.com