PCB007 PRESENTS

April 2012

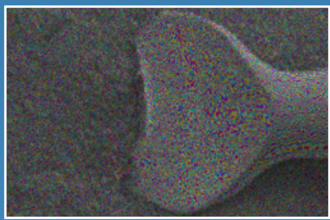
the pcb magazine

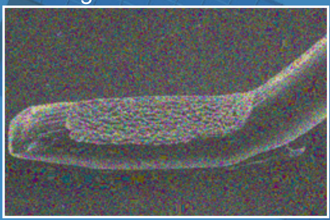
AN I-CONNECT007 PUBLICATION

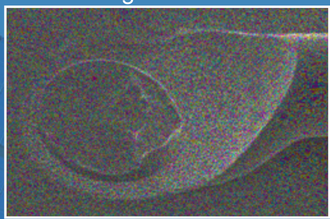
IMAGING:
ADAPTING TO THE CHANGING TECHNOLOGY

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

Soldering


Au-wire wedge

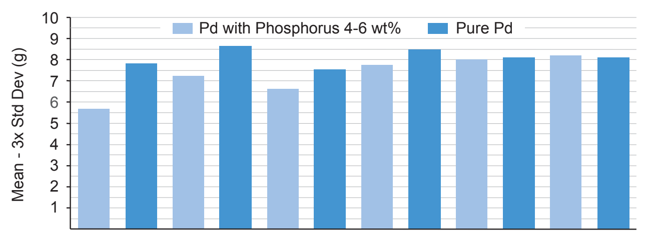

Al-wire wedge


Cu-wire wedge

Process Flow

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

Gold Wire Bond Pull Test



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

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

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

Features and Bene ts

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

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

TOTAL

ATOTECH

FREE BOOK DOWNLOAD

PiHR TECHNOLOGY
BY BOB WILLIS

Contents

"Overall, an excellent read for the experienced, education for the beginner, and a first-class reference book covering all aspects of PiHR."

— Dennis Price
Merlin Circuit Technology, Ltd.

"The definitive, complete and comprehensive guide to Pin in Hole Reflow technology."

— Pete Starkey
I-Connect007

"An excellent book for those embarking on PiHR as a new process and also as a reference manual for more established users."

— Sue Knight
Surface Technology International, Ltd.

PIHRTECHNOLOGY.COM

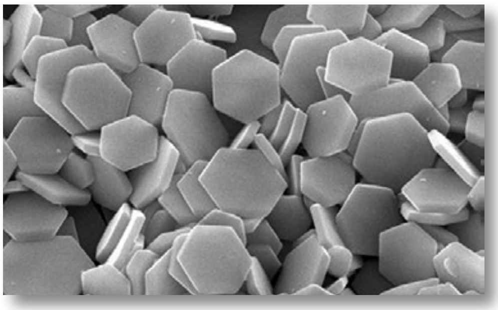
PCB007 PRESENTS

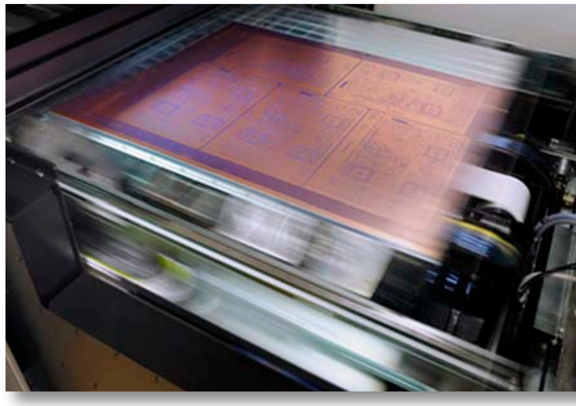
the pcb magazine

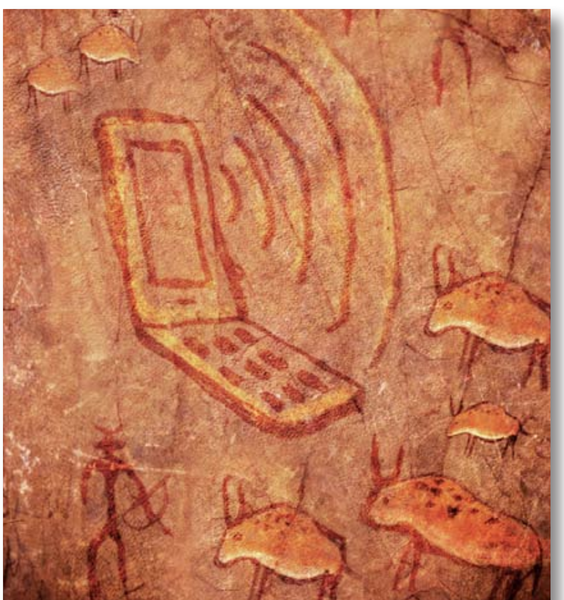
AN I-CONNECT007 PUBLICATION

IMAGING PROCESSES

This month, The PCB Magazine offers expert analysis of imaging technologies, in addition to imaging process issues, such as the pros and cons of photolithographic and digital techniques, and how successive images are aligned and kept in register. Also included are on-the-floor reviews of CPCA 2012, IPC APEX EXPO video interviews and of course, our regular line-up of columnists offering plenty of technical advice, common sense management ideas and market predictions.







PCB007 PRESENTS

APRIL 2012
VOLUME 2
NUMBER 4

THE DEFINITIVE INTERACTIVE MAGAZINE DEDICATED TO THE GLOBAL PCB INDUSTRY

thepcbmagazine.com

the pcb magazine

AN I-CONNECT007 PUBLICATION

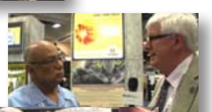
IMAGING PROCESSES

CONTENTS

ARTICLE


VIDEO INTERVIEWS




SHORTS

EXTRAS

COLUMNS


NEWS HIGHLIGHTS

Another Bad PR Month for Foxconn

by Ray Rasmussen
I-CONNECT007

SUMMARY: Coming off a stellar fourth quarter, buoyed by Apple's continued dominance in multiple markets, Foxconn should be riding pretty high. Instead, the über-EMS provider has had a very rough start in 2012, stumbling from one PR blunder to the next. Clearly, the PR machine over at Foxconn has some work to do.

The story that won't die is the one dealing with working conditions at Foxconn's many Chinese factories. The international media have latched onto this, and the more success Apple experiences in the form of soaring profits, rising stock prices, and greater market penetration, the more the press scrutinizes their supply base. No one seems to care much about investigating RIM's or Nokia's suppliers (they probably are, but we don't hear about them). I haven't heard anything about HP or Dell's suppliers, either. Interviews with workers' rights groups continue to expose poor working conditions for Chinese factory workers, singling out Foxconn and their partner, Apple. The stories are focused on the difficult working conditions related to long hours, consecutive workdays and underage labor. All of these articles' claims run counter to Apple's published labor policies for their suppliers. Of course, the picture painted by the NGOs and press is that this behavior is commonplace and a company like Apple only gives lip service to their suppliers' labor policies.

As a result of the bad press, Foxconn recently was featured in an unflattering segment on The Daily Show; CBS Sunday Morning also ran a segment entitled, "The Dark Side of Shiny Apple Products."

Then, to make matters worse, Foxconn CEO Terry Gou made headlines for referring to his employees as "animals." The press had a field day with that one. Although Gou's comments about his diverse workforce were taken out of context, they still generated quite a few articles with arguably sensationalized titles, which hasn't helped the company's cause and continues to rankle Apple and tarnish its image.

If you read Gou's comments, along with the clarification from the company, it seems obvious he didn't mean that his more than one million employees all over the world were animals, or acted like animals. He was pointing out that much like a bunch of animals in a zoo, they have different needs, and that the company needed to learn how to manage such a diverse group. But, the damage was done.

Foxconn is under a microscope these days. They have to get on top of this, or it's going to cost them dearly.

Foxconn has been trying to lay low after all the bad press they've suffered over employee suicides in 2010. They have too many moving pieces, suppliers, customers and employees. Their biggest problem is a connection to the most visible tech company on the planet. It may seem that Foxconn is being unfairly targeted, but sometimes it appears as though they are placing the target on their own back.

I'm sure the other EMS giants are happy to have Foxconn in the spotlight, but I would suspect it also makes them nervous. Trying to distance his company from the bad press, Jabil's CEO Tim Main stated at a recent shareholders' meeting that his company was quite progressive and treated their employees very well. He felt

that Foxconn's labor practices would come back to bite them. We'll see. I would also suspect that all multinationals tailor their approach to their workers on a country-by-country basis. I could be wrong, but I doubt that these multi-national manufacturers have the same standards for all their employees. In any case, I'm sure some are better or worse than others.

Perspective

Do we see things differently with our Western perspective? Do Gou and other Asian manufacturers have a different perspective on what good labor conditions are? Even if this is a case of perspective, some have been refuting claims made in a recent article in the New York Times, "In China, Human Costs Are Built Into an iPad," that has caused a quite a stir. The article attacks Apple for being complicit with their suppliers by ignoring recommendations from independent auditors. Apple CEO Tim Cook was furious, claiming that no other OEM does as much as Apple to ensure good labor practices at their suppliers' factories. Apple's independent auditor, BSR, denies several of the accusations made in the NY Times article and recently posted this letter. It doesn't exonerate Foxconn, but it does suggest that Apple is working to continuously improve worker conditions and is not ignoring the recommendations of their independent auditors. I haven't seen much press on that one.

The bond Foxconn and Apple have forged seems to be pretty solid. Tim Cook was the guy most responsible for the supply chain for most of this decade, so I doubt we'll see those relationships easily disrupted. If I were Foxconn, I'd open my kimono. They already have, to Apple and its auditors, but I'd get a bit more aggressive. I'd acknowledge where I was wrong and I'd work hard to show them all the good things the company is doing.

In response, Apple came out with its list of suppliers, and its corporate responsibility report that said the company will get more aggressive auditing their suppliers. My guess is that Foxconn will likely be at the top of the priority list. Here's the report and here's the supplier list.

The Bigger Picture

For me, in order to get my arms around this story, I have to step back a bit and look at this from a much higher altitude. There's a lot more to this than just what the press is focused on. Below are some statistics to help bring some perspective to the situation. Bear in mind that I am a huge proponent of human rights and worker rights, but I'm also a realist.

According to the CIA FactBook, the unemployment rate in China is around 6.1%. (Note: Official data is for urban areas only; the inclusion of migrants may boost total unemployment to 9%, as there is substantial unemployment and underemployment in rural areas.)

In 2010, the workforce in China was estimated to be about 815 million people, which, at 6% unemployment, equates to about 48 million unemployed Chinese. At 9%, which is the more likely number, there are over 73 million people unemployed in China. This leaves Foxconn and other manufacturers with a huge advantage when hiring. The power over employees resides in the hands of the companies and the Chinese government through their closely controlled labor unions.

Even though it would seem that the cards are all stacked in the companies' favor, even with this high unemployment rate, there is substantial competition for these workers. A recent BBC article highlights a 23% increase in the minimum wage in Sichuan province, for instance. The increase was established to attract more workers. Having a choice of where to work offers the Chinese workforce some control. It's not perfect, but it's something. As we all know, labor rates and living standards are improving in China for at least some of the population (about half) and hopefully, companies like Apple will continue to ensure their suppliers meet basic labor standards.

High turnover rates (20-30%) have also been a huge issue for most manufacturers in China. That issue hasn't diminished. I can't imagine trying to run a business with that much turnover. There's a high cost associated with this. To reduce turnover, companies have to pay more, offer better incentives and create a better working environment. So, the people do have some power, but too much could be deadly.

With an unemployed population of over 70 million people, it's no wonder the Chinese government bends over backwards to incentivize businesses to come to China. Maintaining control of their labor force to ensure a fairly stable labor environment is a major piece of this puzzle. Freedom to organize, at will, would certainly bring the country to its knees (at times) and close the door to company expansions or new company entries to the Chinese market, essentially killing their economic miracle. Without their continued economic expansion, unrest would become a huge issue.

Here's another tidbit to help us see the bigger picture from the CIA World Factbook:

Population below poverty line: 2.8%—21.5 million rural population live below the official "absolute poverty" line (approximately $90 per year); an additional 35.5 million rural population live above that level but below the official "low income" line (approximately $125 per year) (2007).

Based on this scale, there are probably 50 million people living on less than $150 a year. About half of China's people live in cities; the rest are in the countryside. I would suppose (I haven't been able to locate a definitive source, yet) that at least half of China's population earns less than $500 per year. The report also states that a majority of China's population has a high school education or less. These are the people who companies like Foxconn hire to work on their assembly lines.

"I would suppose (I haven't been able to locate a definitive source, yet) that at least half of China's population earns less than $500 per year."

I do appreciate the work of those who have gone before us to create the rules and regulations that provide decent wages and working conditions today. I do believe that workers all around the world deserve the right to be treated fairly. Further, if we can apply pressure on a company like Apple to ensure the people making the products we buy are being treated fairly, then we should do what we can. However, we (the West) must be careful not to push too hard. We've already heard Gou state that he will be adding one million robots to his factories over the next few years (that's a million Chinese workers who won't be needed and will likely remain unemployed). Surely, he will continue to automate these mind-numbing jobs as soon as possible to offset the growing labor costs (wages and the cost of retraining 30% of his workforce year-after-year) and the enormous bad press associated with his low-end workforce. The Chinese government sees this danger, too. They have to walk a very fine line. Push too hard to support worker's rights and you kill the economic engine or you force companies to automate millions of low-end jobs, which could rip the country apart. On the other hand, not working to improve the conditions for their workers could generate unrest.

Well, that's the way I see it. **PCB**

Ray Rasmussen is the publisher and chief editor for I-Connect007 publications. He has worked in the industry since 1978 and is the former publisher and chief editor of *CircuiTree Magazine.* To contact Ray, click here.

FEATURE

PCB IMAGING TECHNOLOGY: Development and Trends

by Eric Janssens
AGFA-GEVAERT N.V.

SUMMARY: The key today is market know-how: knowing and understanding the needs of a PCB shop and translating that into film specifications.

After World War II, when radio tubes were being replaced by semiconductors, connections moved from soldered copper wires to PCBs. It was the dawn of a new industry.

During the '50s and '60s, single-sided boards ruled the world. In the early '60s, double-sided boards entered the scene. A decade later the holes of these double-sided boards were plated. Then multilayer boards appeared. Holes were mechanically drilled, but when high-powered lasers became available, microvias gave birth to sequential buildup boards. It is to be expected that under the ever increasing pressure of new applications, more functionality per square centimeter and especially cheaper, higher performing PCBs will be developed.

During this early era, the cradle of the PCB was in England. Under the influence of Silicon Valley, the PCB industry moved to the U.S. In the 1980s, the rise of Japanese electronics manufacturing resulted in an important PCB industry developing under the Rising Sun. Lately, economics (labor costs) have led the PCB industry exodus to Eastern China. The United States and Europe, each with a market share below 7%, became small players. But Japan defended its 20% market share well. Today, Asia is responsible for 85% of the world's PCB production.

Where do we go next? We see interest in mainland China, while Vietnam and India are making special efforts to acquire a piece of the pie. Additionally, Brazil and Russia may become important players.

By 1985, 6,100 companies were involved in the production of PCBs worldwide. As of 2010, the number of shops had decreased to 2,600. The PCB industry had matured into a limited number of strong companies. It is generally accepted that that figure continues to decrease.

Over time, PCB construction, production location and the professionalism of the manufacturers have changed, but the basic principles have not. After more than 50 years, one still starts with a full surface copper, adds a full surface resist, images and develops the resist, etches the copper, and strips the resist. Not the wisest approach.

> **Today a phototooling film and its relevant chemistry are real masterpieces.**

From Day One, silver halide films were used in the production of PCBs. The first phototools were pencil drawings shot by reprographic cameras on reprographic films. Then, tape replaced the pencil. Blue and red tapes were used for solder and component side. The cameras and the film didn't care.

Gerber opened a new era when it launched the Gerber language and the photoplotter. The design was created on a computer and the phototool was imaged by a vector photo plotter. A light spot, generated by a standard light bulb and shaped by apertures, travelled over the film, drawing the pattern as if it were laying down tape.

The specs of a PCB film and a reprographic film slowly began to differentiate. A PCB film was copied several times whereas a reprographic film, renamed graphic film, was copied only once to an offset printing plate. The printing plate was multi-copied. As a result, film suppliers designed PCB films with improved scratch resistance. With multilayer PCBs, the importance of a better dimensional stability became obvious. In the graphics industry, the four films (one for each printing color) had to match. All layers had to match one another, but they also had to match the drilled holes. Phototooling film 7 mils (175 micron) thick became the standard.

Vector plotters couldn't cope with the speed needed to plot high-resolution phototools. In those days, 4,000 DPI was considered high resolution. The flatbed plotters were replaced by internal and external drum raster plotters. Afterwards, due to accuracy limitations, the internal drum raster plotter gave up. Raster plotters first were equipped with green, then blue and later red lasers. Modifying the color sensitivity of a PCB film was not hard to do. But making sure a good image was formed in nanoseconds, about a million times faster compared to a camera or a vector plotter, was more difficult.

Until the end of the 1980s, phototools were processed in Lith chemistry. The photographic quality was unsurpassed, but the stability was hard to control. This processing required constant monitoring and correction. Rapid access chemistry was far more robust, but didn't supply the expected image quality; in particular, good line width control was a problem. By the turn of the century, hybrid chemistry offered the requested image quality, comparable to Lith, combined with the ease of rapid access.

Today a phototooling film and its relevant chemistry are real masterpieces. Only three companies worldwide are able to address all issues involved. The energy needed to generate a full black latent image in one square meter of Idealine RPF film is 8.0 mJ. That equals the energy generated by a cube of sugar (6 grams) falling from a height of 14 centimeters.

A phototooling film must serve many masters, as evidenced by Table 1.

The problem with designing a phototool is conflicting interests. One can increase the thick-

Photographical	Physical and Chemical	Others
Low Dmin and high Dmax	High dimensional stability	Fitness for AOI
Best line sharpness and roughness	Trouble free loading and und loading in plotter	No impurities
Highest resolution	Good vacuum behavior on plotter and printer	Low dust contamination
Precise line width control	Highest scratch resistance	Consistent quality
Wide exposure latitude	Permanent antistatic	Long shelf live
Wide process latitude	Low anisotropy	Compliant with legislation on ecology, health and safety
High energetic sensitivity	Low chemistry consumption	Price

Table 1: Characteristics of a silver halide film.

ness of the base material in order to improve the dimensional stability and handling, but by doing so, the Dmin goes up; the dimensional changes, even though smaller, occur more slowly, and the price goes up. One can improve the hit rate of a phototool by increasing the thickness of the anti-scratch layer(s). That works, but then the processing kinetics is slowed down, affecting the image quality, and because thicker layers absorb more water, the dimensional stability is poorer.

Another compromise to be made is the silver crystal size. In the emulsion preparation, one can control the size and shape of the silver crystal quite well. Smaller crystals have a better resolving power and thus can handle smaller lines and spaces. But, the smaller a crystal, the more light is needed to image it. Plotting at high resolutions is done with smaller spot sizes, with less energy and with faster dwell times.

After more than 100 years of development, phototool manufactures have a large arsenal of possibilities to use for any given compromise. The key today is market knowhow: knowing and understanding the needs of a PCB shop and translating that into film specifications.

Today, two types of coated film constructions are used: double-sided and single-sided. Double-sided coated films have gelatin layers on both sides. With respect to water absorption, one is compensating for the other. As a result, this type of film will not curl when the relative humidity is changing. The price paid for that is the gelatine load. More gelatin means more water absorption and thus a greater dimensional change. Single-sided coated films have gelatins on the emulsion side only. Often, these films are referred to as plastic back films. The curling of these films is acceptable when the relative humidity changes are limited. In high-end PCB shops, where temperature, relative humidity and dust level are well controlled, the curl disadvantage is by far compensated for by the better dimensional stability.

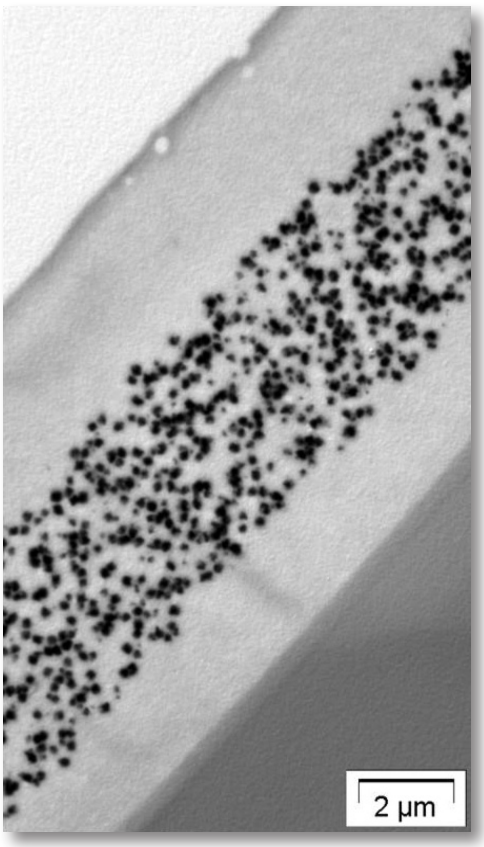


Figure 1: Cross-section of all layers on emulsion side.

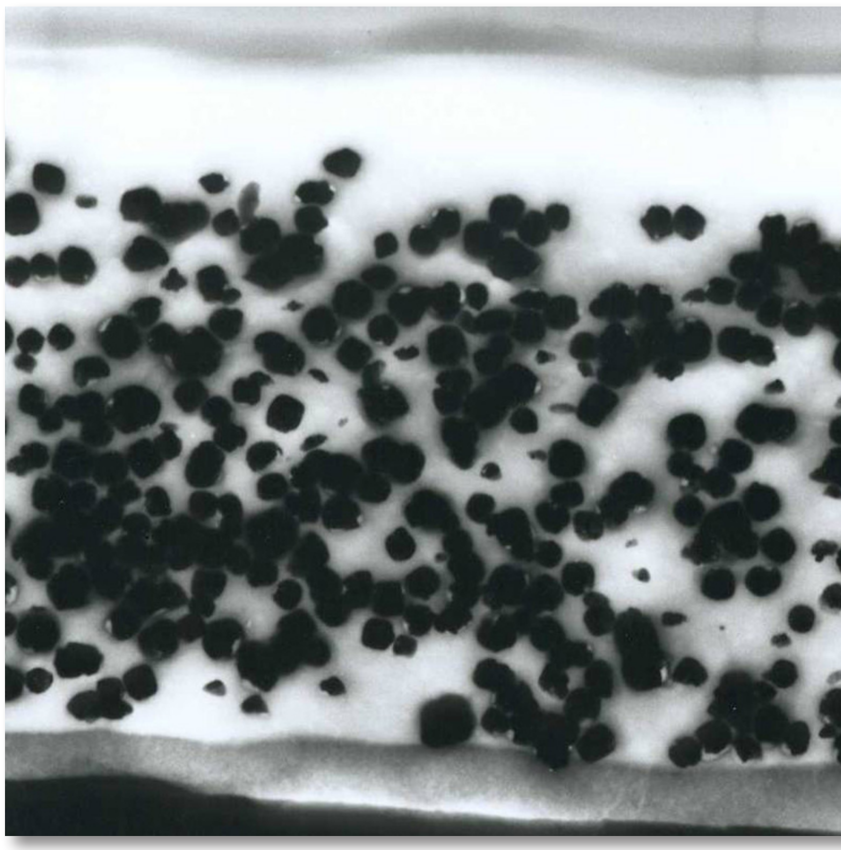


Figure 2: Cross-section of emulsion layer.

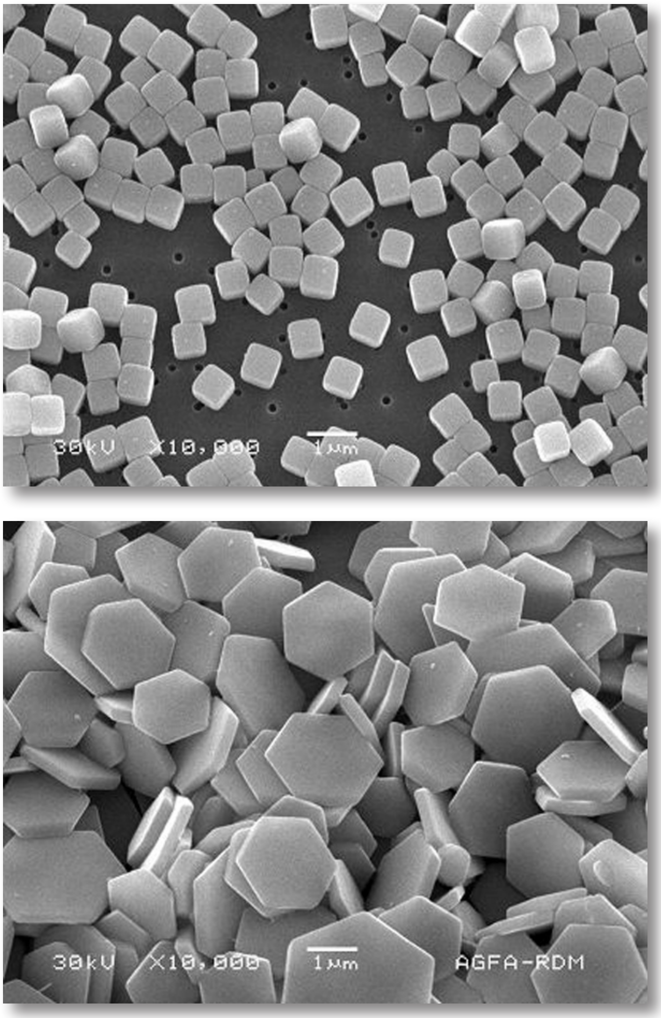


Figure 3 and 4: Isolated silver halide crystals. Each application requires a specific shape and size.

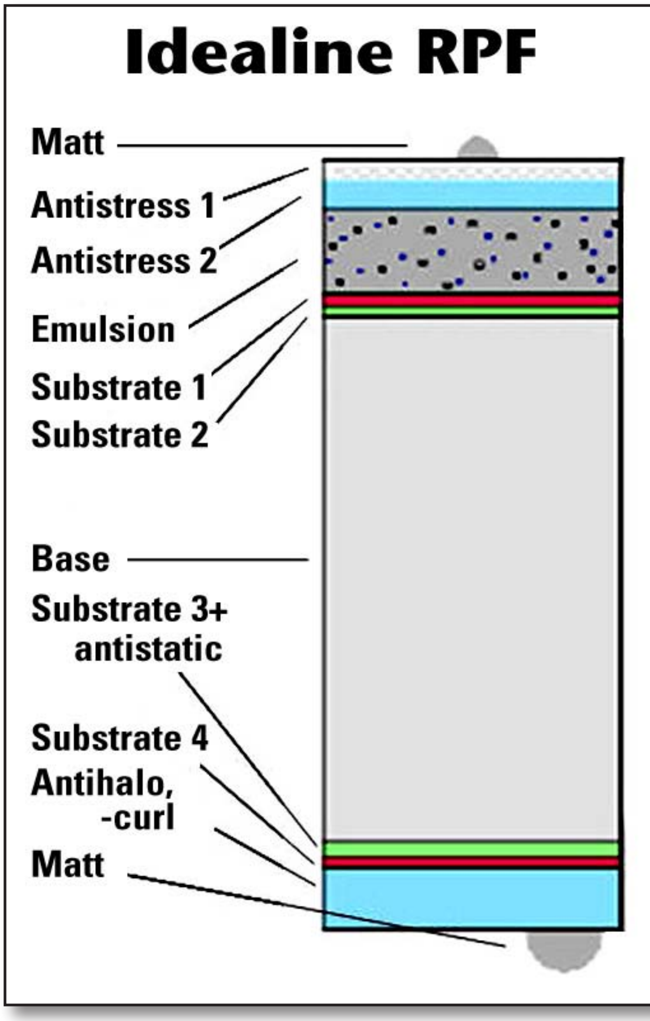


Figure 5: Structure of a double-sided coated film.

A silver halide phototool is based on a 175 micron-thick polyester base. The base is double-sided-coated with two adhesion layers on each side. An adhesion layer is 200 nanometers thin. The matte particles on the emulsion side are spherical transparent plastic pearls with a diameter of 2.2 microns. They are partly embedded in the anti-stress layer. The matte particles control the vacuum build up and behavior during printing. Agfa phototooling films have two gelatin-based, anti-stress layers. The upper one contains a grease component that encapsulates possible dust particles and prevents them from damaging the image underneath. The second anti-stress layer adds robustness and acts as a barrier to prevent contamination of the silver crystals. The total anti-stress thickness is 1.5 microns. The emulsion layer is 2.5 microns thick and consists of silver halide crystals embedded in gelatin. The crystals are approximately 200 nanometers in size. A crystal contains up to one million silver halide compounds.

During imaging, the silver crystals absorb part of the light to create a latent image. In the developer, that latent image will be chemically applied to metallic silver, which is fully black. The fixer then removes the unused silver. The left over light may generate a so-called ghost image. Therefore, an anti-halo layer is added that absorbs the leftover light. With a single-side coated film, the anti-halo layer is coated between the emulsion layer and the base. With

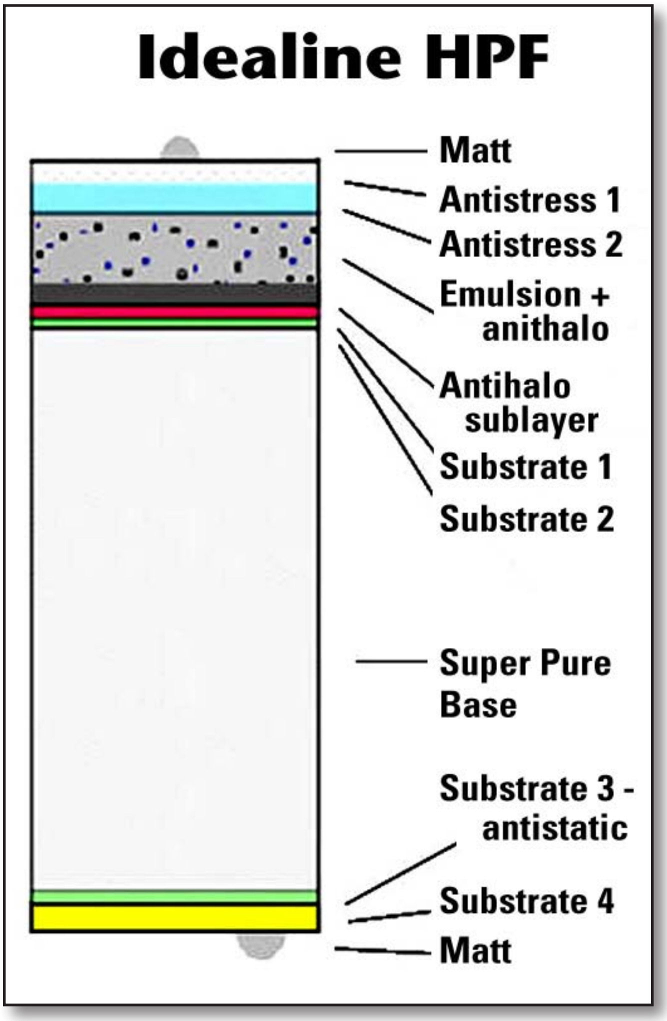


Figure 6: Structure of a single-sided coated film.

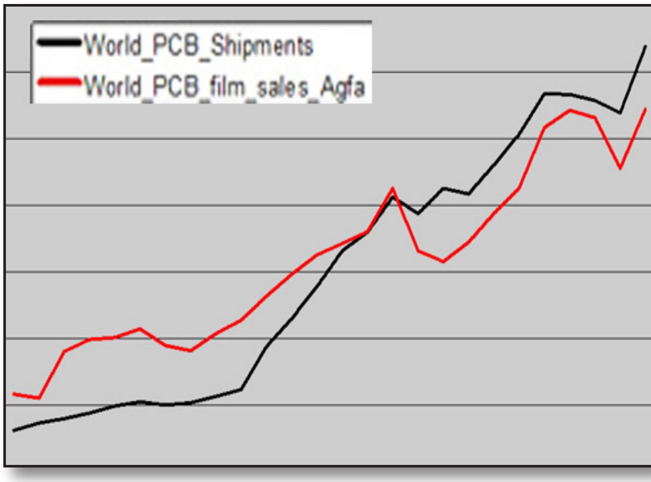


Figure 7: Agfa film turnover (SQM) versus PCB shipment (USD) (Source: IPC, Walt Custer).

Imaging	2000
Dry film resist	514.0 mio sqm
Liquid photo imageable resist	90.9 mio sqm
Liquid photo imageable solder mask	1.6 mio kg
Diazo	3.4 mio sqm
Silver halide film	11.6 mio sqm

Table 2: Total surface imaged worldwide (Source: Quantum Performance Group).

a double-sided coated film, the anti-halo is added to the back layer. Then, PEDOT PPS, a conductive, transparent, flexible polymer, is added to the back layer in order to give the film its permanent anti-static characteristics. Anti-static film does not attract dust particles. Dust on a phototool during imaging and processing can cause "pinholes." During printing, dust absorbs the UV light needed to polymerize the resist.

The matte particles on the back side are spherical transparent plastics with a diameter of 4.6 microns. These matte particles control the loading and unloading in the plotter and assure the films can be separated.

Today, the PCB industry worldwide processes approximately 2.5 billion square metres of imaged or structured surface area per annum, and more than 90% of that surface is imaged using silver halide phototools (a small amount is still imaged using diazo film or by screen printing). The remaining 10% is imaged by other technologies.

The most important alternative printing technology is laser direct imaging (LDI), which offers some benefits that a film cannot, but it is cursed with some structural drawbacks. Still, LDI is a viable way to produce certain PCBs.

One of the drawbacks of LDI is the expensive optics required. Projection systems, based on digital mirror devices, do not have that drawback and are a good candidate to compete with film and LDI.

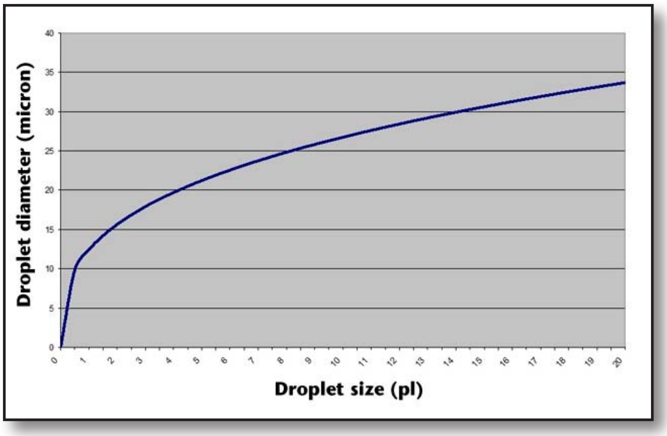


Figure 8: Droplet: volume versus diameter.

All of these technologies are additive. In other words, one starts with too much and adds some more on top of that to then take away what is not needed. The holy grail of PCB production is a reel-to-reel full additive system. Inkjet may do the job one day. Konica Minolta recently announced the KM128SNG-MB print head. The droplet volume is 1 picoliter. The diameter of the droplet in flight is 12 microns. With a good copper preparation and good control of the rheology of the ink lines and spaces, 25 microns may become possible. **PCB**

Eric Janssens is business manager PCB with Agfa-Gevaert N.V., Mortsel, Belgium. He graduated as industrial engineer Electro-Mechanics in 1978 and subsequently joined Agfa-Gevaert N.V. He held several positions in engineering, application and sales. Since 1991 he is involved in phototooling. Since 2004 he has been responsible for the company's worldwide phototool activities. He can be reached at eric.janssens@agfa.com.

Most-Read PCBDesign007 News Highlights This Month

Mentor Graphics Releases New PADS Suite

Key new features of the PADS suite include the ability to associate nets necessary for DDRx interconnect design. Multiple nets can now be grouped and given unique high-speed constraints including length and differential pair rules. Associated nets adhere to these rules during interactive and automatic routing.

Complete IPC Designers Forum Coverage From San Diego

The Designers Forum is over now, but we have interviews with the movers and shakers of the PCB design world. If you couldn't make it to San Diego, you missed a happening show!

Intercept's Pantheon 7 Soon to be 3-D Design Environment

Immediately following the announcement of Pantheon 7, a redesigned user interface version of Intercept Technology's advanced PCB/Hybrid/RF layout software, development is under way for 3-Dimensional design.

DownStream Releases New Version of PCB Post-Processing Solutions

DownStream Technologies, LLC today announced the release of new versions of their industry-leading PCB post-processing solutions, CAM350 and BluePrint-PCB.

ODB++ Solutions Alliance to Provide Open Forum

The ODB++ Solutions Alliance has been established today to provide an open forum for implementers and developers of the ODB++ format, an intelligent, single-data structure for transferring PCB designs into fabrication, assembly, and test.

Mentor Graphics Signs KETIV Technologies as Distributor

Mentor Graphics Corporation has signed KETIV Technologies, Inc., an established leader in delivering CAD/CAM software and services, as a new distributor for California, Arizona, Nevada, Washington, and Oregon.

Direct Logix Releases Control Center Version 5

Direct Logix announces the major release and immediate availability of version 5 of Control Center for Quoting, Engineering Planning, CAM Automation, DFM, and Automated Traveler generation system.

Altium Releases Altium Designer 12

Altium, developer of next-generation electronics design software and services, announces Altium Designer 12, the evolution of the award winning unified electronics design solution Altium Designer.

Scientists Produce Graphene Using Microorganisms

The Toyohashi Tech Graphene Research Group at the Electronics Inspired Interdisciplinary Research Institute (EIIRIS) has reported on an innovative method for producing high-quality graphene by reducing graphene oxide flakes using easily extractable microorganisms from a local river bank.

The method developed by the Toyohashi Tech team was inspired by a recent report that graphene oxide (GO) behaves as a terminal electron acceptor for bacteria, where the GO is reduced by microbial action in the process of breathing or electron transport.

Notably, the Toyohashi Graphene Research Group method is a hybrid approach, where chemically derived graphene oxide flakes are reduced by readily available microorganisms extracted from a river bank near the Tempaku Campus of Toyohashi University of Technology, Aichi, Japan. Raman scattering measurements showed that the GO flakes had indeed been reduced.

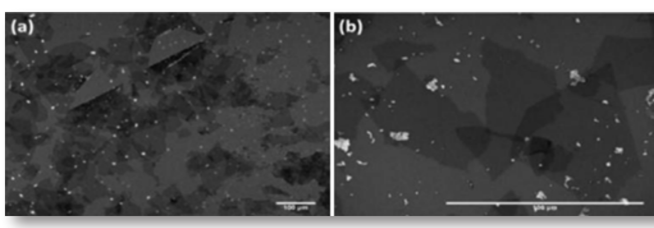


Image of reduced GO sheets on a SiO2/Si substrate: optical microscope image (a) and higher magnification (b).

Statistics without Tears

by Steve Williams

SUMMARY: Most companies have some level of a statistical process control (SPC) program, but few truly understand how to apply it to produce significant benefits. The PCB fabrication industry as a whole still has not yet unleashed the power of SPC.

Introduction

Process efficiency is a critical tool under any conditions, but it takes on even more importance during times of economic uncertainty, such as our current environment. The irony of statistical process control (SPC) is that while it is one of the most powerful process improvement tools available to manufacturers, it is also the one that is usually executed most poorly. Although most companies have some level of an SPC program, few truly understand and apply it in a way that provides significant benefits. And, while I wrote about the need for effective SPC ten years ago, and with apologies to the select few companies that really do a good job with this today, the PCB fabrication industry as a whole still has not yet unleashed the power of statistical process control.

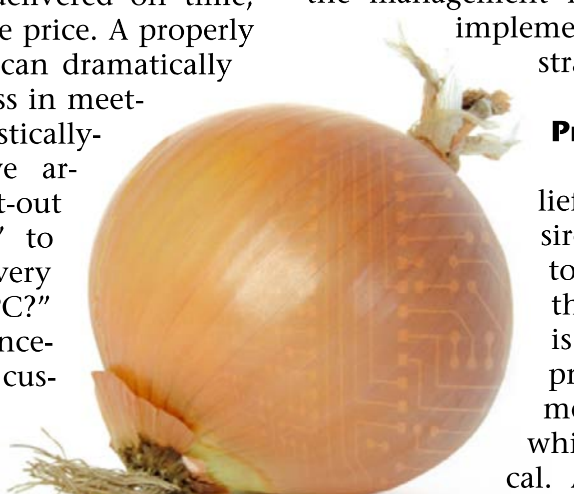
Why SPC?

A customer has an expectation of consistency in product quality, delivered on time, and at a fair and competitive price. A properly implemented SPC program can dramatically increase the degree of success in meeting these expectations. Statistically-challenged companies have argued everything from a flat-out "we don't believe in SPC" to "our customer return rate is very low, so why do we need SPC?" In addition to the performance-related issues that reach a customer, like late delivery and defective product, the cost of inefficiencies are directly passed along to the customer. Scrap (or overrun to cover anticipated scrap), rework, and inefficient processing are factored into the pricing model, so the lower the efficiency, the higher the total cost to the customer. In other words, customers pay for your inefficiencies.

Early in my career, a wise old mentor told me, "Never argue about what can be measured." This has stuck with me, and in my quest for continuous improvement, translated into "How can we get better if we don't know where we are now?" followed by, "How can we know where we are now without metrics?" An effective SPC program should be a key component of continuous improvement and a valuable management information system. The following are essential, basic attributes that any customer is going to want to see in an SPC program.

Implementation Plan

This is the high-level policy that overviews the company philosophy and management support of the SPC program. It should detail the steps used in both identifying critical processes and those required to progress from the data collection stage to process capability index (C_{pk}) analysis. The plan should also contain specific C_{pk} goals and timelines, as well as describe the management review process to measure implementation progress and action strategies.

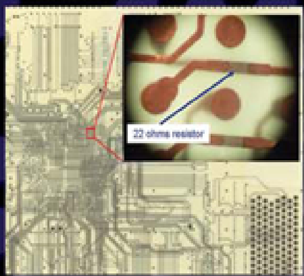


Process Characterization

Contrary to popular belief, the customer has no desire to tell the supplier how to run their business. What they do demand, however, is that you characterize your processes and have a documented method to identify which ones you consider critical. A common mistake is for a company to try to chart every

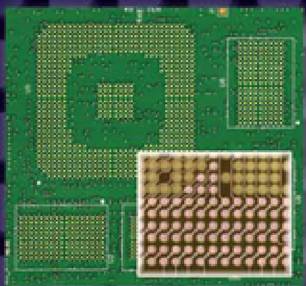
OhmegaPly® Embedded Resistors

Decades of Innovation



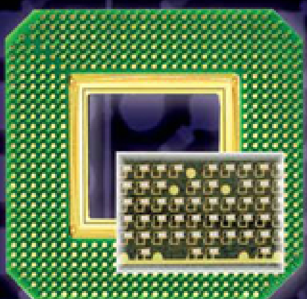
2010's

ORBIT ®
Built -In-Trace
HDI resistor



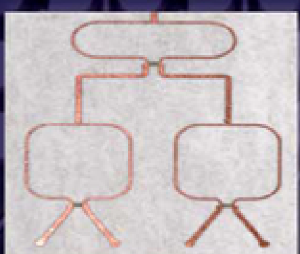
2000's

Series termination
under BGA device



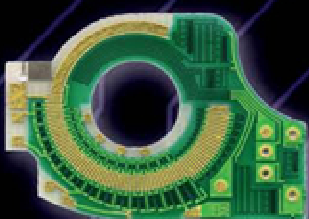
1990's

Parallel termination
of interposer board



1980's

Power divider
RF application



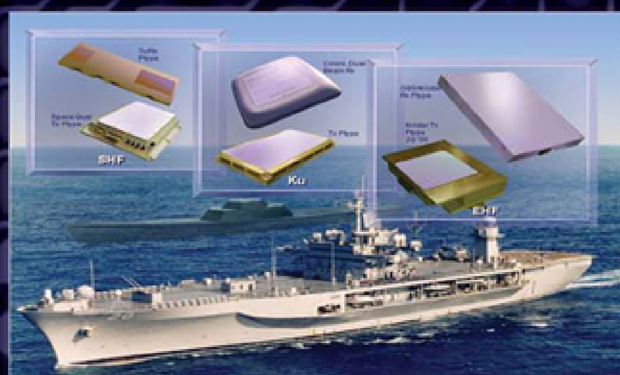
1970's

Potentiometer
surface resistor

Unsurpassed Reliability

Globalstar Satellite

Mars Express Lander



Phased Array Antenna

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

Ωhmega
Ohmega Technologies, Inc.

www.ohmega.com

single process. One company I visited proudly stated that they had 131 control charts in their SPC program. A customer does not want to see wallpaper; that many processes cannot possibly be controlled and managed properly. Customers want to see the processes controlled that the supplier has deemed critical to their business.

Another common observation is that many companies limit SPC control to the chemical processes, largely because most are using lab analysis software that includes an integrated statistical module. Ignoring the key mechanical, imaging and non-manufacturing processes limit the potential benefits that can be reaped by an organization.

Operator Level Involvement

There is a tendency for companies to relegate the SPC program to the process-engineering department. While this approach is better than nothing, it is not optimal as the greatest benefit would be achieved through active participation of the personnel actually running the process and producing product. A control chart's true function is to provide real-time feedback to control and improve a process. The program will fail if the data displayed on the charts do not help front line operators make better process decisions. Figure 1 shows a typical control chart for a process that is in statistical control.

Whatever the level of participation, SPC requires specialized formal training over and above the standard job function and operational training all employees receive. Statistical theory and methods, process capability, and control chart maintenance, interpretation and reaction are key topics to be covered in depth.

C_{pk} Analysis

A frequent scenario witnessed when I visit a company is to be rushed past a beautiful display of control charts, like the one shown in Figure

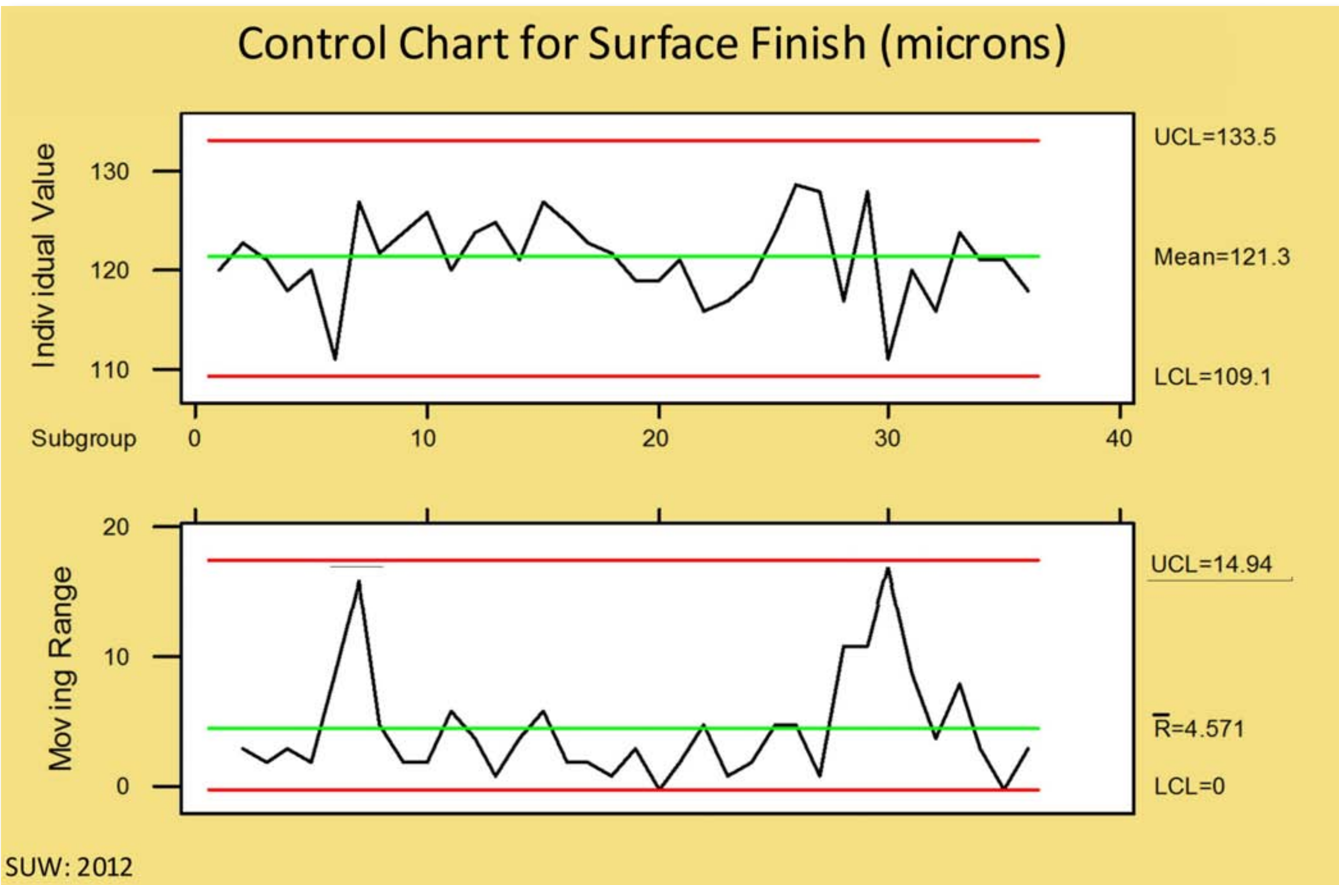


Figure 1: Typical control chart for a process that is in statistical control.

1, followed by some beads of sweat forming on my tour guide's brow when I stop and actually step closer to examine the data. I know they are thinking, "I hope Steve knows less about this than I do." When asked to see the time-trended C_{pk} analysis, I am invariably confronted with that glazed over, deer-in-the-headlights syndrome. Data collection and simple run charting is not SPC; process capability is the goal, and monitoring C_{pk} levels is the method.

This is where many SPC programs hit the wall; C_{pk} levels are calculated once, or not at all, and the process continues to run with no real understanding of capability. Once the proper spec limits have been defined and the process is fairly stable, incremental process improvement can be achieved through C_{pk} analysis. As my good friend Vince Lombardi once said, "Calculating process C_{pk} levels is not a sometime thing, it is an all-the-time thing."

Embrace the Concept

Truly understanding and properly applying SPC in the PCB manufacturing environment can realize significant gains in cost control, performance consistency, and most importantly, customer satisfaction. I was reminded of this just last year at our 93rd high school reunion, when another old friend (no pun intended) W. Edwards Deming told me "Steve, the result of long-term relationships is better and better quality, and lower and lower costs." As with most things Deming, it is advice worth taking. **PCB**

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

An Update on Imaging Technologies

by Karl H. Dietz
KARL DIETZ CONSULTING, LLC

SUMMARY: Imaging technology continues to evolve in the manufacturing of electronic devices, from contact printing using phototools, exposure equipment and photoresist, to alternatives such as DI and LDI, the best known and most widely practiced.

Introduction

The term imaging in manufacturing of electronic devices such as circuit boards usually refers to the formation of a circuit pattern from digital data by means of photolithography, typically by contact printing, involving the use of a phototool, exposure equipment and photoresist. Alternatives to contact printing have also found acceptance, notably digital, direct imaging technologies, of which laser direct imaging (LDI) is the best known and most widely practiced. It might be best to use the broader term DI (direct imaging) to label this technology since not all DI units employ lasers. Hayao Nakahara (N.T. Information Ltd) estimates that (cumulative) installations of DI units used in PCB fabrication were about 1,230 at the end of 2011.

The benefits of DI technology may be summarized as follows. Since phototool generation and conditioning are omitted, there is the advantage of shorter lead time. Small lots can be customized at no extra cost (e.g., with added date and lot number information). There may be an advantage in fine line imaging of surfaces with poor co-planarity because of the depth of focus of the laser beam. The biggest advantage may be the ability to scale, i.e, to change the dimension of each individual exposure for best fit to reference points on an underlying pattern of a multilayer structure. This is of particular value in soldermask LDI applications.

Inkjetting

Inkjetting is a unique form of DI which has been accepted in PCB fabrication as a viable alternative to screen-printing of legend print. MicroCraft and Orbotech are well known suppliers. Other applications such as inkjetting of an etch resist pattern or inkjetting of a soldermask pattern are less common. Camtek[1] has introduced a soldermask inkjet system that allows the selective coating thickness adjustment, depending on features to be covered and board surface location. The system uses a proprietary solder ink formulation, a fact that might limit

> **"Since phototool generation and conditioning are omitted, there is the advantage of shorter lead time."**

the acceptability of the system. Since Camtek acquired Printar, the company that pioneered soldermask inkjetting, it is a reasonable assumption that Camtek's system evolved from the Printar technology.

Use of Texas Instruments' DMD™ (digital micromirror devices)

The number of companies that offer DI systems that make use of Texas Instruments' DMD™ (digital micromirror devices) is growing. They include:

• ORC DI-Impact (formerly by Pentax). The DI-Impact offers a minimum feature size of 15 um L&S, position accuracy (repeatability) of +/- 7.5 um, and a side-to-side registration of +/- 10 micron. The exposure time is about 35 seconds for 340x510mm, depending on the photospeed of the resist. Use of 405 nm wavelength. High-pressure mercury lamp light source. Double-sided exposure, automated loading/unloading.

• Hitachi's DE imagers DE-H, DE-S, and DE-F series differentiate in their resolution capabilities. The imagers use a 405 nm diode laser, and make use of Texas Instrument's DMD™ micromirror array model XGA. The DE-H has the highest resolution, capable of producing 10 micron lines and spaces. The overlapping pixels have an addressability (center-to-center spacing) of 1.2 micron. The DE-S is capable of doing 20 micron L/S (2.6 micron addressability), and the DE-F does 40 micron L/S.

• Miva 2600X (MIVA Technologies GmbH, Germany). Direct imager, using Texas Instrument's DMD™ micromirror array and LED light source. Highest resolution model: 2600X-16, 16,000 dpi, image time 125 sec (15mj/cm2; 18x24"). Also offered by Printprocess AG (Switzerland), www.printprocess.com as system Apollon-DI. Distributed by KuperTek (www.KuperTek.com) in the USA.

• Maskless Lithography (USA)

• Aiscent Technologies Inc. (Canada)

A variety of light sources can be used with DMD™ technology, both laser and non-laser, since the pixel size and shape is defined by the micromirror and not by the foot print of a laser beam.

It should be noted that the early laser systems that used wasteful, short-lived argon-ion lasers (Figure 1) have been replaced by more efficient solid-state lasers. Only two of the argon-ion laser emission lines were actually used for LDI.

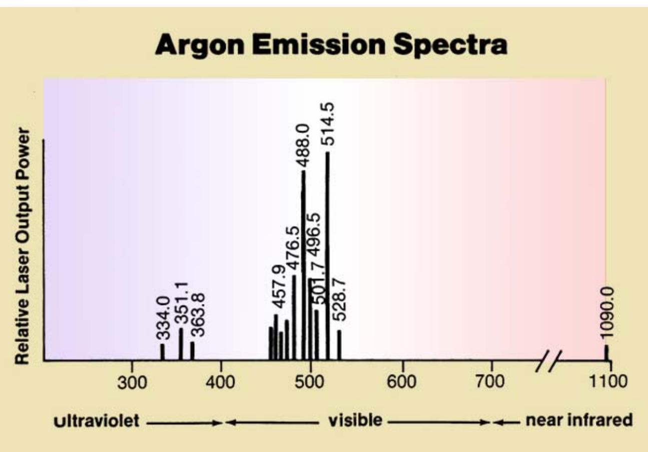


Figure 1: Argon-ion laser emission spectrum.

Use of LEDs

One of the fastest growing light sources for both conventional and DI applications in 2011 have been LEDs (light emitting diodes). It is worth reviewing the intrinsic advantages of LED light sources and looking at some of the exposure systems that are being offered. But first, a brief look at the packaging (interconnect) options for LEDs and LED arrays, in particular high-intensity LEDs.

High-brightness LEDs produce a lot of heat that needs to be removed, otherwise the junction temperature gets too hot, which is detrimental to the light output and the LED lifetime. Additionally, it can result in an undesirable spectral shift. Figure 2 shows a simplified LED package. Most LED packages are more complex,

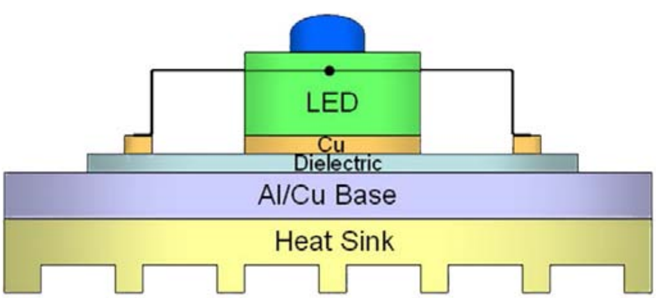


Figure 2: Illustration of a simplified LED package.

with large arrays of LEDs, and possibly more than one circuit layer to fan out interconnections. The thermal laminate for such a package may also be referred to as metal core PCB (MCPCB); sometimes the terms thermal interface material (TIM) or IMS (insulated metal substrate) are used.

LEDs are known to have a longer life, lower power consumption, and easier maintenance compared to alternative power sources such as mercury lamps or lasers.

Dainippon Screen Manufacturing Co. (Japan) introduced its LED powered DI system at the 2011 TPCA Show in Taiwan (Figure 3).

The company started commercialization in January 2012 and expects to sell about 100 units during the first year. Dainippon Screen claims that the Ledia 5 is the fastest DI system due to its high intensity, multi-wavelength UV-LED light source. It can use standard dry film photoresist and photoimageable soldermasks. Resolution down to 30 micron lines and spaces is achieved.

Miva Technologies (Germany) introduced its LED Direct Imaging System 2600X in 2010. Miva is represented in the U.S. by KuperTek and cooperates with Printprocess AG (Switzerland) in Europe. The 2600X-5 model achieves a minimum line width resolution of 2.5 mil (62 micron) and requires a 30-second exposure time @ 15 mJ/cm^2. It works with conventional dry film resist and soldermask, as well as LDI dry films, liquids and soldermasks. LED arrays with emissions at 365 nm, 390 nm, and 405 nm are available. The system makes use of DMD™ dynamic photomasks.

Bacher (Germany), a well known supplier of exposure units, is also offering an LED powered exposure unit called SupraLight®. This is a conventional contact printer. Illumination uniformity across the exposure area is +/- 5%. Resolution of 50 micron features is achieved. There is no temperature change of the phototool during long production runs. Dry film photoresist exposure time is 6-8 seconds depending on the resist's photospeed.

Figure 3: Ledia 5 Multiwavelength UV-LED direct imager.

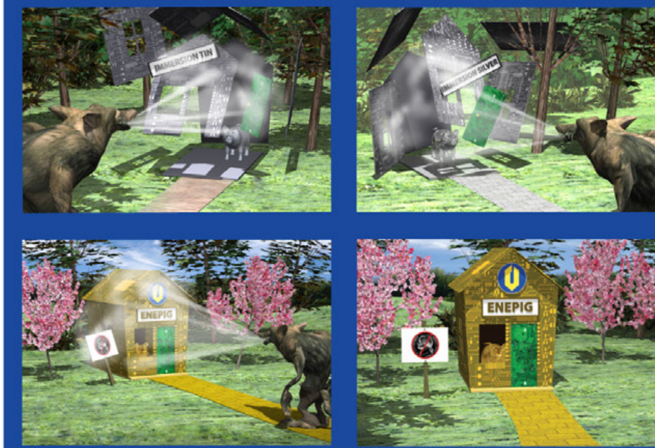

ENEPIG

Figure 4: Maskless Lithography's Model 2027 direct imager.

Limata GmbH[2](Germany), also through representation by Walter Lemmen GmbH, has introduced its UV-LED direct imager at productronica 2011. The UV-P100 UV-LED Direct Imager was developed for prototype and short run production. It can use conventional dry film and soldermask. The light source life is >10,000 hours. Max panel size: 650 x 540 mm2. Resolution: 50 micron. Automated load/unload are available. Side-to-side registration makes use of cameras and registration holes.

Gray-Level Imaging Technology

Maskless Lithography, Inc.[3] was the first to introduce gray-level imaging technology to the direct imaging of PCBs. The Model 2027 uses a mercury arc light source and DMD™ arrays (Figure 4).

Eric Hansotte describes the gray-level/micromirror principle as follows:

"Each of the multiple DMD images is carefully aligned and oriented so that its mirror columns are parallel to the stage motion during the exposing scans. This means that, at some time during the scan, each point on the substrate passes under an imaged column of 768 DMD mirrors. As it passes it receives doses, in serial, from some fraction of the 768 mirrors in the column. Whether or not a pixel receives a dose from an individual mirror depends on whether that mirror is on ("flipped" to direct light through projection lens) as the point passes under its projected image. Thus, in theory any point on the substrate can receive partial energy in discrete gray-level increments up to a maximum of 768, or the total number of mirrors in a DMD column. This gray-level control of exposed image pixels allows for fine placement of feature edges using relatively large projected mirror pixels (34 um)." **PCB**

References

1. Revolutionary solution for the PCB Industry: Digital Ink Jet Solder Mask Printing, Tomer Segev, Proceedings 2011 Technical Conference, P-26, APEX EXPO IPC, April 12-14, 2011, Las Vegas, NV

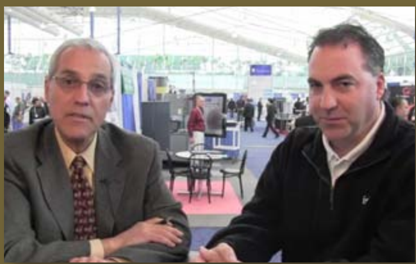
2. "Die neue Art der Elektronikfertigung: UV-LED Direktbelichter der Limata GmbH," Produktion von Leiterplatten und Systemen (PLUS), November 2011, pg.2593.

3. High Speed Digital Imaging Using Gray Level with Micromirror Array, Eric J. Hansotte et al, Proceedings 2011 Technical Conference, S 06-1, APEX EXPO IPC, April 12-14, 2011, Las Vegas, NV

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

Isola Introduces Four New Products

by Real Time with...IPC APEX EXPO 2012

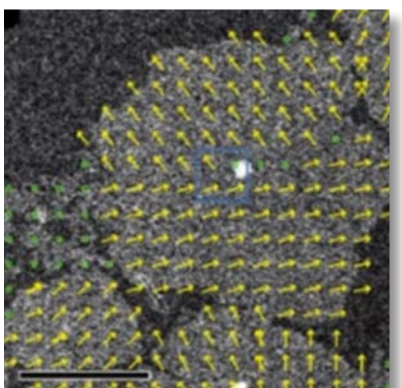


Rich Pangier, Director of OEM Marketing at Isola, discusses the company's new products being introduced at IPC APEX EXPO 2012 with Editor Ray Rasmussen.

realtimewith.com

Better Organic Electronics

Future prospects for superior new organic electronic devices are brighter now thanks to a new study by researchers with the U.S. Department of Energy (DOE) at Lawrence Berkeley National Laboratory (Berkeley Lab). Working at the lab's Molecular Foundry, a DOE nanoscience center, the team has provided the first experimental determination of the pathways by which electrical charge is transported from molecule-to-molecule in an organic thin film. The results also show how such organic films can be chemically modified to improve conductance.



Scanning transmission electron microscopy image of an organic thin film deposited on a silicon nitride membrane. Yellow arrows indicate the lattice orientation of each crystalline domain. Green circles mark polycrystalline areas.

"We have shown that when the molecules in organic thin films are aligned in particular directions, there is much better conductance," says Miquel Salmeron, a leading authority on nanoscale surface imaging who directs Berkeley Lab's Materials Sciences Division and who led this study. "Chemists already know how to fabricate organic thin films in a way that can achieve such an alignment, which means they should be able to use the information provided by our methodology to determine the molecular alignment and its role on charge transport across and along the molecules. This will help improve the performances of future organic electronic devices."

Organic electronics, also known as plastic or polymer electronics, are devices that utilize carbon-based molecules as conductors rather than metals or semiconductors. They are prized for their low cost, light weight and rubbery flexibility. Organic electronics are also expected to play a big role in molecular computing, but to date their use has been hampered by low electrical conductance in comparison to metals and semiconductors.

For a full description of the team's methodology and its applications, click here.

CPCA 2012:
The Impact of Current Events

by Marcy LaRont
I-CONNECT007

I was excited to attend the CPCA show this year. I haven't been since 2007, and China manufacturing has continued to evolve at a rapid pace during the past four years. (For an interesting and detailed discussion on China, the society, the economy, and manufacturing, I recommend tuning into The Changing Face of China and Manufacturing Perspectives: Europe, US and China, from Real Time with CPCA.

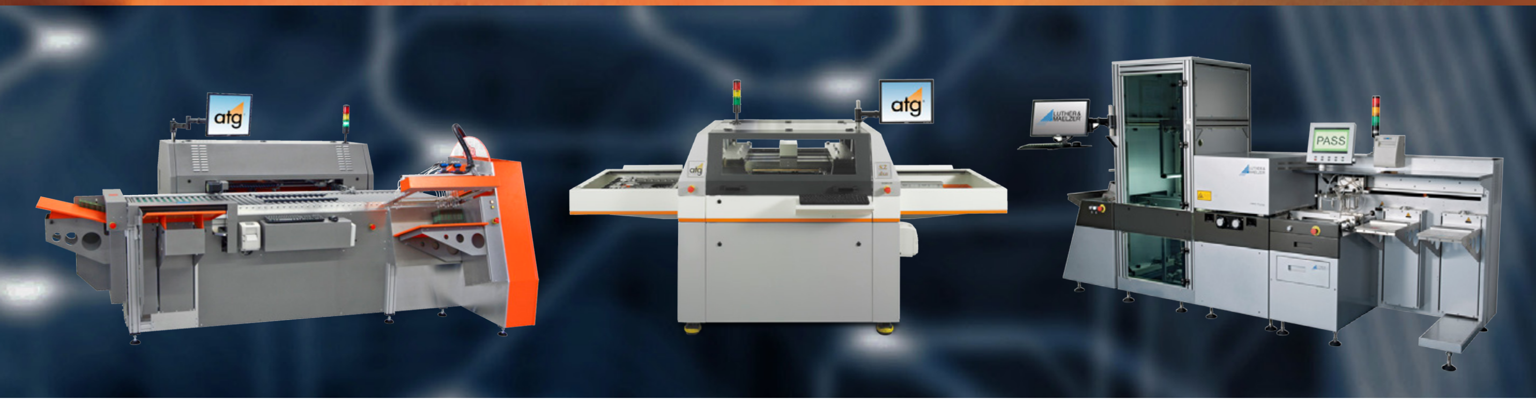
I arrived on Day 2 of the show, missing what was reportedly the best of the three show days according to several exhibitors we spoke to. The halls were not overly filled. The crowd, including booth personnel, seemed to be very young. Not everyone was displaying their machines, and not all machines present were actually working, something that seems to be more and more standard across the entire trade show circuit. Drills were well represented, with a couple of exciting new machines on display. And both Maskless Lithography and Orbotech kept their impressive machines running at continually busy booths, perhaps speaking directly to the robustness of the digital imaging market. Yet for me, overall, the show was disappointing.

The venue was the "new" Shanghai World Expo Center. It was cold inside the show, something one can only assume was a conscious cost-saving decision on the part of the organizers. Exhibitors were bundled up in coats and gloves. There was just one small restaurant inside the exhibition, an Italian café, which is something that anyone who has been to China would likely have stayed away from if there had been any other choice in food. And the personal facilities were not as clean as advertised.

But logistical annoyances aside, what really stood out was the fact that people were missing. There was a much smaller-than-normal international visitor contingent. And some of the multinational companies I expected to see didn't bother to exhibit this year. A representative from one of those companies made the passing comment, "You couldn't trip over a decision-maker here," a sentiment that may have been overstated, but then again, maybe not, based on the crowd that appeared to be college-aged.

So, why was CPCA 2012 so underwhelming? I believe there are two primary reasons.

Your Single Source for Your Electrical Test Needs



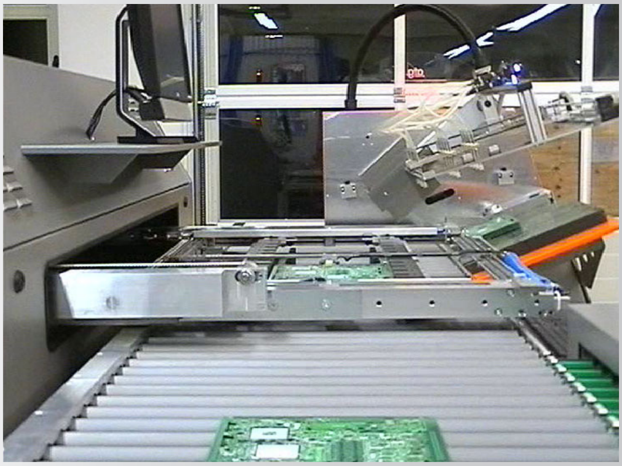
A5a
Fully automatic flying probe test system

S2
High speed Flying probe test system

LM800 Picomat
Grid test with Sniper optical board alignment

atg Luther & Maelzer is a leading supplier of electrical testing solutions for the pcb industry. We offer solutions for every application ranging from main stream technology all the way to HDI products such as smart phones and tablet PC technology.

Our atg flying probe tester products are renowned for speed, accuracy and automation while the Luther & Maelzer grid tester product range is known world wide for high throughput, repeatability and flexibility.



atg

LUTHER & MAELZER

www.atg-lm.com
Phone: +49 (9342) 291-0
E-mail: sales@ atg-lm.com

The CPCA found itself as the end cap of a staggeringly busy tradeshow season that began in October 2011. To refresh, this global tradeshow line-up went as follows:

- TPCA, November 9-11, 2011
- productronica, November 15-18, 2011
- HKPCA, November 30-December 2, 2011
- (US/Europe – Christmas/New Year Holidays – late Dec 2011)
- (China/Asia – Chinese New Year Holidays – late January 2012)
- IPC APEX EXPO 2012, February 28-March 1, 2012
- CPCA, March 13-15, 2012

Quite simply, that is far too many tradeshows for many of the same key players to exhibit. Despite my assertions in my coverage of TPCA that many would naturally wait for the larger HKPCA or CPCA shows to see everything all at once, that just didn't seem to pan out this time around for the CPCA Show.

The second reason this year's CPCA Show just didn't meet expectations is more subtle and far more important. In a casual discussion about the show and booth traffic, one exhibitor commented, "It's not like it used to be. Chinese customers and manufacturers will come to us now to see our equipment. And we go to see them. It used to be that everyone had to come to the CPCA Show to see a certain segment of the China market. Now everyone travels everywhere."

Travel visas in and out of China are far easier to get than in times past. The Chinese government has been and continues to be supportive of multinational corporations expanding operations into China. And with China's own economic woes, it only seems more likely we will see greater openness in the back-and-forth between China and the international business world.

Thus, things have progressed naturally, as is to be expected in any society where the bulk of its economic growth is pinned to capitalist business expansion and growth, even if it is a special brand of capitalism with a uniquely Chinese twist to it. Customers and suppliers regularly travel to conduct their business. And for places like China, perhaps it makes the tradeshow circuit slightly less essential than it has been in the recent past. But we all have to change and adapt our businesses to survive, and the CPCA Show will no doubt survive and thrive going forward.

But as for the CPCA Show 2012, this just wasn't its year. **PCB**

Marcy McAllister LaRont has worked in the PCB industry for more than twenty years, and has held positions in EH&S, purchasing, production control, customer service and sales. Prior to joining the I-Connect007 team in 2005, she worked for Data Circuits (Merix), Trend Circuits/Praegitzer Industries, and Hadco/Sanmina-SCI. LaRont currently resides in Hong Kong. Contact LaRont at marcy@iconnect007.com.

185HR

Laminate and Prepreg

RoHS Compliant

Next Generation Phenolic Epoxy for Demanding High Reliability Applications

185HR - Laminate Properties

Property	Units	FR406	Competitive Phenolic FR-4s	185HR
Tg, DSC	°C	170	>170	>170
Td, 5% (TGA-ASTM)	°C	300	340	340
CTEz-axis (50-260C)	%	4.0	2.7 - 3.5	2.7
T-260	Min.	10	>30	60
T-288	Min.	>2	>10	>15
Dk, 100 MHz	-	3.91	-	4.19
Dk, 1 GHz	-	3.86	-	4.07
Dk, 2 GHz (Bereskin)	-	3.83	4.0 - 4.8	4.04
Dk, 5 GHz (Bereskin)	-	3.81	3.9 - 4.8	3.92
Dk, 10 GHz (Bereskin)	-	3.81	3.9 - 4.7	3.92
Df, 100 MHz	-	0.0134	-	0.0155
Df, 1GHz	-	0.0171	-	0.0188
Df, 2 GHz (Bereskin)	-	0.0177	0.019 - 0.025	0.0195
Df, 5 GHz (Bereskin)	-	0.0182	0.021 - 0.26	0.0229
Df, 10 GHz (Bereskin)	-	0.0182	0.022 - 0.027	0.0230
UL	-	FR-4	FR-4	FR-4, Family with 370HR

Dk Df data at 54 % resin

185HR Product Attributes

- Tg >170 C
- Td 340 C
- CTE z-axis 50 to 260 C - 2.7 %
- Lead Free Assembly compatible
- IPC 4101 C Slash Sheets / 21 / 24 /26 / 98 / 99 / 101 / 126
- UL Family with 370HR
- Global Availability - Manufactured in North America and Asia Pacific
- Supported by Isola Sales, Technical Service and Analytical Services
- Lead Free Assembly Compatible across a wide range of designs
- Stocked material in NA for Quick Turn requirements

For more detailed product information:
www.isola-group.com/185HR

Isola Corp., 3100 West Ray Rd., Suite 301, Chandler, AZ 85226,
+1-480-893-6527, www.isola-group.com

isola
Supplying the base for innovation

Top Ten Most-Read Flex007 Highlights This Month

All Flex Upgrades Capabilities with LDI Purchase

All Flex Flexible Circuits announced this week it has purchased a Paragon™ 9800 laser direct imaging (LDI) system that will improve image quality with fine-line, tight registration accuracy, and high-speed performance.

Merlin Flexible Circuits Receives SC21 Bronze Award

Flexible and flex-rigid PCB manufacturer Merlin Flexible Circuits, based in Sussex, has been awarded a prestigious SC21 Bronze award at a presentation in Edinburgh.

Cosmotronic Earns NADCAP Accreditation

TC Cosmotronic, Inc. has received NADCAP accreditation for the manufacture of rigid, flex, rigid-flex, and high-density interconnect PCBs.

Printed Circuits Inc. Acquires ProX5 from IMPEX

Printed Circuits, Inc. has purchased a ProX5 scanner/measurement machine from IMPEX Leiterplatten, St. Michael, Austria. This will be the very first ProX5 unit to be placed in the United States.

Merlin Flex-Ability Earns AS9100 to Rev C

"Our team has put considerable effort into strengthening our systems, processes and procedures in order to achieve this accreditation and to provide our customers with the very best service possible," said Managing Director Mark Merifield.

Printed and Potentially PE to Reach $9.4 Billion in 2012

IDTechEx find that the market for printed and potentially printed electronics in 2012 will be $9.4 billion. This includes devices not yet printed today but which are moving towards being printed. Of this market, 30% of the devices studied are made predominately by printing, and 6% are on a non-rigid substrate.

Stevenage Circuits Names Les Browne Director of Sales

Robert Brown, Managing Director of Stevenage Circuits, a UK-based PCB fabricator, announced the appointment of Les Browne (formally with DYCONEX) as the company's director of sales.

Arlon Posts 7.8% Sales Increase Driven by China Telecom

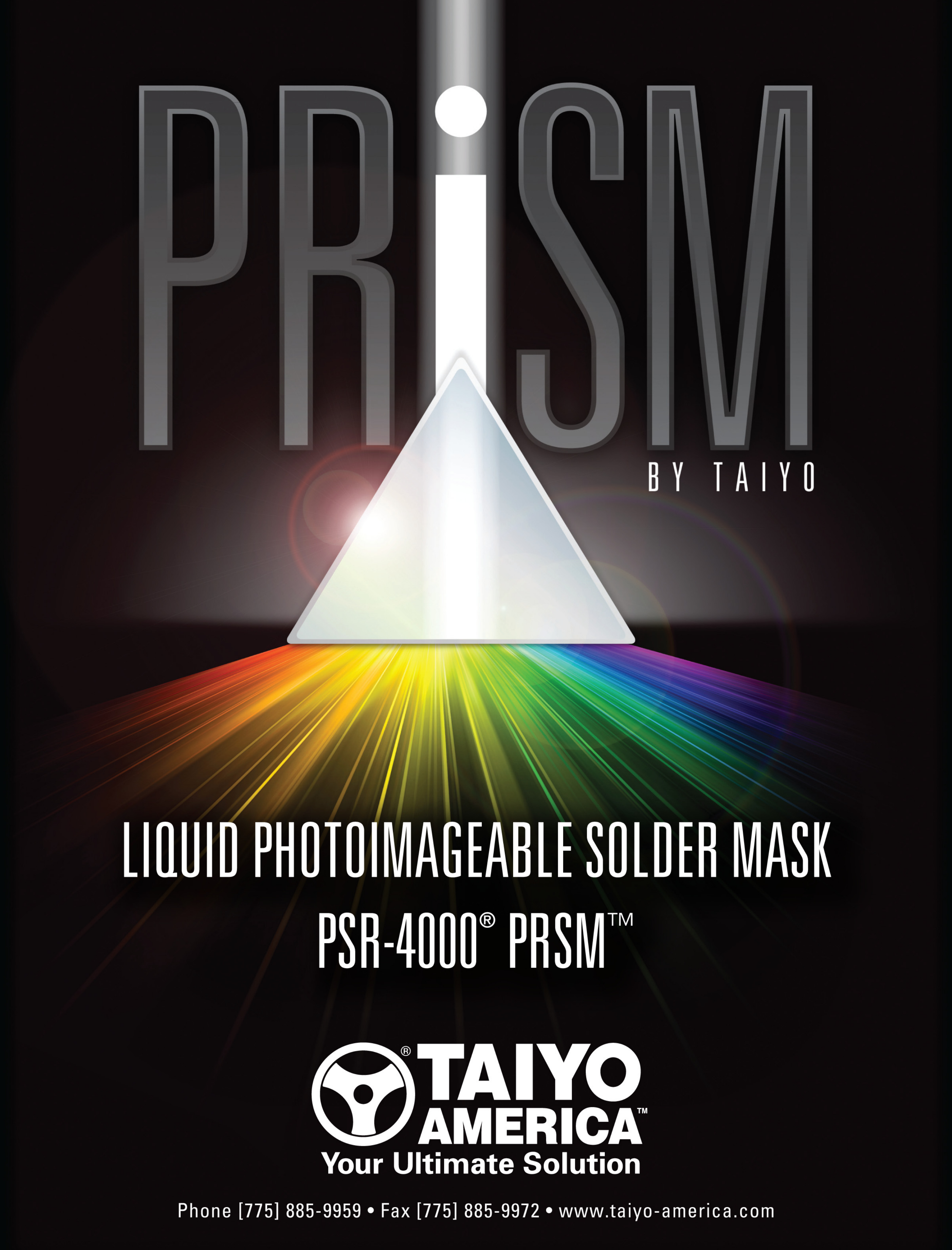
The sales increase was primarily due to increased sales of PCB materials related to the telecommunications infrastructure in China, as well as increased sales of flex heater and coil insulation products for the general industrial market.

Teknoflex Achieves Further UL Component Recognition

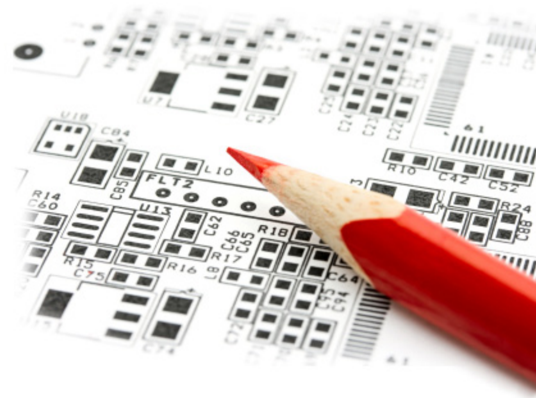
Teknoflex, the UK's largest dedicated flexible and flex-rigid circuit manufacturer, has recently been awarded an extension to its UL component recognition scope.

UniPixel Releases Q4 and Full Year 2011 Results

"2011 was a year of extraordinary progress, as UniPixel continued to transition from a development and IP-based company to a scalable manufacturing company," said Reed Killion, UniPixel's president and CEO. "We now have consumer electronics OEMs around the world extensively testing our UniBoss and Diamond Guard products. During the year we also strengthened our globally-oriented management team in preparation for the next stage of our growth."


PRISM
BY TAIYO
LIQUID PHOTOIMAGEABLE SOLDER MASK
PSR-4000® PRSM™
TAIYO AMERICA™
Your Ultimate Solution
Phone [775] 885-9959 • Fax [775] 885-9972 • www.taiyo-america.com

Stackup Planning and the Fabrication Process



by Barry Olney
IN-CIRCUIT DESIGN PTY LTD | AUSTRALIA

SUMMARY: Before starting a PCB design, we need to plan the PCB stackup for optimized performance, ensure that the selected substrate materials are available and clearly document the stackup so that it can be fabricated to engineering specifications.

Back in 1987, when I first started working on high-speed designs, the PCB stackup did not seem that important. If a board would not route, I simply added another couple of signal layers—problem solved. But that was running at a maximum frequency of 166 MHz, which at the time seemed fast.

Times have changed. Now, multi GB/s designs are becoming the "norm" and the stackup configuration, characteristic and differential impedance control are crucial to the performance and reliability of the product.

Rick Almeida of Downstream Technologies, Inc. pointed out in his article, "What to Look Forward to in 2012" (*The PCB Magazine*, January 2012) that "The growing importance of stackup design and analysis will be recognized. Today, there is a growing need to plan and design the stackup properly, as well as optimize it for better overall performance and communicate and document stackup information between engineering and fabrication."

I have previously written on the topic as well, but perhaps some designers are not quite at this level, yet. Therefore, let's look at some basic stackup configurations and briefly cover what is required for the fabrication process.

Dielectric Materials

Before starting a PCB design, we need to plan the PCB stackup and ensure that the selected substrate materials are available from our chosen fabrication—a step that is regularly missed. Changing the stackup towards the end of the design process could mean changing trace widths and clearances to achieve the correct impedance, which

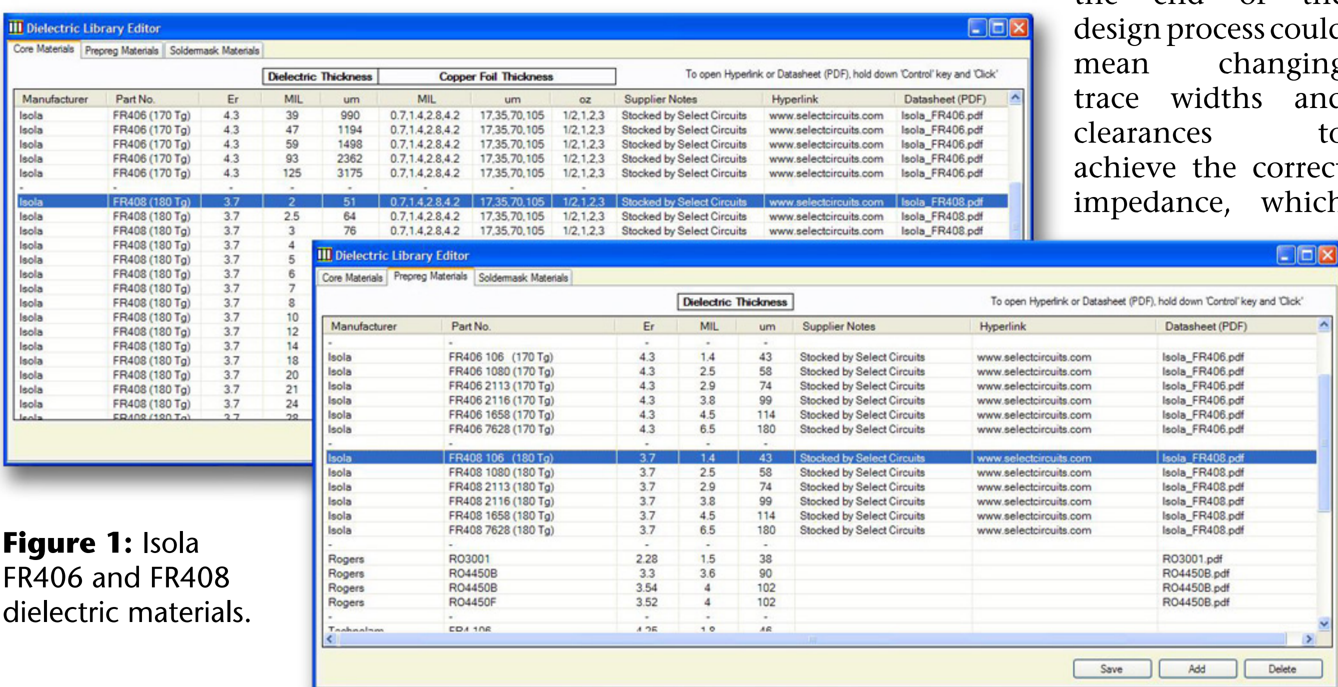


Figure 1: Isola FR406 and FR408 dielectric materials.

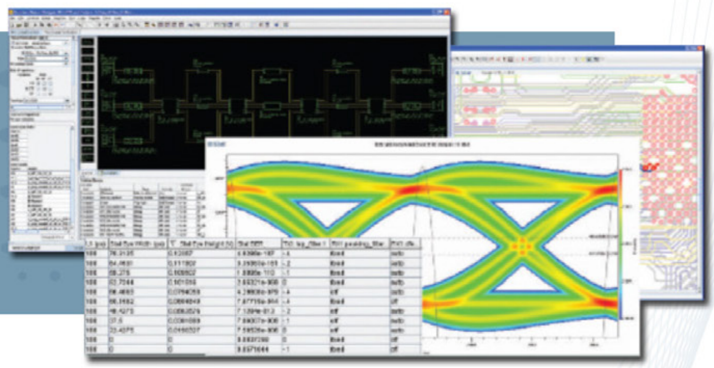
Consulting Services and Methodology Training
Automated Analysis Methodologies
Award-Winning EDA Tools
IBIS-AMI Modeling

Advanced Signal Integrity Solutions

for Full System Level Design

Quantum Channel Designer®

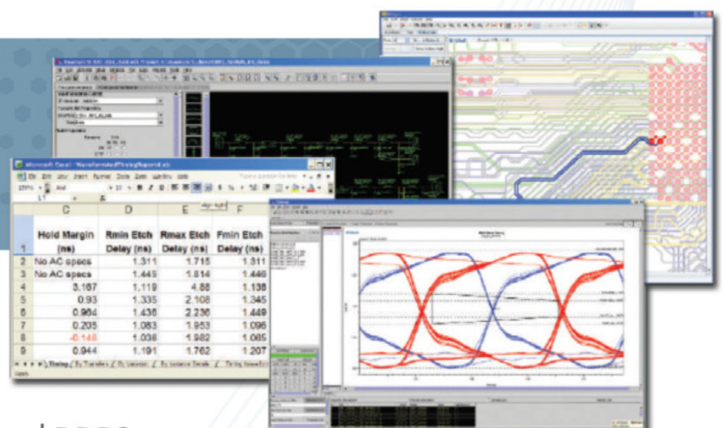
Multi-Gigabit Serial Link Design and Analysis



- Most Certified IBIS-AMI Models & Kits
- Broadest Correlation
- Automated Standards Compliance
- Intuitive, Superior Use Model

Quantum-SI™

Automated SI Analysis for High-Speed Parallel Interfaces



- Integrated Signal Integrity, Static Timing and Crosstalk Analysis
- Design Kits Offering ready-to-run setups for popular interface standards like DDR2 and DDR3

To learn more about available design kits for vendors including Intel, IBM, TI, Xilinx, Altera and more, visit www.sisoft.com/elearning today.

SiSoft™
We Are Signal Integrity
Signal Integrity Software, Inc.
www.sisoft.com
(978) 461-0449

FR406/8 Materials	Final Thickness	Combined Prepreg Materials = Total Thickness
106	1.4	106 + 106 = 2.8 mil
1080	2.5	1080 + 1080 = 5 mil
2113	2.9	2113 + 2116 = 6.7 mil
2116	3.8	2113 + 2116 + 1080 = 9.2 mil
1658	4.5	2 x 2116 + 1658 + 7628 = 18.6 mil etc.
7628	6.5	*There are many combinations to achieve the desired thickness.*

Table 1.

could create a lot of unnecessary work.

If we use the same materials that the fab shop stocks to build our stackup, then the impedance will be more accurate. If we just choose a convenient number, for core thickness, for example, then this may be up to 3% different off from what is available; hence, the impedance will vary by 3%.

The most widely used dielectric material is FR-4 and may be in the form of core or prepreg (pre-impregnated) material. Isola's top selling materials are FR406 and FR408. While FR406 sets the industry standard for basic multilayer PCB fabrication, FR408 is a high-performance FR-4 epoxy dielectric for improved signal performance. Its low dielectric constant and low dissipation factor make it an ideal candidate for broadband circuit designs requiring fast signal speeds or improved signal integrity. Also, the high glass transition temperature makes it compatible with ROHS compliant components and most FR-4 processes. Figure 1 illustrates the available core materials in the default Dielectric Library Editor of the ICD Stackup Planner (download from www.icd.com.au).

The core material is a thin dielectric (cured fibreglass epoxy resin) with copper foil bonded to both sides. For instance: Isola's FR406/8 materials include 2, 2.5, 3, 4, 5, 6, 7, 8, 10, 12, 14, 18, 20, 21, 28, 31, 36, 39, 47, 59, 93 and 125 mil cores. The copper thickness is typically ½ to 2 oz (17 to 70 um). If you look at the datasheets from Isola (or any other manufacturer) these sizes are not very clear, so it is best to contact your fab shop for the details of the materials they stock.

The prepreg material is a thin sheet of fibreglass impregnated with uncured epoxy resin which hardens when heated and pressed during the PCB fabrication process. Isola's FR406/8 materials include 1.4, 2.5, 2.9, 3.8, 4.5, and 6.5 mil prepregs that may be combined to achieve the desired prepreg thickness. These are the final thicknesses after processing (Table 1).

The most common stackup, known as the Foil Method, is to have prepreg with copper foils bonded to the outside on the outermost layers (top and bottom), then core alternating with prepreg throughout the substrate. An alternate stackup is called the Capped Method, which is the opposite of the Foil Method and was used by old-school military contractors, but may still be fabricated today.

The total substrate thickness is generally 62 mils (1.6 mm) but may vary according to the application: 20, 31, 40, 47, 62, 93 and 125 mils are other not-so-typical thicknesses. Obviously, as the layer count increases the total board thickness increases. Twelve layers is the limit for 62 mil substrates.

The configuration of the PCB stackup depends on many factors, but whatever the requirements, one should ensure that the following rules are followed in order to avoid a possible debacle:

- All signal layers should be adjacent to and closely coupled to a reference plane, creating a clear return path and eliminating broadside crosstalk
- There is good interplane capacitance to reduce inductance at high frequencies

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

UNITS: MIL — ICD STACKUP PLANNER – www.icd.com.au — 2/27/2012 — Total Board Thickness: 59.2

Differential Pairs > Diff Pair

Layer		Material	Dielectric		Copper	Trace		Current	Impedance	Edge Coupled	Broadside Coupled	
Number	Name	Type	Constant	Thickness	Thickness	Clearance	Width	(Amps)	Characteristic(Zo)	Differential(Zdiff)	Differential(Zdbs)	Description
		Dielectric	3.3	0.5								Soldermask
1	Top	Conductive			0.7	20	12	0.42	54.82	101.71		Signal
		Dielectric	4.3	8								Prepreg
2	GND	Conductive			1.4							Plane
		Dielectric	4.3	39								Core
3	VCC	Conductive			1.4							Plane
		Dielectric	4.3	8								Prepreg
4	Bottom	Conductive			0.7	20	12	0.42	54.82	101.71		Signal
		Dielectric	3.3	0.5								Soldermask

Figure 2: Four-layer board.

UNITS: MIL — ICD STACKUP PLANNER – www.icd.com.au — 3/1/2012 — Total Board Thickness: 58.4

Differential Pairs > Diff pair | New Pair

Layer		Material	Dielectric		Copper	Trace		Current	Impedance	Edge Coupled	Broadside Coupled	
Number	Name	Type	Constant	Thickness	Thickness	Clearance	Width	(Amps)	Characteristic(Zo)	Differential(Zdiff)	Differential(Zdbs)	Description
		Liquid Photo Imageable	3.3	0.5								Soldermask
1	Top	Conductive			0.7	20	12	0.42	55.74	103.99		Signal
		FR408 2116 (180 Tg)	3.7	3.8								Prepreg
		FR408 2116 (180 Tg)	3.7	3.8								Prepreg
2	GND	Conductive			1.4							Plane
		FR408 (180 Tg)	3.7	39								Core
3	VCC	Conductive			1.4							Plane
		FR408 2116 (180 Tg)	3.7	3.8								Prepreg
		FR408 2116 (180 Tg)	3.7	3.8								Prepreg
4	Bottom	Conductive			0.7	20	12	0.42	55.74	103.99		Signal
		Liquid Photo Imageable	3.3	0.5								Soldermask

Figure 3: Four-layer board with dielectric materials.

- High-speed signals should be routed between the planes to reduce radiation
- The substrate should be symmetrical with an even number of layers, which prevents the PCB from warping during manufacture and reflow
- The stackup should accommodate a number of different technologies
- Cost (the boss's most important design parameter) should also be addressed

It is not always possible to configure the stackup to have both tight coupling of the planes and tight coupling of the signal layers to the planes, as this depends on the number of layers and the available materials. Four- and six-layer boards typically have this issue. Fortunately, the lower layer count boards are generally used for designs below 100MHz so the interplane capacitance may not be as important at these frequencies.

In the four-layer board (Figure 2) the signal layers are < 10 mils from the plane, which helps reduce crosstalk, and the single-ended (Z_o) and differential impedances (Zdiff) are 54 and 101, respectively. A good range of impedance (Z_o) is from 50 to 60 ohms. Keep in mind that lower impedance will increase the dI/dt and dramatically increase the current drawn (not good for the PDN); higher impedance will emit more EMI and also make the design more susceptible to outside interference.

In order to communicate our intent to the PCB fab shop, we need to produce documentation detailing the required materials. In Figure 3, the centre core of 39 mils thickness has been replaced by a core from the Dielectric Materials Library which fortunately is also 39 mils thick. Also, in order to get a total of 8 mils thickness of prepreg between layers 1 and 2, as well as between layers 3 and 4, two sheets of FR408 2116 material are used, each totaling 7.6 mils. As mentioned, we cannot always get the exact value and this is where the inaccuracy occurs. The impedance Z_o drops by 1.6 ohms (2.8%) and Zdiff by 2.4 ohms (2.3%). And, the total board thickness reduces.

The six-layer board in Figure 4 again has close coupling between the signal layers and the planes—reducing crosstalk. The bulk of the board thickness is again made up of the centre core material. Broadside coupling between signal layers is reduced by keeping signal layers 3 and 4 far apart, and close to the reference planes. Ideally, these signal layers should be routed diagonally (or at 45 degrees) to each other to minimize the coupling. Routing pairs should be layers 1 and 3 (GND reference) and

UNITS: MIL — ICD STACKUP PLANNER – www.icd.com.au — 3/8/2012 — Total Board Thickness: 62

Differential Pairs > Diff Pair

Layer Number	Layer Name	Material Type	Dielectric Constant	Dielectric Thickness	Copper Thickness	Trace Clearance	Trace Width	Current (Amps)	Impedance Characteristic(Zo)	Edge Coupled Differential(Zdiff)	Broadside Coupled Differential(Zdbs)	Description
		Dielectric	3.3	0.5								Soldermask
1	Top	Conductive			0.7	12	6	0.25	53.52	101.04		Signal
		Dielectric	4.3	4								Core
2	GND	Conductive			1.4							Plane
		Dielectric	4.3	6								Prepreg
3	Inner 3	Conductive			1.4	12	6	0.42	53.33	100.28		Signal
		Dielectric	4.3	35								Core
4	Inner 4	Conductive			1.4	12	6	0.42	53.33	100.28		Signal
		Dielectric	4.3	6								Prepreg
5	VCC	Conductive			1.4							Plane
		Dielectric	4.3	4								Core
6	Bottom	Conductive			0.7	12	6	0.25	53.52	101.04		Signal
		Dielectric	3.3	0.5								Soldermask

Figure 4: Six-layer board.

UNITS: MIL — ICD STACKUP PLANNER – www.icd.com.au — 3/8/2012 — Total Board Thickness: 62.6

Differential Pairs > Diff Pair

Layer Number	Layer Name	Material Type	Dielectric Constant	Dielectric Thickness	Copper Thickness	Trace Clearance	Trace Width	Current (Amps)	Impedance Characteristic(Zo)	Edge Coupled Differential(Zdiff)	Broadside Coupled Differential(Zdbs)	Description
		Liquid Photo Imageable	3.3	0.5								Soldermask
1	Top	Conductive			0.7	12	6	0.25	53.52	101.04		Signal
		FR406 (170 Tg)	4.3	4								Core
2	GND	Conductive			1.4							Plane
		FR406 2113 (170 Tg)	4.3	2.9								Prepreg
		FR406 2113 (170 Tg)	4.3	2.9								Prepreg
3	Inner 3	Conductive			1.4	12	6	0.42	52.95	99.7		Signal
		FR406 (170 Tg)	4.3	36								Core
4	Inner 4	Conductive			1.4	12	6	0.42	52.95	99.7		Signal
		FR406 2113 (170 Tg)	4.3	2.9								Prepreg
		FR406 2113 (170 Tg)	4.3	2.9								Prepreg
5	VCC	Conductive			1.4							Plane
		FR406 (170 Tg)	4.3	4								Core
6	Bottom	Conductive			0.7	12	6	0.25	53.52	101.04		Signal
		Liquid Photo Imageable	3.3	0.5								Soldermask

Figure 5: Six-layer board with dielectric materials.

4 and 6 (VCC reference). High-speed signals should be routed on the stripline layers 3 and 4 to reduce EMI.

Figure 5 illustrates the completed stackup with all the dielectric materials defined, ready for pasting into the PCB specification and sent off with the Gerber and NC Drill data to the fab shop for manufacture.

Please see my previous article "The Perfect Stackup for High-speed Design" for information on additional stackups.

Points to remember:

- Control impedance to ensure performance and reliability of the product
- Keep the signal layers close to the planes to reduce crosstalk
- Avoid broadside coupling by separating the internal signal layers with bulk core material
- Combine prepreg and core materials from the datasheets, in the stackup, in place of the defaults to improve accuracy
- Ensure that the selected substrate materials are available from your chosen fab shop **PCB**

References:

1. "What to Look Forward to in 2012" by Rick Almeida, The PCB Magazine January 2012

2. Electromagnetic Compatibility Engineering—Henry Ott

3. "The Perfect Stackup for High-speed Design", by Barry Olney, The PCB Magazine November 2011

4. The ICD Stackup Planner and ICD PDN Planner can be downloaded from www.icd.com.au

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

FEATURE

Circuit Imaging Technologies—a Retrospective

by Joseph "Joe" Fjelstad
VERDANT ELECTRONICS

SUMMARY: Printed circuit imaging technologies have made remarkable strides over the 70+ years that the industry has been in volume production. Improvements made in printing and imaging technologies have enabled product designers to hold to their long-standing mantra of faster, smaller, lighter, better and cheaper.

Imaging and hole formation are the two primary gatekeeper technologies used in the manufacture of printed circuits. Arguably, this has been true since the origins of printed circuit manufacturing, going back to the 1940s. While there have been measurable and important changes and improvements in the materials used over the years, the limits of the technology have been largely paced by circuit feature dimensions and plated through-hole diameters. While much has changed in terms of the technologies used in the manufacture of printed circuits over the last seven decades, what has remained constant is the relentless pursuit of ever finer features, in an ongoing effort to make ever smaller, denser and higher-performing electronic products, while reducing cost.

If one looks back to the earliest days of printed circuit manufacturing, one will find that the operative word is printing. This is because the earliest printed circuits were in fact printed, using combinations of conductive and resistive inks, to which were affixed the electrical components required to complete the circuit assembly. Circuit feature sizes were coarse by today's standards, but then so were the components used in those early electronics assemblies. In the days before the transistor, the vacuum tube served as switch for logic circuits. Vacuum tubes were large components and generated a great deal of heat, as a result substrates tended towards refractory materials such as ceramic and circuits were printed and fired to the ceramic surface.

This first approach to manufacturing was, in retrospect, an obvious one. It was consistent with the techniques that had been developed in the past. In fact, it is arguable that printing is the oldest of man's technologies, dating back approximately 35,000 years. This argument is supported by the discovery of the Chauvet caves of Southern France in 1994, where were found the oldest known pictorial creations of humankind on the cave walls. There are actually two different printing techniques demonstrated by our ancient ancestors on these cave walls. The first is transfer printing and the second is stencil printing. The former method was accomplished by placing their hands on a wet pigment and transferring their print to the cave wall. The latter method was performed by placing their hands flat against the wall, and presumably, based on what is known of the methods of the aboriginal people of Australia, spraying the pigment over the backs of their hands using their mouths to expel it and create a stencil outline of their hands.

That digression aside, printing has remained an integral part of the circuit manufacturing process since that time. The evolution of the printing process as applied to printed circuit manufacture has been one which has taken many twists and turns. To put things in proper perspective, and appreciate the creativity of our technological forefathers, one should avail oneself of a pamphlet published by the U.S. Government printing office in 1946, entitled, Printed Circuit Techniques, which is believed to be the first-ever publication on printed circuit technology. In the text, more than 25 different methods for manufacturing a printed circuit are described. Included were imaging and etching techniques, discussions of printing and reducing inks containing metal salts to make conductors and even flame spraying of metals through stencils to create circuit patterns. Echoes of many of those technologies can be heard today in the developing stock of technologies used for circuit manufacture.

For purposes of review and consideration, there are several basic methods for creating circuit patterns on insulating base materials and a number of not-so-common methods. All of these methods are illustrated in brief in the accompanying graphic. Presently the most common methods for imaging used in manufacturing are print and etch and print and plate, also called patterned plating. While the print and etch process is straightforward, patterned plating methods are more nuanced and can include the use of etch-resistive metals over plates or the employment of so-called differential etch techniques. The latter method when used in combination with suitable imaging technologies can achieve very fine circuit features.

> ***"Presently the most common methods for imaging used in manufacturing are print and etch and print and plate, also called patterned plating."***

Circling back to imaging technologies and their capabilities, there have been remarkable strides made in the last few years to create feature sizes smaller than those first offered for semiconductors on how sizes more than two orders of magnitude larger than those used for early ICs. Technology roadmaps for interconnection such as the IPC's Electronic Interconnection Roadmap are projecting need for circuit features as small as 10µ in the coming years and technologies are rising to the challenge. One example is Rainbow Technologies' recently introduced imaging solution which can reliably resolve features down to 10µ line and space on both sides of panels 610x460mm (24"x18") in size at rates of up to 200 pounds per hour. This represents an unprecedented leap in both productivity and dimension capability.

Not to be forgotten are the various direct printing techniques for imaging circuits which can be done both additively and subtractively. In the former case, circuits are being printed using catalytic inks that can be plated subsequently, and, in the latter case, thin metal foils can be ablated directly using lasers. Obviously, the techniques used must be consistent with the end use objectives of the circuit and assembly techniques to be applied.

In summary, printed circuit imaging technologies have made remarkable strides over the 70+ years that the industry has been in what can be called volume production, though obviously volume is a relative term today compared to the

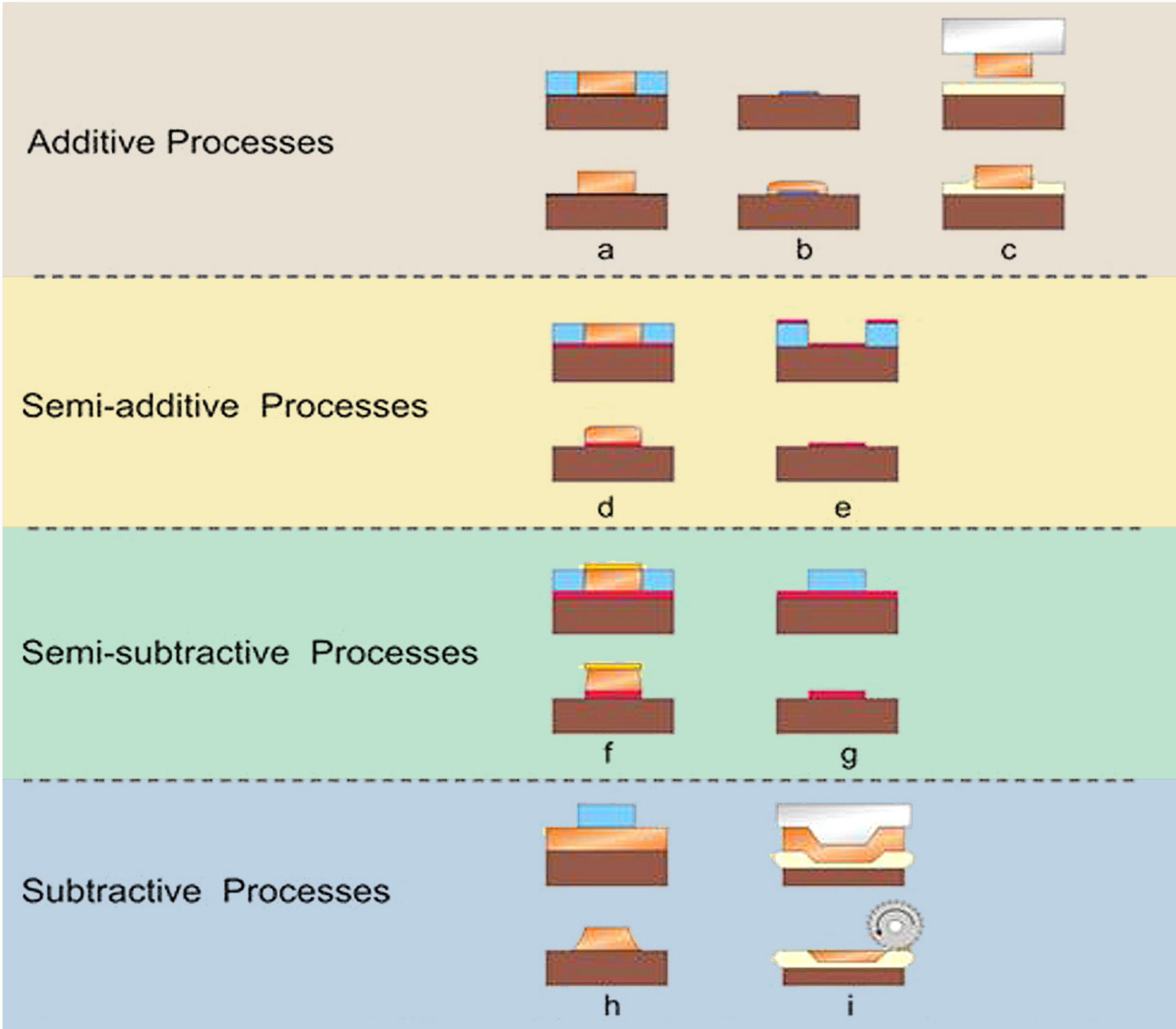


Figure 1: Examples of various processing methods for creation of single conductor layer circuits.

past. Improvements made in printing and imaging technologies have enabled product designers to continue to hold to their long-standing mantra of faster, smaller, lighter, better and cheaper.

The red layers in Figure 1 are thin conductive films which can be produced by any one of several different methods and of many different metals, though copper is most common. This image is reproduced from the book Flexible Circuit Technology, 4th Edition, where the various methods illustrated are discussed in more detail. The book can be downloaded for free at www.flexiblecircuittechnology.com. **PCB**

Verdant Electronics Founder and President Joseph (Joe) Fjelstad is a four-decade veteran of the electronics industry and an international authority and innovator in the field of electronic interconnection and packaging technologies. Fjelstad has more than 250 U.S. and international patents issued or pending and is the author of *Flexible Circuit Technology.*

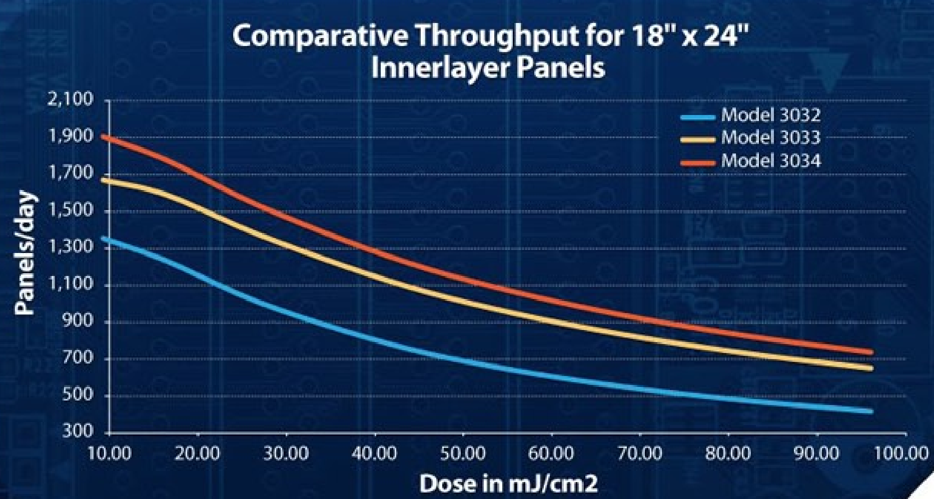
COST OF OWNERSHIP

Maskless LITHOGRAPHY

DIGITAL IMAGING AT PRODUCTION SPEEDS™

INDUSTRY LEADING COO

- Maskless Lithography provides customers with the lowest cost per core using standard and less expensive material sets



Most-Read MilAero007 News Highlights This Month

Reliability Requirements Challenged by Budget Cuts

Designing circuit boards that go into space requires close attention to detail, given the high cost of launches and the even greater cost of rework in instances where maintenance is possible. The complexity of these projects is becoming even more difficult to manage as budgets get tighter.

Merlin Flexible Circuits Receives SC21 Bronze Award

Flexible and flex-rigid PCB manufacturer Merlin Flexible Circuits, based in Sussex, has been awarded a prestigious SC21 Bronze award at a presentation in Edinburgh.

Cosmotronic Earns NADCAP Accreditation

TC Cosmotronic, Inc. has received NADCAP accreditation for the manufacture of rigid, flex, rigid-flex, and high-density interconnect PCBs.

MacDermid Final Finish Approved by Northrop Grumman

MacDermid has recently been approved for the Northrop Grumman ENEPIG specification with its Planar Ultra 46 final finish process.

Merlin Flex-Ability Earns AS9100 to Rev C

"Our team has put considerable effort into strengthening our systems, processes and procedures in order to achieve this accreditation and to provide our customers with the very best service possible," said Managing Director Mark Merifield.

Westak Earns AS9100 Rev C Certification

Westak has earned the AS9100 Rev C certification for its manufacturing location in Sunnyvale, California. "We have been working with a number of aerospace customers over the years, but believe this certification will strengthen those relationships and allow Westak a competitive advantage with new accounts moving forward," says Louise Crisham, CEO, Westak.

Rogers Highlights Laminate Materials at Satellite 2012

Rogers Corporation will be highlighting two of its leading PCB laminate materials for commercial and military satellite-communications (satcom) electronic applications at the upcoming Satellite 2012 Exhibition.

Stevenage Circuits Appoints Brown as Managing Director

Stuart Spink, chairman of Stevenage Circuits, has announced the appointment of Robert Brown as managing director of the company.

Vital Role of Aerospace and Defense Industry to U.S. Economy

"The data speaks for itself, America's aerospace and defense industry is a sector that punches far above its weight," said AIA President and CEO Marion C. Blakey. "And it's not just the numbers, which are impressive by themselves—it's how this industry makes a difference in the lives of all Americans."

Military Ground Robot Market: $12.3 Billion By 2018

Military ground robots promise to provide the core technology for management of conflict as strict nationalism in military matters is being replaced by the new era of terrorist threats.

For more, go to MilAero007.com

PTH Voids: Getting to the Root Cause, Part 1

by Michael Carano
OMG ELECTRONIC CHEMICALS

SUMMARY: Getting to the root cause of any technical issue is the first step in troubleshooting. Start with the desmear process and consider the type of resin you are trying to process.

Introduction

If manufacturing of complex printed wiring boards did not present enough of a challenge, fabricators today are faced with additional hurdles. One added hurdle has been the adoption of the higher-performance materials utilized for lead-free assembly. In addition, material sets necessary for microwave, harsh environment, telecommunication and medical applications are further driving the use of higher T_g, high T_d resins.

Critical Processes

Printed wiring board fabricators and end users must manage ever-increasing complexities in PWB design, technology requirements and long-term reliability. It is extremely critical that key processes such as desmear and electroless copper provide optimum reliability through a defined process window. This is important because the designer selects the interconnect technology and feature sizes required for the intended application. These various selections, including the resin material selection, impact the PWB reliability. While the designer must work within the parameters of form factor, weight and overall thickness, significant discretion with respect to wiring density, layer counts, hole-size diameters, copper thicknesses and material properties is often exercised.

The material, hole sizes, copper layer thicknesses and electrical requirements can significantly impact the manufacturability, reliability and performance of the PWB. In addition, end-users are pushing the envelope on reliability by requiring multiple thermal excursions in the form of higher temperatures and longer dwell times at temperature. The higher temperature values are in response to lead-free soldering. In order to ensure that greater reliability requirements are met, fabricators are being forced to re-qualify through the manufacture of more complex test vehicles. These test vehicles are most often designed to introduce more stressful conditions through the interconnect and barrel of the via. Adhesion of the electroless copper to the inner layer copper foil interconnects as well as the surface foil is severely tested under these conditions. Dielectric thicknesses between the layers and copper foil thicknesses (1/2 ounce versus 1 ounce) affect reliability.

Hand in hand with PTH reliability is the ability to plate a void-free electroless copper deposit. Indeed, achieving void-free deposit is critical with respect to subsequent PWB processing. It is well known that significant thermal reliability is added to the PWB when one is able to deposit 1 mil (25.4 microns) of electro-

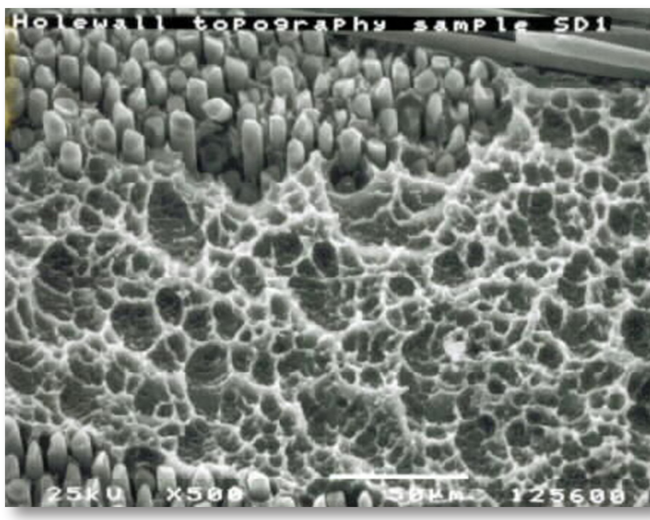


Figure 1: Highly roughened topography after permanganate desmear process.

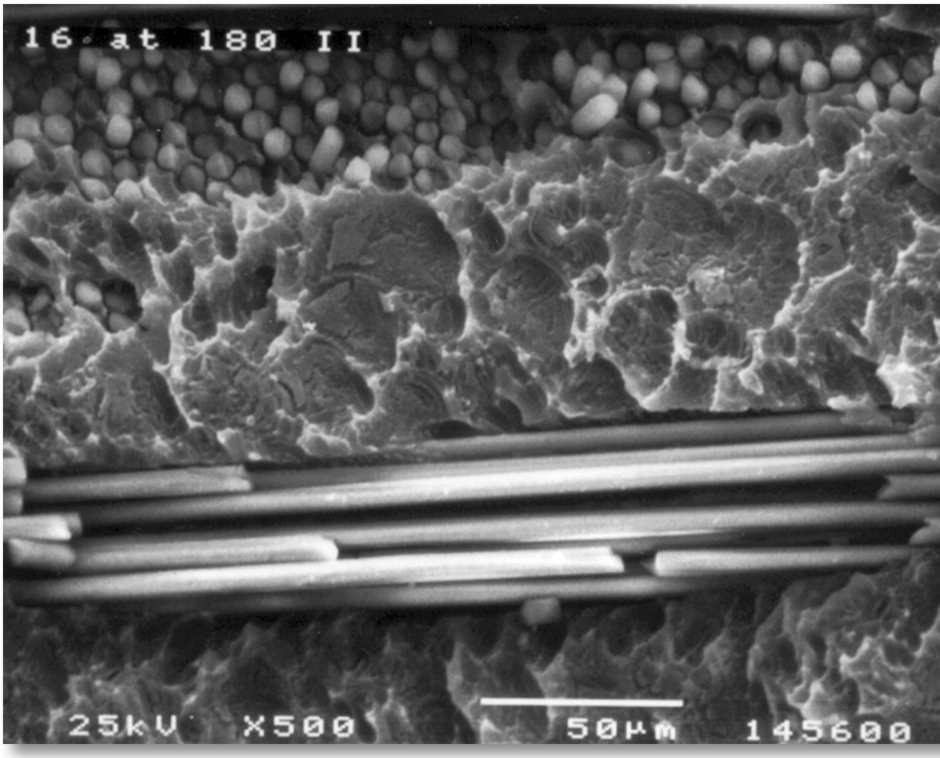


Figure 2: High T_g. Note: less microroughening than shown in Figure 1.

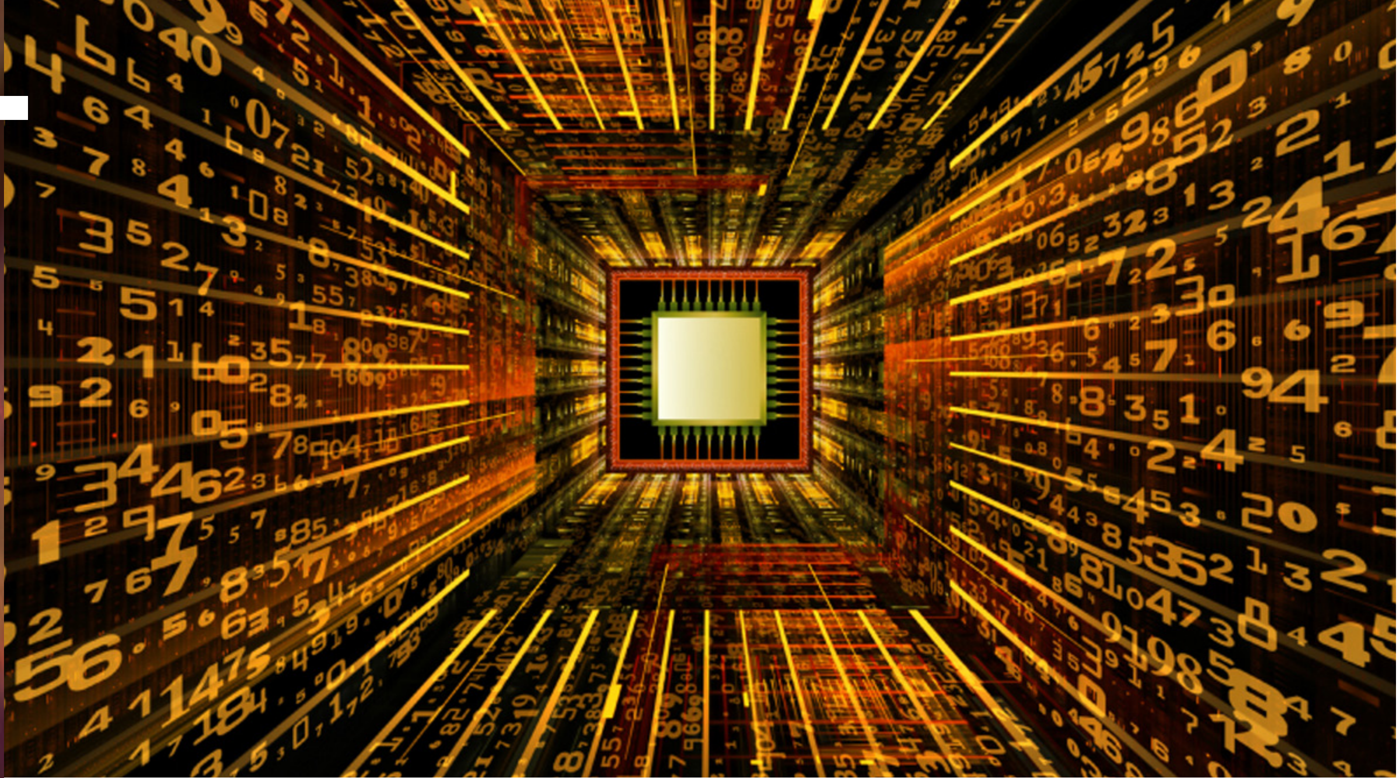
lytic copper in the vias. If there are significant voids in the electroless copper deposit, the uniformity and leveling of the electrolytic copper deposit will be less than desirable. Certainly, this fact influences grain size of the copper, as well as copper deposit thickness, which in turn affects the ability of the electroplated copper to withstand multiple thermal excursions. Process parameters, deposition rates, resin topography, palladium catalyst interaction with the resin, and copper deposit characteristics all have an impact on the capability of the metallization process to achieve a void-free deposit.

I will now focus on one approach to achieving void-free electroless copper deposits. It is important that we look at the entire process, especially those process steps involving drilling, lamination, desmear, PTH preplate and the electroless copper solution itself. In last month's column, I approached the issue of dealing with higher performance resin materials. These materials exhibit greater resistance to desmear treatment. This, in turn, makes achieving a suitable topography (Figure 1) extremely difficult. The material represented in Figure 1 is a lower T_g material and is easily modified by the alkaline permanganate chemistry.

Clearly, there is sufficient micro-roughening of the material to ensure a tightly adherent bond of electroless copper to the resin surface. Indeed, this type of roughened surface enhances the seeding of the palladium catalyst onto the resin. Consequently, this enables the deposition to electroless copper to take place with excellent coverage. In contrast, the higher-performance materials (those with a high T_g and high T_d) are less prone to attack from desmear chemistry (Figure 2). Thus, the fabricator is left to deal with less topography. A less than optimum topography adsorbs less catalyst, potentially leading to a copper void, as well as other potential issues.

This is precisely why fabricators must work closely with their process and material suppliers to develop robust processes that ensure a high-quality copper deposit. Keep in mind, just because the material one is using presents difficulty with respect to desmearing, there is no need to panic. While it is true that a topography shown in Figure 2 makes it more difficult to achieve a continuous void-free electroless copper deposit, there are adaptations that are available to mitigate this type of situation. In the next few columns I will detail these solutions in more detail. **PCB**

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



PCB IMAGING TECHNOLOGIES

by Stuart Hayton
MUTRACX

SUMMARY: Today's digital cameras are outstanding, allowing everyone to produce high-quality images easily, quickly and without waste. The same phenomenon is coming in the use of inkjet to digitally image PCBs. There is little doubt that our industry will go digital; it is a question of when, not if. I am not alone in my opinion that we are now approaching the "tipping point."

Early last year I was invited to write an article for this publication on the use of inkjet techniques in the primary imaging of PCBs. Whilst I remain convinced that inkjet will revolutionise the industry, in this follow-up article we'll take a wider look at the way fabricators image their boards today and how things are likely to shape up over the coming months and years.

Satisfying the appetite of today's digital world is one of our industry's primary drivers. It is therefore easy to be fooled into believing that digital processing dominates the manufacturing processes for fabricators across the globe. In fact, nothing could be further from the truth. A staggering 160 million square feet of silver halide film is used to generate phototools for tracking layers, solder mask and legend. The film business is dominated by a handful of vendors—AGFA, Kodak, and FujiFilm. According to their own figures, these three companies account for more than 90% of the market's film demands, and this film is employed in the production of 96% of manufactured PCBs. Each acknowledge the importance and growth of laser direct imaging (LDI) and its detrimental effect on film usage, yet still claim that the film business remains steady, and although it has not grown, it has (so far at least) escaped any decline.

Obviously, it is difficult to qualify these claims, but their basic logic seems to make sense. Annual PCB revenues continue to grow. Last year's global PCB market was estimated to have generated some US$58 billion in revenue, an increase of around 5% on the year before. If we factor in even modest price erosion of, say, 2%, then it is easy to argue that the volume of PCBs produced in 2011 increased by more than 8%, which is more than enough for the film business to coexist with LDI and maintain annual volumes.

This theory seems to be upheld by the plotter manufacturers. A quick walk around the recent CPCA Show in Shanghai would have been

Why the apparent reluctance to move away from film and plotters?

enough for any visitor to notice several low-cost plotter manufacturers that have entered the arena alongside the industry stalwarts of Orbotech, Ucamco and First EIE. Wondering if there is still a real demand for plotters in today's market, I decided to talk to Jean-Paul Birraux, sales and marketing manager of First EIE and Ucamco's Managing Director Karel Tavernier. I was interested to learn that both Jean-Paul and Karel claimed that the market for new plotters had been reasonably stable in the last three to four years, and believed the annual consumption of new, non-Chinese manufactured plotters to be in the region of 120-140 machines per year. Just as with film, the split of these sales is pretty equal between the three major players.

Both claimed two main market drivers for plotter sales—an increased capacity requirement in the East, and the need for high-resolution for HDI applications, which amounts to around 50% of new plotter sales and some additional and not insignificant upgrade business. This is perhaps the most surprising aspect of the conventional lithography story—even in today's market with the general acceptance of LDI, half of new plotters sold are of the highest resolution and intended for applications where many industry observers would expect to find direct imaging of some kind.

So, why the apparent reluctance to move away from film and plotters? My own opinion is that for some GMs and COOs, there will always be a desire to use tried and tested technology when faced with a choice of some other, perhaps slower, more expensive, more problematic alternative. I used to sell plotters, so I know firsthand that this is the case and it's something that vendors in the conventional lithography food chain are keen to promote. To be fair, plotters are faster, more accurate, and a better value today than they've ever been. The same goes for exposure units, and maybe that's the point. LDI just can't give the same levels of throughput as conventional lithography and it's a little bit trickier to use well.

That said, the plotter vendors are not resting on their laurels. Orbotech has for a long time had an extensive portfolio of PCB imaging equipment, including plotters, LDI and inkjet. FirstEIE has moved into inkjet as well and Ucamco has recently announced collaboration with Dainippon Screen, of Japan, to market their Ledia5 UV LED direct imaging system.

So, what of LDI? I can still remember working for a photoplotting service company in the early 1990s when there was ample talk about a new direct imaging technology that would kill the plotting business and replace films forever. As we've just learned, direct imaging in 2012 still only accounts for 4-5% of PCBs today. There is no doubt that the hot item in PCB imaging for the last few years has been LDI, a market which until recently was dominated by Orbotech and four Japanese players: Hitachi, ORC, FujiFIlm and Dainippon Screen.

LDI is unquestionably an enabling technology. Its advent allowed small volume manufacturers in the West to get to market quicker, whilst at the same time bringing significant benefits in registration-associated yield. The driver today, however, is miniaturisation and all of the challenges of HDI PCBs with stacked microvias.

Orbotech's Paragon remains the dominant machine, with over 600 installations across the globe. It is no longer a one-horse race, though, with FujiFilm in particular making an impact with its Inprex high-volume in-line system and some significant sales from Hitachi. More noticeable is the number of new players in this space that is being repositioned as digital imaging (DI). I've already mentioned Ucamco's arrangement with Dainippon Screen. Add to this new products from Maskless, Miva, Printprocess and even drill machine maker

Schmoll, not to mention the fact that the last twelve months have seen the introduction of Chinese competitors in this arena.

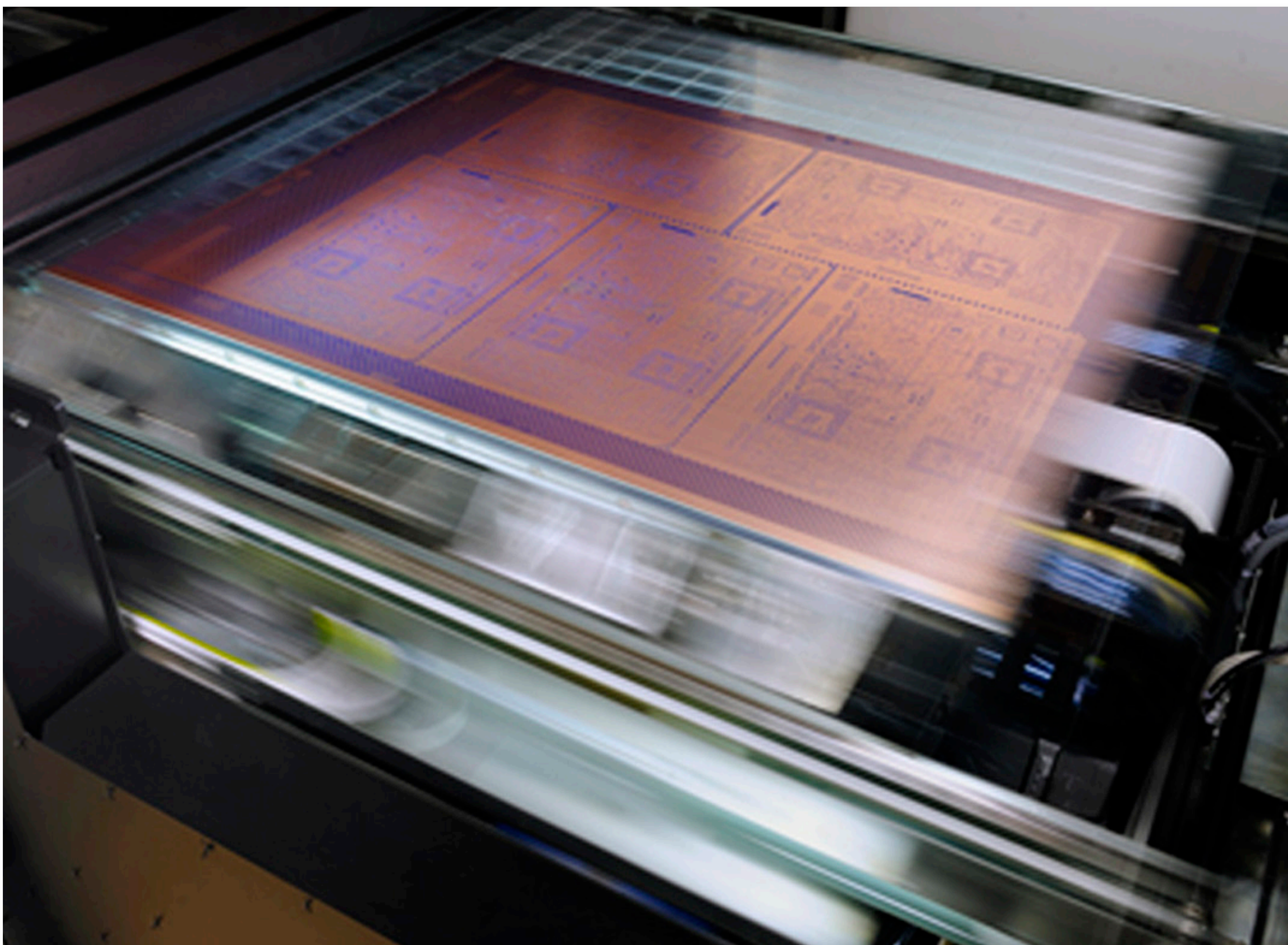
By all accounts, the DI market in 2011 was more than 300 units. The suggestion is that this year that number will increase to more than 350, of which the vast majority are employed in the imaging of signal layers. Approximately 25% of new LDIs sold are used to image solder mask. DI in this particular application has the potential to bring enormous yield rewards, but is hampered by much lower throughputs. I spoke to several DI vendors who were unanimous in the opinion that resolving this issue would almost certainly double the number of units sold.

So, that seems to be our reference, today LDIs are outselling film plotters by around 2:1, with the potential of 4:1 if higher production throughputs can be achieved in soldermask.

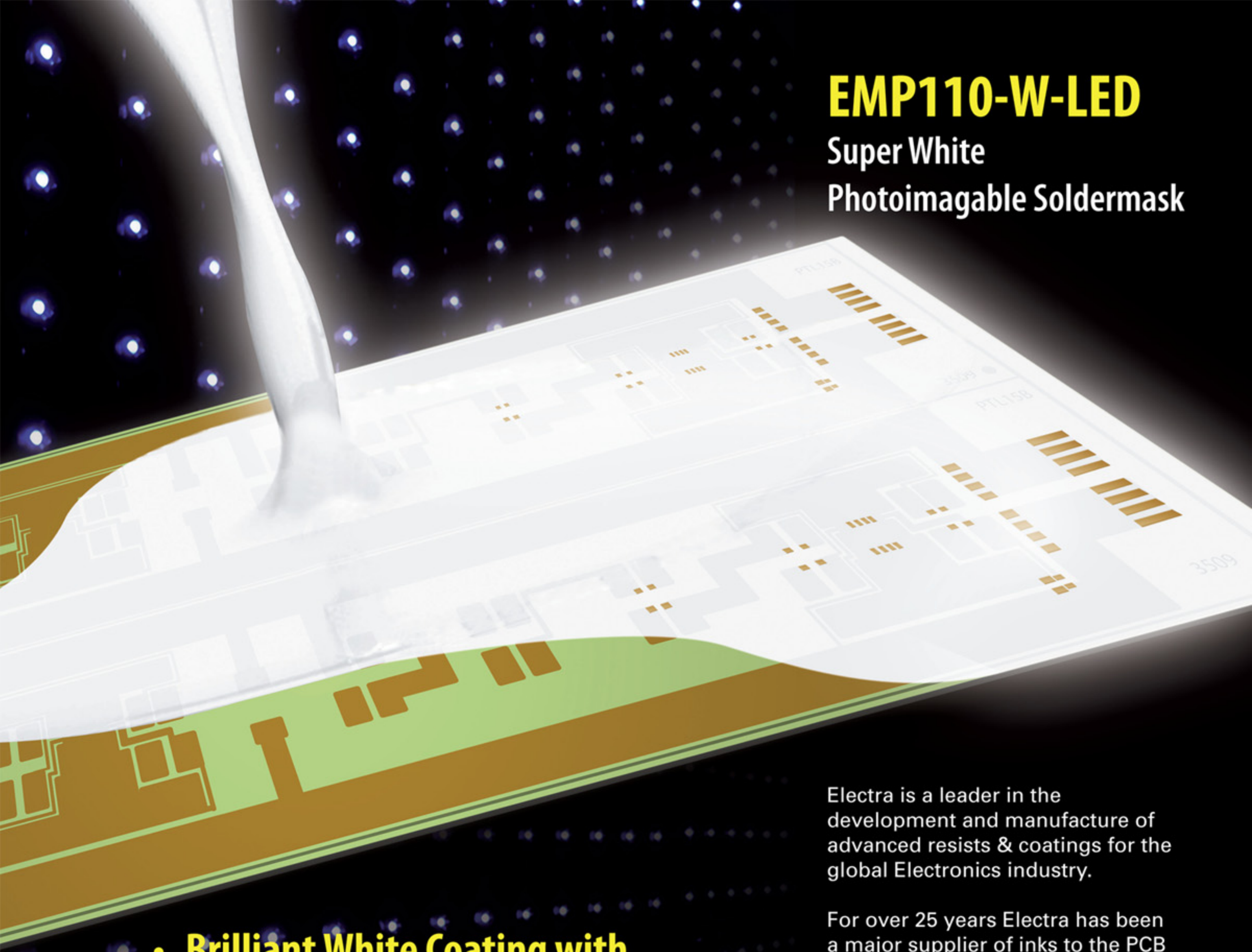
What about the future? What comes next? As you know from my last article, I've been working on Lunaris, an inkjet solution for the primary imaging of inner layers. At the same time, inkjet has become more common place in legend applications (Orbotech, Microcraft, FirstEIE), and even soldermask with Camtek's Greenjet.

There are many parallels between the industry's uptake of inkjet and the slow adoption of LDI. Early product offerings simply weren't up to the task at hand, and then of course there's the issue of ink and the question of whether or not sufficient volumes are required to incentivise chemical manufacturers to make the necessary investment.

In my mind, there is one striking difference, something that I believe will mean that inkjet



Lunaris fully automatic inkjet based inner layer imager.



will revolutionise the way that circuit boards are made. Inkjet is a truly digital solution, removing the need for conventional lithography completely, whilst simultaneously allowing inspection to take place earlier in the process and before the point of no return is passed.

Let's look what happened to the film camera business. For a long time professional and amateur photographers pointed their cameras, pressed a button and when the film was fully used, sent it off to be developed. Days later, the pictures would be returned and the results seen for the first time—resulting in fond memories often mixed up with underexposure, overexposure, missing heads, and fingers in front of the lens! The first digital cameras were a little crude, but they brought the advantage of immediately being able to view the captured image and to retake if necessary before going to print.

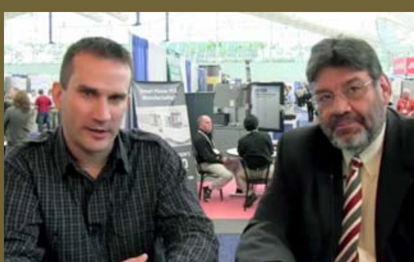
Making the first inspection of the photo immediate changed the way we use cameras forever. Today's digital cameras are outstanding, allowing everyone to produce high-quality images easily, quickly and without waste. I personally believe that we will see the same phenomenon in the use of inkjet to digitally image PCBs. Current print heads can reliably produce 3-mil line and space technology. Smaller droplet heads are around the corner and these will open up some, if not all, of the HDI market. There is little doubt that our industry will go digital; it is a question of when, not if. I am not alone in my opinion that we are now approaching the "tipping point." **PCB**

Stuart Hayton is the global sales and marketing director for MuTracx. He has more than 25 years experience in the PCB industry in technical, commercial and executive management roles. In 2008, Stuart moved to the Netherlands to be involved with the Océ Technologies Lunaris project, an industrialized solution for the digital imaging of PCB inner layers using inkjet technology.

VIDEO INTERVIEW

Fab and Design Speaking the Same Language

by Real Time with...IPC APEX EXPO 2012



Colonial Circuits Director of Engineering Michael Tucker talks about working with designers to give them an understanding of how boards are manufactured. He also discusses buying American whenever possible.

realtimewith.com

Most-Read Supplier/New Product Highlights This Month

Isola & Circuit Foil Launch Ultrathin 40 Micron Laminate

The product is a major improvement over existing technologies, as the laminate is glass reinforced and does not have thermo-mechanical mismatch issues commonly associated with unreinforced films.

Orbotech Presents Digital Production Tools for PCB Mfg

"Delivering enabling, digital-based tools for our customers to maximize operational efficiency, while keeping pace with the rapid advancement of the electronics industry is at the core of creating a new PCB world," said Orbotech West President Gaby Waisman.

Isola Commercializes I-Speed Laminate

Joining Isola's UL family of high speed digital products, I-Speed is a next generation lead-free compatible, low-loss, laminate product.

Burkle's Diversification Effort Pays Off

The machine manufacturer partially succeeds in compensating the depression in photovoltaics business through the other parts of its product range.

atg Luther & Maelzer Launches A5a at IPC APEX EXPO

atg Luther & Maelzer will use the IPC APEX EXPO to introduce its A5a fully-automatic Flying Probe Test System to the North American market.

Rainbow Technology Premiers New PCB Production Process

Rainbow Technology Systems will be showing a revolutionary new process for PCB production at this year's IPC APEX EXPO 2012, the premier trade show for the U.S. electronics sector.

Polar Instruments Names Prochaska N.A. Sales Manager

Polar Instruments Inc., a specialist provider of tools for PCB design, fabrication and test, announces the appointment of Cheryl Prochaska to the position of North American Sales Manager.

Frontline Unveils InSight PCB: Web-Based Pre-CAM Solution

"InSight PCB™ revolutionizes pre-CAM work," said Avi Glasberg, President. "With its simplicity and application power, InSight PCB™ significantly reduces load from CAM to Sales and Engineering and saves time for CAM operators. Sales people can prepare quotes without consulting CAM experts while incoming orders can be shifted from sales to engineering directly."

Maskless Adds Non-Exclusive Asian Distributorship

"Maskless digital imaging products are receiving broad acceptance globally, and it is clear that we need additional sales support as we expand into the overall Asian marketplace. I am excited to be announcing the addition of C SUN to our global distribution network," commented Bill Elder, President and CEO.

Park Electrochemical Unveils New Products

Park Electrochemical Corporation announced the introduction of its new N6800-22 High Speed/Low Loss and N6800-22 SI(R) High Speed/Very Low Loss digital electronic materials products. N6800-22 and N6800-22 SI are available globally in both prepreg and laminate forms.

For more, go to PCB007.com

Different Copper Foils for Different Reasons

by John Coonrod
ROGERS CORPORATION

SUMMARY — Various copper foils have their own unique sets of capabilities and limitations. Understanding these attributes can be very important for the circuit fabricator as well as the OEM.

Introduction

The many different classes of copper foils used in the PCB industry have their own capabilities and limitations. When you get into the details, copper foil issues for PCB applications can be overwhelming. In a broad sense, there are two types of copper foil used: rolled copper and electro-deposited (ED) copper. Within these two types there are a tremendous amount of variants. To add to the complexity, several different treatments may be applied to these coppers, and for a variety of reasons.

Rolled Copper

In general, rolled copper has been used extensively in the flexible PCB industry for decades. The grain structure and smooth surface is ideal for dynamic, flexible circuitry applications. Another area of interest with rolled copper types exists in the high-frequency PCB industry. It has been proven that copper surface roughness can impact high-frequency insertion loss and a smoother copper surface is advantageous. As with all material issues there are always pros and cons, and with the benefit of rolled copper being smooth, it is sometimes offset with the fact that less mechanical tooth can cause challenges for bond strength.

Additionally, rolled copper can be related to the horizontal grain structure, which can be more challenging for the etching of tight conductor spaces. Beside the grain structure, there can be imaging issues with lower adhesion of the photoresist to the smooth rolled copper surface. Having a poorer bond of the photoresist to the smooth copper surface can be a disadvantage for imaging and developing dense circuit patterns.

Electrodeposited Copper

With respect to ED copper, there is a huge diversity of foils regarding surface roughness, treatments, grain structure, etc. As a general statement, ED copper has a vertical grain structure that can be advantageous for obtaining tight-etched spacing and well defined conductor walls. The standard ED copper typically has a relatively high profile or rough surface as compared to rolled copper, and can benefit bond strength.

Classifications within ED Copper

There are different classes of ED copper. As a basic classification, they are: high-profile, standard profile, low-profile and very low-profile. The high-profile coppers are often used in applications where high bond strength is required. Some issues associated with high reliability benefit from the high-profile copper foils as well. On the other end of the spectrum is the very low-profile coppers, which are often used to minimize electrical conductor losses associated with insertion loss for high-frequency applications. During the past several years there have been many of these very low-profile copper foils offered with reverse treat, where the bond enhancement layer is on the outside of the copper

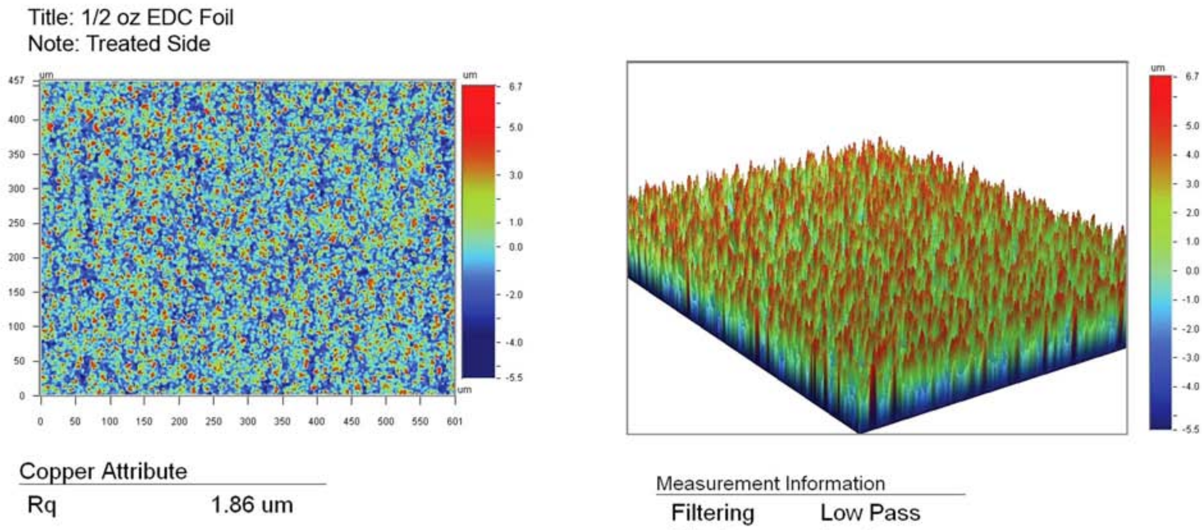


Figure 1: Mapping of the copper surface of a high-profile copper when scanning the treated side.

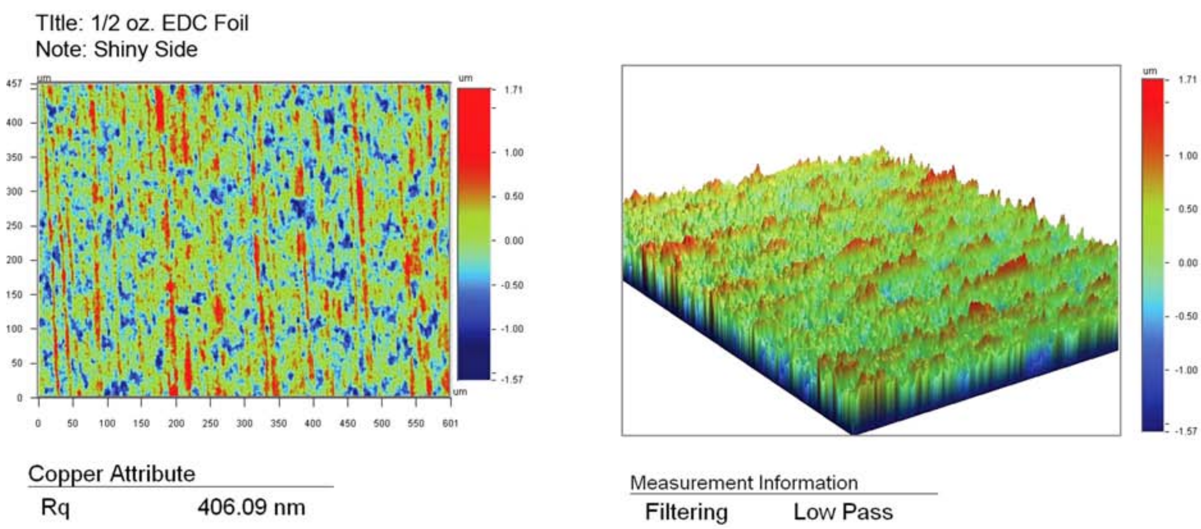


Figure 2: Mapping of the same copper, but scanning the shiny side of the copper, which is the side the circuit fabricator will have exposed to them.

of a laminate. This enables the circuit fabricator to have access to the treatment layer of the copper and also offers significant benefits with photoresist adhesion. Having a good copper treatment surface for the adhesion of photoresist enables better capabilities for imaging and developing of dense circuit features.

There is an art to measuring copper surface roughness and many different test methodologies exist. Additionally, several different characteristics of the copper profile may be reported. Most often the R_a or the R_z numbers are report-

ed by the copper foil suppliers. The R_a values are basically the average peak-to-valley numbers within the measurement scan area. The R_z number is similar to the R_a, where it is the average of the peak-to-valley numbers in the scan area, except this value is from five consecutive points measured. The measurement that is regarded as the most appropriate in regard to high-frequency electrical performance is the R_q value. This is also considered the root mean square (RMS) of the peak-to-valley measurement within the scan region.

Historically, the copper-surface roughness has been measured with a physical stylus procedure. This is a well known test method, but it may report a smoother profile, due to the limitation of the stylus tip in reaching the depth of a valley. There are also laser profilometer measurement methods that are generally regarded as more accurate; however, there are several different technologies and techniques to consider. An example of a laser profilometer measurement is shown in Figures 1 and 2.

Conclusion

There are many different types of copper foils, each with its own set of capabilities and limitations. Understanding these attributes can be very important for the circuit fabricator as well as the OEM. As always, it is highly recommended that you contact your materials supplier when considering different materials—especially copper foils. **PCB**

John Coonrod is a market development engineer for Rogers Corporation, Advanced Circuit Materials Division. About half of his 25 years of professional experience has been spent in the flexible PCB industry doing circuit design, applications, processing and materials engineering. Coonrod has also supported the high-frequency, rigid PCB materials made by Rogers for the past 10 years. Reach Coonrod at john.coonrod@rogerscorporation.com.

VIDEO INTERVIEW

ENIG and ENEPIG: Latest Processes and Standards

by Real Time with...IPC APEX EXPO 2012

George Milad, Uyemura's national account manager for technology and chairman of the IPC Plating Committee, explains to Technical Editor Pete Starkey how recent improvements in ENIG chemistry have reduced process costs and eliminated black pad defects. Milad also gives an update on IPC standards for ENIG and ENEPIG.

realtimewith.com

NEW!

THE FOURTH EDITION IS NOW AVAILABLE!

This newest digital edition of ***Flexible Circuit Technology*** has been expanded, adding timely and crucial information to existing chapters on Materials, Design, Implementation, Manufacturing and Assembly. Register now to download your ***FREE*** copy of Joe Fjelstad's ever-popular book.

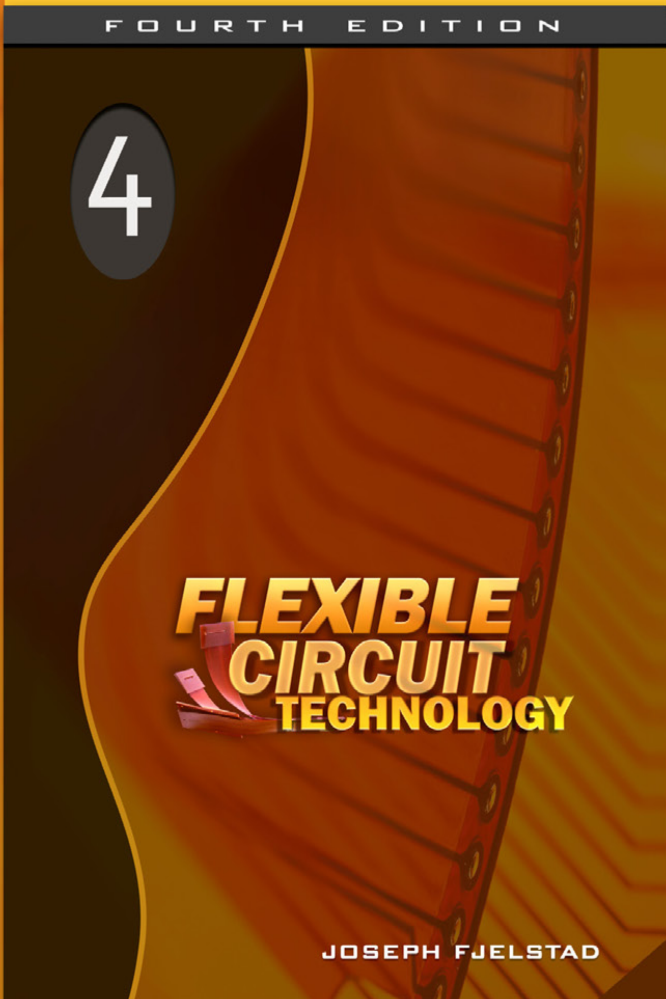


Table of Contents

Free PDF Download

www.flexiblecircuittechnology.com

This publication was made possible through the generosity of these great sponsors:

AIRTECH PCB DIVISION

epec ENGINEERED TECHNOLOGIES

CIRCUITS LLC

imagineering inc.

LENTHOR ENGINEERING

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

DUPONT
The miracles of science™

all flex Flexible Circuits, LLC
1991-2011 20TH ANNIVERSARY

BRP
BR PUBLISHING, INC

1 MIL LINES & SPACES AND BEYOND: Reclaiming Technological Leadership

by Gray McQuarrie
GRAYROCK & ASSOCIATES

SUMMARY: If we want our PCB businesses to thrive, we must abandon a cost-cutting strategy, adopt a growth strategy with innovation as its cornerstone and create technological breakthroughs that cause turbulence for competitors. We must lead micro fine-line fabrication processing that approaches semiconductor scales.

Today it is commonplace to produce 1 mil lines and spaces in large volume, but not here in the United States. What happened? We aren't the technology leaders anymore because a commodity mindset happened. I have asked a number of people in our industry this question: "Could we be the leader in micro fine-lines and produce such product at high volume in this country?" The answer I received was, "No, because of cost."

The Upside of De-stabilization

In his internet article "Selecting PCB Suppliers," Lee Ritchey stated, "It is rare that the lowest price PCB bid turns out to be the lowest cost." I remember once when a CEO got in an argument with a customer, saying, "We will never be the lowest-cost supplier." The customer was mortified and began taking steps to find an offshore supplier almost immediately.

If we think price is the same as cost, then we are blind to the value of coming up with innovative technologies that enable our customers. Instead, we are attracted to stability. When this happens, we have a commodity mindset. When we decouple price from cost, we are on the right track to coming up with technology that is destabilizing to our competitors and provides our customers a strategic technological advantage. This is how we can build a strong, profitable business. Back to our questions: How did we lose our technology edge? Consider Table 1.

Too many of us want stability. We want our customers to be there forever and we want to run the same parts day after day. As much as

> *When variation is the enemy, innovation is the enemy, too.*

we may want this, it is a fantasy. The fact is the stable mindset that might have worked back in the '60s, '70s, and '80s is gone forever.

With a desire to stay stable, many of us have tried to remain competitive in the wake of significant pricing pressure. We've become very good at taking costs out and making safe evolutionary changes. However, this alone hasn't been enough to save us. The world has been advancing at an exponential rate technologically. That is why "variation as the enemy" has been so attractive to us and so damaging. When variation is the enemy, innovation is the enemy, too.

When our industry was in a crisis (actually, it still is in crisis) those who survived had to make revolutionary changes. They had to learn new technologies, reconfigure their operations, take on entirely different markets, or do all three — and more. This is an extremely stressful and dangerous way to adapt.

The Importance of Turbulence

What we need to become is turbulent. We need to become unpredictable. We need to be the ones inflicting change on others and not the other way around. The way we do that is by becoming the technology leader. It means we have to rediscover how to be innovative. It means we have to invest in innovation with a very clear strategic intent.

Before we get into 1 mil lines and spaces and beyond, consider the innovation construct outlined below.

1. Understand current technology trends and how you can define a role where you enable your customer to provide higher-valued products.
2. Search for opportunities that might be lurking right under your nose. A good place is to start is with your low-yielding problem jobs.
3. Build the solution in the most efficient way around the need and not around trying to fit the solution to common manufacturing paradigms.
4. Figure out what can be done in the shortest possible time that will provide the highest value to the customer at the lowest cost.

As much as I would like to take credit for this innovation process, much of it was derived from my discussions with Joseph Fjelstad of Verdant Electronics. People like Joe are what I like to call iconoclasts. If we want to move from stability and towards turbulence, we need iconoclasts like Joe.

Let's proceed step by step with this innovation construct. In order to understand the need and desire for 1 mil lines and spaces, we need to understand the technological landscape of our customers. The goal of new devices still remains cramming as much functionality in the smallest space possible. For example, a multichip module is several chips bonded to a substrate material, which, if it is a PCB substrate, is designated as MCM-L. If the chips on the MCM are bonded to a PCB material in the form of a ball grid array (BGA) then it would be called an MCM-PBGA. In order to get maximum density the chips are likely flipped and attached using solder balls rather than fanned out with wire bonds that would increase the size. Using the flip chip technology allows the MCM-PBGA to be a small chip scale package, or CSP. If the size of the package needs to be even smaller, then these chips can be stacked one on top of the other to create a package-on-package solution, or POP. Here again, the chips can be three-dimensionally wire bonded, or connected in a series of flip chips.

Finally, all of the functionality could be crammed onto a single chip. This often leads to the need for a custom chip or an applica-

	Small Size Step	Large Size Step
High Frequency of Steps	Evolutionary	Turbulent
Low Frequency of Steps	Stable	Revolutionary

Table 1: Frequency and Size of Innovation Step.

tion specific integrated circuit, or ASIC. Since an ASIC can create delays in new product development and be of limited supply, a better option might be a field programmable gate array, or FPGA, where the specific functionality of the chip can be programmed. FPGAs are available from a number of manufacturers.

It's important to understand the different chip die configuration sets so that we can communicate and understand our customer better. For example, how the dies are configured in the package defines the constraints and design requirements for the substrate. Think of the substrate as a miniature high-density PCB. It is here where micro fine lines make sense and are required. It is here that our ability to make these fine lines defines the ultimate density that is achievable. It is here where we need to focus and start to apply our ingenuity.

Even with the progress that has already been made on PCB substrates, the density demands are so great today that there is a need for a silicon interposer (see Figure 1). Think of the silicon interposer as a multilayer silicon PCB. The most advanced interposers are three dimensional, where the IOs are fanned out slightly and vias are created within the silicon so that different layers can be connected. These vias are called through-silicon vias, or TSVs. It is here where the smallest lines, with the greatest aspect ratios, using the smallest vias are achieved in order to create the most compact density. But it comes at a cost and still requires a substrate interface to create the final BGA. As you can see, there is a lot to understand. The packaging world is changing very rapidly and it is very turbulent. It demands innovation up and down the supply chain. Innovation isn't a choice. You have to do it if you want to survive.

The second step to our innovation construct is vital, because it can uncover very profitable opportunities that can be quickly realized. I knew of a fabricator that had horrible yields on a small part with 1.5 mil lines and spaces going in every conceivable direction. The fabricator was trying to use a subtractive pattern plate process, which was the wrong process for this level of technology. This was a part in very high demand that commanded a premium and nobody wanted to build it!

One of the advantages achieved by exploring the current state-of-the-art is that it uncovers immediate opportunities. For example, a standard manufacturing process for BGA substrates used in smartphones could provide huge enabling opportunities for medical devices that need small, densely packed circuits. These lateral shifts may not seem that difficult to do, but they are vital to innovation and they are getting

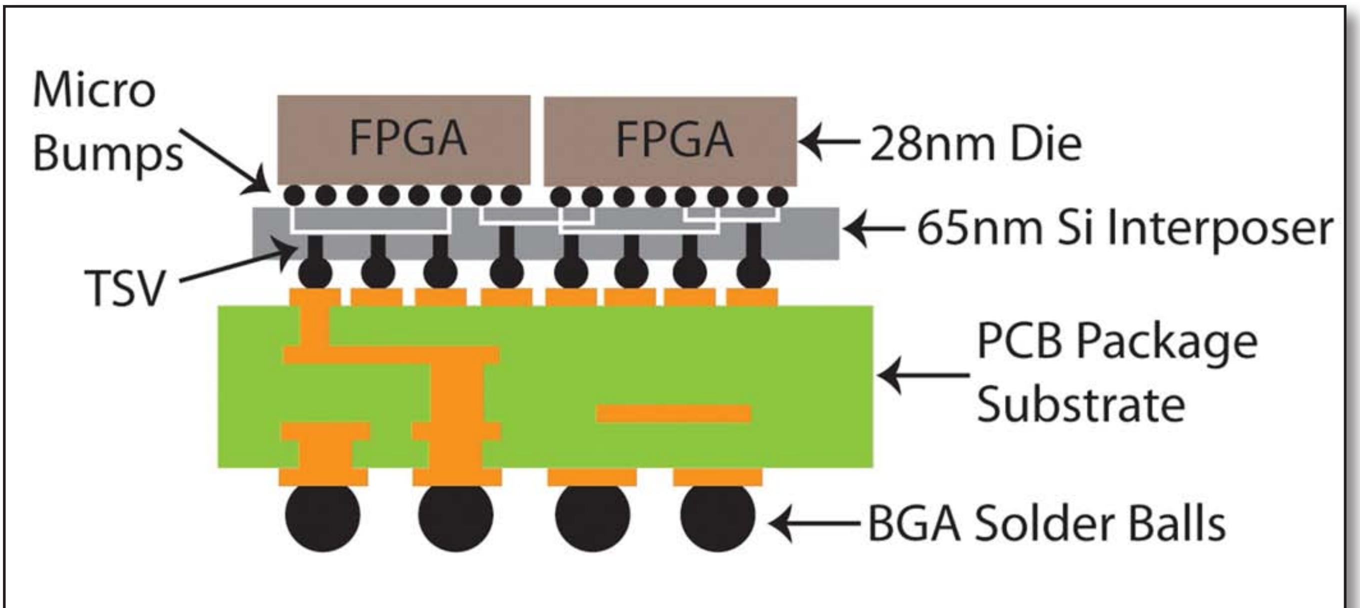


Figure 1: A BGA cross-section showing the dies, silicon interposer and PCB substrate stack.

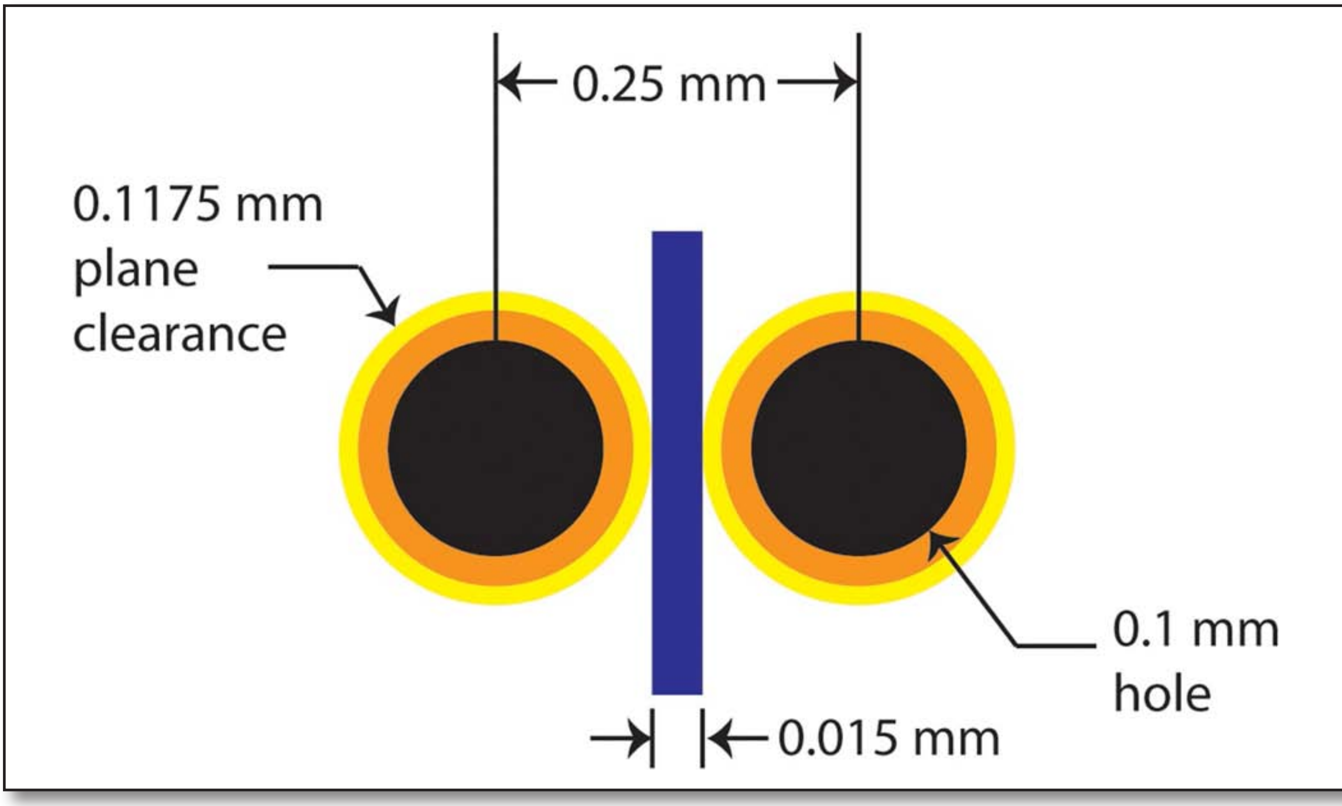


Figure 2: Trace routing width through a 0.25 mm BGA pitch.

rewarded. You don't want to start from scratch. You want to try different combinations of what is already known.

The third step in our innovation construct is to figure out the best technological solution. For example, when we are talking about the substrate for a BGA, it used to be that 0.4 mm pitch was state-of-the-art. The pitch of a BGA is the spacing of the centers of BGA pads (which are going to have solder balls attached to them) so you can punch in a number on a computer and bam, you have the grid you want. Traces have to go in between these pads. For a 0.4 mm pitch, you would have a 0.25 mm solder ball. This would require 0.125 mm (about 5 mil) micro via hole, a .23 mm pad, with a total of .35 mm to allow for clearance. The result is a 0.05 mm or 2 MIL or 50 micron trace.

Let's consider the feasibility of making a 0.25 mm pitch BGA, which would be at the bleeding edge of technology. Let's say the smallest we can go with our laser drill is 0.1 mm or 2.9 mils. This would allow for a pad and a safe area for the hole to be 0.1175 mm total. This would allow for a trace width of 15 microns (see Figure 2). Is there a market for this greater density? This is where you must get close to your customers and understand what they are trying to achieve and what packaging solutions they are thinking about now, and in the future.

Can you produce a 15 micron trace on a PCB substrate? A highly recommended process as outlined to me by John Andresakis of Oak-Mitsui and George Dudnikov of Founder PCB is this:

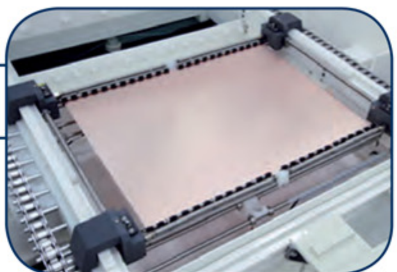
1. Start with a seed layer of copper such as laminating Oak-Mitsui's MicroThin material, which would leave a 1 to 5 micron copper seed layer after the carrier has been peeled away.

2. Using a laser direct image (LDI) in clean room conditions, process a dry film resist of a thickness that will be used to establish the conductor height.

3. Put the panel in a specially designed electroless bath (talk to your suppliers) and plate. This really defines a semi-additive approach.

4. Strip the resist and then…

New Contact-Free Transport System for Spraying and Immersion Processes



To meet the latest production requirements regarding packaging substrates, chip carriers and flex circuits

Due to contact free transport:
- No damage of the fine circuit structures
- No shadowing
- Uniform process results for top and bottom

Usable for different processes
- Clamping frames adaptable to particular processes
- Suitable for panel thicknesses $< 50\ \mu m$

Convincing advantages **... ask for more information!**

www.schmid-group.com

5. No need for a tin resist! Simply flash-etch the entire bare panel. The seed layer is so thin that it takes minimal etch time to remove. This is called sacrificial etching and it leads to very straight side walls.

In this last step you really should use something called a vacuum etcher, which has a series of vacuum plumes strategically placed, allowing etchant to be vacuumed off the panel continuously so that no puddles form. This creates a uniform and consistent etch.

The End Result

In step 4 of our innovation construct, realize that there are many alternative processes. The fewer the process steps the easier the process will be to control, and the higher quality result you will obtain. What is described above is just one recommended process; there are other issues to consider than just the fine lines in making a BGA substrate. List all possible processes, rank the best ones and try them; do this rapidly.

If you complete this four-step innovation process well, you will have the best chance of becoming the technological leader and the lowest-cost producer that your customers want. And, this will ensure that your business will remain in the United States. **PCB**

Gray McQuarrie is president of Grayrock & Associates, a team of experts dedicated to building collaborative team environments that make companies maximally effective. McQuarrie is the primary inventor of the patent, *Compensation Model and Registration Simulation Apparatus for Manufacturing PCBs.* He has worked for AlliedSignal, Shipley, Photocircuits, Monsanto and others. For more information, e-mail McQuarrie at gray@grayrock.net.

VIDEO INTERVIEW

C.A. Picard Talks AOI versus Visual Inspection

by Real Time with...IPC APEX EXPO 2012

C.A. Picard's Gary Weidner discusses the pitfalls of visual/eye inspection versus automated optical inspection, particularly when it comes to pin holes, solder masks, nomenclatures, and other board defects.

realtimewith.com

TOP TEN

Most-Read News Highlights from PCB007 this Month

1 IPC Honors Volunteers for Contributions to Industry

IPC presented Presidents, Special Recognition, Distinguished Committee Leadership, and Committee Service Awards at IPC APEX EXPO at the San Diego Convention Center. Awards were also given to individuals who made significant contributions to IPC and the industry by lending their time and expertise through active committee service.

2 IPC Adds New Officers, Members to Board of Directors

The Nominating and Governance Committee of the IPC Board of Directors presented nine candidates for election at the IPC Annual Meeting on Tuesday, February 28, 2012. Three candidates were elected as board officers and will serve two-year terms beginning February 2012. Six candidates were elected as board directors and will serve four-year terms beginning February 2012.

3 IPC Names Winners of Best International and U.S. Papers

IPC has announced the winners of the Best U.S. and International Papers of IPC APEX EXPO 2012. Selected through a ballot process by a Technical Program Committee, the papers were presented at IPC APEX EXPO, February 28–March 1, at the San Diego Convention Center.

4 DDi Corporation Posts Strong Q4, FY 2011 Results

"In the fourth quarter, we improved net income to $0.32 per share, increased our EBITDA more than 10% sequentially and year-over-year, and grew our cash to $31.2 million, while paying a quarterly dividend of $0.10 per share and significantly investing into our business," said Mikel Williams, president and CEO.

5 IPC Reveals PCB Industry Results for January 2012

"Both rigid PCB and flexible circuit sales followed normal seasonal patterns in January, with sales down from December," said Sharon Starr, IPC Market Research Director. "The good news is that rigid PCB orders are up and the book-to-bill ratios for both rigid and flex improved again this month. They are now just above parity, which suggests a return to modest growth in the next few months."

6 The Rebirth of Printca Finalized

Effective February 15, 2012, the main assets and activities of the company Printca A/S under bankruptcy have been transferred to a newly-founded company, Printca Denmark A/S.

7 Founder PCB Certified on Shocking's ESD Protection Solution

"We pride ourselves on being at the leading edge of PCB manufacturing innovation, and this new approach to ESD protection is consistent with our strategy of delivering solutions that help our customers compete in the dynamic consumer electronics market industries," said George Dudnikov, CTO and GM at Founder PCB.

8 Cosmotronic Earns NADCAP Accreditation

TC Cosmotronic, Inc. has received NADCAP accreditation for the manufacture of rigid, flex, rigid-flex, and high-density interconnect PCBs.

9 Aspocomp Reports 26% Sales Growth in 2011

"2011 was in all respects a successful year for Aspocomp. Our net sales grew by 26% to EUR 23.6 million, which boosted our operating result to EUR 4.1 million, representing more than 17% of net sales. Cash flow from operations after investments amounted to EUR 2.8 million," said CEO Sami Holopainen.

10 Cicor Maintains Sales Volume Amid Challenging Market

Cicor announced for the 2011 financial year, after taking into account all negative external influences, a slight decrease of its net revenue by 2.3% to CHF 178.7 million (2010: CHF 183.0 million).

The China Trap

by Barry Matties
I-CONNECT007

SUMMARY: The economic recovery being reported in the U.S. right now is built on a weak foundation—no fundamental change has been implemented. It could just be a spike from deficit spending. Those expecting the good old days to return may be waiting for a long, long time. New realities are reaching our core and we may finally realize that only we can help ourselves.

Each year the U.S. exports large volumes of raw materials to China, and such activity has been going on for many years. It started at a much lower volume but continued to grow. Then we hit a tipping point—we, the U.S., were no longer competitive enough to continue manufacturing many of our own goods in the U.S., at a low enough cost, to compete against low-cost producers. This cycle really began with consumers being excited as the retail price of products dropped and they saved money. Remember when a DVD player became really affordable?

As the demand for lower prices and higher profits continued, the demand for a quality product never diminished. Initially, China was not really geared to produce high-quality products so the next thing we exported was knowledge. It arrived in the form of partnerships with America's large corporations who began relocating employees there to teach the Chinese how to produce a high-quality product. It obviously worked—the quality of many products manufactured in China is world-class. Case in point, I am sitting in Starbucks in Shanghai, surrounded by mostly foreigners, typing this column on my iPad2, which, of course, is manufactured right here in China.

Another way China was able to fuel and accelerate growth was through the blatant theft of intellectual property. The so-called copy or knock-off market, though not as overt these days, is still thriving behind closed doors. It is amazing how many imitation iPhones, iPads, and iPods you can easily find in China. The copy market extends into just about all industries. Even equipment that was displayed at the recent CPCA trade show, some claim, is based on theft.

Is there any way to prove it? The owners of the intellectual property (IP) say yes; the same software bugs that exist in the original software, which they created, can be found in the imitation equipment's software. This is not an unusual claim. The imitation equipment

Yangshan Deep-Water Port, Shanghai is the largest cargo port in the world. Click for interesting video clip.

Knock-off electronics are easy to find.

is then sold for less money, thereby taking revenue away from the rightful owners of the technology. This behavior only adds to the struggle our industry faces in America and in other parts of the world. It also leaves the world with a negative impression regarding the Chinese government's willingness to uphold laws. You have to wonder why a company like Apple, and so many others, would continue to do business in a country where there is little or no regard for IP.

Even so, we continue to send more of our raw materials to China, along with the expertise needed to continue to teach the Chinese how to improve manufacturing efficiency to satisfy our continued thirst for low-cost, high-quality products. Their finished products are shipped back to the U.S. and sold for less than we could sell them for if we had produced them. We can no longer effectively compete and our manufacturing jobs are rapidly disappearing. Raw materials and expertise are not the only things America is exporting. As we purchase products from China, we export profit to China as well.

With the loss of so many jobs and the dollars associated with those jobs, we could no longer afford the cost of our own government and societal needs without increased deficit spending. So part of the answer was to borrow money from China, or, in other words, the profits we exported to China that they made by selling us finished goods from the raw material we exported to them. This was the brilliant solution, at least in their minds, that our government officials came up with to help offset our self-inflicted deficit in America. It's a trap that we stepped into on our own. When Americans hear news about how indebted we are to China, it concerns them, and rightly so. They can now see and feel the economic strength and influence China has on our daily lives.

Many blame the Chinese for the loss of jobs, but I think it was our own doing; part of the blame lies with consumers who sought more value for less money. It was also a corporate choice to avoid the over-regulation in America that has forced U.S. companies to move offshore. Many, like Steve Jobs, had and have strong opinions regarding this matter. Early on in this trend, critics made claims that the workers in China were being treated poorly, that kids were working in sweat shops, and that wages were not fair. We know that was—or in some cases still is—the truth, but I have not seen evidence of this in the Chinese factories that I have toured. What I see is a trend driven by international pressure and the workers themselves to raise the standards.

We are now closing in on another tipping point. With our manufacturing and management knowledge being continuously transferred to China (increasing China's debt leverage over the U.S.), China's middle class is rapidly expanding. What that means is a self-sustaining China market is flourishing. And that is exactly the message the Chinese government is spreading. They are actively encouraging the population to go out and buy products, and it's working.

No matter where you go in the developed regions in China, a voracious appetite to consume is evident. More privately-owned cars are on the road than ever before including a large number of high-end luxury cars, more cell phones are in use than in any other region of

Exotic cars are a common sight. Ferrari China

the world, and more Buicks are sold here than anywhere else. Jobs are abundant and salaries are increasing. China is flexing its economic muscle. I have written about the economic war in past columns, but it is clear that the winner right now is China. It does not mean it's perfect in China; it just means that the U.S. is in a really tough spot with no easy path back.

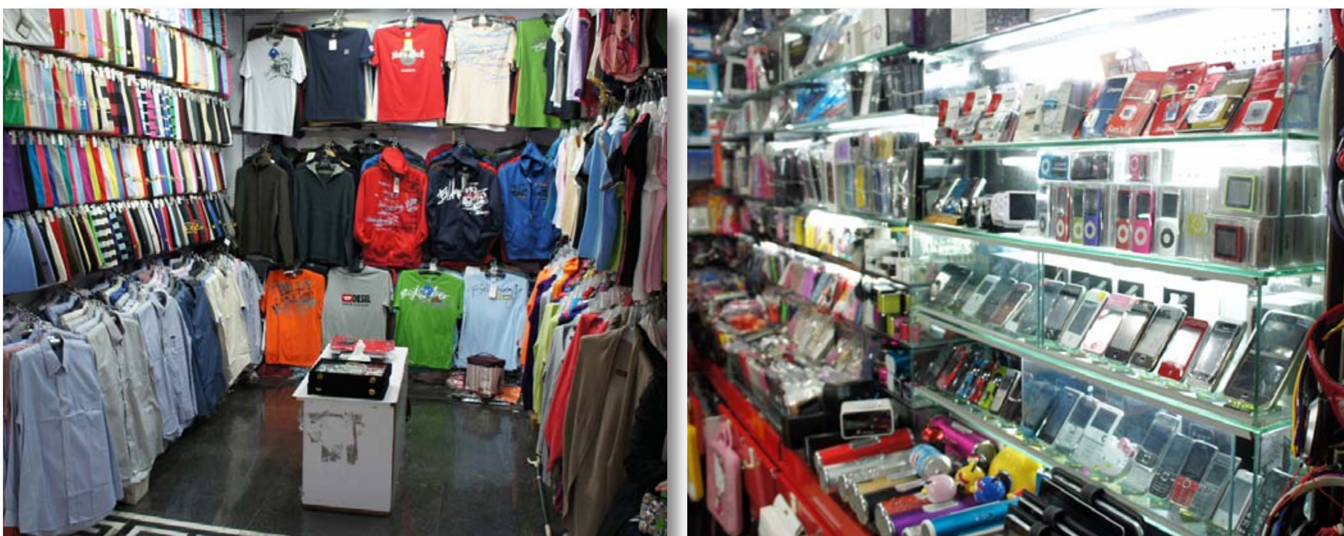
Not too many years ago, there were thousands of PCB fabricators in the U.S. Now the number is barely more than 300, according to www.ThePCBList.com. It's not because the demand for PCBs has decreased; in fact, just the opposite is occurring. More boards are being purchased today than ever before, most of which are manufactured in China. What happens if China cuts us off?

It's not the first time that question has been asked, but has it ever really been answered? How dependent is our military on foreign suppliers? Does that leave us at a vulnerable disadvantage if our supply lines are cut off? Of course, right now the U.S. is still a valuable trading partner with China, but it's the next tipping point we need to pay attention to...and I hope we are paying attention.

We are now really feeling the effects of what we have done...real estate has declined; good paying jobs are still hard to come by; college students can't afford the rising cost of higher education; the public school system is collapsing; retirement funds for so many, including those that were ready to retire, have vanished; and healthcare costs will continue to be a burden for so many who now can't afford it. We can't afford higher taxes to continue to support a failed system—our infrastructure is old and needs updating. So many people are stuck in their lives, and this stagnation has nearly paralyzed the entire country.

The economic recovery being reported in America right now is built on a weak foundation because no fundamental change has been implemented. It could just be a spike from deficit spending. Those expecting the good old days to return, well, they may be waiting for a long, long time. The new realities are reaching our core and we may finally realize that we can only help ourselves.

With that new reality we can feel a swell of patriotism spreading in America. We must keep in mind that isolationism is not the answer.



Knock-off shops like these are easy to find selling "name brand" merchandise.

We must be able to really compete in the new world order in a way that also leaves wealth in the hands of Americans. That may mean tough choices about leveling playing fields, changing the way we do business, and reducing our dependency on foreign countries—especially when it comes to energy. But the idea of having low-paying jobs for the masses and continued deficit spending, well, we already know it doesn't work. We have lived through that experiment and it is painful.

In America we hear a lot of talk of over-regulation, but we do not hear a lot of talk about accountability—accountability for mis-managing public funds, writing bad loans, and continued fraud in social programs. The waste is enormous, and if we ran our businesses like that we would be out of business. Instead, we have the burden as business owners and taxpayers of paying for the sins of others.

It is not all good news for the central government in China. They certainly have their many challenges of fraud, corruption, and social and political issues they must also deal with. Are they loosening up? Maybe. Recently, in a small province in China, an uprising took place against local politicians who were involved in taking land from citizens, for little compensation to the owners, and selling that land for a large profit to developers. The people had had enough.

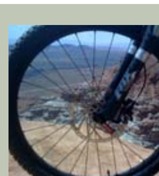
They organized, protested, and held elections to choose new local officials. Sounds like democracy to me. That is a big change that might be slowly catching on, to some degree, as society in China continues to gain strength and confidence. One interesting note around this event is that it made the national news in China. A Chinese friend told me that in the past this activity would not have been allowed to happen, let alone be featured on the national news. The change may have really started years ago when one man stood in front of a tank in protest for the whole world to see—at least the whole world outside China.

For those who have visited China, even in just the last five years, the country has witnessed an amazing, rapid transformation. For those who have not visited, I think it might be very difficult to even imagine what is really going on. I recently sat down for an interview with WKK

High-end luxury cars are abundant. Bentley China

Distribution President Hamed El-Abd and Bob Neves, Chairman/CTO, Microtek Laboratories during the recent CPCA show in Shanghai. Our discussion covered many issues and opportunities in China, from the theft of intellectual property, education, politics, currency, and the environment, to the overall business climate. These two gentlemen know China and its electronics industry better than most Westerners; Hamed has lived in Hong Kong for 20 years, and Bob has been in China for nearly 10 years. You can view our discussion here.

Only time will tell how our past choices will impact our future. We should all truly understand the implications of our current and future choices. **PCB**



Barry Matties started in PCB manufacturing in the early 1980s and in 1987, co-founded *CircuiTree Magazine,* which sold in 1999 as the leading industry publication. In the early 2000s, he and business partner Ray Rasmussen acquired PCB007, followed by *SMT Magazine* in July 2010. With his many years of business leadership skills, Matties now produces this column, relating 25 years of successful business leadership, marketing and selling strategies that really work. Contact Barry at barry@iconnect007.com.

EVENTS

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

79th China Electronics Fair
April 10-12, 2012
Shenzhen Convention & Exhibition Center
Shenzhen, China

Photonix 2012
April 11-13, 2012
Tokyo Big Sight, Japan

FINETECH JAPAN
April 11-13, 2012
Tokyo Big Sight, Japan

NEPCON Korea 2012
April 11-13, 2012
Coex Convention & Exhibition Center
Seoul, South Korea

Marine South
April 11-12, 2012
MCB, Camp Lejeune, North Carolina

IPC Tin Whiskers Conference
April 17-19, 2012
Ft. Worth, Texas, USA

NEW:UK 2012
April 18-19, 2012
The NEC Pavillion, Birmingham, UK

16th Annual Atlanta Expo and Tech Forum
April 19, 2012
Duluth, Georgia, USA

Green Lighting Shanghai
April 25-27, 2012
Shanghai World Expo Exhibition & Convention Center, Shanghai, China

Electronics New England 2012
April 25-26, 2012
Boston Convention & Exhibition Center
Boston, Massachusetts

NEPCON China 2012
April 25-27, 2012
Shanghai, China

SMT/HYBRID/PACKAGING 2012
May 8-10, 2012
Nuremberg, Germany

PCIM Europe
May 8-10, 2012
Nuremberg, Germany

MASTHEAD

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

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

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

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

MANAGING EDITOR: **LISA LUCKE**
(209) 304-4011; lisa@iconnect007.com

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

MAGAZINE PRODUCTION CREW:
PRODUCTION MANAGER: **MIKE RADOGNA**
mike@iconnect007.com
MAGAZINE LAYOUT: **RON MEOGROSSI**
AD DESIGN: **MIKE RADOGNA, BRYSON MATTIES, SHELLY STEIN**
INNOVATIVE TECHNOLOGY: **BRYSON MATTIES**
COVER: **MIKE RADOGNA**

PCB007 Presents

the pcb magazine

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

April 2012, Volume 2, Number 4 • The PCB Magazine© is published monthly, by BR Publishing, Inc.

ADVERTISER INDEX

Next Issue: Finishing Processes Explained

The PCB fabricator's role is to produce interconnecting substrates for electronic devices. Then, it is the assembler's task to gather the component parts and join them through soldering processes. How can the PCB fabricator facilitate the assembly process, and promote solderability where solder joints are required? What are the capabilities and limitations of PCB finishes? What are the latest developments?

In the next issue of The PCB Magazine, these questions and more are answered and explained, by experts from Semblant's Steven Lowder, Uyemura's George Milad, Joe Fjelstad, Michael Carano and others who have plenty to say on the topic of finishing processes.

Don't miss our columns department, offering plenty of technical advice, common sense management ideas and market predictions. Not a subscriber? Click here to get The PCB Magazine delivered to your inbox every month.