PCB007 PRESENTS

SPECIAL EDITION

APEX EXPO IPC 2012

POST-SHOW COVERAGE

March 2012

the pcb magazine

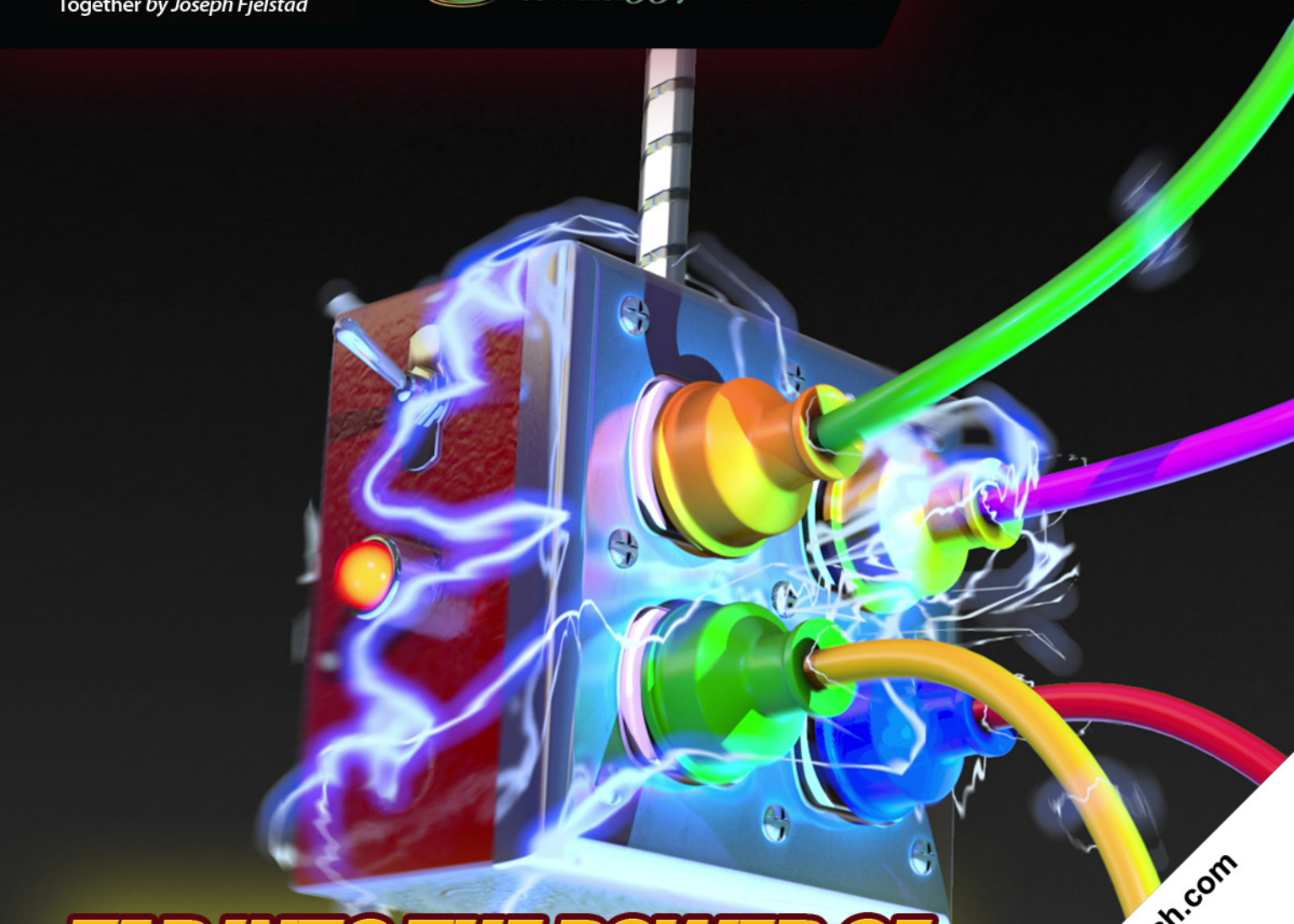
AN I·CONNECT007 PUBLICATION

12 Connecting Innovation Through Standards Development *by Dave Torp*

40 Flying by the Seat of its Pants: A View of EDA from the File Format Standards Perspective *by Abby Monaco*

76 Standards Boost Business for an Innovation Nation *by S. Joe Bhatia*

86 Industry Standards —The Glue That Holds Industries Together *by Joseph Fjelstad*



TAP INTO THE POWER OF STANDARDIZATION

www.atotech.com

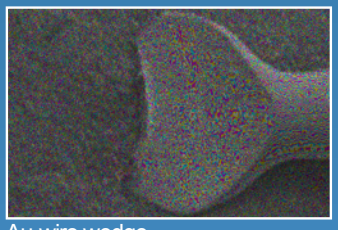
ATOTECH

Technology for Tomorrow's Solutions

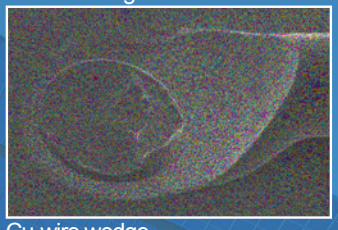
Final Finishing
Universal Finish SolderBond (ENEPIG and ENEP) for Soldering, Au-Wire and Cu-Wire Bonding



Soldering


Au-wire wedge


Al-wire wedge

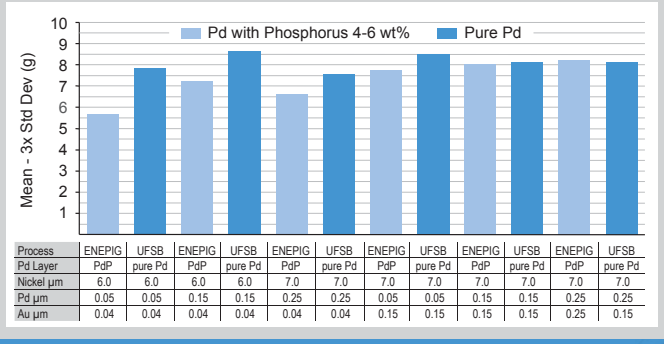

Cu-wire wedge

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

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.

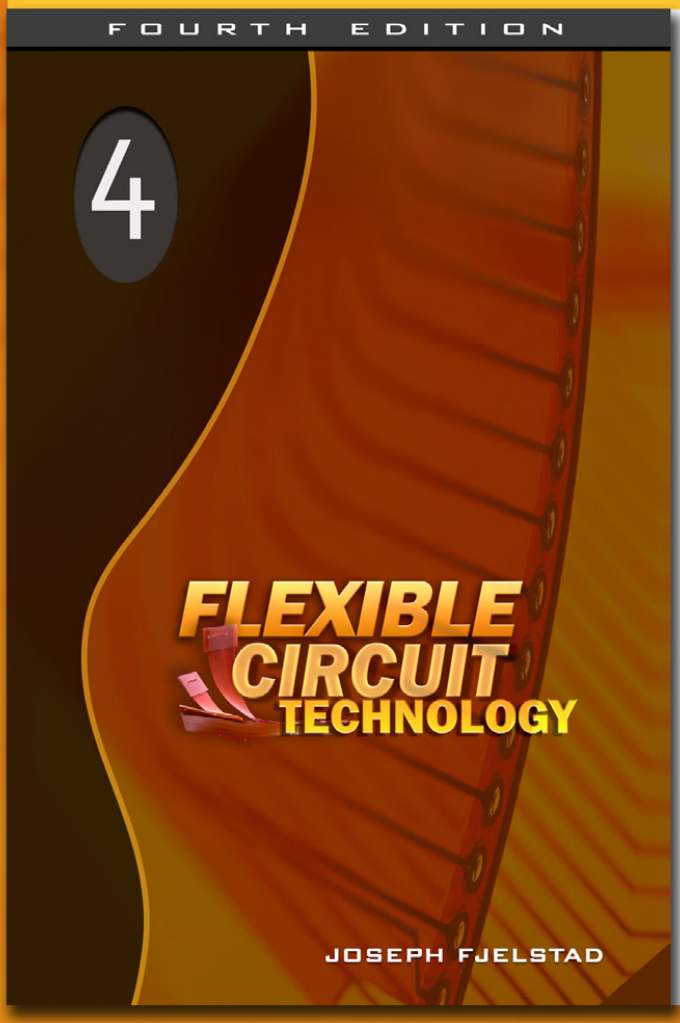


Table of Contents

Acknowledgements		8
Foreword		12
Chapter 1	**Flexible Circuit Technology Overview**	14
Chapter 2	**Flex Circuit Technology—Past, Present & Future**	26
Chapter 3	**Flex Circuit Drivers, Benefits & Applications**	48
Chapter 4	**Process Challenges and System Applications in Flex**	94
Chapter 5	**Flexible Circuit Materials**	126
Chapter 6	**Implementing Flexible Circuit Technology**	168
Chapter 7	**Practical Design Guidelines for Flex**	184
Chapter 8	**Guide to Reliable Static and Dynamic Flexing**	262
Chapter 9	**Flex Circuit Manufacturing Process**	280
Chapter 10	**Flexible Circuit Assembly**	354
Chapter 11	**Printed Electronics**	380
Chapter 12	**Solderless Assembly Processes for Flexible Circuits**	448
Chapter 13	**Stretchable Circuits**	478
Chapter 14	**Inspection and Test of Flex Circuits**	514
Chapter 15	**Plated Through-Hole Reliability**	542
Chapter 16	**Documentation Needs for Flexible Circuits**	572
Chapter 17	**Flexible Circuit Standards and Environmental/ Regulatory Requirements**	586
References		608

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 HIC

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

PCB007 PRESENTS
the pcb magazine
AN I-CONNECT007 PUBLICATION

STANDARDS, SPECIFICATIONS AND APPROVALS

Standards and specifications are fundamental to the PCB industry both locally and internationally. These carefully crafted industry tools harmonize fabrication processes and safety requirements and define reproducible levels of product quality and reliability. However, several questions must be addressed: Which standards are relevant to PCBs? Who is responsible for writing and enforcing them and what is there to gain by participating in the process? The March edition of The PCB Magazine navigates the who, what and why of today's industry standards.

12 Connecting Innovation Through Standards Development
by Dave Torp



28 IEC Cooperative Activities Yield Maximum Benefit for the Global Electronics Market
by Dieter Bergman and Marc Carter



40 Flying by the Seat of its Pants: A View of EDA from the File Format Standards Perspective
by Abby Monaco



76 Standards Boost Business for an Innovation Nation
by S. Joe Bhatia



86 Industry Standards—The Glue That Holds Industries Together
by Joseph Fjelstad



IPC APEX EXPO 2012 Special Section

52 Post-Show Coverage and Reviews

69 Show Floor Photos

73 Golf Tournament Collage

IPC APEX EXPO VIDEO INTERVIEWS:

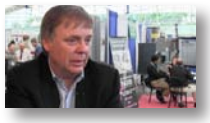
55 Market Outlook



59 PC Hall of Fame Award Winner Denny Fritz



61 iNEMI Update with CEO Bill Bader



65 IPC Addresses Conflict Minerals



69 New IPC Board Chair Sets Sights on Future



72 Mikel Williams' Government Relations Update



THE Future OF Surface Finish

SPF™ (Semblant Plasma Finish)

- Replaces or Enhances Gold/Silver
- Reduces Process Costs
- Environmentally Responsible
- Mitigate Creep Corrosion
- Pb-Free Solderability
- Flux Containment

Semblant

semblantglobal.com

PCB007 PRESENTS

MARCH 2012
VOLUME 2
NUMBER 3

THE DEFINITIVE
INTERACTIVE MAGAZINE
DEDICATED TO THE
GLOBAL PCB INDUSTRY

thepcbmagazine.com

the pcb magazine

AN I-CONNECT007 PUBLICATION

STANDARDS, SPECIFICATIONS & APPROVALS

CONTENTS

ARTICLES



88 **Registering Multilayers for Lamination: The Challenge of Registering New Materials on High-Density Interconnection PCBs**
by Victor Lazaro and Paul R. Waldner

104 **The Future of American PCB Shops, Part II: Electronics Assembly and PCB Platform**
by Harvey Miller



106 **CPCA Show 2012 Preview**

SHORTS

17 **Corporate Recognition Awards Honor DDi Corp. & Flextronics**

24 **AT&S Leads Hermes Consortium**

35 **Adams Named Prototron Circuit's National Sales Manager**

39 **Cosmotronics Earns NADCAP Accreditation**

46 **Polar Instruments Names Prochaska N.A. Sales Manager**

51 **Mentor Graphics Releases New PADS Suite**

EXTRAS



114 **Events Calendar**

115 **Advertisers Index & Masthead**

COLUMNS



8 **The Wrap Up**
by Ray Rasmussen

18 **Introduction to Board Level Simulation and the PCB Design Process**
by Barry Olney

36 **Rigid-flex Manufacturing Part III: Z-axis Expansion**
by Dale Smith



48 **How William Shatner Changed the (PCB) World**
by Steve Williams



82 **It's Not Your Father's FR-4 Anymore**
by Michael Carano

100 **Advances in Copper Plating, Part B**
by Karl Dietz



112 **Was it Good for You?**
by Barry Matties

NEWS HIGHLIGHTS



26 **Flex007 Highlights**

35 **MilAero007 Highlights**

74 **PCBDesign007 Highlights**

103 **Supplier/New Product Highlights**

110 **Top Ten Most-Read News Highlights from PCB007 This Month**

The UK's leading PCB equipment supplier

PRODUCTS

High Speed Moving Probe testers



MICROCRAFT

High Accuracy legend Printers



MICROCRAFT

Affordable Universal Testers



MASON

Ink Jet Peinter



CircaPrint
a printing legend

Economical Drill and Rout Machines



KEJIE

Wet Process Lines



U.C.E.

Laser Processing systems



PROCESS PHOTONICS

Fully Automatic Laser Plotter



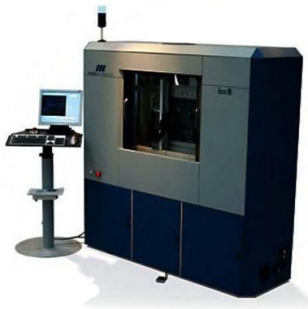
WIDE

SERVICES

The UK Leading bare Board Test Service



Mania Machine Service/Support



Spindle Repair Service



Excellon Athorised Service Provider



Long Barn, Ham Barn Farm, Farnham Road, Liss, Hants GU33LG
Tel: +44 (0)1420 537020
Fax: +44 (0)1420 538950
Web: www.vikingtest.com
email: sales@vikingtest.com

Viking

COLUMN

The Wrap Up

by Ray Rasmussen
I-CONNECT007

SUMMARY — Real Time with…IPC, our video program, produced in conjunction with IPC, taped more than 190 interviews, with many of the industry's premier companies and top professionals. This was an amazing effort by the I-Connect007 crew to schedule and produce this impressive number of interviews for our global audience.

All-in-all it was a really good show; not a great show, but a good one. The who's who of the industry descended on San Diego to hear and see all the latest and greatest technology in the industry. A solid technical program drew up to 1,700 folks who then flowed onto the show floor, delighting the exhibitors. Most of the folks we talked to were quite pleased with the quality of the traffic this year and satisfied with their investment.

Initially, the weather didn't cooperate, with Monday and Tuesday wet and drizzly, disappointing many of those from out of town who were expecting San Diego to be warm and sunny. By the end of the week, the expected weather finally showed up with even warmer weather predicted after most of us had already gone home.



Pudles to Lead IPC

Steve Pudles, CEO of Spectral Response, has taken on the role of Chairman of the IPC Board of Directors. Steve has served IPC in a variety of volunteer positions. His goals as chairman are to expand membership in IPC and continue to foster IPC standards throughout the electronics industry. Currently, the IPC Board is interviewing candidates for President of the IPC to replace former President Denny McGuirk.

Printed Electronics Exclusive Coverage

Betting on the future and in support of their recent efforts into printed electronics (PE), IPC for the first time dedicated a section of the show floor to this emerging technology. It's encouraging to see this bet on this closely related, and in some cases, game-changing industry. Aligning with this effort, we invited PE Technology Roadmap Subcommittee Chairman Dan Gamota to conduct a series of interviews to outline IPC efforts, as well as introduce some of those companies developing PE materials or equipment. Dan's interviews demonstrate the effort and position this exciting new industry can play within the world of circuits.

Market Outlook

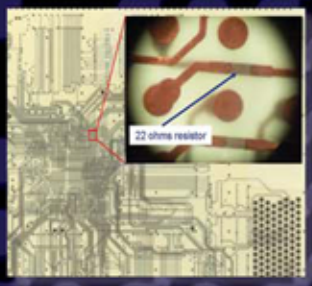
Walt Custer again walks us through the market dynamics driving our industry in his Market Outlook_interview. Expectations for the U.S. are flat for the first half of 2012, with the second half finally experiencing some growth. Nothing stellar is predicted, but we'll take what we can get. Not sure why Walt didn't give his presentation at this year's conference like he's done for the past 20-plus years, but we have it in this issue, so give it a look.

Hall of Fame Award

Denny Fritz has been inducted into the Raymond E. Pritchard Hall of Fame. A quote

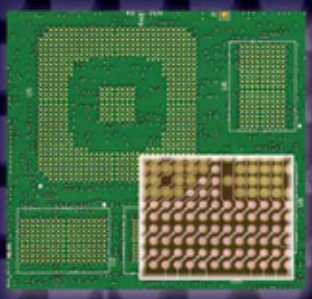
OhmegaPly® Embedded Resistors

Decades of Innovation



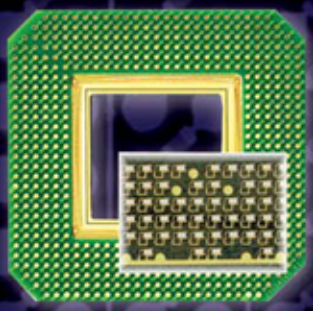
2010's

ORBIT ®
Built -In-Trace
HDI resistor



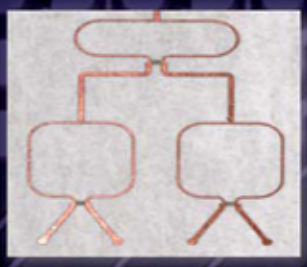
2000's

Series termination
under BGA device



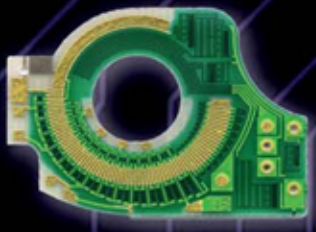
1990's

Parallel termination
of interposer board



1980's

Power divider
RF application



1970's

Potentiometer
surface resistor

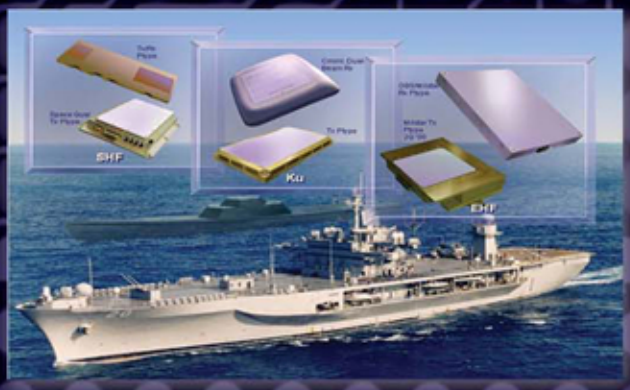
Unsurpassed Reliability



Globalstar Satellite



Mars Express Lander



Phased Array Antenna

Ohmega
Ohmega Technologies, Inc.

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

© Copyright 2011 Ohmega Technologies, Inc.

from an IPC write-up about Denny said: "Back then, companies were fierce competitors. Many were run by wild-West entrepreneurs who were real characters," says Fritz. "IPC meetings were exceedingly open and these guys were ready to tell their competitors how they built boards, which often provided the basis for IPC standards. Their secret wasn't in how they made the boards, but the level of service they provided their customers."

A very long-time member and supporter of the IPC, this well-deserved award couldn't have gone to a better guy. Congratulations, Denny.

Real Time with…IPC

Real Time with…IPC, our video program produced in conjunction with IPC, produced more than 190 interviews with many of the industry's premier companies and top professionals. This was an amazing effort by the I-Connect007 crew to schedule and produce this impressive number of interviews that capture the essence of the show for our global audience. Thanks so much to our sponsors who made this event possible: Aegis Software, Assembleon, BBG, Bürkle, Isola, Indium, Mentor Graphics, MicroCraft, Nordson Asymtek, P. Kay Metal, Prototron Circuits, Rehm Thermal Systems, Saturn Electronics, and Sensible Micro Corporation.

What really helped make this Real Time with… event exceptional isn't the number of interviews, but the quality. This year our team of editors went to great lengths to match up a guest editor's technical or business expertise





with a corresponding interviewee. Thanks to: Kelly Dack (PCB designer), Mark Thompson (PCB technologist), Bob Neves (reliability expert), Dan Beaulieu (PCB business consultant), Dan Gamota (printed electronics), Dr. Jennie S. Hwang (SMT, PCB assembly technologies), Mike Carano (PCB chemistries), Dick Crowe (PCB lamination), Joe Fjelstad (PCB, PCBA, flex circuits), Ron Lasky (PCB assembly materials), Stuart Hayton (PCB equipment, and business) and the I-Connect007 editorial staff: Andy Shaughnessy (PCB design), Pete Starkey (PCB and assembly) and Steve Gold (PCB business). Great job, everyone; we killed it!

Special thanks to our own Jo Ann Sotelo, Program Manager; Bryson Matties, video production team manager; and the entire production crew for producing so many high-quality videos in record time for viewers around the world.

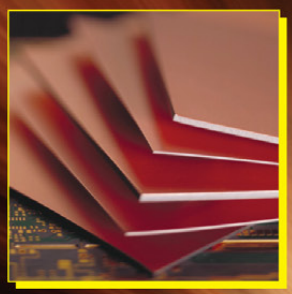
We're back in San Diego again next year, so start making your plans. The conference and EXPO is scheduled for February 19-21, 2013. See you there!



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.

Tactical Advantage.

When Reliability Counts.
Mission-critical applications have relied on Rogers' microwave materials for years. The superior electrical and mechanical characteristics of Rogers' military-grade laminates provide the stable, consistent performance over time and temperature that's so critical for aerospace and defense applications.



The rock solid performance of Rogers' microwave materials deliver the high performance and high-reliability demanded by the most mission-critical applications. Can you afford to use anything less?

To learn more about Rogers' aerospace and defense microwave material solutions, visit us at:
www.rogerscorp.com/military



Photos used by permission from Northrop Grumman Corporation.

Read Our New ROG Blog
www.rogerscorp.com/acm



	Features	Benefits
RT/duroid® 6202PR	Reduced planar resistor variation	Lower manufacturing costs due to decreased tuning
	Low thermal coefficient of dielectric constant	Stable electrical performance versus temperature
	Low coefficient of thermal expansion	High reliability in complex multilayer designs
RT/duroid® 5880LZ	Dielectric constant of 1.96	Lowest dielectric constant microwave PCB material
	Low z-axis coefficient of thermal expansion	Plated through hole capable
	Light weight	Advantage for airborne applications
RT/duroid® 6035HTC	Highest thermal conductivity (1.44 W/mk) for 3.5Dk printed circuit board laminates	Excellent power handling capability
	Low loss 0.0013	Excellent high frequency performance
	Low profile, thermally stable copper options	Lower insertion loss, and performance reliability in high temperature applications

The world runs better with Rogers.®

ROGERS CORPORATION
Advanced Circuit Materials

USA - AZ, tel. +1 480-961-1382 • EUROPE - BELGIUM, tel. +32 9 235 3611 • www.rogerscorp.com/military



Connecting Innovation through Standards Development

by Dave Torp
IPC

SUMMARY — As a nonprofit association, IPC is an integral part of the electronics supply chain, providing technical conferences, trade shows, market research, and most notably, standards and certification programs to the entire electronics industry.

IPC — Association Connecting Electronic Industries® — has been disseminating electronics knowledge and expertise since 1957. IPC was founded by companies from the PCB industry, including suppliers, designers and fabricators. Although IPC has grown tremendously, its roots are deeply planted within the printed board community. IPC, as a nonprofit association, is an integral part of the electronics supply chain, providing technical conferences, trade shows, market research, and most notably, standards and certification programs to the entire electronics industry.

Standards development is a core competency within IPC and has been from the onset. Most people who work within electronics manufacturing are first exposed to IPC through an encounter with one of our standards. IPC standards provide form, fit and functional requirements, guidance and best practices for the electronics technical community of users, suppliers, testing facilities, government agencies as well as academia. IPC standards activities focus on documentation of knowledge and best practices for the electronics industry.

The process IPC uses to develop standards is based upon industry best practices. IPC is accredited by the American National Standards Institute (ANSI) as a standards developer and uses *ANSI Essential Requirements* as the basis for standard development activities. *ANSI Essential Requirements* provides a great framework for standard development organizations (SDO), and by using ANSI methodologies, a specific set of criteria is used in the standard development process. IPC believes the incorporation of ANSI best practices ensures that the standards development process is an open process; therefore, IPC standards development committees are open to anyone who would like to participate. There are no membership requirements for participation on standards development committees for IPC.

The IPC standards development process encourages balance within the standards devel-

opment committees. Often, like-minded individuals who work on standard development have similar objectives. For example, individuals who work on aerospace and defense-related electronics for OEMs have similar needs. IPC encourages individuals from all segments of the industry to participate on committees so that a wide range of opinions and expertise can be shared within an open process. Diversity in participation of users and suppliers, in conjunction with associated industry groups, such as governmental agencies, academia and other associations, makes for robust standards that can be adopted on a large scale.

> ***"The IPC standards development process encourages balance within the standards development committees. Often, like-minded individuals who work on standard development have similar objectives."***

All participants of standards development committees have a voice. Through consensus-building activities within the standards development process, contributions, input and feedback are received and considered, sometimes through public notification. At times in the standards development process, the discussion on issues may become contentious and argumentative; however, the process benefits from the airing of different perspectives, concerns and interests, and eventually a certain amount of harmonization occurs. This process leads to the development of robust standards that withstand the test of time. Many IPC standards are widely adopted by the electronics industry and become the foundation for communications within and between electronics markets.

Standards for Design

Within the printed board community, organizations use IPC standards every day in the design, fabrication, test and inspection of printed boards. The design series of IPC standards are used as guidelines for the design and layout of PCBs. The most noteworthy of the design standards is IPC-2221A, *Generic Standard on Printed Board Design*. This standard provides guidance to printed board designers on materials, mechanical properties, electrical properties, thermal management and component assembly issues as well as interconnect issues within the design of the printed board itself. There also exists a series of design standards for a variety of special, more focused board configurations. For example, there are standards for design elements that specifically relate to flexible circuits (IPC-2222) and high-density interconnect (IPC-2226). The design series of standards provides essential knowledge of best practices for printed board designers.

Some IPC standards supplement design standards and typically provide application-specific information which is critical, but extremely difficult to come by. An example of a supplemental standard is IPC-2152, *Standard for Determining the Current Carrying Capacity in Printed Board Design*. The information within IPC-2152 helps designers understand the conductor sizing based on temperature, voltage and current for a given set of conditions. This is critical information for board designers that is very difficult to find anywhere else in the world. To determine current carrying capacity for conductors empirically or through experiments takes a considerable amount of time and money. This is one of the most important reasons that standards are adopted within the electronics industry. Through the use of standards, information allows designers to lay out boards with a high level of confidence, minimizing the number of redesigns required. Ultimately, standards help make products more reliable, in a more efficient manner.

Standards for Fabrication

For board fabricators, there are two essential IPC standards: IPC-A-600H and IPC-6012C. IPC-A-600H, *Acceptability of Printed Boards,* describes acceptability criteria of PCBs. IPC-A-

600H provides an in-depth look at externally and internally observable printed board requirements. Requirements are broken down into three different classes: Class 1 includes limited life products; Class 2 includes products for which extended life is required and uninterrupted service is desired; Class 3 includes products for which continued high performance or performance on-demand is critical and product down-time cannot be tolerated. The flexibility and applicability of IPC-A-600H is wide ranging across the entire electronics community, from the manufacturers of consumer electronics to users of high-performance weapons systems. IPC-A-600H has hundreds of pictures, illustrations and drawings to aid understanding of basic requirements for best practices for all three classes of product types.

IPC-A-6012C, *Qualification and Performance Specification for Rigid Printed Boards,* is one of the printed board industries most widely used standards. It is an essential standard for understanding printed board requirements that influence form, fit and function. Within IPC-6012C, boards are defined by types. Six different types of boards are identified and each type of board is progressively complex. For example, Type 1 defines attributes for a simple single-sided printed board. Type 6 defines attributes for a complex, multilayer metal core printed board with blind or buried vias. IPC-6012C is the electronics industry's most comprehensive document for rigid printed board requirements. These requirements are found in every high-performance printed board fabricated around the world. IPC-6012C provides a common language for users, fabricators, materials suppliers and equipment suppliers to agree upon use requirements without having to create customized requests for quotes.

IPC standards allow users to specify requirements within procurement and quality documents. The specified use of requirements within IPC standards is voluntary. There are no laws requiring the use of IPC standards — just a desire to conform to industry best practices. Standards like IPC-6012C provide a common reference for establishing agreed upon criteria between users and suppliers. If there are discrepancies between users and suppliers, IPC standards provide sufficient detail to facilitate problem resolution faster and more equitably than if standards did not exist. In extreme cases of disagreements between these two parties, standards are used to define the best practices for reaching a resolution.

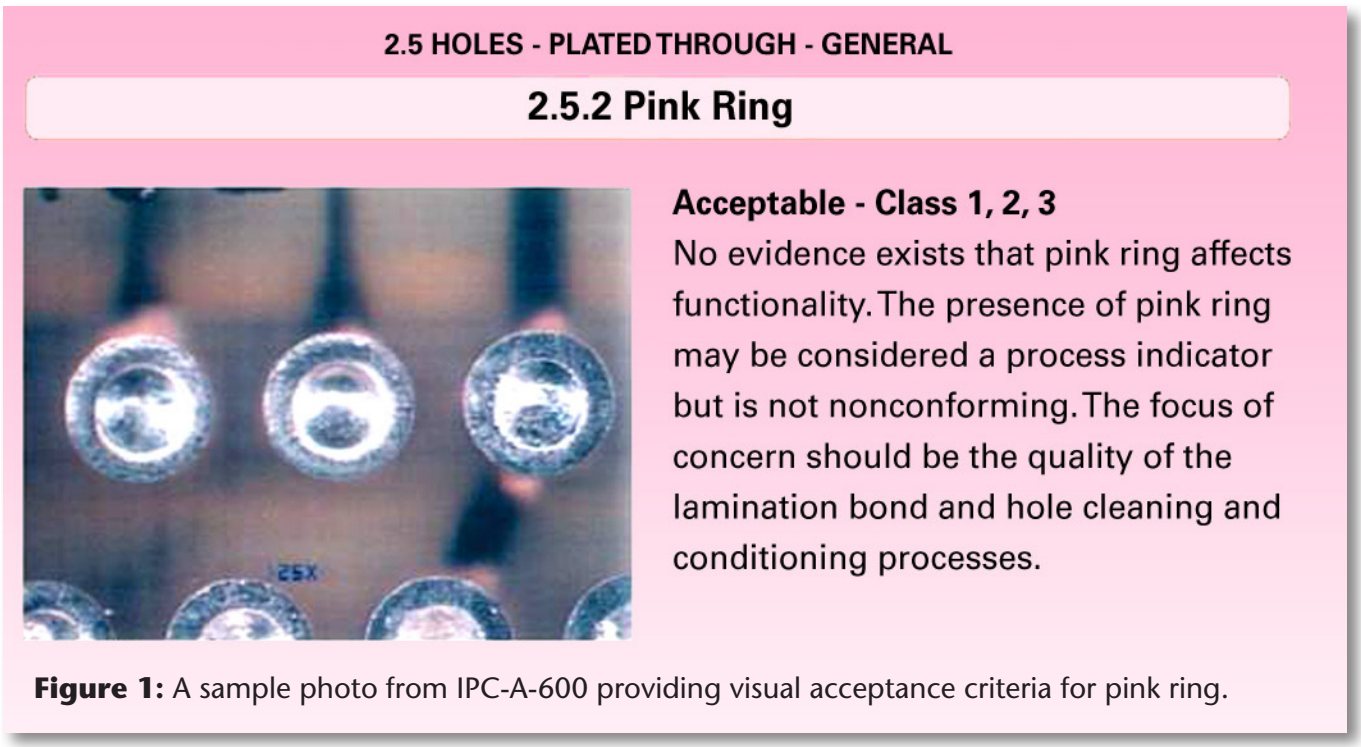


Figure 1: A sample photo from IPC-A-600 providing visual acceptance criteria for pink ring.

OMG
OM Group
Electronic Chemicals, LLC

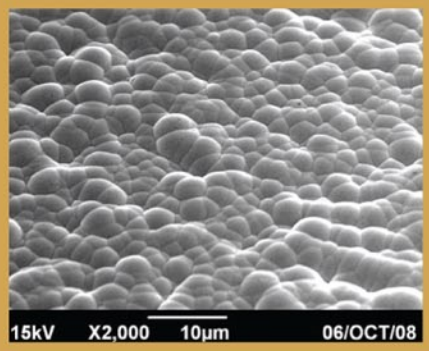
OMNi Guard 930 EN and OM Gold

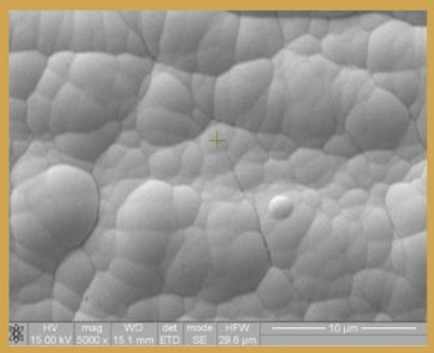
Squeezing the costs out of PWB fabrication without sacrificing quality or reliability.

OMG OMNi Guard and OM Gold

- OMG 930 EN ease of control– no dummy plating
- New immersion gold designed to inhibit grain boundary attack on EN
- Lower concentration OM Gold process reduces precious metal costs significantly
- OMG 930 EN consistent phos control from 0-5 MTOs

OM Gold supplied through a partnership with Japan Pure Chemical.





Final Finish
TECHNOLOGY

OMNi Guard 930 EN

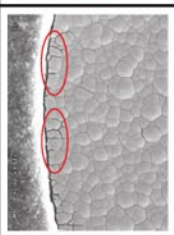
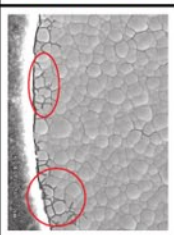
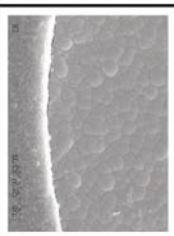
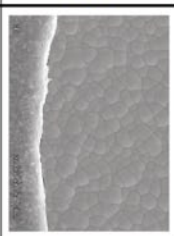
930 EN consistent and uniform grain structure up to 5 metal turnovers (MTO)

Uniform and consistent grain structure promotes excellent solderability over the range of metal turnovers

Resistant to hyper-corrosion of Immersion Gold

Advantage of OM 930 and OM Gold

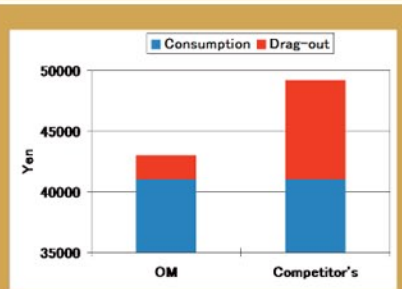
OM doesn't attack Ni surface under low Au content.

Au bath	Conventional IG	Conventional IG	OM	OM
Au content	2g/L	1g/L	1g/L	0.5g/L
N bath	Conventional Ni	Conventional Ni	930	930
SEM				

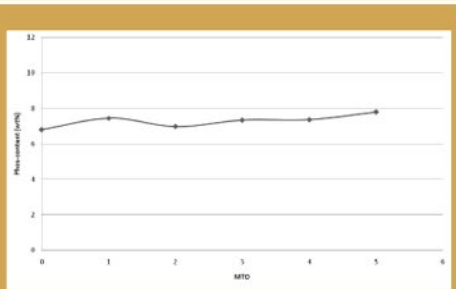
Ni attack is heavier in low Au

Ni attack not observed at 1g/L

New Combination OM/OMG 930 can Further decrease



Au cost for 10g plating



Phos-content OMNi Guard 930 EN

Reduce Your Precious Metal Costs

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

Table 3-10 Cap Plating Requirements

	Class 1	Class 2	Class 3
Copper Cap - Minimum Thickness	AABUS	5 µm [0.0002 in]	12 µm [472 µin]
Filled Via Depression (Dimple) - Maximum	AABUS	127 µm [0.005 in]	76 µm [0.003 in]
Filled Via Protrusion (Bump) - Maximum	AABUS	50 µm [0.002 in]	50 µm [0.002 in]

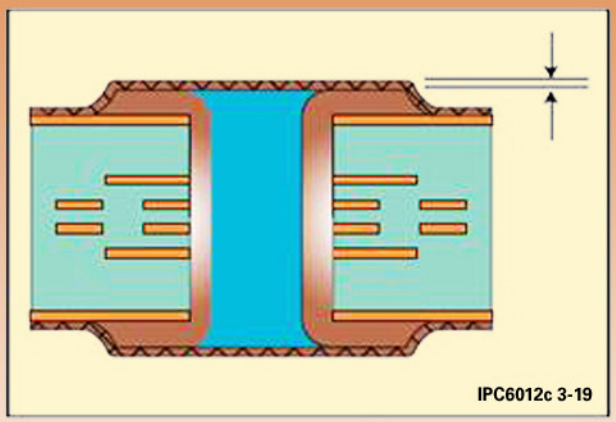


Figure 3-19 Copper Cap Thickness

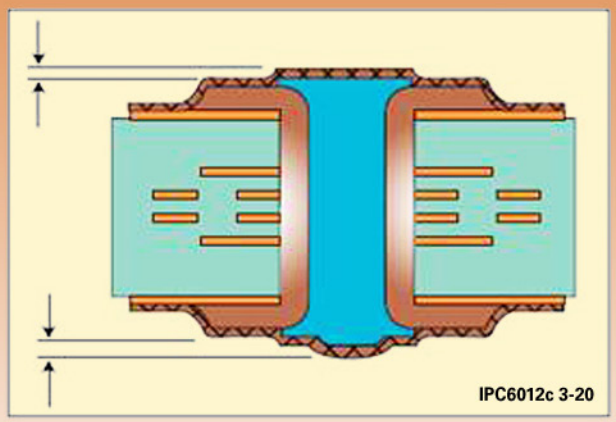


Figure 3-20 Copper Cap Filler Via Height (Bump)

Figure 2: Cap plating requirements as specified in IPC-6012C.

Innovation, Technology and Standards

Standards also drive innovation within the electronics industry, which in turn drives advances in technology. Because the electronics industry advances very rapidly, it is extremely difficult for standards to keep up with technology. There is a certain amount of data and analysis required before a technology can be considered a best practice. Within the electronics community, technology is driven by higher speeds, greater density interconnections, smaller feature sizes and lower cost. These technology drivers push traditional printed board fabrication to the physical limits of materials and processes. Often, new breakthroughs in technology are required to meet the needs of a rapidly advancing market. Standards provide a method to build upon prior technology and reserve a place to store new technology after it has overcome the initial resistance. Standards allow technology innovators to come up to speed faster than they would have without them and the time to market is reduced by having a pathway to known best practices through IPC standards.

The Human Factor

The most important asset of any standards development organization is the people who participate in the standards development activities. Standard development communities gather together to discuss and resolve issues and gaps within the current set of standards. The people who participate on IPC standards development committees are some of the best and brightest people in the industry. Most of them are natural-born problem solvers.

The intangible benefit of participation in standards development committees is access to some of the world's most gifted people when it comes to printed board design, fabrication, test, inspection and assembly. A cast of tens of thousands of subject-matter experts make up IPC's standards development network and more than 230 standards development committees, subcommittees and task groups are responsible for generating, reviewing and overseeing the incorporation of new technology into the standards. Who are these people? They are leaders within their companies who understand the value of expertise.

IPC recognizes our committee members in a variety of ways; most importantly, they are acknowledged in each of our standards. Problems are solved through the networking activities within IPC standards development communities. Some companies hire consultants, while others send their most talented, creative minds to standards development meetings to help solve complex problems. It's the network of subject-matter experts within the IPC that drives innovation for next-generation technology. It's the IPC committee volunteers who help connect innovation through standard development activities. **PCB**



David Torp has been the vice president of standards and technology for IPC since 2007. Prior to joining IPC, he was a senior staff engineer at Plexus. He also served as vice president of marketing and business development at Kester and held various engineering positions at Rockwell Collins and Underwriters Laboratories. Torp holds a Bachelor of Science degree in Chemical Engineering from Iowa State University.

Corporate Recognition Awards Honor DDi Corp. & Flextronics

IPC – Association Connecting Electronics Industries® bestowed its highest corporate honors to DDi Corporation and Flextronics. During luncheons held in conjunction with IPC APEX EXPO®, IPC awarded the IPC Peter Sarmanian Corporate Recognition Award to DDi Corporation and the IPC Stan Plzak Corporate Recognition Award to Flextronics. For many years, both companies have consistently provided staff resources to standards development and other committee initiatives.

Named for a former IPC Board Chairman, the Peter Sarmanian Corporate Recognition Award honors an IPC member company in the PCB industry that has taken a leadership role and made contributions to the industry while demonstrating support of IPC through participation in technical and/or management programs.

DDi Corporation has been an IPC member since 1996 and its staff is active on 22 technical committees. Mikel Williams, President and CEO, serves on IPC's Board of Directors and chairs the Government Relations Committee; an additional 70 employees have earned certified IPC specialist or certified IPC trainer status.

Commenting on DDi Corporations's involvement, Williams said, "We encourage broad participation of DDi employees in IPC. It's good for any industry to have active companies that are pushing forward with agendas that benefit the entire industry. In printed circuit boards, IPC is the conduit for these efforts."

Named after a former IPC Board Chairman who was a founding member of the IPC Electronics Manufacturing Services Industry Management Council, the Stan Plzak Corporate Recognition Award was presented to Flextronics for its initiative in enhancing the industry while demonstrating support of IPC through participation in technical and/or management programs.

An IPC member since 1993 with 90 member facilities across the world, Flextronics has staff active on 39 IPC technical committees. In addition, nearly 1,000 Flextronics employees have earned certified IPC specialist or certified IPC trainer status. In 2010, the company became IPC's first global corporate member.

"We've actively promoted the adoption and utilization of IPC standards at Flextronics," said Dr. Dongkai Shangguan, Vice President of Advanced Technology, at Flextronics International and an IPC Board member. "We have a global agreement with IPC that lets all Flextronics employees access IPC standards. That facilitates widespread adoption of these standards within our company."

For more information on IPC's Corporate Recognition Awards and other honors presented at IPC APEX EXPO, contact Anna Garrido, IPC Director of Marketing and Communications, at +1 847-597-2804.

COLUMN

Introduction to Board Level Simulation and the PCB Design Process



by Barry Olney
IN-CIRCUIT DESIGN PTY LTD | AUSTRALIA

SUMMARY — Board-level simulation reduces costs by identifying potential problems at the conceptual stage, so that they can easily be avoided, and then catching any further issues, during the design process, eliminating the potentially disastrous final stage changes.

High-speed digital multilayer boards can be designed to work right the first time, with little additional effort, providing you follow a tried and proven process that results in a reliable, manufacturable design that conforms to specifications and is produced on time and to budget.

Typically, a high-speed, computer-based design takes three iterations to develop a working product. However, the product life cycle these days is very short, and therefore, time-to-market is of essence. One board iteration can cost more than $25K. And this is only in engineering time and does not consider the cost of delaying the products' market launch. This missed opportunity could cost hundreds of thousands of dollars.

If changes are made late in the design process, then it takes more time, people, material and therefore, money. The idea of board-level simulation is to identify issues early in the design process and rectify them before they become a major problem.

As seen in Figure 1, design changes that occur in the conceptual stage cost nothing; during the design stage requires just a little extra time; during the test stage means that you have to go back one stage and re-design; and design changes during production, or worse still, in the field, can costs millions to fix and possibly damage the company's reputation.

This is where board-level simulation cuts costs: By identifying potential problems at the conceptual stage so that they can easily be avoided, and then catching any further issues

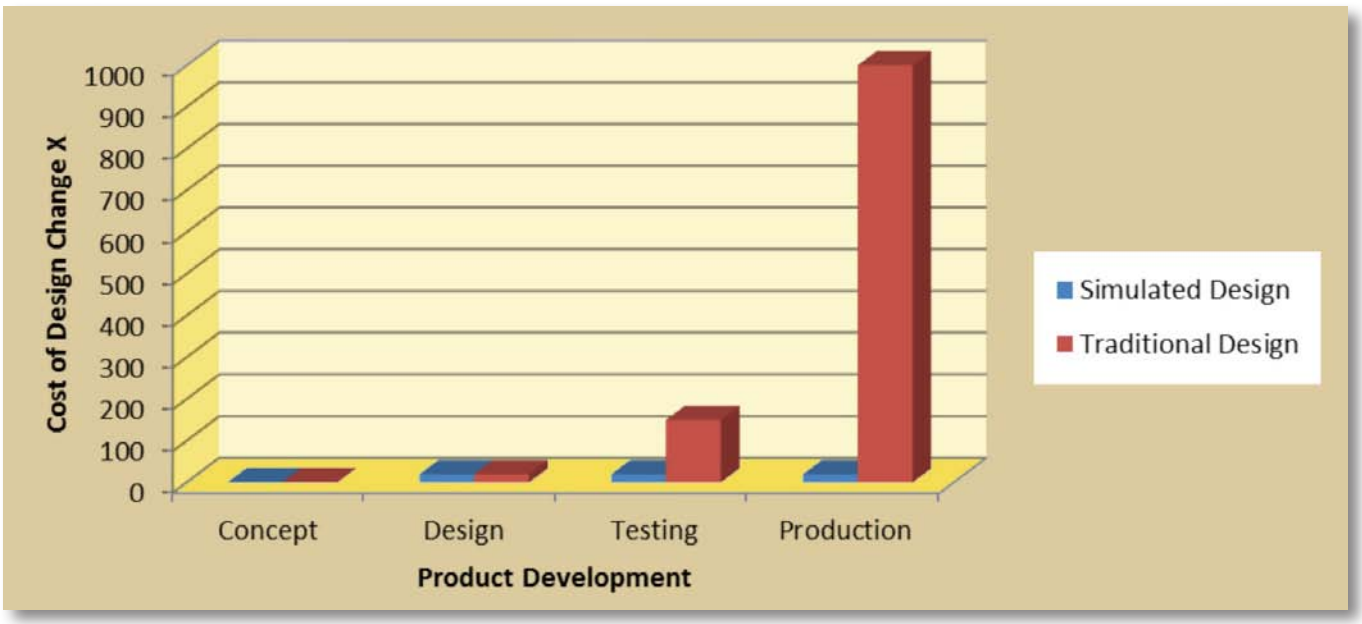


Figure 1: The cost of design change during the product development.

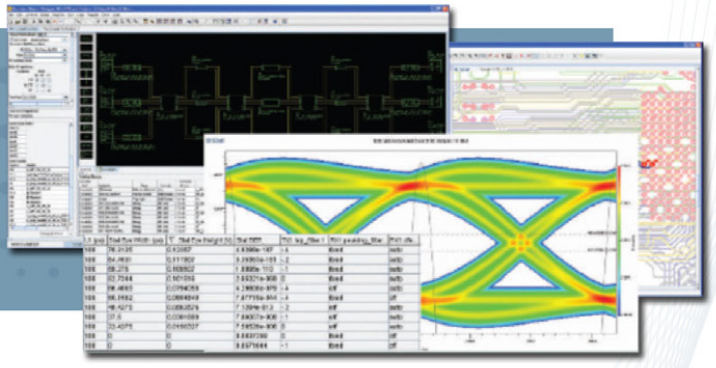
Consulting Services and Methodology Training
Automated Analysis Methodologies
Award-Winning EDA Tools
IBIS-AMI Modeling

Advanced Signal Integrity Solutions

for Full System Level Design

Quantum Channel Designer®

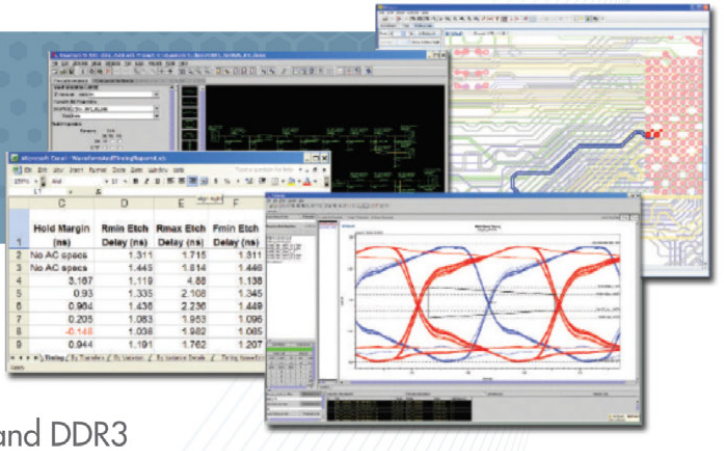
Multi-Gigabit Serial Link Design and Analysis



- 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

Copyright © 2012 Signal Integrity Software, Inc.
SiSoft, Quantum Channel Designer and Quantum-SI, are trademarks of Signal Integrity Software, Inc.
All other trademarks are the property of their respective owners.

during the design process, which eliminates the potentially disastrous final stage changes. Of course, we also need to keep our eye on the ball during the entire design process by catching any small issue before it becomes a major problem.

Board level simulation and the design process cover the following:

1. Stackup planning
2. PDN planning
3. Design rules and DFM
4. Pre-layout simulation
5. Mixed analog digital technologies
6. Critical placement
7. Interactive placement and routing strategies
8. Critical routing
9. Post-layout simulation (already done)

I will be reviewing these topics in more detail over the coming months, but here are some initial factors to consider for each:

1. Stackup Planning

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.

• Good interplane capacitance to reduce inductance at high frequencies.

• 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.

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 100 MHz so the interplane capacitance may not be so important at these frequencies.

It is imperative that we plan the board stackup from the beginning, ensuring that both single-ended and differential impedances conform to the technology requirements. And, be sure to ensure that the selected materials are available from our chosen fabrication shop — this step 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 could create a lot of unnecessary work.

At this stage we need to plan where the power planes will be in the stackup, considering that every signal layer needs to have a reference plane (either ground or power) adjacent to it in order to provide a clear return path.

2. PDN Planning

The design of the power distribution network (PDN) is also a very important part of the conceptual design process, ensuring that we have a stable power delivery system before we even start placing a chip on the board.

The idea of AC analysis of the PDN is to keep the effective impedance of the PDN as low and flat as possible over the required frequency range for reliable product performance. A switch mode voltage regulator module (VRM) provides low impedance up to about 30 KHz. Then, bypass tantalum capacitors lower the impedance up to about 10 MHz. Ceramic capacitors then provide high-frequency decoupling up to several hundred MHz. The power-to-ground plane capacitance of the PCB provides an ideal capacitor in that it has no series lead inductance and little equivalent series resistance (ESR), which helps reduce noise at extremely high frequencies.

This is a trial-and-error process and needs to be done with the assistance of an analysis tool. The ICD PDN Planner (available for download from www.icd.com.au) in Figure 3 illustrates a typical decoupling scheme showing how the VRM, capacitors and the planes are used to reduce the effective impedance of the PDN.

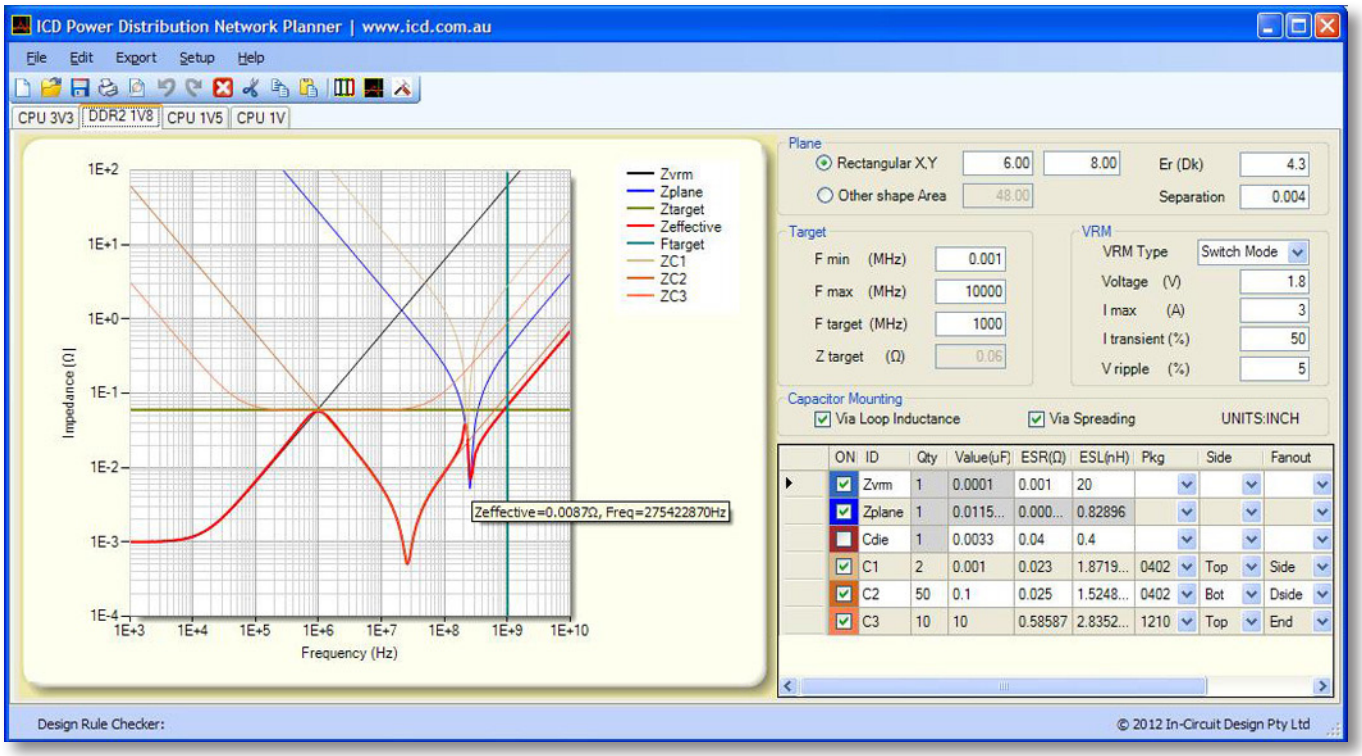


Figure 2: The ICD PDN Planner illustrates a typical decoupling scheme for 1.8V DDR2.

3. Design Rules and Design for Manufacturability (DFM)

Design rules should be set up and attached to critical nets in the schematic. This allows the engineer to transfer his desired intent, with regard to placement and routing, to the PCB designer without the information being lost in the process. One should keep in mind that if rules are modified in the PCB database then they need to be back-annotated to the schematic or they may be lost.

Design rules should be based on the manufacturer's device specifications and the relevant industry documents and standards that are available. For instance, DDR2 specifications (JESD79-2E) can be downloaded from JEDEC. Also, there are design guides and design technique documents available for download from Micron, Xilinx, Altera, etc., which are very helpful.

The IPC–2220 series of design documents, developed over the years by the industry, contain essential information for the design of SMT and mixed-signal multilayer boards and should be used to set up the basic design rules (www.ipc.org).

The technology rules are based on the minimum pitch of the BGA components employed and are basically the largest trace, clearance and via allowable whilst minimizing PCB fabrication costs.

Once these rules have been established, calculate the stackup required for the desired characteristic impedance (Z_o) and the differential impedance (Zdiff). These are typically Z_o = 50 ohms and Zdiff = 100 ohms. The ICD Stackup Planner can be used for these calculations. 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. So, a good range of Z_o is 50 - 60 ohms.

Design for manufacturability (DFM) is the practice of designing board products that can be produced in a cost effective manner using existing manufacturing processes and equipment.

DFM is gaining more recognition as it becomes clear that the cost reduction of printed circuit assemblies cannot be controlled by manufacturing engineers alone. The PCB designer

now plays a critical role in cost saving and the DFM needs to be practiced throughout the entire design process. Again, identifying an issue too far into the process can cost time and money.

4. Pre-Layout Simulation

Pre-layout simulation allows the designer to predict and eliminate signal integrity, crosstalk and EMC issues early in the design process. This is the most cost effective way to design a board with fewer iterations rather than starting with the post-layout simulation.

One can quickly simulate complex interconnect scenarios including integrated circuits, transmission lines, connectors and passive components and identify which scenario is best suited to a particular design.

The value and placement of the series resistors and VTT pull-ups for data, address and command nets depends on the distances between the loads, number of loads and the stackup of the board, and are best determined by simulation. The series terminator may not be required if a single memory chip is used and the trace length is short.

5. Mixed Analog and Digital Technologies

In the past, we have only had to deal with mixed analog and digital technologies, but radio frequency (RF) and analog mixed-signal (AMS) technologies, which serve the rapidly growing wireless communication market, are essential in today's PCB design. A digital system is also an RF system with significant noise and therefore interference potential.

Digital technologies with different voltage levels can also cause interfere with each other. The technology being used has changed quite dramatically over the years, from TTL devices (with high thresholds) to today's high-speed Gb/s devices (with noise margins as low as 500mV). The trend is toward lower processor core voltages, which conserves power. But, reducing the core voltage also reduces maximum operating frequency and the level of acceptable crosstalk.

In analog circuits, external noise sources are usually the primary concern. However, in digital circuits, the internally generated noise sources are of major concern.

When planes are used to distribute power supplies, crosstalk (coupling) can occur through the ground return path for these signals. This is called common-mode impedance coupling. Essentially, a returning signal causes a ground potential rise due to the DC resistance of the plane. This problem can be very significant, especially in analog circuitry when digital signals are present.

Although a single-point ground may be desirable in low-frequency analog circuits, it may be the primary source of noise coupling and emissions in a digital circuit. When mixed analog and digital circuits are used on the same PCB, the best approach, for grounding practices is to use one solid and common ground plane and ensure that the routing of the traces does not create crosstalk.

6. Critical Placement

To obtain an excellent route completion rate on a complex design, placement is extremely important. If the board is difficult to route, it may just be the result of poor placement.

The pre-layout simulation of high-speed signals gives us an advantage when it comes to placement. The critical nets must be of a certain length. For instance, the pre-layout simulation

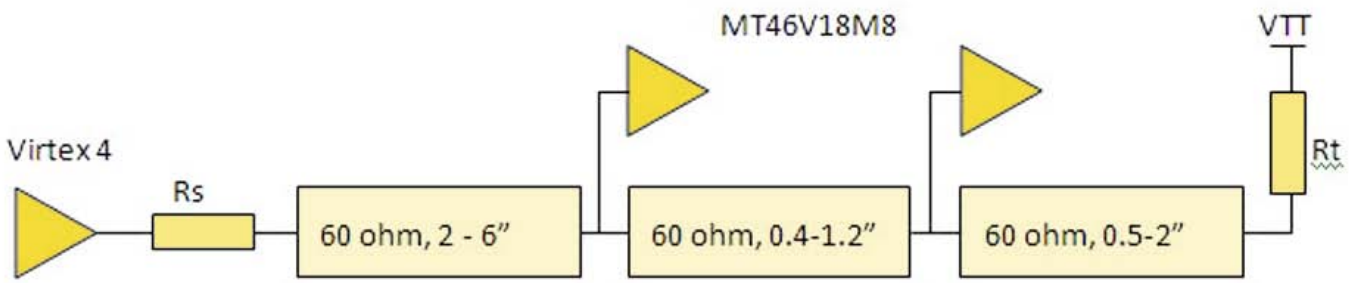


Figure 3: Example of DDR2 data signals simulated to find the best placement and routing strategy.

might suggest that a balanced T section route of 1.5 inches on all address, command and control lines of our DDR2 memory would be the best approach, and that data and strobes be no more than 1.2 inches matched to within 25 mils. Obviously, this does not give us much room to play with, so the correct positioning of these devices initially is of paramount importance. Placing the processor in the centre of the board allows all signals to be fanned out and promotes star wiring which is best for high-speed signals.

7. Interactive Placement and Routing Strategies

When the engineer creates the schematic he does so in logical, functional blocks — and that is exactly the way we should place and route the PCB. Cross-probing between the schematic and PCB database can be done in all major EDA packages and is recommended for design integrity.

For example, I often see PCB designers randomly placing resistors on the board. It is only a resistor — or is it? Well, that resistor could be a static pull-up that can be placed anywhere within reason or it could be a series terminator for a high-speed signal that needs to be placed within 200 mils of the driver. One should know the function of every component in order to make a valued decision with regard to effective placement, and cross-probing achieves this.

Critical placement should (of course) be done first. Then position the highest frequency components nearest the connectors and graduate the placement from high to low speed devices, with the low frequency analog devices positioned farthest from the connectors in a corner. Highlight the components on the schematic and move them into position on the PCB, one by one, making sure that you have the entire functional block in the one area. This is a tedious process, but it is the only way to ensure the placement is optimum.

Interactive routing is done in much the same way. Firstly, cross-probe and select the critical signals from the schematic, fanout, and route on the board. Route in priority from the most critical to least critical nets. I often use the autorouter to finish off the non-critical nets, but this of course depends on the quality of the autorouter. I certainly would not attempt this with a basic router.

8. Critical Routing

I previously mentioned that the pre-layout simulation determines the placement of the critical components which in turn determines the critical routing topology. So, it is a simple process once you have the design methodology right, although at times one loses a bit of hair when tuning nets to length. Having so many matched-length signals on the size of a postage stamp is a daunting thought.

For DDR designs, differential clocks should be routed first, because all other signal lengths/delays are referenced to tClk, followed by data and strobe signals. Next, route the address, command and controls signals. It is best, although the pre-layout simulation has given us a routing strategy, to route a couple of each signal group first, then run the post-layout simulator to check the waveforms, skew and radiation on this sample group before continuing with the routing. The pre-layout simulation gives us a good heads-up, but the physical board simulation can be quite different in some cases.

9. Post-layout Board Level Simulation

A preliminary batch mode simulation is initially completed on the design. Default IC characteristics, crosstalk of 150mV maximum and EMC to FCC, CISPR Class A and B are set up in the simulator. The batch mode simulation automatically scans large numbers of nets on an entire PCB, flagging signal integrity, crosstalk and EMC hot spots.

The post-layout simulation analysis can then be prepared using supplied specifications. This is an extensive interactive board level simulation which takes the analysis to the next level — simulating trouble spots identified by the batch analysis in order to further resolve the issues with greater accuracy.

Crosstalk is typically picked up on long, parallel trace segments. These can be on the same layer, but may also be broadside coupled from the adjacent layer. It is for this reason that orthogonal routing is recommended on adjacent layers (between planes) to minimize the coupling area.

Flight times of the critical signals are then examined. One could compare the matched lengths of each signal, but delay will vary depending on the meander pattern.

Since all products must comply with strict electromagnetic compliancy (EMC) regulations, all critical high-speed signals should be simulated to determine the amount of expected radiation.

Board-level simulation is engaged too often toward the end of the design cycle. Simulation should be done throughout the entire design process to ensure that the design is on track. A simulation service such as that provided by In-Circuit Design can complement your design team. It will save you time, money and frustration, but more importantly, from an engineer's or designer's point of view, give you confidence in the performance and reliability of the product. **PCB**

References:

1. Advanced Design for SMT – Barry Olney
2. PCB Design Techniques for DDR, DDR2 & DDR3 – Barry Olney
3. Design for EMC – PCD Magazine Jan 96, Barry Olney
4. The Perfect Stackup – Barry Olney
5. Controlling the Beast – Barry Olney
6. Matched Length Routing – Barry Olney
7. High Speed Signal Propagation – Howard Johnson
8. Electromagnetic Compatibility Engineering – Henry Ott
9. 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.

AT&S Leads Hermes Consortium

Miniaturization combined with ever increasing functionality and enhanced reliability has become part of our way of life, not least in the smartphone sector. The Hermes Consortium's task is to industrialize new miniaturization technologies.

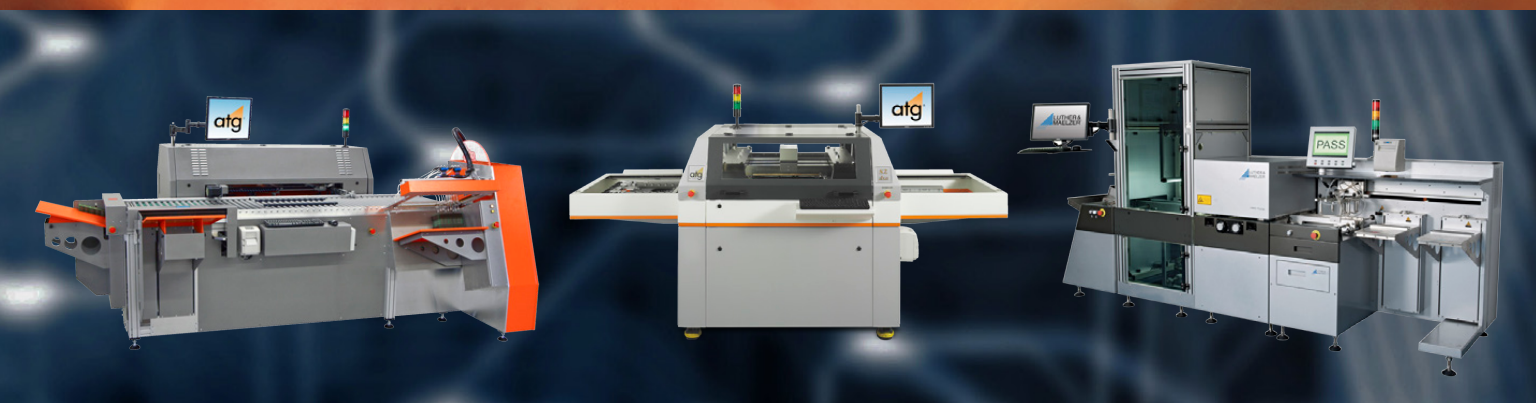
In spring 2008 AT&S brought together eleven renowned European global players in one of the largest ever EU-sponsored projects. The participants came from different stages in the value chain in the automotive, aeronautical and other industrial sectors.

Hermes goes far beyond what is currently possible in connectivity. Series production processes are used to embed active components such as chips, as well as passive components such as resistors and capacitors in the interior of the PCBs. This creates additional space on the circuit boards, and the improvements in connectivity increase energy efficiency and extend the product's useful life.

One of the Consortium's main goals is to set standards for the industry, and to generate advantages over the competition in Asia. Industrial implementation of this technology opens up a wide range of potential applications in medicine (including new generations of hearing aids and pacemakers) and in functional modules for GPS, WLAN, Bluetooth and cameras. At the same time, AT&S is strengthening its position with existing customers as an innovation and technology leader, and is well placed to offer new solutions.

ECP technology has been brought to market in the course of the project following the successful construction of the production line in Hinterberg. AT&S picked up the Fast Forward Award 2011 for its ECP® technology. In the next few years, the complexity inside and on the surface of the modules will continue to grow, and the 3D capabilities of embedding technologies will pave the way for the next generation of mobile electronic devices.

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

Top Ten Most-Read Flex007 Highlights This Month



MFLEX, AT&S Collaborate on HDI Rigid-Flex Circuits

The resulting alliance will enable the parties to offer best-in-class HDI rigid-flex printed circuits for electronic devices, and positions both companies to tap new opportunities in additional applications beyond their current offerings, as well as continue to pursue increasing market opportunities in their core businesses.

Flexible, Printed Electronics Will Usher in "Organic Age"

Flexible electronics, now being printed, is the future of new applications in sensors, displays, power and lighting according to experts gathered at the FlexTech Alliance 2012 Flexible Electronics & Displays Conference & Exhibition.

Printed Circuits Ramps Up Rigid-Flex Manufacturing Capacity

Printed Circuits has expanded its rigid-flex manufacturing capacity with the installation of a new Plasma Etch MK II plasma etching system.

MFLEX Reports Strong Fiscal 2012 Q1 Results

Commenting on the company's business outlook, CEO Reza Meshgin noted, "We anticipate continued strong demand for our flex assemblies for both smartphones and tablets in the second quarter of fiscal 2012."

All Flex Offers Free Flexible Circuit Design Course

All Flex Flexible Circuits announces the addition of an online design course, "Designing Flexible Circuits," authored by John Michael Pierobon, an electronics industry expert and writer who has also taught numerous technical courses throughout the world.

FlexTech Alliance Names 2012 FLEXI Award Winners

Thin Film Electronics, PARC, Western Michigan University, and the Georgia Institute of Technology receive honors in innovation, R&D, and technology leadership in education.

Schweizer Electronic Reports Stable Fiscal 2011 Sales

Despite a seasonally weaker fourth quarter, Schweizer Electronic AG was able to keep its annual sales stable at 105.4 million Euro, compared to the previous year, and within expectations.

DYCONEX Launches New Information Platform

Since 2010, DYCONEX has been actively participating in the FP7-SME-2008-1 IMAGINE Project, which targets strengthening the research capabilities of SMEs and is funded by the European Union (Grant Agreement no. 231997).

Printed Electronics Asia 2012 Highlights Growth

IDTechEx will host the Printed Electronics Asia conference and tradeshow on October 2-3, 2012 in Tokyo, Japan. The event, part of the IDTechEx global series of events on the topic, focuses on a region that has invested more in printed electronics than any other territory.

Printed Electronics Europe to Tackle Hot Topics

IDTechEx is hosting its 8th annual Printed Electronics Europe event in Berlin, Germany, April 3-4, 2012. The event will focus on the commercialization of printed electronics, with adopters from many verticals discussing needs and experiences.

INTEGRAL TECHNOLOGY

Circuit boards in smartphones are under a lot of stress.





BEND HEAT DROP



Zeta® products relieve it.

ZETA®

NEXT GENERATION HDI

STOP PAD CRATERING NOW!

949.587.3370 www.HDIoptionstoday.com



IEC Cooperative Activities Yield Maximum Benefit for the Global Electronics Market

By Dieter Bergman and Marc Carter

SUMMARY — IEC standards provide a distillation of best practices on a global basis that have been reviewed by an array of technical experts. For a standard to earn the IEC logo, the information it includes must undergo a rigorous international review, discussion and approval process, which makes for the generation of robust and well-defined standards.

Introduction

The International Electrotechnical Commission (IEC) is the world's central organization for the preparation of electrical, electronic and related technology standards. It is one of three sister organizations (IEC, ISO, ITU) that develop international standards, and since its founding in 1906, has amassed thousands of standards in its library. IEC activities are governed by a Standardization Management Board that is responsible for the overall management of technical work. The standards work of the IEC is carried out through technical committees and subcommittees composed of the representatives of the full member national committees, each dealing with a particular subject. Many standard development organizations serve on IEC technical committees and supply technical resources and subject matter expertise through national committees. Technical committees are created by the Standardization Management Board. Each technical committee has a chairman and secretariat.

Organization

IEC has 177 technical committees and subcommittees. The technical committee membership is composed of the IEC National Committees, all of which can take part in any technical committee as a participating member or observer. IEC technical committees and subcommittees (Working Groups) prepare documents on specific subjects which are then submitted to the national committee for vote and approval as an international standard. Each national committee of IEC handles the participation of experts for its country.

Standardization Process

The IEC process for development of international standards is rigorous, well-defined and specific, ensuring that each nation's representatives have every opportunity for input

at multiple stages along the way. The dedication of participants on IEC technical committees for creation of standards is highly regarded throughout the electronics industry. The standards that pass through this process help promote a common understanding between participants in the electronics industry, regardless of where they are located. The possibility of more specific or more stringent regional or industry standards is retained, but goods and services produced by all participants will meet the minimums defined in the IEC standards.

For the printed board market, the most applicable IEC technical committee is TC-91, initially established in 1990 for surface mount technology. In 2000, TC-52 for printed boards was merged into TC-91. The scope of TC-91 is to prepare international standards on electronics assembly-relevant technology and in the field of printed board assemblies.

Within the IEC portfolio of standards, the PCB community is highly interested in the 61249 series of standards regarding laminates. This series of standards is currently under review for incorporation of lead-free materials, as well as RoHS-compliant materials into the standard (TC91-WG4). Within the test and measurement for printed boards and materials, the IEC 61189 series of standards is being updated to include new components with new logic descriptions (TC91-WG10). More detailed information regarding specific Working Group activities are included later in this article under “Current Activities.”

Given the IEC’s extensive process to ensure all national committees’ input, it is understandable that organizations with a more direct operational focus (IPC for example) will create standards for immediate, specific application use by their members. These pre-existing documents often provide the basis for eventual IEC standards. A partial listing of documents originating with collaborating organizations that have been or are in the process of being incorporated into IEC standards follows.

IEC also collaborates with many standards development organizations such as IPC, JEITA, JPCA, IEEE and JIC as required, to take content from the electronics industry’s most frequently accessed standards and incorporate content into IEC standards. This collaboration activity provides for a global harmonization for best practices for printed circuit card design, materials, processes, test and measurement.

Activity

Current action plans for TC 91 are summarized under each working group (WG) listed below:

WG 1: Requirements for electronic components

Working Group 1 is responsible for:

- Describing how the assembly process stresses the surface mount devices (SMDs)
- Finding a suitable classification for the SMDs to meet the different process conditions
- Describing the tests with which this classification can be demonstrated

Current activities:

- IEC 61760-4: Classification, packaging, labeling and handling of moisture sensitive devices

WG 2: Requirements for electronics assemblies

Working Group 2 is responsible for establishing international acceptance for standards and guidelines in surface mounted assembly technology. Areas of immediate work will include evaluating standards or guidelines for the following:

- Assembly process definitions (methods and options)
- Solder material, fluxes and cleaning standards
- Solder joint and workmanship standards
- Substrate material selection and specification
- Requirements for writing specifications for SMT
- In process test and measurement methods
- Assembly equipment capability evaluation guidelines

Current Activities:

- IEC 61190-1-2 am1: Attachment materials for electronic assembly
  - Part 1-2: Requirements for soldering pastes for high - quality interconnects in electronics assembly
- IEC 61191-1: Printed board assemblies
  - Part 1: Generic specification Requirements for soldered electrical and electronic assemblies using surface mount and related assembly technologies
  - Part 2: Sectional specification - Requirements for surface mount soldered assemblies
- IEC 62588: Marking and Labeling of Components, PCBs and PCBAs to Identify Lead (Pb), Lead-Free (Pb-Free) and Other Attributes

WG 3: Measuring and test methods for electronics' assemblies

Working Group 3 is responsible for the development of measurement and test methods specific to surface mounting technology and not covered by other technical committees of the IEC.

Current Activities:

- IEC 60068-2: Environmental testing
  - Part 2-58: Tests - Test Td- Test methods for solderability, resistance to dissolution of metallization and to soldering heat of surface mounting devices (SMD)
  - Part 3-13: Supporting documentation and guidance on test T: Soldering.
- IEC 61189-5: Test methods for electrical materials, interconnection structures and assemblies – Test methods for printed board assemblies
  - Part 5-1: Guidance
  - Part 5-2: Soldering Flux
  - Part 5-3: Soldering paste
  - Part 5-4: Solder alloys and fluxed and non-fluxed solid wire
  - Part 5-5: Surface insulation resistance (SIR)
- IEC 61189-11: Test methods for electrical materials, interconnection structures and assemblies
  - Part 11: Measurement of melting temperature or melting temperature ranges
- IEC 62137-4: Electronics assembly technology
  - Part 4: Environmental and endurance test methods for solder joint of area array type package surface mount devices
- IEC 62739-1: Test method for erosion of wave soldering equipment by molten lead free solder alloy
  - Part 1: Erosion test method for the metallic material without surface treatment

WG 4: Printed board and materials

Current Activities:

- IEC 61193-3: Quality Assessment Systems
  - Part 3: Selection and use of sampling plans for printed board and laminate end-product and in-process auditing
- IEC 61249-2: Materials for printed boards and other interconnecting structures; Reinforced base materials clad and unclad
  - Part 2-27: Bismaleimide/triazine modified with non-halogenated epoxide woven glass laminate sheets*
  - Part 2-30: Non-halogenated epoxide modified cyanate ester woven glass laminate*
  - Part 2-39: High performance epoxide and non-epoxide, woven E-glass laminated sheets for lead-free assembly*
  - Part 2-40: High-performance, modified, non-halogenated epoxide woven E-glass laminated sheets for lead-free assembly* [* of defined flammability (vertical burning test), copper-clad]
- IEC 61249-4: Materials for printed boards and other interconnecting structures – Sectional specification set for prepreg materials, unclad (for the manufacture of multilayer boards)
  - Part 4-18: High performance epoxide woven E-glass prepreg for lead-free assembly*
  - Part 4-19: High performance, non-halogenated epoxide woven E-glass prepreg for lead-free assembly*
- IEC 61249-6: Materials for printed boards and other interconnection structures
  - Part 6-3: Reinforcements - Woven fiberglass fabrics

Source documents and corresponding IEC numbers

SCOPE	Original Document	IEC equivalent or similar
Scope: Commonly-used terms and definitions in the electronics industries	**IPC-T-50 Terms and Definitions for Interconnecting and Packaging Electronic Circuits**	**IEC 60194** Printed Board Design, Manufacture and Assembly - Terms and Definition
Scope: This standard applies to components and assemblies that contain Pb-free and Pb-containing solders and finishes. This standard describes the marking of components and the labeling of their shipping containers	**IPC/JEDEC J-STD-609 Marking and Labeling of Components, PCBs and PCBAs to Identify Lead(Pb), Pb-free and Other Attributes**	**IEC/PAS 62588** Marking and labeling of components, PCBs and PCBAs to identify lead(Pb), Pb-free and other attributes
ASSEMBLY		
Scope: This standard prescribes practices and requirements for the manufacture of soldered electrical and electronic assemblies.	**J-STD-001 Soldering Electrical and Electronic Assemblies**	**IEC 61191** Printed board assemblies - - 1: Generic specification-soldered assemblies - 2: Requirements: surface mount assemblies - 3: Requirements: through-hole assemblies - 4: Requirements: for terminal assemblies
ACCEPTANCE		
Scope: This standard is a collection of visual quality acceptability requirements for electronic assemblies.	**IPC-A-610 Acceptability of Electronic Assemblies**	**IEC 61192** Workmanship requirements for soldered electronic assemblies -1: General -2: Surface mount assemblies -3: Through-hole mount assemblies -4: Terminal assemblies
ASSEMBLY SUPPORT		
Scope: This document covers procedures for repairing and reworking printed board assemblies. It is an aggregate of information collected, integrated and assembled by the Repairability Subcommittee (7-340 of the Product Assurance Committee of the IPC.	**IPC-7711 Rework of Electronic Assemblies**	**IEC 61192-5** Workmanship requirements for soldered electronic assemblies – Part 5: Rework, modification and repair of soldered electronic assemblies
ASSEMBLY MATERIALS		
FLUX/SOLDER		
Scope: This standard prescribes general requirements for the classification and testing of soldering fluxes for high quality interconnections.	**J-STD-004 Requirements for Soldering Fluxes**	**IEC 61190-1-1** Attachment materials for electronic assembly -1-1: Requirements for soldering fluxes
Scope: This standard prescribes general quality control requirements for the characterization and testing of solder pastes used to make high quality electronic interconnections.	**J-STD-005 Requirements for Soldering Pastes**	**IEC 61190-1-2** Attachment materials for electronic assembly -1-2: Requirements for solder pastes
Scope: This standard prescribes the nomenclature, requirements and test methods for electronic grade solder alloy.	**J-STD-006 Requirements for Electronic Grade Solder Alloys and Fluxed and Non-Fluxed Solid Solders**	**IEC 61190-1-3** Attachment materials for electronic assembly -1-3: Requirements: solder alloys and fluxed and non-fluxed solid solders, electronic soldering applications
COMPONENTS		
Scope: This standard identifies the classification level of nonhermetic solid state surface mount devices (SMDs) that are sensitive to moisture-induced stress for proper packaging, storiage, and handling	**J-STD-020 Moisture/Reflow Sensitivity Classification for Nonhermetic Solid State Surface Mount Devices**	**IEC 62190 Moisture/Reflow Sensitivity Classification for Nonhermetic Solid State Surface Mount Devices**
Scope: The purpose of this document is to provide SMD manufacturers and users with standardized methods for handling, packing, shipping, and use of moisture/reflow sensitive SMD packages which have been classified to the levels defined in J-STD-020.	**J-STD-033 Handling, Packing, Shipping and Use of Moisture/Reflow Sensitive Surface Mount Devices**	**IEC/PAS 62169 Handling, Packing, Shipping and Use of Moisture/Reflow Sensitive Surface Mount Devices**
Scope: This test method defines the procedures for performing acoustic microscopy on non-hermetic encapsulated electronic components.	**J-STD-035 Acoustic Microscopy for Non-Hermetic Encapsulated Electronic Components**	**IEC 60749-35 Semiconductor devices - mechanical and climatic test methods - Part 35: Acoustic microscopy for non-hermetic, encapsulated electronic components**
LAMINATE		
RIGID		
Scope: This specification covers the requirements for base materials, herein referred to as laminate or prepreg, to be used primarily for rigid or multilayer printed boards for electrical and electronic circuits.	**IPC-4101 Specifications for Base Materials for Rigid and Multilayer Printed Boards**	**IEC 61249-2 Materials for printed boards and other interconnecting structures - Part 2 Reinforced base materials clad and unclad**
ASSEMBLY		
Scope: This document provides information on land pattern geometries used for the surface attachment of electronic components. The intent of the information presented herein is to provide the appropriate size, shape and tolerance of surface mount land patterns to insure sufficient area for the appropriate solder fillet, and also to allow for inspection and testing of those solder joints.	**IPC-SM-782 Surface Mount Design & Land Pattern Standard Replaced by IPC-7351**	**IEC 61188-5 Printed boards and printed board assemblies - Design and use** - Part 5: Attachment (land/joint) considerations. Multiple subsections by component configuration
INTERFACES		
Scope: This standard specifies record formats used to describe printed board products with detail sufficient for tooling, manufacturing and testing requirements.	**IPC-D-350 Printed Board Description in Digital Form**	**IEC 61182-1 Printed boards - Electronic data description and transfer - Part 1: Printed board description in digital form**
Scope: This standard describes a data format for transmitting bare board electrical test information.	**IPC-D-356 Bare Substrate Electrical Test Data Format**	**IEC 61182-7 Printed boards - Electronic data description and transfer** - Part 7: Bare board electrical test information in digital form
Scope: This standard specifies data file formats used to describe printed board and printed board assembly products with details sufficient for tooling, manufacturing, assembly, inspection and testing requirements.	**IPC-2511 Generic Requirements for Implementation of Product Mfg. Description Data & Transfer Methodology**	**IEC/PAS 62119 Generic requirements for implementation of product manufacturing description data and transfer methodology** (Equivalent to IPC-2511A)
This standard specifies the XML schema that represents the intelligent data file format used to describe printed board and printed board assembly products with details sufficient for tooling, manufacturing, assembly, and inspection requirements.	**IPC-2581 Generic Requirements for Printed Board Assembly Products Manufacturing Description Data and Transfer Methodology**	**IEC 61182-2 Printed board electronic description and transfer -** Part 2: Product description standard for printed board, printed board assembly and testing in XML

- IEC 62326-20 Ed.1: Electronic circuit board for high-brightness LEDs
- IEC/TS 61249-3: Materials for printed boards and other interconnecting structures
  – Part 3-1: Copper-Clad Laminates for Flexible Boards (Adhesive and Non-adhesive types)
- IEC/TS 62326-7: Printed boards
  – Part 7-1: Performance guide for single- and double-sided flexible printed boards

WG 5 (TC 91/TC 93): Terms and definitions

Working Group 5 is responsible for establishing terms used in design, manufacture, assembly, testing and reliability of printed boards, printed board assemblies, and related materials and components including clear and concise definitions for those terms that represent agreed international preferred terminology.

Current Activities:

- IEC 60194 Ed. 6.0: Printed board design, manufacture and assembly - Terms and definitions.

WG 6: Device embedded substrate

Working Group 6 is responsible for preparing guidelines for new types of printed boards.

Current Activities:

- IEC 62326-14: Printed boards - Device embedded substrate - terminology/ reliability/design guide
- IEC 62326-15: Printed boards - device embedded substrate - general electrical test guide for device embedded substrate with active devices, passive components (capacitor, resistor, inductor, etc.), integrated passive device (ipd), and discrete packages.
- IEC 62326-18: Printed boards - device embedded substrate - test method
- IEC/TS 62326-16: Printed boards - device embedded substrate - scope and definition
- IEC/TS 62326-19: Printed boards - device embedded substrate - design guide

WG 10: Measuring and test methods for printed boards and printed board materials

Working Group 10 is responsible for:

- Reviewing the test methods of IEC 61249-1 (base materials) of IEC 62326-2 (printed boards) and those test methods of IEC 60068-2 that apply to base materials, printed boards and printed board assemblies
- Analyzing the effectiveness of the existing methods and identify methods which have hazardous materials or environmental impact
- Developing proposed revisions of methods that have been identified as hazardous or having an environmental impact
- Recommending change or deletion of methods no longer considered effective
- Starting work on new methods

Current Activities:

- IEC 61189-2: Test methods for electrical materials, printed board and other interconnection structures and assemblies
  – Part 2-628: Mechanical test methods for printed board and assembly materials - Glass transition temperature of base material by dynamic mechanical analysis (DMA)
  – Part 2-629: Test method of adhesion between a dielectric and inkjet printed circuits to be used for printed electronics sheet (board) application
  – Part 2-716: Electrical test methods for printed board and assembly materials - Relative permittivity and dissipation factor (500MHz to 10GHz)
  – Part 2-720: Electrical test methods for printed board and assembly materials - Relative permittivity and dissipation factor (1MHz)
- IEC 61189-3: Test methods for electrical materials, printed boards and other interconnection structures and assemblies
  – Part 3-913: Test methods for interconnection structures (printed boards) - Test methods for electronic circuit board for high-brightness LEDs

...More than
Immersion Tin

...More than
Immersion Silver



Uyemura ENEPIG

Provides a Final Finish that is as close to Indestructible as a Final Finish can be!



Uyemura's Electroless Nickel prevents Cu diffusion, strengthens thru-holes and has the **greatest line definition for tight lines and spaces of any EN.**

Plus: Exclusive Uyemura NPR nickel eliminates bridging and operates at 176°F for best possible soldermask compatibility.

Uyemura Electroless Palladium allows finish to be solderable and wire bondable, without damaging the nickel.

Uyemura's Exclusive Immersion Gold provides a low-porosity deposit – and further boosts wire bondability.

UYEMURA
USA

Tech Center:
Southington, CT • ph: (860) 793-4011
(800) 243-3564 • fax: (860) 793-4020

Corporate Headquarters:
Ontario, CA • ph: (909) 466-5635
(800) 969-4842 • fax: (909) 466-5177

www.uyemura.com

WG 12: Design of printed boards and board assemblies

Working Group 12 is responsible for developing and coordinating design methodologies and principles that correspond to the concept of automated or manual design for competitive manufacture. Included in the principles are design for fabrication, assembly, test, reliability, and the environment.

Current Activities:

- IEC 61188-8: Component shape data specification for CAD library
  - Part 8-1: Generic descriptions of the 2D and 3D description

JWG 9 (TC 86/TC 91): Optical functionality

Joint Working Group 9 is responsible for optical PCBs and assemblies and is a joint working group between TC 91 and TC 86. A recent roadmap of optical assembly technology has been prepared and reported at the TC 86 plenary.

Current Activities:

- IEC 62496-2
  - Part 2-1: Attenuation and isolation measurement procedures for optical characteristics
- IEC 62496-3
  - Part 3-1: Flexible optical circuit boards using unconnectorized optical glass fibres
- IEC 62496-3: Performance standards
  - Part 3: General and guidance

JWG 11 (TC 91/TC 93): Printed board electronic data description and transfer

Joint Working Group 11 is responsible for digital descriptions of completely designed printed boards and printed board assemblies with emphasis on the manufacturing requirements of the final product under IEC Publication Series 61182. A close liaison is maintained with TC93 Design Automation Technology, which has the responsibility for computer formats and is administered by IEEE, and includes software testing procedures used to transfer information between computers.

Current activities:

- IEC 61182-2: Generic requirements, complete XML schema structure/full Mode: This document is the most basic document to the work of data description and transfer
- IEC 61182-2-1: Design requirements, three design mode functions/original CAD
- IEC 61182-2-2: Board fabrication requirements, three board fab mode functions/multilayer

Summary

The benefit of IEC standards is that they provide a distillation of best practices on a global basis that have been reviewed by an array of international technical experts. The information included in IEC standards has been reviewed, discussed and approved several times before a standard is considered worthy to bear the IEC logo. The rigorous international standards review and modification processes are designed to generate robust and well defined standards generation. **PCB**



Marc Carter is the director of Technology Transfer at IPC — Association Connecting Electronics Industries, and responsible for oversight of IPC technology and printed electronics roadmap activities, and supporting IPC efforts in regulatory compliance, training, and support of the developing printed electronics industry. Carter has been actively employed in electronic fabrication, assembly, and applied material science industries for approximately 36 years and joined IPC in 2011.



Dieter Bergman joined IPC as technical director in 1974. In 1985, he was named to the IPC "Hall of Fame," one of IPC's highest awards. As technical director, he was responsible for the coordination of standards, specifications and guidelines development, round robin test programs. In 1984 he became director of technology transfer in order to foster the interchange between design and manufacturing.

Most-Read MilAero007 News Highlights This Month



Merlin Flexible Circuits Achieves AS9100 to Rev C

Rev C expands the scope of AS9100 and is designed to meet the stringent and complex demands and requirements of the defense and commercial aerospace sectors.

Compunetics Certified by Defense Logistics Agency

Compunetics, Inc. has been certified by the Defense Logistics Agency Land and Maritime (DLA), located in Columbus, Ohio, to supply PCBs to the requirements of MIL-PRF-31032.

Fineline to Distribute FTG's Aerospace/Defense Products

Fineline Group has been appointed by Firan Technology Group as its representative and distributor for its aerospace and defense electronics products.

Reports of Counterfeit Parts Quadruple Since 2009

Information and analysis provider IHS reports that counterfeit parts have soared dramatically in the last two years, presenting huge challenges for electronics manufacturing and especially the military and aerospace industry at a time when the U.S. government is tightening regulations covering fake components.

Military Aircraft Avionics Market Set to Reach $16.94B in 2012

Visiongain's analysis indicates that the military aircraft avionics market will reach a value of $16.94 billion in 2012, as militaries around the world continue to develop and upgrade their avionics capabilities for a number of military settings and roles.

Adams Named Prototron Circuit's National Sales Manager

Dave Ryder, president of Prototron Circuits of Redmond, Washington and Tucson, Arizona, has announced the appointment of long-time sales professional Russ Adams as National Sales Manager.

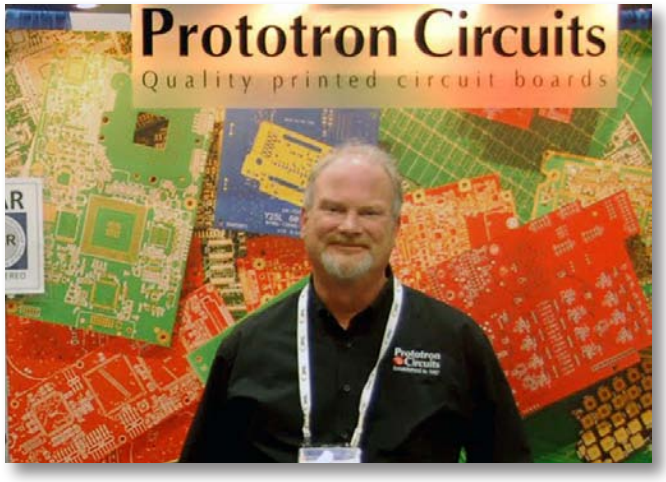
In this new position, Adams will be responsible for the company's outside sales effort, managing the entire sales team, including both direct and independent representatives.

Celebrating 25 years in business, the company has a national sales team covering United States and Canada.

In the PCB business for 34 years and involved in the sales of PCBs for over 20 years, Adams worked for several other PCB companies in the Northwest before coming to Prototron Circuits 10 years ago.

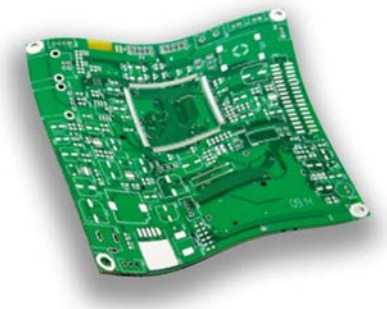
Commenting on his new position, Adams said, "I am thrilled to be given this opportunity. Prototron is by far the best company I've ever sold for and to be able to manage what I consider the best sales team in the business is a great honor. Our 25th year promises to be our best ever and I am very happy to be a part of that."

Ryder added," Russ has truly earned this position. He has worked hard as the Western Sales Manager for the past three years, growing that territory to where it is today, and I am sure that he will be able to do the same great job for us throughout the country."



COLUMN

Rigid-flex Manufacturing Part III: Z-axis Expansion



by Dale Smith
DAS FLEX CIRCUIT CONSULTANTS, LLC

SUMMARY — In this installment in the series on rigid-flex manufacturing, Z-axis expansion of PCB materials is examined—what it is and why we should be concerned about it. What are the expansion rates for materials that are used in rigid-flex construction, and which have desirable low Z-axis expansion rates versus those that can cause plated through-hole reliability issues?

What is Z-axis Expansion?

Z-axis expansion is an expansion of the materials when exposed to thermal conditions (hot and cold) relating to the Z direction (thickness) of the rigid-flex circuit board. As thickness increases, the Z-axis expansion rate for the materials can become problematic for plated through hole interconnect reliability. If the Z-axis expansion rate of the materials is greater than the expansion capability of the electrodeposited copper plating, after repeated thermal exposure cycling, the plating will eventually fracture. This can lead to electrical failures (opens or intermittent opens). Figure 3.1 is an example of a PTH failure after solder float.

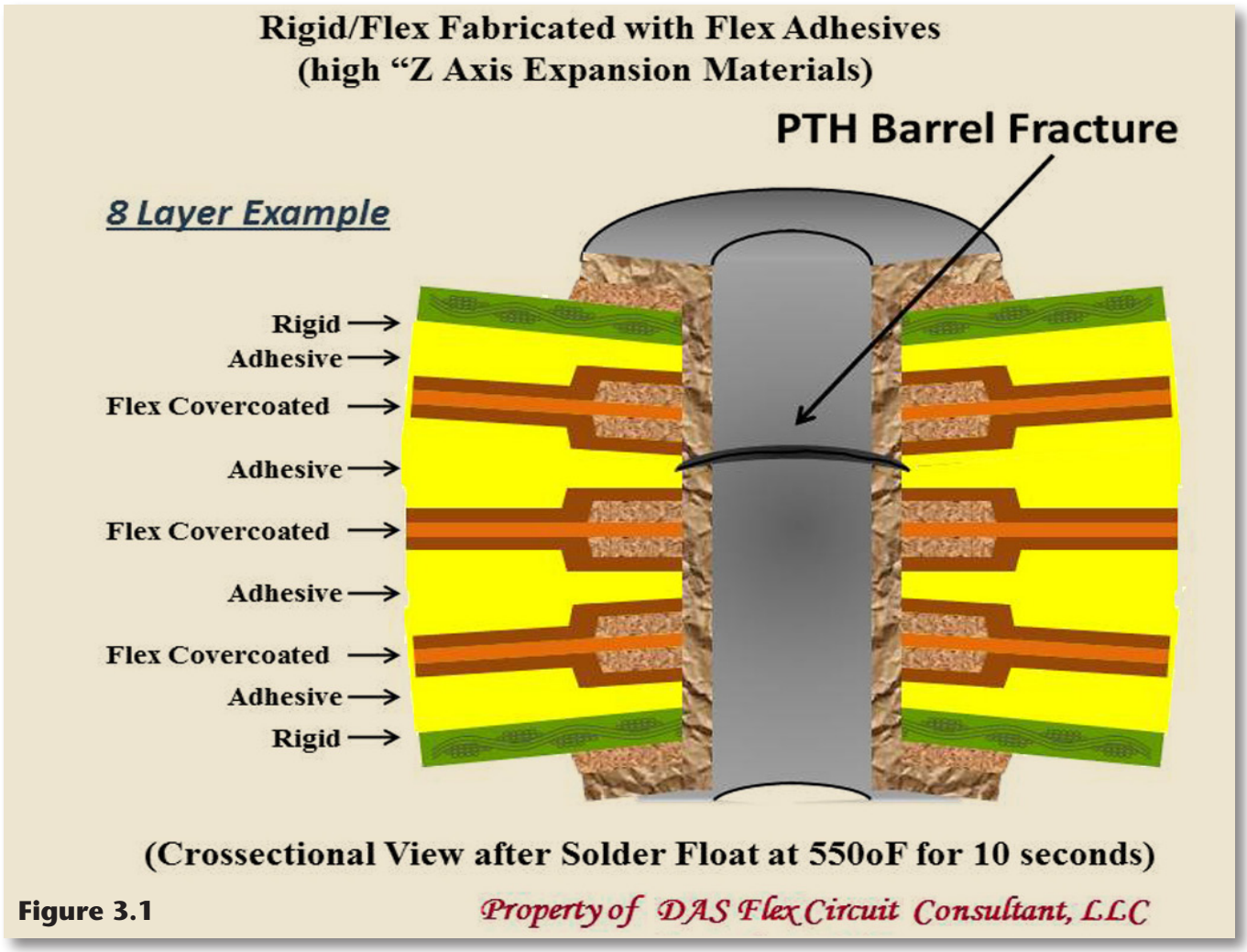
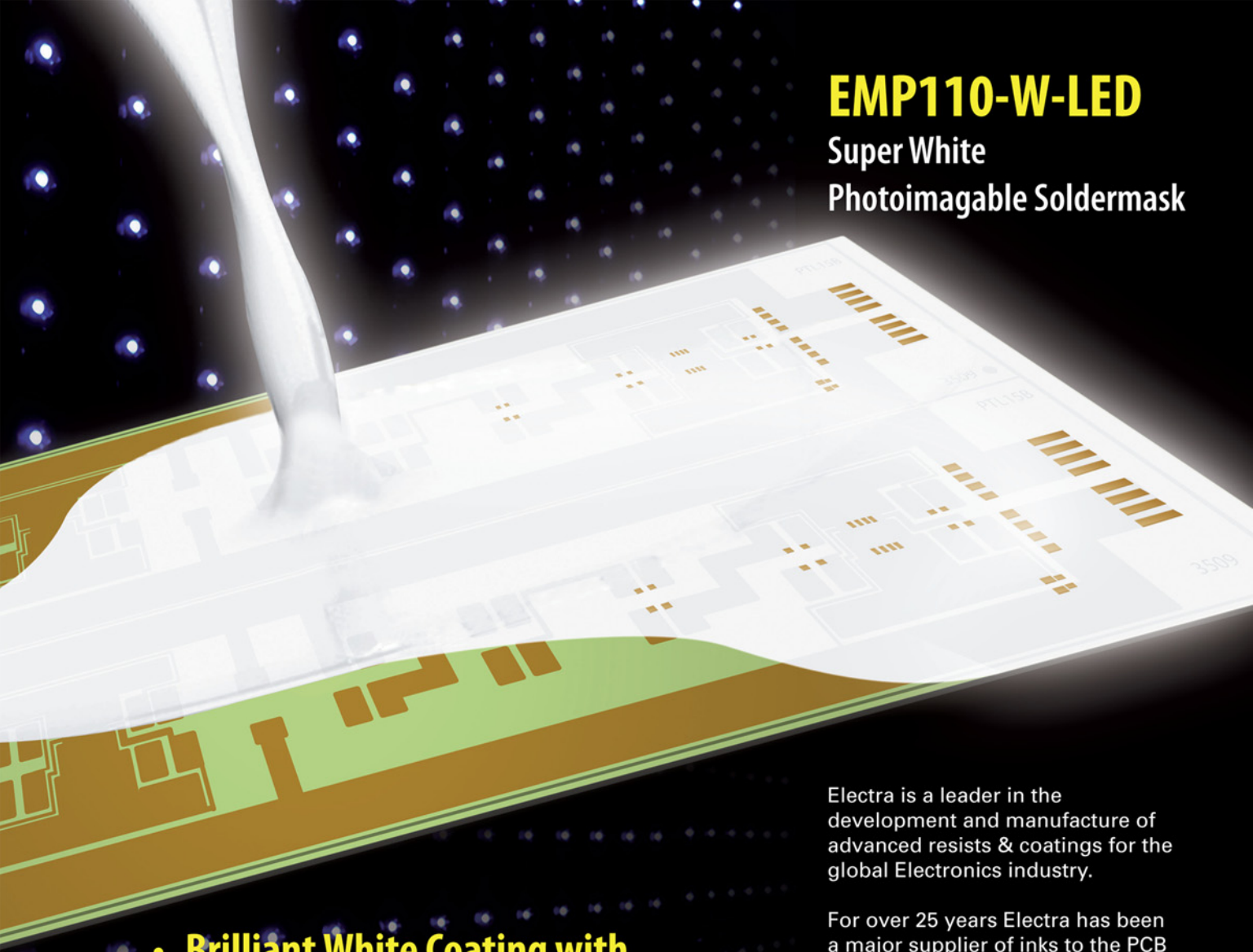


Figure 3.1



EMP110-W-LED

Super White Photoimagable Soldermask

- **Brilliant White Coating with High Blocking Power**
- **Tailored Reflectivity for White LEDs**
- **Non Yellowing - High Resistance to Thermal and UV Ageing**
- **Rapid Tack Dry for Use on High Heat-sink Substrates**
- **Screen Print and Spray**
- **Lead-free Compatible**

Electra is a leader in the development and manufacture of advanced resists & coatings for the global Electronics industry.

For over 25 years Electra has been a major supplier of inks to the PCB market with worldwide distribution and support.

Electra offers customers unparalleled choice from our extensive range of products and OEM approvals.
Products include; Conventional & LDI Photoimageable Soldermasks, Photoimageable Flex-Coverlay, Polymer Thick Film materials for surface and embedded applications, automotive dielectrics, 100% solids hole-plug & Peelable masks formulated for lead-free solder processes.

ELECTRA

www.electrapolymers.com

"Z Axis" Expansion Rates for Rigid/Flex Materials

Material Type	Tg (oC)	Z-Axis CTE (ppm/oC)	
Isola™ Epoxy FR406	170	60	
Isola™ Polyimide P95	260	55	
No Flow Prepreg GF	140	65	
No Flow Prepreg GI	200		55
Copper	>1000	17	
Bonding Adhesive	40	400	
Covercoat Adhesive	40		400
Flex Clad Adhesive	40		400
DuPont™ Kapton®	>200	100	
DuPont™ AP® Polyimide	>220	100	

Key Notes:

1) *Items highlighted in Red have excessive "Z axis expansion" rates*
2) *Tg is important because the CTE can dramatically increase when the material is exposed to temperatures above their Tg.*
3) *Values will vary depending upon manufacturer. Above values are for comparison only*

Property of DAS Flex Circuit Consultant, LLC

Figure 3.2

Z-axis Expansion Rates for Rigid-flex Materials?

Since Z-axis expansion plays a significant role in plated through reliability, it is important to understand the expansion rates of the materials available for rigid-flex construction as shown in Figure 3.2.

How Can We Minimize Z-axis Expansion in Rigid-flex PTHs?

From the Z-axis expansion chart, we can identify which materials offer best performance (have low CTE expansion rates) and which materials will perform poorly (have high CTE expansion rates) which would require special tooling techniques to keep their presence out of the plated through-hole zone. Here are some recommendations:

1. Use adhesiveless flexible metal-clad laminates for the flex layers (i.e., DuPont™ Pyralux AP® product line). No high CTE adhesive is used to bond the copper to the dielectric.
2. Use no-flow prepreg to glue all layers together (polyimide preferred).
3. Use Isola™ rigid metal-clad laminates for rigid layers (polyimide preferred).
4. Keep covercoat and any flexible bonding adhesives out of the PTHs. These are commonly referred to as adhesiveless or "bikini" construction.

What is Adhesiveless or "Bikini" Construction?

Although the terms sometimes are used interchangeably, technically they have different meanings:

Adhesiveless

Refers to the type of flexible metal-clad laminate material used. As mentioned in a previous column, this is typically an all-polyimide copper-clad dielectric. There are no 1980s-era acrylic or epoxy adhesives with high CTEs used for bonding the copper to the dielectric.

Bikini

Since there is no adhesiveless covercoat currently available today that is similar to the adhesiveless flexible metal-clad laminate mentioned above, special tooling methods must be employed to keep the covercoat adhesive out of the plated through holes while still providing a dielectric over the flexible sections. This can be accomplished by tooling out the material from the rigid section(s) or by applying patches of covercoat to the flexible sections only. Either method results in no flexible adhesives (with high CTE) in the rigid section where the plated through holes are located, while providing a flexible dielectric in the flexible sections. The covercoat material that was tooled out or intentionally omitted will be replaced with no-flow prepreg.

Combination of Both

A combination of adhesiveless flexible metal-clad laminate and bikini methods results in no undesirable high CTE flex materials present in the plated through holes. PTHs are now similar to a rigid board, with the exception of the flexible polyimide layer(s), which has very good CTE properties.

In my next column, Rigid-flex Manufacturing, Part 4, I will explain how to construct a rigid-flex with materials and techniques for yielding a robust plated through hole interconnect. **PCB**



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

Cosmotronic Earns NADCAP Accreditation

TC Cosmotronic, Inc. announced it has received NADCAP accreditation for the manufacturing of rigid, flex, rigid-flex, and high-density interconnect printed circuit boards.

"NADCAP accreditation is another validation of Cosmotronic's on-going commitment to our dfense and aerospace customers to provide products of the highest quality in a manufacturing environment that meets or exceeds world-class standards," according to Cosmotronic President John Wright.

TC Cosmotronic, Inc. received NADCAP accreditation for demonstrating their ongoing commitment to quality by satisfying customer requirements and industry specifications.

"Achieving NADCAP accreditation is not easy: it is one of the ways in which the aerospace industry identifies those who excel at manufacturing quality product through superior special processes. Companies such as Cosmotronic work hard to obtain this status and they should be justifiably proud of it," said Joe Pinto, Vice President and Chief Operating Officer at the Performance Review Institute. "PRI is proud to support continual improvement in the aerospace industry by helping companies such as Cosmotronic be successful and we look forward to continuing to assist the industry moving forward."

FEATURE

FLYING BY THE SEAT OF ITS PANTS:

A View of EDA from the File Format Standards Perspective

by Abby Monaco
INTERCEPT TECHNOLOGY, INC.

SUMMARY — Sometimes, specifications drive standards; other times, standards develop because something is adopted by a large majority. Regardless, the reality is that standards and specifications change over time. How does this apply to PCB design and manufacture, and what are the factors that drive those changes?

What made bell-bottoms cool? Why are most interior doors 80 inches in height? Sometimes, specifications are the reason something becomes a standard. Other times, something becomes a standard because it is adopted by a large majority. But as these two examples show, standards and specifications change over time.

Look around, and you'll see all shapes of kids mincing around in skinny jeans. Some new homes are being built with eight-foot doors now, not just because our children will tower over us when we're old, but also because the nine-foot ceiling is back in fashion. A new standard is budding and the 1970s are fading into memories and being left behind for the history books.

Just how does this apply to PCB design and manufacture? As many past committees and consortia have demonstrated, the PCB industry is locked in a staring contest between the engineer and manufacturer, and the No. 1 factor that makes both finally blink is cost. The cost can be man-hours, or it can be parts and machines. An engineer on the cutting edge dreams up all kinds of possibilities, but whether these can be built once or 1,000 times for the same cost is dictated by the manufacturer. However, even more than



Figure 1: Ca. 1975 Computervision CADDS 1 CAD/CAM System, with a NOVA 1200 computer, 32K memory, 14 MB disk, four digitizing/plotter tables, and a photoplotter.

the design itself, a major driving factor in the industry is the ability to share a PCB design in a format that is universally accepted.

Surprisingly, specified formats and universal acceptance follow opposing tracks. If one was to discuss standards and specifications pertaining to the PCB, CAD/CAM/CAE, or EDA industries, it is necessary to first take a look at how these industries were born, how they evolved, and where they are today.

> ***"As many past committees and consortia have demonstrated, the PCB industry is locked in a staring contest between the engineer and manufacturer, and the No. 1 factor that makes both finally blink is cost."***

The History

The only way to offer a well-painted view is to hearken back to 1957 when six long-forgotten producers of PWBs organized themselves as the Institute of Printed Circuits (IPC). These producers were frustrated by the inefficiencies and error-prone processes that made their PWBs expensive and difficult to sustain. Along with the formation of IPC, another major development was happening: Heinz Joseph Gerber, founder of Gerber Scientific Instrument Company (the PCB division is now Ucamco), was inventing his way into the standardization of manufacturing processes for the PCB industry.

In the 1960s and 1970s, Joe Gerber brought automation to the U.S. garment industry with his invention of the Gerber cutter S-70, a machine that cuts apparel quickly and effectively while using less cloth. From there, he broke into the printed circuit world with his introduction of the photoplotter. This invention made it possible to manufacture PCB designs with greater accuracy and fewer failures. But the photoplotter was run using phototools that were manually produced using stick-on tapes.

As the manufacturability of PCBs gained ground, it became clear that the most difficult part of the process was lack of accuracy due to human error. The first solution to this was a digitizer, a device consisting of a table-like surface with constrained cursor, whereby an operator could enter coordinate data from a large-scale PWB layout on Mylar film and have it turned into computer-readable form. At this point, hand drafting shifted into an electromechanical hybrid; computing languages and hardware were beginning to evolve to the point that computers became a viable alternative to manual drafting.

During this time, Calma developed the GDS software system, a mini-computer-based graphics system built around a digitizer. As Calma claimed the semiconductor and integrated circuit design markets, Computervision's CADDs systems began to succeed in the PCB industry. Companies such as Scientific Calculations (SciCards) and Telesis followed suit. The common factor that made these systems popular, however, was that the hardware and software were bundled together.

With their success, these hardware and software companies were burdened with sustaining both the hardware and software development under one roof. New companies were formed, founded on the concept of developing software separately from the hardware. Mentor Graphics, founded in 1980, embraced this very principle; they commissioned the then-upstart company, Apollo, to build their hardware while they focused solely on developing the CAD software. Valid Logic Systems, born from prior acquisitions, and Daisy were also founded in 1980. Shortly after, the electronic design automation (EDA) industry was bestowed with its official name. Throughout the 1980s, scores more software and hardware firms began to come online: Protel, OrCAD, Cadence Design Systems, and Intergaph were among some of the pivotal companies that emerged.

As these new companies were forging ground with computer-aided-design (CAD) software, the idea of computer-aided engineering (CAE) was also developing. The concept of schematic entry and simulation had been around since the 1960s, and its origins are largely credited to SPICE, but widespread adoption of both CAD and CAE really took hold in the 1980s, when they

finally reached a level of affordability and productivity that worked for a larger spectrum of consumers.

As PCB designs became more automated, so did the manufacturing processes. The concept of computer-aided-manufacturing (CAM) budded in the 1970s, but it didn't take off until machines and data input formats reached a level of sophistication that didn't require much operator skill or time (i.e., cost). As manufacturers struggled with the inaccuracies of Gerber and GDS, efforts to establish improved formats such as EDIF and IPC-D-356 surfaced.

In the 1980s and 1990s, a great number of new output formats developed. In the midst of this activity, Valor Computerized Systems was founded in 1992 on the concept of CAM and design for manufacture (DFM). The CAM methods were and still are a challenge, so Valor attempted to sidestep the problem with its DFM system that verified the output format before it was built. It is fair to say that EDA came into full swing in the late 1980s and 1990s. The business activity alone was mind-boggling, with new companies popping up and then being consumed by larger companies in an instant.

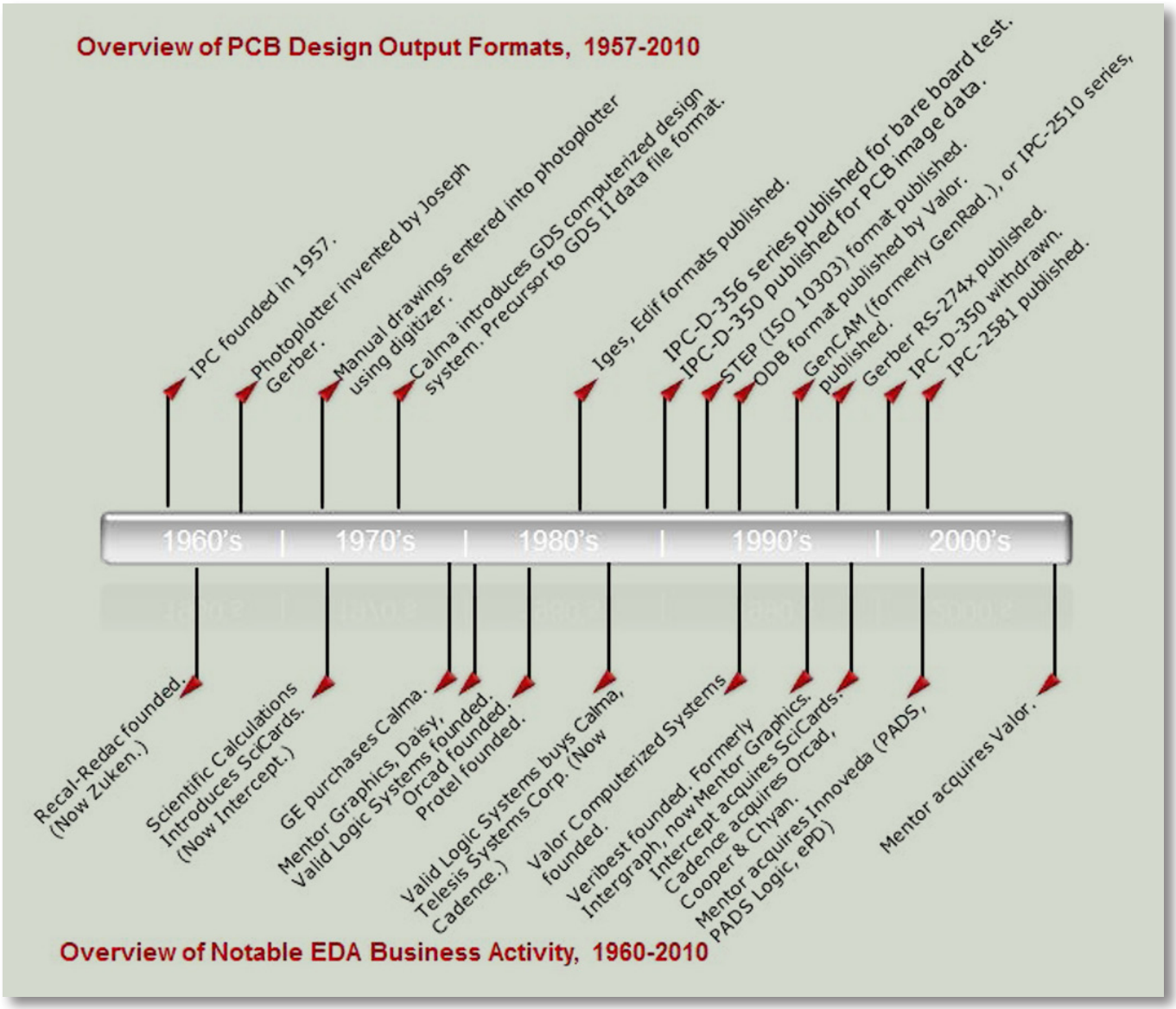
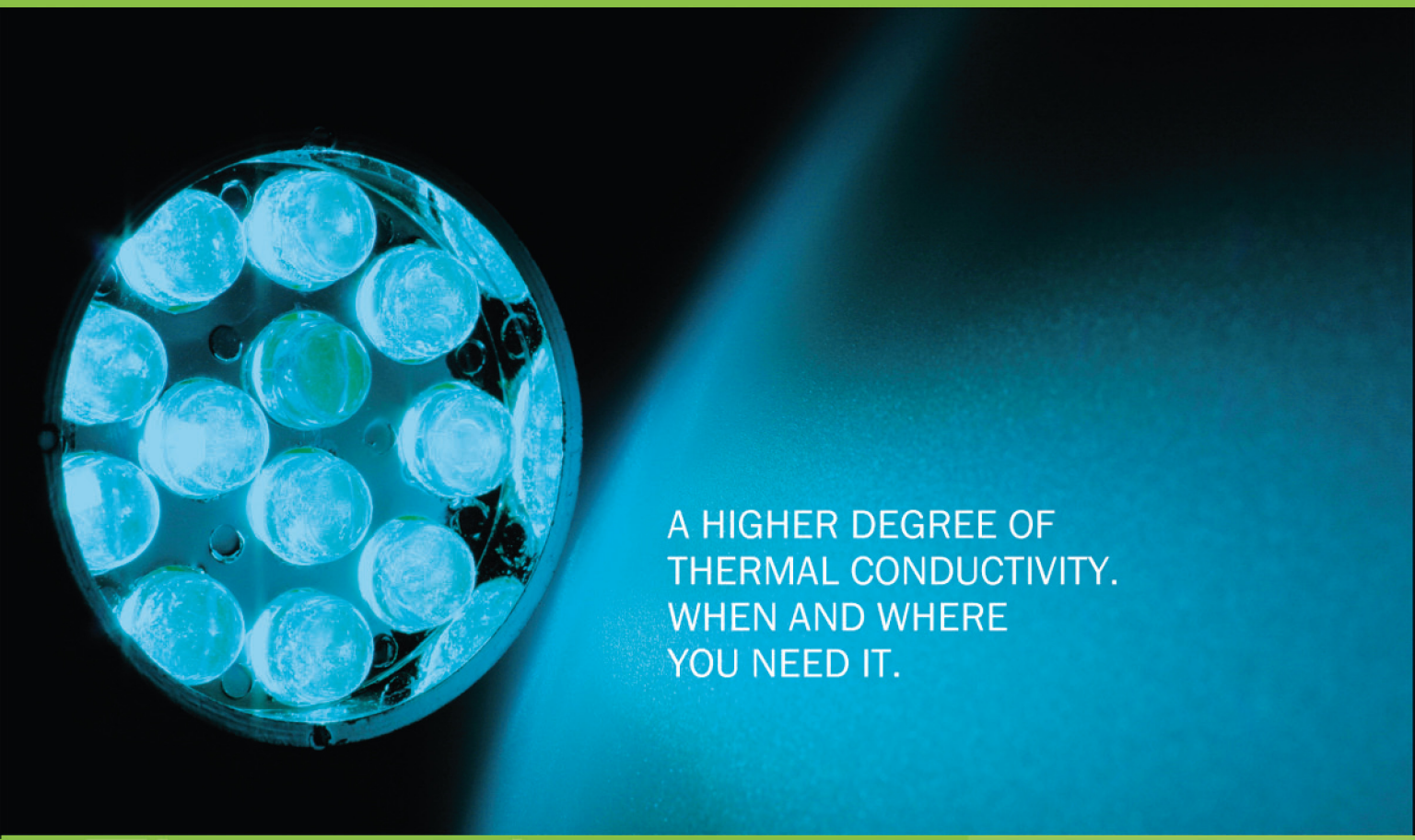


Figure 2: Timeline of PCB design output formats as they coincided with events in the EDA business market, 1957-2010.



Thermal Management

VT-4A1 / VT-4A2 / VT-4A3 THERMAL MANAGEMENT

Ventec offers the highest quality aluminum base laminates and prepregs in three types.

- VT-4A1: 1.6W/m.K, Ceramic Filled, Halogen Free
- VT-4A2: 2.2W/m.K, Ceramic Filled, Halogen Free
- VT-4A3: 3.0W/m.K, Ceramic Filled, Halogen Free
- VT-4B3: 3.0W/m.K, Non-glass reinforced, Fully Formable
- Excellent Electrical and Mechanical Characteristics
- Flame Retardant (UL94 V0)
- UL: E214381

Our global resources and local distribution network put the products you need in your hands at the right time and at the right cost.



Wherever technology takes you, we deliver.

www.ventec-usa.com
www.ventec-europe.com
www.venteclaminates.com

ROHS COMPLIANT & LEAD FREE MATERIALS

Lead Free

VT-481	*150. Tg Phenolic Cured*
VT-47	*180. Tg Phenolic Cured*

Halogen Free & Lead Free

VT-441	*150. Tg Phenolic Cured & Halogen Free*
VT-447	*175. Tg Phenolic Cured & Halogen Free*

Traditional FR-4

VT-42	*140. Tg Dicy Cured, Unfilled*
VT-42S	*150. Tg Dicy Cured, Unfilled*

Low D.K. & Lead Free

VT-461	*200. Tg Phenolic Cured & Dk 3.7, TD>350 ˚C*

Special Applications

VT-4A1/ VT-4A2/ VT-4A3	*Thermal Management Prepreg and Aluminum Base Laminates*
VT-901	*250. Tg Polyimide Resin System UL94 V0*
VT-90H	*250. Tg Polyimide Resin System UL94 HB*
VT-42C	*130. Tg CTI 250, 400 & 600 Volts*
VT-42F	*125. Tg Dicy Cured, Semi Flex*

Rigid Flex/NO Flow Prepreg

VT-45NF	*170. Tg Dicy Cured*
VT-47NF	*170. Tg Phenolic Cured High Td*
VT-447NF	*170.Tg Halogen Free & Lead Free*
VT-901NF	*200. Tg Polyimide Resin System*

Hole Fill Prepreg

VT-901PP HW	*250. Tg Polyimide Resin System*
VT-47PP VF	*180. Tg Phenolic Cured*

VENTEC

Among the most memorable M&A stories: VeriBest created a firestorm, and was swallowed up by Mentor; Intercept was founded in 1990 and acquired SciCards in 1998; Cadence was founded in 1991 through its purchase of Valid, then went on to acquire Cooper & Chyan Technology, OrCAD, and others in the late 1990s; Mentor acquired Innoveda and much later, Valor Computerized Systems. One of the smaller companies leftover, P-CAD, was swallowed up by Protel, which was renamed Altium in 2001.

Throughout 2000-2010, the PCB design software industry went from more than 20 vendors to just a few: Cadence, Mentor, Zuken, Intercept, and Altium were the major players left standing. All but Altium were expanding their PCB design software into a full design environment solution, where the CAD/CAM/CAE applications were now sold together as one package. But as these companies were struggling to get their acquisitions talking smoothly, Altium swooped in to offer a cost-effective "shrink-wrapped," total environment solution to the larger portion of the market that couldn't afford or didn't need an advanced design system.

The last few major events leading to the present were Mentor's acquisition of Valor in 2010, followed by Carl Icahn's purchase of even more shares of Mentor. In 2011, all three of Icahn's board nominees won seats on the board. What this means for the future remains to be seen.

The Power of the File Format Standard

As the PCB industry evolved into ever-larger circles to encompass CAD, CAM, and CAE under the umbrella term of EDA, a powerful undercurrent of this activity was database formats. With the dawn of the computer age, the concept of a transferable digital database format emerged. The Gerber and CADDs systems used proprietary formats developed to run the exact hardware they sold with the software; however, as hardware and software broke into independent platforms, it was clear that data had to be transferable. Gerber became known more as a file format rather than a

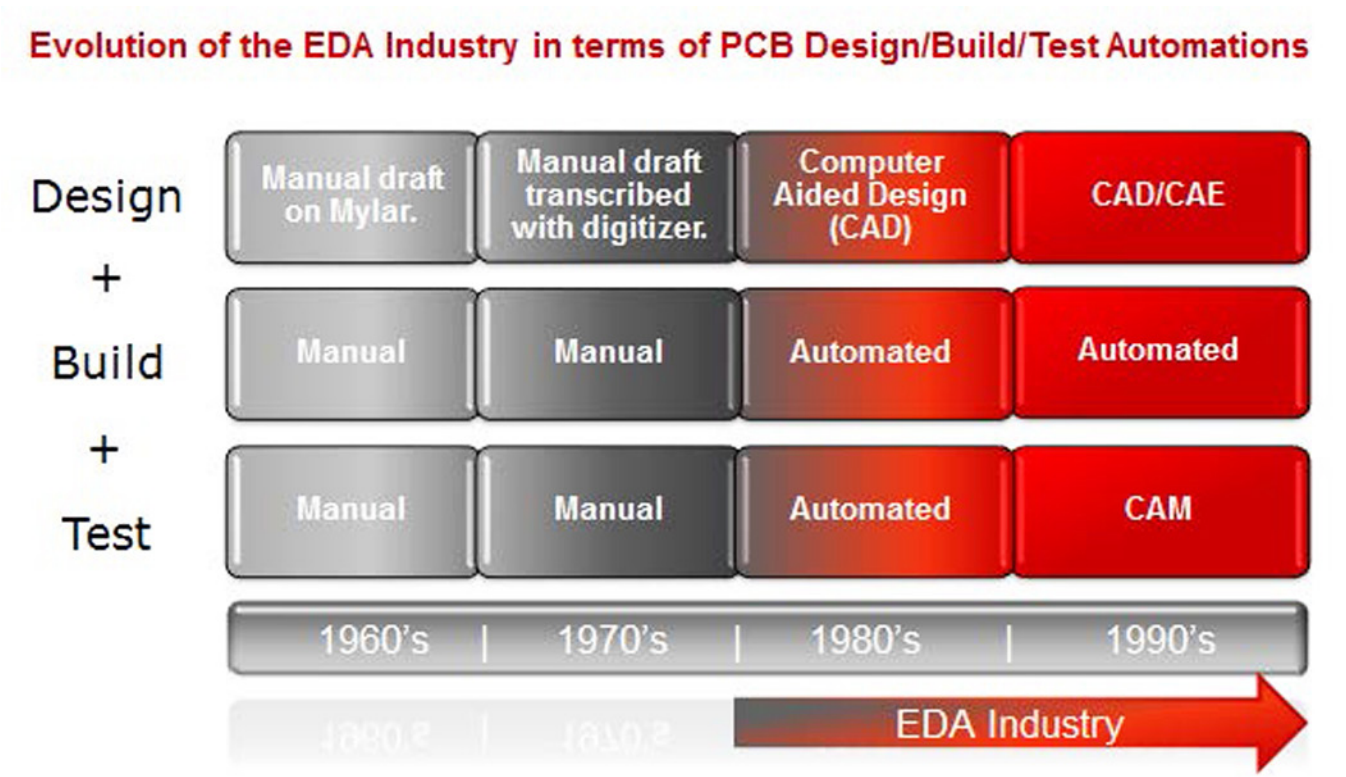


Figure 3: Evolution of the EDA industry in terms of the realization of the CAD, CAM, and CAE sectors.

photoplotting system. Furthermore, as more companies entered the EDA sphere, it became increasingly important to have vendor neutral formats.

Computervision Cadds, Calma GDS (later GDS II), Applicon, Prisma, Daisy, SciCards, Unicad, and Telesis are but a few names that became synonymous with a file format, but none of them broke into the manufacturing side; manufacturers continued to require Gerber output from the CAD systems. This was a problem because Gerber was not an intelligent format; it was originally meant to transcribe imaging data. Manufacturers had to go through all kinds of conniptions to reverse-engineer Gerber back into something intelligent that they could test with their CAM systems.

Efforts cropped up to develop more accurate, vendor-neutral industry formats. EDIF (Electronic Design Interchange Format) was introduced around 1985 and became a moderately used standard. In 1990, IPC made another attempt to improve upon the Gerber format by publishing its IPC-D-350 series, which was intended to replace Gerber. Prior revisions of the IPC-350 series had not gained much adoption. They also published the IPC-D-356 series, which was pivotal for the automation of bare board test. But the IPC-D-350 series was eventually withdrawn after Gerber formally published its improved RS-274X format in 1998. Though it is common knowledge throughout the industry that Gerber is an inferior format, it remains the de facto standard.

As IPC was working to standardize formats, Valor revolutionized the CAM industry with the introduction of its powerful DFM checking system. This system used the ODB (eventually ODB++) format, which gained prevalence among manufacturers mostly through Valor's success automating the DFM process. IPC continued its work to standardize a vendor-neutral format for CAM by publishing its IPC-2510 series, more commonly known as GenCAM. Intercept Technology was the only certified EDA vendor on GenCAM. Meanwhile, use of the ODB++ format continued to grow.

The last major format proposal for the PCB industry took place in 2004, with IPC-2581. The IPC-2581 format is intended to provide a vendor-neutral, XML format database that contains even greater intelligence than its prior attempts, the IPC-D-350 and GenCAM. The format incorporates features of the ODB++ format now owned by Mentor Graphics Corp. Since its proposal in 2004, there has been no move by the industry to adopt it until recently. In late 2011, Cadence worked to form the IPC-2581 Consortium whereby software and manufacturing vendors would agree to adopt and support the IPC-2581 format.

"IPC continued its work to standardize a vendor-neutral format for CAM by publishing its IPC-2510 series, more commonly known as GenCAM. Intercept Technology was the only certified EDA vendor on GenCAM. Meanwhile, use of the ODB++ format continued to grow."

Conclusion

Time and again, the PCB industry has adopted a format developed by a vendor seeking a profit. These vendors have a lot at stake if they can't distribute their format widely among both designers and manufacturers, so it stands to reason that they are the most motivated to put resources into developing improved formats. As a proprietary format becomes widely accepted, organizations such as IPC step in and attempt to standardize it in a vendor-neutral IPC format.

It is an honorable pursuit, and one that is crucial to industries such as architecture, construction, and manufacturing. But time and again, IPC is superseded by the pace of the EDA business market; while the published standards may be preferable in their correctness and usefulness, the overhead required by software and manufacturing businesses to develop support for them can't be justified because there is a vendor-proprietary format already entrenched.

Sometimes IPC is able to prevail; its IPC-D-356 series for bare-board test, for example, is

still widely used. But as it tried to solve Gerber data inefficiencies with its IPC-D-350 series, the ODB++ database and Gerber's subsequent publishing of RS-274X trumped those efforts. It is difficult to say whether the IPC-2581 will have a higher adoption rate than its predecessors.

While standards and specifications are necessary in any trade, it goes without saying that adoption of these standards is not guaranteed. We are all flawed, and thus our creations are flawed. The human being seeks perfection because perfect is something we are not. There is no perfect format. But a format's ability to help us get our jobs done faster, more efficiently, and for a lower cost will forever trump its lack of perfection. It is humbling to conclude that it all comes down not to what is right, but what is cost-effective from an industry perspective.

For a complete list of references, please click here. **PCB**

EDA: Where Electronics Begins. Clive "Max" Maxfield and Kuhoo Goyal Edson. Copyright 2001, techBITES INTERactive, Inc. ISBN 0-9714063-0-8. Photo reprinted with permission. Copyright Bob Sherman http://bobshermanspage.com/WorKTrips.htm



Abby Monaco, CID, is a product manager for Intercept Technology Inc. Prior, Monaco has worked for Intercept as a software translation expert, migrating and reverse engineering boards from third-party software into Pantheon PCB/hybrid/RF layout software. She has held positions in technical documentation, software quality assurance and software support. Monaco's column, Software Bytes, appears bi-monthly in PCBDesign007.

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.

Prochaska joined Polar in April 2008, with responsibility for supplying Polar's range of signal integrity software to customers throughout North America.

"This new appointment recognises the passion with which Prochaska has delivered value to our growing base of customers through the Polarcare programme of maintenance, service and technical support for licensed software users. In her new role, as North American Sales Manager, Prochaska will extend her responsibilities to ensure that our sales and service activities match the value and comprehensive support that customers expect from Polar," comments Ken Taylor, President Polar Instruments Inc.

Taylor continues, "Prochaska brings extensive expertise in delivering customer support for business-to-business (B2B) software, which she has perfected by working in the fast-moving consumer goods (FMCG) and financial sectors. She has a thorough understanding of the mechanisms which are essential to provide customers with the highest levels of communication, information and support, which allow them to gain maximum value from their investment in Polar signal integrity tools."



Polar Instruments is a market leader in designing and manufacturing tools to simplify and enhance the design, fabrication and testing of PCBs. Their innovative tools include the industry-standard Controlled Impedance Test System (CITS) which provides the global PCB industry with an easy-to-use test system for high-speed digital and RF boards, as well as class-leading tools for fast and accurate design and testing of controlled impedance in PCBs. Polar also leads the industry in tools for automated PCB layer stackup design and documentation. Polar Instruments was established in 1976 and now has operations in the U.S., UK, Europe, and Asia Pacific. For more information, visit www.polarinstruments.com.

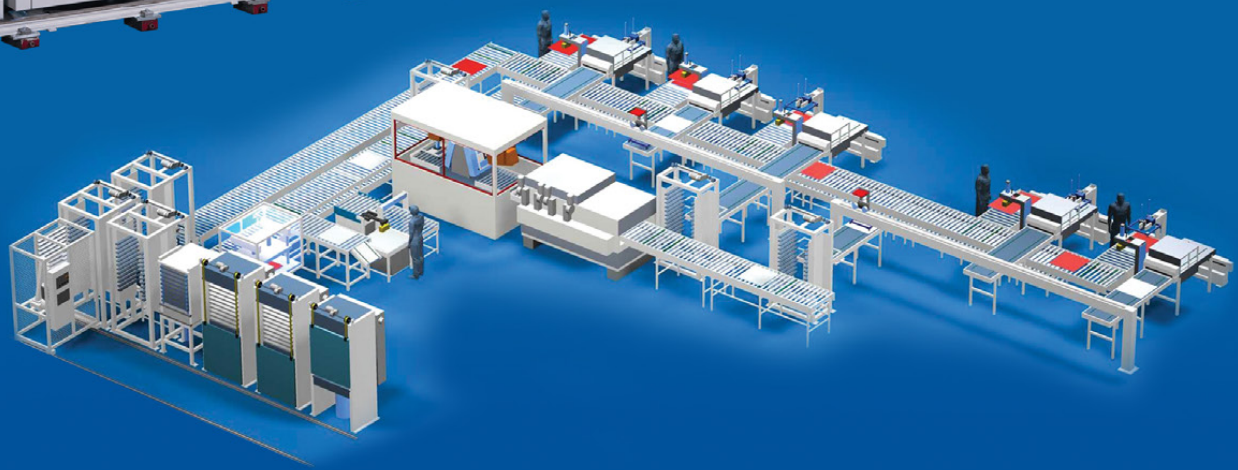
WorkCell² Vacuum Lamination System

... A Modular Design That GROWS With You

With the ability to expand as production needs dictate, Bürkle's modular WorkCell² provides a precision Vacuum Lamination System for every phase of business – from start-ups to high-volume production houses – we have the right system for you.







The Bürkle WorkCell² is part of Bürkle's wide range of laminating and coating systems. Bürkle is a member of the Bürkle North America alliance with Schmoll-Maschinen, Bürkle GmbH, IMPEX, LHMT and Bacher Systems. For more information contact Bürkle North America via e-mail: sales@burkleusa.com.

Alliance partner web sites:

- www.schmoll-maschinen.de
- www.buerkle-gmbh.de
- www.bacher-systems.com
- www.impex.co.at
- www.lhmt.de

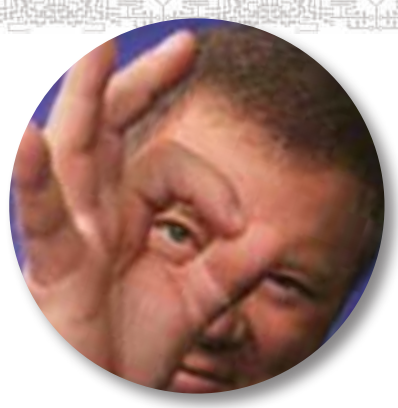
BÜRKLE NORTH AMERICA, INC.

WWW.BURKLEUSA.COM

Bürkle North America, Inc.
12802 Valley View St., Ste. 12/13
Garden Grove, CA 92845
TEL: 714-379-5090
FAX: 714-379-5092

POINT OF VIEW

How William Shatner Changed the (PCB) World



by Steve Williams

SUMMARY — With William Shatner headlining as the keynote speaker at IPC APEX EXPO 2012, I thought it would be appropriate to delve into how this man has impacted our technology, and directly, the PCB industry, with his role as Captain James Tiberius Kirk of the Starship Enterprise.

Denny Crane!

I have to admit, I'm a fan. I have been most of my life (with the possible exception of those T.J. Hooker years). I rediscovered my enjoyment of William Shatner when he reinvented himself as Denny Crane, the always amusing, frequently offensive, and often brilliant senior lawyer on the TV series Boston Legal, which ended a few years ago. One of his endearing qualities was his willingness to turn himself into a caricature of the overbearing, overacting persona that has provided ample fodder for comedians and impressionists throughout his career. We can all learn a lesson from him: Don't take ourselves too seriously!



Beam Me Up, Scotty

I still remember spending my Saturday afternoons watching Star Trek with my dad and brother, mesmerized by all the fantastical science fiction gadgets and technology that was used in the show. This was the late 1960s (to this day I can't believe the original Star Trek only ran for three seasons), and I never imagined any of this technology would be realized in my lifetime. As the series was set some 300 years in the future, this was an understandable assumption. However, in a short 45 years, it is amazing to see just how much of today's PCB-driven technology was influenced by the gadgets that outfitted the crew of the Starship Enterprise. Let us explore a few of these…

Warp Drive

The Starship Enterprise was equipped with warp drive speed, which is a faster-than-light (FTL) propulsion system that allows travel at velocities greater than that of light, by many orders of magnitude. Any discussion of real-world application of this technology must begin with Dr. Marc D. Rayman, the chief propulsion engineer at NASA's Jet Propulsion Laboratory. Dr. Rayman has stated that he became interested in propulsion through Star Trek, and has revealed how NASA's deep space probe's ion propulsion system was inspired by a Star Trek episode. William Shatner once claimed that "those pesky trekkies are everywhere in the space program," which can be validated by merely looking at the name of the first space shuttle: Enterprise.

Communicator

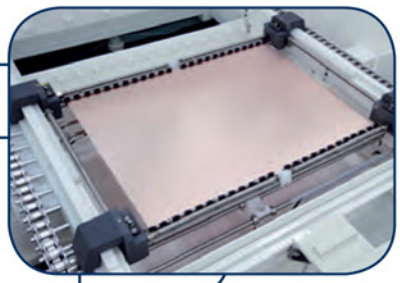
Communicators were handheld flip-open devices used for voice communication between individuals via the ship's communication system. The top section held a transceiver antenna and the bottom contained user controls, a speaker and a microphone.

The cell phone was invented by Martin Cooper, the chief engineer at Motorola. Mr.





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 µm

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

www.schmid-group.com

Cooper states that the communicator was his inspiration for the cell phone, and that Star Trek also introduced the first concept of voice recognition dialing.

When the UI (user interface) for the Palm operating system was first developed, product design architect Rob Haitani stated that his first sketches were highly influenced by the UI of the Enterprise bridge panels. The later model, Treo, which was a combination phone/PDA, had a similar form factor to the original communicators that featured a speakerphone so you could "stand there and talk into it like Captain Kirk."

Tricorder



The tricorder was a multifunction handheld device used for sensor scanning, data analysis, and recording data. The "tri" referred to the device's three scanning functions: geological, meteorological, and biological. This technology now exists via software for tricorder functionality in today's handheld smartphone devices. Modern GPS technology can be compared to the tricorder's functionality as well. Specialty technology companies have developed applications for electromagnetic field, light meter, and mass spectrometer sensing, and Sandia National Laboratories has developed lab-on-a-chip handheld technology for biological or chemical analysis. Medical non-invasive image scanners, robotic laser surgery, and handheld patient diagnostic equipment were all inspired by the Star Trek tricorder technology used by Dr. "Bones" McCoy.

Hypospray

"Bones" also used a device called the hypospray to administer injections of medication without the use of needles. An interesting note is that the concept for the hypospray was developed when producers of the original Star Trek series discovered that hypodermic syringes were not allowed due to broadcast restrictions. The modern counterpart to this technology is the jet injector, which delivers a high pressure stream of liquid through the skin, commonly used for vaccinations.

Computer

Computer was the user-friendly, interactive computer software system LCARS (library computer access/retrieval system) on the Enterprise that used a graphical user interface before we could ever have imagined what a GUI was. The first home computer, the Altair 8800, was invented by fellow Trekkie Ed Roberts, and named after the solar system Altair (Altair 6) from a Star Trek episode. This led to Bill Gates writing the computer programming language BASIC for the Altair computer and later forming Microsoft. This build-it-yourself personal computer was pretty basic, but it would inspire a couple of guys named Steve (Jobs and Wozniak) to improve upon the model and usher in the age of the personal computer, with Apple Computer taking the user-friendly concept to an unbelievably high level.

In later generations of the Star Trek franchise, a device was introduced called the PADD (personal access display device), which was basically a handheld computer. Apple's iPad and other tablets have been compared to the PADD from a functionality perspective. The average smartphone of today has 1000x the computing power of early supercomputers whose electronics filled up entire wings of buildings. Apple's latest iPhone 4S comes with Siri, the artificial intelligence/speech recognition software that is right out of Star Trek's Computer and PADD technology, allowing us to interact with the device using natural language.



VISOR

Later generations of the series also featured a wearable device called the VISOR that allowed the blind character Geordi La Forge to "see" by directly sending scanned information to his brain. The underlying functional mechanics of the VISOR has been developed in the real world through a combination of microchip implants behind the retina and goggles equipped with LED readouts.

Life Imitates Art

There seems to be an endless list of products that trace their origins back to Star Trek, including flat-panel TVs, Tasers, Bluetooth headsets, touchscreens, language translators, and bionic implants. To say that William Shatner is the face of technology might be a galactic stretch, but the influence of his iconic series cannot be overlooked. As we think of innovation and technology in our companies, do not be afraid to "boldly go where no man has gone before." I will end this column with an excerpt from a contentious dialogue between Captain Kirk and Mr. Spock:

Capt. Kirk: All right, you mutinous, disloyal, computerized half-breed. We'll see about you deserting my ship.

Spock: The term "half-breed" is somewhat applicable, but "computerized" is inaccurate. A machine can be computerized, not a man.

Capt. Kirk: What makes you think you're a man? You're an overgrown jackrabbit. An elf with a hyperactive thyroid.

Spock: Jim, I don't understand...

Capt. Kirk: Of course you don't understand. You don't have the brains to understand. All you have is printed circuits. **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).

Mentor Graphics Releases New PADS Suite

Mentor Graphics Corporation has released the latest version of the PADS product suite, the world's most widely used desktop PCB design solution. These additional features in the release add significant value to electronics companies as they strive to deliver the most competitive, high-performance products to market faster.

Key new features 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 assuring that tight, high-speed performance specifications are met. Another feature allows a trace to "hug" an existing obstacle like another trace or board outline. This feature will reduce routing time and more effectively use board space. Drafting operations have also been enhanced with new 2D line styles and snap capabilities.

"The new associated nets functionality in PADS 9.4 reduces design time," stated Johnnie Howman, director at Icarus Design. "I can now easily define the high-speed associated nets, assign constraints, and be assured that they will be routed to those rules."

The PADS 9.4 software is a significantly enhanced, next-generation PADS release which adds unique DFM analysis functionality to the PCB design space, giving the designer the ability to perform over 100 fabrication and assembly checks in PADS Layout. This analysis is based on the Valor® technology, the industry leader in fabrication and assembly DFM analysis. Complimenting ODB++ intelligent data transfer from design into manufacturing, the new ODB++ viewer provides the ability to overlay and compare traditional CAM documents, such as Gerber files for PCB design-through-manufacturing data integrity. Detecting and correcting manufacturing issues early in the design process helps decrease overall time to market by reducing design/manufacturing re-spins. With the scalable PADS 9.4 flow, engineers and designers can cost-effectively design the industry's most complex, highest performance, and densest PCBs.



Sunny San Diego a Good Move for APEX

by Andy Shaughnessy
I-CONNECT007

SUMMARY — Andy reveals, "The show had a good vibe, and I didn't see anyone looking for work, résumé in hand. I talked to everyone I could, and everyone was upbeat about their companies and the prospects for the electronics industry overall."

When IPC announced that APEX EXPO was moving from Mandalay Bay in Las Vegas to the San Diego Convention Center, many people were worried that attendance might drop. I don't know what the final numbers are for this year's show, but I doubt there was any decrease in traffic. I saw packed aisles on the exhibition floor for the first two days of the show, and the technical conference classes and the committee meetings drew decent crowds, too.

The show had a good vibe, and I didn't see anyone looking for work, résumé in hand. I talked to everyone I could, and everyone was upbeat about their companies and the prospects for the electronics industry overall.



The show occupied two floors at the Convention Center; all of the heaviest assembly equipment vendors were located downstairs, while the consumables and makers of smaller machines were upstairs in the Sail Room, so-called because a massive sail that is draped over the glass roof. The result is a show floor that's often flooded with California sunshine.

The spacious Real Time with…IPC video studio was one of the most happening places to be. We had about two dozen staffers, editors and guest editors on hand, who mainly subsisted on Subway sandwiches and Snickers bars. All told, I-Connect007 recorded almost 200 interviews with some of the industry's most knowledgeable process engineers, scientists, professors, analysts, PCB designers, design engineers, and managers.

Meanwhile, anyone and everyone stopped by to watch us shoot these interviews. You know you're doing something right when you can't find a seat in your own booth!

Much of my time was spent wrangling our guest editors. My colleague, Jo Ann Sotelo, organized and maintained the interview schedule. Her job was to get the editors and their interviewees into the studio, usually one interview every 15 minutes. As soon as the interview was over, I'd herd the editors over to a laptop to write short summaries of their chat. This process kept us from having to watch 200 interviews ourselves, which would take forever and get in the way of our goal of having the each day's interviews live by the end of that day.

In between my editor-wrangling duties, I conducted about 20 interviews. Mike Creeden

of the design bureau at San Diego PCB raised a serious question: Where is the next generation of young PCB designers, and how do we ensure that they're being adequately trained and educated?

We filmed a great demo with Blake Stevens of HZO, a company that makes a waterproof coating called WaterBlock. Blake cued up a Taylor Swift video on his WaterBlock-coated iPhone and dropped it into a bowl of water. The video played the entire time. You know that everyone from Apple to the DoD is interested in this coating, which was the hit of the CES show, too.

But, all work and no play is no way to operate. Before the show started, Guest Editor Kelly Dack and his wife, and my girlfriend and I took a 37-foot sailboat for a half-day cruise, and managed to keep from sinking. Next year, to paraphrase Robert Shaw in Jaws, we're going to need a bigger boat. We'll also start planning the trip in the fall and leave from Mission Bay, so we don't spend an hour getting out into the Pacific.

And during the Wednesday night reception on the show floor, the Real Time Players band entertained the crowd. If you've never seen the Harpalicious Twins, Kelly Dack and Bürkle President Dick Crowe trade harmonica solos, you don't know what you're missing.

We couldn't have done any of this without a little help from our friends. Our old buddy Steve Gold used a week of vacation from his "day job" to help us out; he's either really devoted to PCBs or just crazy. (During an interview, Steve asked a group of engineers what had changed in the industry during his two-year absence, and they replied, "Nothing.")

Steve was our go-to guy: He took care of anything and everything we asked of him, and he gave me a great deal on his cherry red Gibson SG guitar.

I must hand it to IPC for pulling this off. You'd never know that the organization is in the midst of a search for a new president. Kim Sterling, Anna Garrido, Alicia Balonek, Susan Filz, Anne Marie Mulvihill, Tony Hilvers and the rest of the IPC crew deserve a big high-five. They get blamed when things go wrong, but they don't get many "attaboys" when all goes well.

I hope to see you next year in San Diego. Let me know if you want to go sailing!



Andy Shaughnessy is the editor of PCBDesign007, the top online source of information for everyone whose livelihood depends upon PCB design. Andy has been a journalist for 15 years, including tenure as editor-in-chief of Printed Circuit Design & Manufacture magazine. He is also a past conference chair for the PCB Design Conference series. Andy began covering PCB design and EDA in 1999. His articles have appeared in EE Times, PC Fab and Circuits Assembly. To contact Andy, click here.

Let the Sun Shine In!

by Dan Beaulieu
D.B. MANAGEMENT GROUP

SUMMARY — People were buying, the mood was positive and the sun was shining — through the windows of the San Diego Convention Center, and more importantly, on the PCB industry.

Finally, the sun shone on the PCB industry and we were ready for it. It was San Diego, it was bright and sunny the top floor of the San Diego Convention Center, thanks to windows that gave us plenty of light, contributing the positive mood of the week. People and companies were upbeat, and optimism was in the air.

Just about everyone I talked to said it was a great show for them, made even better by the fact that people were buying. I talked to a number of people who said that people who visited their booths were serious about doing business. The same went for those folks that I interviewed, who talked about new products, new services and new technologies, all focused on making the PCB industry in North America a more relevant player in the global market. There was a great deal of talk about innovation

Let the Sun Shine IN! *continues*

and how it will help raise our North American market to a level where we represent the cutting, and even bleeding edge in PCB technology.

If there was a common theme, it was this: We must do everything we can to not only save, but also actively promote our board shops here in North America. For the first time in many years, I sensed a feeling of respect for our American shops as well as a sentiment of protection for them, not only today, but for the future as well.

PCB shop owners I spoke to talked of concern for the sustenance of a viable vendor base. Vendors I talked to, the good ones anyway, understood that. They feel a sense of duty and obligation to support our American companies. Without exception, every single person I talked to was not only concerned about the future of the American PCB industry, but told of plans they were making on how to support our industry in Canada and even Europe as well.

This IPC EXPO should be considered a success, at least in terms of quality (I have not seen the attendance numbers yet so maybe in terms of quantity as well). Best of all, we're back here to sunny San Diego again next year. It's a good thing.



Dan Beaulieu is a well-known industry writer and columnist who has written sales and marketing columns for Circuitree, Printed Circuit Design & Manufacture, U.S. Tech, and Printed Circuit News. He is also a feature article author who writes about various board shops and how they perform in the industry. Dan has more than 30 years experience in the PCB industry. He is considered one of the industry's top marketing and sales experts, as well as a leading strategist in the industry.

New Era for IPC, Educational Opportunities

by Mike Carano
OMG ELECTRONIC CHEMICALS

SUMMARY — Subject matter offered in the workshops included reliability concerns with PCBs and assemblies, optimization of surface mount and through-hole assembly operations, Lean manufacturing, and lead-free solderable finishes, just to name a few. There was no lack of learning opportunities throughout the week.

This year's IPC APEX EXPO should receive few or no complaints. The move in venue to San Diego, California from four previous years in Las Vegas, Nevada was certainly a welcome change.

What I can tell you from my own unscientific sampling of the exhibitors and attendees is that overall trade show traffic was greater than last year. IPC staff stated that attendee registrations were running a bit more than 22% above the 2011 show. IPC staff also related that there were just over 1,700 attendees for the various professional development courses presented during the week — an 8% increase over last year! The numbers are due, in part, to the quality of the instructors and content presented. Consequently, no courses were canceled due to a lack of attendees. Subject matter offered in the workshops included reliability concerns with PCBs and assemblies, optimization of surface mount and through-hole assembly operations, Lean manufacturing, and lead-free solderable finishes, just to name a few. There was no lack of learning opportunities throughout the week.

Behind the scenes, many committee meetings did great work to advance IPC standards and other key value-added activities for IPC membership. This included the kick-off of the 2013 IPC Technology Roadmap work. In addition, the IPC Government Relations Subcommittee provided an update on the various initiatives affecting IPC member companies to a standing-room-only crowd.

Again, there was no lack of opportunity for those interested in getting involved in IPC and the electronics industry.

I had the opportunity to speak with four very large equipment suppliers during the three-day event. Each told me booth traffic was greater than in previous years. Yes, there are signs of an improving economy; however, to see this much interest so early in the calendar year is not typical.

In addition, a new era begins for IPC. Membership continues to grow, more new standards are published each quarter, and the IPC reach is far and wide. Steve Pudles has now ascended into the chairmanship of the IPC Board of Directors and has pledged to continue IPC's standing in the world of electronics. He is working hard to fill the position of resident of the IPC to replace Denny McGuirk who left in November of 2011 to join SEMI.

From the fabrication side, however, I was disappointed to see so few North American and European bare board fabricators in attendance. Of course, a few large U.S.-based firms sent several folks, but that was about all that I saw outside a few niche fabricators. This is in sharp contrast to the Asian trade shows where at least 100 fabricators use booth space to promote capabilities. Perhaps that will change someday and we will see more North American PCB fabricators take up space on the IPC APEX EXPO show floor!



Michael Carano is with OMG Electronic Chemicals, a developer and provider of processes and materials for the electronics industry supply chain and has been in the PWB, general metal finishing photovoltaic industries for almost 30 years. Carano holds nine U.S. patents in topics including plating, metallization processes, and PWB fabrication techniques. To contact Michael, click here.

Market Outlook

by Real Time with...IPC APEX EXPO 2012





Industry market guru Walt Custer offers his insight into the markets for electronic products. Turmoil around the globe has some markets in decline or flat while others, driven by certain segments, are growing.



realtimewith.com

VIDEO INTERVIEW

Reflections on IPC APEX EXPO 2012

by Dan Feinberg
FEIN-LINE ASSOCIATES, INC., P. KAY METAL, INC.

SUMMARY — As I think back to the first show in Boston, Massachusetts, founded under the newly-formed Suppliers Council, which I was honored to have chaired, I cannot help but consider the ups and downs and changes our industry has seen in the past two decades.

I am just back from this year's IPC APEX EXPO show. This year we were back in San Diego for the first time in many years, which gave me reason to reflect.

As I think back to the first show in Boston, Massachusetts founded under the newly-formed Suppliers Council, which I was honored to have chaired, I cannot help but consider the ups and downs and changes our industry has seen in the past two decades. Back then, suppliers had very little say on the direction of IPC; we were only associate members after all, but that was changing. The Board of Directors, which included no suppliers, took the advice of a forward-looking Tom Damerich and Tony Hilvers, and asked me, in my position as President of a major supplier, to form the Suppliers Council. The first thing I did was schedule a brainstorming session to choose the most important actions that we, as suppliers, could do under the auspices of the IPC that would help strengthen the industry and make progress.

Attending the brainstorming session were the presidents of most of the major suppliers to the industry at that time: DuPont, Shipley, Rohm and Haas, MacDermid, Lea Ronal, key laminate suppliers, Excellon, and of course my company, Morton Electronic Materials (Dynachem) and others. After a full day of brainstorming and narrowing down a relatively long list of ideas, we produced a short list with the IPC sponsoring its own trade show at the top of that list—a show run by and for the industry, where costs would be reasonable and money would assist the industry rather than enrich a trade show company. We then had to convince the IPC Board to support this. Remember, up to that point the IPC was primarily a standards and educational organization, something it still does very well today. As such, it was virtually 100% focused on circuit board fabrication.

Let's just say that the Board needed a modest amount of convincing, but the first IPC EXPO was born. In the following years, we had shows in Boston, San Jose, San Diego, Anaheim, Las Vegas and now back to San Diego. As things progressed, the IPC started a sister show, one for assemblers and the general EMS industry, a segment in which the IPC had become very involved. Eventually, it made sense to combine the two shows into today's format of IPC APEX EXPO.

Okay, so how did 2012 stack up? Well, the show is not as large as the events in the 90s, but then again the manufacturing segment here in North America is smaller. If sheer numbers of attendees and booths is your measurement criteria, then this show is not as impressive as its ancestors are. However, if you compare it to the last few years, I would say that this one was quite good. This year's show included a number of new product introductions and the traffic at exhibitors' booths was modest, but steady. I did not see any dead times; even on the last day of the show there were interested prospects visiting booths and asking for information. The visitors were responsible people who seemed to have authority and sounded like they were ready to make investments. Each exhibitor will have to answer this for himself or herself, but those who I spoke and worked with seemed to feel that this was a very good show. A "10" on the scale? No, but I would give it a good solid "7+" — well worth exhibiting and attending.



Dan Feinberg is president and CEO of Fein-Line Associates, Inc., a consulting firm focused on the electronic interconnect and assembly industries and a founding partner in the Quantum Performance Group, LLC, a market research firm. Formerly, Dan was president of Morton Electronic Materials (Dynachem), a $240 million division of Morton International, Inc., and a group vice president of the parent company Morton International, until its acquisition by Rohm and Haas in 1999. To contact Dan, click here.

TECHNICA, U.S.A.

Fulfilling the Manufacturing Needs
Throughout the Electronics Industry



ROOTED IN THE ELECTRONICS INDUSTRY FOR OVER 25 YEARS

Technica, U.S.A. • 2431 Zanker Road, San Jose CA 95131 • PH: 800-456-2970 • FAX 408-943-8115

www.technica.com

Printed & Flexible Electronics: No Longer Waiting For the Train to Arrive

by Daniel Gamota
PRINTOVATE TECHNOLOGIES, INC.

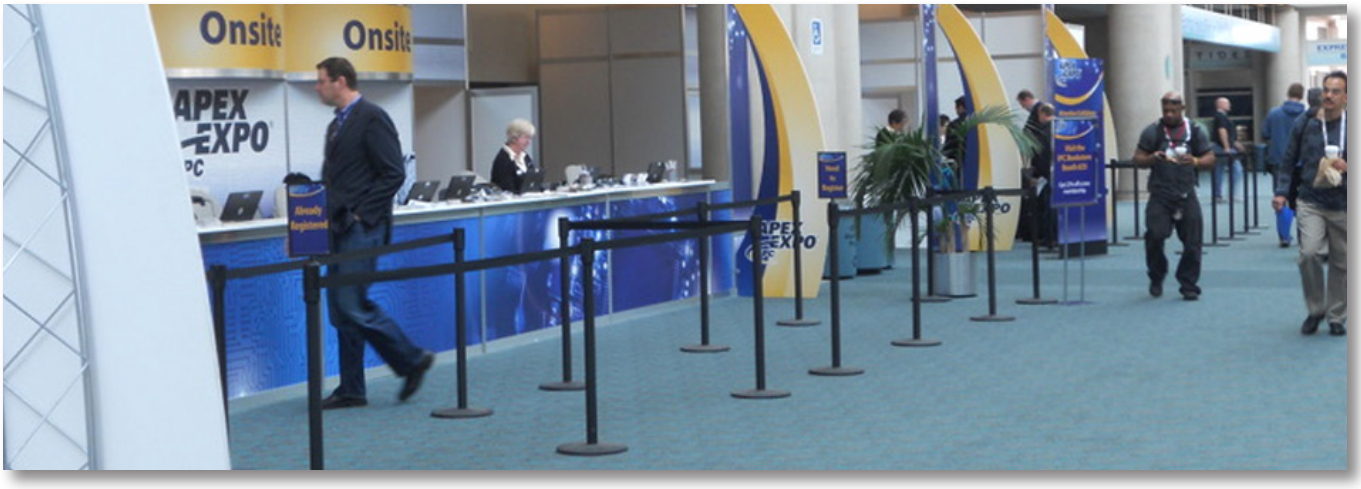
SUMMARY — Real Time with…IPC was able to speak to leaders in this emerging field that represent the most critical components necessary to mitigate risk and to ensure industry sustainability. Interviewees commented on recent activities in the field that have resulted in measurable growth as products approached commercialization in a variety of industries.

Ten years from now people will remember IPC APEX EXPO 2012 as the event at which printed and flexible electronics began its journey to go, "where no technology has gone before." Real Time with…IPC interviewed 15 individuals that represent different elements of the printed and flexible electronics ecosystem. IPC and Real Time with… have been closely monitoring the growth of the industry for more than 10 years as product opportunities moved beyond membrane switches and other electronic product "touch points and interfaces." Moreover, their stakeholders have recognized the potential new revenue streams that could be captured from the emerging field. IPC APEX EXPO 2012 showcased the technology platform, products, and leaders of the industry.

In late 2010, IPC organized several events to gauge the interest level among its stakeholders (e.g. functional ink, flexible substrate, assembly equipment, printing equipment, and testing hardware suppliers) to actively participate in the formation of the necessary supporting elements to launch printed and flexible electronics. Since these information gathering meetings, the architects of the printed and flexible electronics strategy at IPC, David Torp and Marc Carter, have established the foundation — standards, guidelines, and roadmap — on which the industry can grow.

Real Time with…IPC was able to speak to leaders in this emerging field that represent the most critical components necessary to mitigate risk and to ensure industry sustainability. Interviewees commented on recent activities in the field that have resulted in measurable growth as products approached commercialization in a variety of industries. Product families demanding attributes only provided by printed and flexible electronics were mentioned for the aerospace, automotive, consumer, and medical markets. Some interviewees mentioned that printed and flexible electronics were now part of their internal company roadmaps and were considered as a "green" enabler.

Although the interviewees projected excitement for the future of the industry, they cautioned that significant efforts remained before companies would realize financial rewards. On several occasions, the interviewees stressed the importance for the formation of a



robust supply chain and the further development of the value chain. They also highlighted the need for establishing curriculum to be provided in academia, as well as workforce training, offered to companies debating future product offerings. The majority of those interviewed felt that recent technical journals highlighting the benefits of printed and flexible electronics positioned the industry in the spotlight, but felt that most engineers still were unfamiliar with the technology. Another common statement was that, in general, the electronics industry appreciation for the technology was lacking due to the over-promising and under-delivering of the industry.

IPC has committed to leading the formation of the critical structural elements to support the printed and flexible electronics industry. It will provide the mechanisms necessary for efficient coordination between various organizations necessary to succeed. Real Time with…IPC will supply updates for the various initiatives underway that will ultimately increase the number of printed and flexible electronics enabled products. We look forward to IPC APEX EXPO 2013 where Real Time with…IPC will continue to update the electronics industry on the printed and flexible electronics journey.



Dan Gamota is president of Printovate Technologies, which offers new product introduction and manufacturing design services for printed and flexible electronics enabled energy, lighting, communications, and sensor products. Recently, he was selected as the Technical Advisor to the U.S. Technical Advisory Group to IEC TC119 – Printed Electronics Standards. He is chair of the IPC Printed Electronics Standards Committee, IEEE Organic and Molecular Electronics Standards Group, and is chair of the group that publishes the iNEMI Large Area Flexible Electronics Roadmap.

PC Hall of Fame Award Winner Denny Fritz

by Real Time with…IPC APEX EXPO 2012





Longtime IPC member Denny Fritz enjoys a conversation with Editor Ray Rasmussen at IPC APEX EXPO 2012. Fritz won the Raymond E. Pritchard Hall of Fame Award for industry service and commitment to the IPC.



realtimewith.com

VIDEO INTERVIEW

IPC APEX EXPO 2012 Showcases the Future of Interconnection Manufacturing Methods and Equipment

by I-Connect007 Staff

SUMMARY — IPC APEX EXPO's return to San Diego, California was welcomed by many attendees, even though the reputation of the sunny city was called into question by early-week clouds and rain. Regardless, there was a palpable, upbeat feeling to the event.

The tone of the conference, by chance or design, seemed to be set, in part, with the keynote talk given by William Shatner. He stressed the importance of risk, both in product development and in life, but also challenged the audience to pay more attention to the implication of their work on the environment.

After a few years in the background, this year seemed to be a coming-out party for two technologies likely to prove increasingly important in the future: printed electronics and embedded device technologies. The buzz sessions on these two "up-and-comers" were equally well attended, but unfortunately were squared off against one another on the schedule, thus eliminating the ability to sit in on both sessions. The technologies are each compelling in their own right and there is even overlap in terms of what might be done in the future with these viable methods.

Another well-attended buzz session was the IPC Electronic Interconnection Roadmap event. The IPC roadmap has been a valuable operational tool for the interconnection industry for nearly two decades, providing information on what is available and what is ahead to solve current and future generation electronic interconnection needs.

The show appeared to have been much appreciated, as other reports in this edition will confirm and attest. Next year should prove even better, based on this year's experience. We should, at that time, have a better idea of what progress will have been made by next-generation solutions shown in their incubation stage at this year's event.

Impressions of IPC APEX EXPO 2012

by Patty Goldman
I-CONNECT007 GUEST EDITOR

SUMMARY — People at the show, if not wildly enthusiastic, were thinking positively. The show floor, if not jam-packed with people, seemed busy. The technical paper sessions I attended, if not standing room only, were well attended and were all very good papers.

I've been attending and participating in IPC events for quite awhile now (yeah, I'm a dinosaur) and, as always, there are many more things to do than time to do them. I figure there are at least four tracks at these events, of which the show itself is just one. Others include the technical conference paper sessions, professional development courses and workshops, and the technical standards committee meetings, which are the true heart of an IPC meeting.

However, there is also that fifth track, somewhat nebulous, but no less important, which is the relationship building and camaraderie that one sees and feels, the formation and/or renewal of industry contacts and friendships, the exchange of information and knowledge that comes from working together toward a common goal. Quite possibly, it has the broadest attendance of all the tracks and certainly is no less important than the others are.

There is always some sort of a "buzz" during the meetings and show—this year it was a warm, optimistic buzz. No wild excitement,



just a steady, determined sound. People at the show, if not wildly enthusiastic, were thinking positively. The show floor, if not jam-packed with people, seemed busy. The technical paper sessions I attended, if not standing room only, were well attended and (as part of the Technical Program committee, I'm in the know on this) were all very good papers. I heard that one of the workshops had 54 attendees (that's a lot!). The people in the committees and subcommittee meetings that I attended or peeked into seemed busily at work; the one I chaired was pretty well packed. The committee luncheons were full, and judging from the number of awards presented, many new or updated standards have been recently completed and issued.

As for that fifth track, well, I can only say that I went through a couple hundred business cards, about half during the meetings and technical sessions and half while wandering the show or at the I-Connect007 booth; maybe half were given to old friends and the rest to new ones. I don't think I'm alone when I say that I find IPC meetings and shows to be uplifting, renewing, and re-charging.

But, hey, haven't we discussed IPC standards, the electronics industry, and the world economy and what's new and old, enough? Apparently not! Aren't we tired of it? Apparently not! Will we do it again? Apparently so! Do we love this industry, or what?

What do you think?

Patricia Goldman has been in the PCB industry for 30+ years, working in such varied areas as R&D of imaging technologies, wet process engineering, and sales and marketing of PWB chemistry, photoresist and glass reinforcements. Involved with IPC since 1981, Patty has chaired many committees and subcommittees and also served as TAEC chairman. Godlman is also the author and co-author of numerous technical papers on bondable gold, pink ring, replacing electroless copper, and fiber weave effect.

iNEMI Update with CEO Bill Bader

by Real Time with...IPC APEX EXPO 2012





iNEMI initiatives continue to help pull the industry forward. Projects focused on medical electronics, miniaturization, and the environment lead the organization's efforts in 2012 along with continued roadmapping efforts.



realtimewith.com

VIDEO INTERVIEW

Three Roadmaps Addressing Three Levels of Electronics Industry Manufacturing: *Is There a Disconnect Among Them?*

by Harvey Miller
FABFILE ONLINE

SUMMARY — An expert panel discussed the challenges of charting the future of electronic industry interconnection technologies during IPC APEX EXPO 2012. The implications for electronic equipment manufacturing could not be more profound. Following is an interpretation of three messages, addressed selectively to focus on that rather disturbing question: Is there a disconnect?

At IPC APEX EXPO 2012 in San Diego, a very qualified panel discussed the challenges of charting the future of electronic industry interconnection technologies. The implications for electronic equipment manufacturing could not be more profound. Following is an interpretation of three messages, addressed selectively to focus on that rather disturbing question posed in the title: Is there a disconnect? Could it be that economic stakes in the presently dominant manufacturing technologies serve to censor more specific consideration of a future that may have to be very different? (It's not polite to threaten established, still-going businesses. At Eastman Kodak, it wasn't polite to talk about silver halide loss of photographic dominance. In time, the question became irrelevant.)

A Little Background...

Before I discuss the three roadmaps and their implications, important background is necessary to fully appreciate and understand the forces that have driven, and will continue to drive, the future of the electronics industry.

• 1960: The first MOS transistor was built by Kahng and Atalla at Bell Labs.

• 1964: Patent granted to Jack Kilby for the first miniaturized integrated circuit.

• 1965: Moore's law describing a long-term trend whereby the number of transistors that can be placed inexpensively on an integrated circuit doubles approximately every two years. This empirical law has become an imperative ever since, and the entire electronic industry depends on it for new product development. Its continuation drives IC lithography and other fabrication advances that underlie the ITRS.

Fisher's Characterization: ITRS, iNEMI, & IPC, "Expect," "Need," & "Believe," Respectively

Jack Fisher pointed out the weight of $30 billion in making things happen. That's the annual R&D expenditure budget of the semiconductor industry. That is what makes Moore's Law happen, because if it stopped, so would the advance of the trillion dollar electronics hardware industry and the multi-trillion dollar software/applications/media industries that rest on that hardware.

The International Technology Roadmap for Semiconductors (ITRS) does its work through working groups enrolled from over 1,500 members—companies, academia, and governments. Launched in 1991, ITRS has more than 50 years of semiconductor and IC development history to guide its 15-year future horizon for all the levels required to develop and manufacture ICs: Design, test, process integration, front-end process, lithography, interconnect on chip, assembly/packaging, and factory integration. The result of all that work is a seamless set of recipes that enables equipment and consumable suppliers to plan.

The table below is a small sample of ITRS roadmapping for just one device family, high-performance microprocessors.

Note that by 2026, cost per function will decline by more than two orders of magnitude. Imagine $50 iPads. I can't wait! However, I'll bet that the OEMs can.

OEM Perspective

OEMs, the semiconductor industry's customers, sit at the ITRS tables too. They need to know circuit speed improvements, thermal dissipation, and cost of new IC generations, all basic to new equipment designs for medical, mil/aero, computing, telecom, industrial, etc.,

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

The data, while believed to be accurate and based on analytical methods considered to be reliable, is for information purposes only. Any sales of these materials will be govererned by the terms and conditions of the agreement under which they are sold.

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

Disconnect *continues*

markets served. The OEMs respond competitively to the needs of their markets. They have their own roadmapping organization, International Electronics Manufacturing Initiative (iNEMI) to address end equipment needs, plus the demands from the public sector for sustainability, and their customers for reliability.

Fabricators & Assemblers

That takes care of ITRS' "expect" and iNEMI's "need," which brings us to the guys who actually make the hardware: The fabricators and assemblers who are subject to all those semiconductor industry and OEM expectations and needs. They have their own roadmapping activities through IPC. Fisher says their results are based on "belief." Maybe "hope" would be a better word.

I can't do justice to the other speakers, so I'll summarize just two key points from the talks by Dieter Bergman and Joe Fjelstad, respectively.

Bergman makes the very important point that there is great need for coordination in roadmapping among electronic industry tiers. Serendipitously, Denny McGuirk, former President of IPC, is now President of Semiconductor Equipment and Materials International (SEMI). He's the guy who can help make coordination happen.

Finally, Fjelstad emphasized the need for innovation, for flexibility. Surprise! Take it from Alice in Wonderland. That is where we all live.

References:

1. 2012 IPC International Technology Roadmap
2. http://www.google.com/patents



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

Tales from the Show Floor

by Steve Williams
I-CONNECT007 GUEST EDITOR

SUMMARY — If professional development course registration is any type of measure, this year's event was a smashing success, including this editor's course, which saw an all-time high, exceeding 50 attendees. Other instructors reported similar success.

I wasn't sure what to expect this year in San Diego in terms of attendance and exhibitor participation; after all, this year's show was on the heels of several held in Sin City, where extracurricular activities are a huge draw. It doesn't get much sexier than Las Vegas. However, San Diego proved a worthy opponent and more than held its own this year; I haven't seen the attendance numbers yet, but would venture to say it will meet, or exceed, last year's numbers.

If professional development course registration is any type of measure, and I think it is, this year's event was a smashing success. My little course had an all-time high, exceeding 50 attendees, with other instructors reporting similar success. I took a straw poll with my attendees, and about two-thirds had been to last year's show, with the majority stating that they felt 2012 was a bit better in terms of the show and, in general, it was less expensive to send a group of people to California for a week. The city provided an outstanding amount of quality choices for restaurants, entertainment,

and sightseeing opportunities. Lousy economy or not, people were clearly spending money, which is a very good thing for the industry!

I had a couple of drinks with a very good friend who owns a PCB shop on the East Coast. As a business owner in our industry, he is very active in IPC and a leader in activities that move our industry forward. I mentioned that I was moderating a panel the next day on the state of American manufacturing, which spurred the conversation in a different direction. My friend was pleased with the show itself and had made some major equipment purchases. However, he voiced his extreme disappointment in the lack of participation of PCB manufacturers in forums, leadership meetings, and councils. Considering the large number of shops in California, of particular note was the absence of local representation in the leadership groups charting the course for the industry.

The show floor was busy and well stocked with exhibitors and I could sense a cautious optimism and excitement in the air. I noticed a definite lack of PCB manufacturing booths, but this slack was picked up by an increased focus on equipment manufacturers. It is clear that the show is morphing from a showcase of PCB manufactures to equipment OEMs, which is a good thing, just different.



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

IPC Addresses Conflict Minerals

by Real Time with...IPC APEX EXPO 2012





IPC Board and Government Relations Committee member Bhawnesh Mathur and IPC's Fern Abrams talk about the issues facing the industry regarding conflict minerals.



realtimewith.com

VIDEO INTERVIEW

Musings on IPC APEX EXPO 2012

by Dr. Ronald C. Lasky
INDIUM CORPORATION

Summary — Busy booths, quality attendees, and high numbers at the workshops impressed many at this year's IPC APEX EXPO.

I am one who does not enjoy the smoke filled casino atmosphere of Las Vegas, so I much preferred this year's San Diego venue. The convention center is between the Gaslamp Quarter and the ocean — delightful. I had great meals at The Fish Market and the Omni Hotel. In addition, the staff at the convention center was very helpful and friendly.

The numbers of people on the tradeshow floor seemed to me to be the most in recent memory. From the people that I saw at Indium Corporation's booth, I think the quality of the show attendees was very high. Similarly, my workshop, Pb-Free Assembly for High Yields and Reliability, had about 40 people in attendance. My three paper presentations (Correlation of SIR, Halide/Halogen, and Copper Mirror Tests; Minimizing Voiding in QFN Packages Using Solder Preforms; and Choosing a Low-Cost Alternative to SAC Alloys for PCB Assembly: Preliminary Work) continued this trend as each attracted 35 to 55 people. The technical discussions after the workshops and papers were quite stimulating and demonstrated the high technical level of the attendees.

In the past, I have been interviewed for Real Time with… and have found it to be a rewarding experience. This year, in addition to being interviewed, I sat in the interviewer chair as well. I was a little nervous being on the other end of the microphone, but I quickly got the hang of it. I was glad to have the opportunity to help my colleagues share the great things they are doing.

For me, this was the best IPC APEX EXPO ever!



Dr. Lasky is a senior technologist at Indium Corporation and a visiting professor at Dartmouth College. He has more than 20 years experience in electronic and optoelectronic packaging at IBM, Universal Instruments and Cookson Electronics. Dr. Lasky has authored or edited five books on science, electronics and optoelectronics and numerous technical papers. He has also been an adjunct professor at several colleges and has taught over 20 different courses on topics ranging from electronic packaging, materials science, physics, mechanical engineering and science and religion.

"Second Star from the Right and Straight on 'Til Morning"

by Pete Starkey
TECHNICAL EDITOR, I-CONNECT007

Summary — William Shatner's keynote address was packed with eager attendees at IPC APEX EXPO. A thread of serious environmental consciousness ran throughout Shatner's performance. As well, many thought-provoking and inspirational questions were asked and answered with wit, insight, and wisdom.

Our Real Time with...IPC schedule was so crammed full that, although I enjoyed interviewing a whole lot of fascinating people, so little time was left for me to go walkabout and see IPC APEX EXPO as a visitor that I missed most of it.

However, the significant event I was able to attend was the William Shatner keynote. The anticipation was so great, that an hour before it started, folks were already queuing six-deep for 50 yards along the upstairs hallway of the

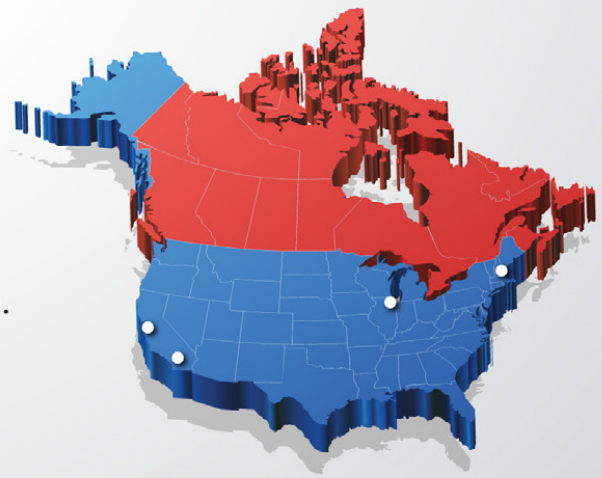
Global leaders in testing.
Industry pioneers in printing.



MicroCraft's history of Service and Support for Probe and Grid Testing, Fixturing and Jetprint is unmatched in the industry.

Don't forget we also offer Testing and Fixturing Services, offshore data prep and specialized Inkjet printing services.

We have multiple 24-hour locations to serve you. To learn more or get a quote, please contact us today at sales@usamicrocraft.com.



MicroCraft

www.usamicrocraft.com

Straight On 'Til Morning *continues*



San Diego Convention Center. When the doors to Ballroom 6 were eventually opened and the crowd flooded in, the seating provision of maybe 1,500 was completely overwhelmed, and the partitions had to be opened to allow the anteroom to accommodate the standing-room-only overflow.

Remarkably sharp for an 80-year-old, but clearly weary from travelling, Shatner made a slow start, but soon had his audience hanging on every word. Acknowledging that IPC APEX EXPO was a technology show, he put on a serious face as he announced that the world is coming to an end: "Look what damage technology has done to the planet — we rely on technology to help put things right." He continued, "I am scared about the future: We have a window of time in which human beings can live and we have to teach the children the future is in their hands. Technology is the thing that can save us, but we've got to do better than we did in the past!"

A little surprising for a keynote presentation, Shatner broke off from his preamble and invited questions from the floor. Many were, predictably, Star Trek oriented, which he fielded with well-prepared and good-humoured responses. Some were more profound: "What made you what you are?" to which he replied, "The mysteries of what exists out there in the universe are mind-boggling. What are the answers? We don't understand shit! Nothing is everything. We are limited by the powers of our imagination. All of the characters I have played represent some aspect of me. I can't be you!"



Reflecting upon the meaning of life, with reference to an old Star Trek episode, Shatner stated, "Life is risk. We risk everything all the time. Risk is why we're here on Starship Enterprise." And, he had this to say about how easy it was to say no: "No is safe. Yes carries risk and commitment — the way to live your life. Yes to new technology. Yes, because you can't learn without making mistakes."

A thread of serious environmental consciousness ran throughout Shatner's performance. As well, many thought-provoking and inspirational questions were asked and answered with wit, insight, and wisdom.

"Where do we go from here?" was supposed to be the cue for the play-out music, but the question came unexpectedly early. Shatner quoted Captain Kirk (who it turns out was actually quoting J. M. Barrie's Peter Pan): "Second star to the right and straight on 'til morning."



Pete Starkey, a longtime industry editor based in the UK, joined I-Connect007 as Technical Editor in 2008. Previously, Pete served as Technical Editor for both the Good PCB Guide (UK) and Printed Circuit News. Most recently, he was Products Editor and Europe Editor for CircuiTree. Pete has more than 30-years experience in the PCB industry, with a background in process development, technical service, and technical sales. To contact Pete, click here.



New IPC Board Chair Sets Sights on Future

by Real Time with...IPC APEX EXPO 2012



Steve Pudles, CEO of Spectral Response, is the new chairman of the IPC Board of Directors. Steve has served the IPC in a variety of volunteer positions. His goals as chairman are to expand membership in IPC and continue to foster IPC standards throughout the electronics industry. Currently, the IPC Board is interviewing candidates for president of the IPC to replace former president Denny McGuirk.



realtimewith.com

VIDEO INTERVIEW


HITACHI
EVERETT CHARLES TECHNOLOGIES
ELECTRA
Maskless
HIGH POWER LED DIRECT EXPOSURE
ATOTECH
OEM
OAK-MITSUI
Excellon
Excellon


EQUIPMENT
You're Not Alone With Taiyo
INSULECTRO
CHEMCUT
NUJAY
MICROTEK
Independent Test Laboratory
Ucamco
UYEMURA

OEM PRESS SYSTEMS INC.

WORLD'S FINEST PRESS SYSTEMS

INNOVATIVE ENGINEERING DESIGN INCORPORATED INTO EVERY PRESS SYSTEM

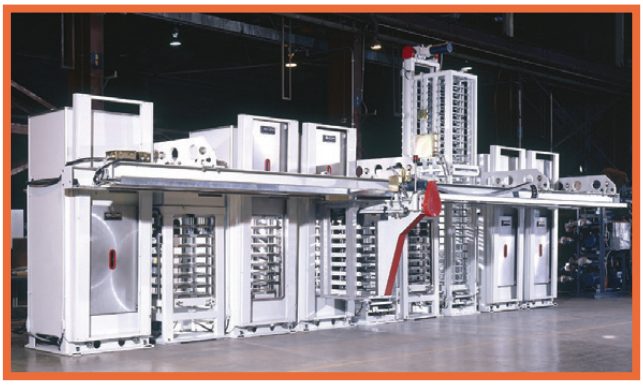
- **OEM Vac-Q-Lam Press design delivers even pressure distribution across the product area**
- **"Low Profile" OEM Vac-Q-Lam Press design (no pit required)**
- **Full material handling and press loading/unloading system**



Copper Clad Laminate Vacuum Press
Designed for full sheet production



OEM Fleetline Vacuum Press Systems
with high temp. capability and tight temperature uniformity, designed to deliver optimum process control capability



High Volume Press Systems
with fully automated press loading and unloading



Quick-turn Vacuum Press Systems
designed to accommodate high mix prototype production

Integrated Vacuum Lamination Press and Material Handling Systems with fully programmable process control capability

CUSTOM PRESS SYSTEMS • TURNKEY FACILITY DESIGN AND INSTALLATION
WORLDWIDE SERVICE AND SUPPORT

OEM Press Systems, Inc. • 311 S. Highland Ave. • Fullerton • CA 92832 • Ph: (714) 449 7500 * Fax: (714) 449 7510
E-mail: **mary.quinlan@oempresssystems.com** • Web: **www.oempresssystems.com**



VIDEO INTERVIEW

Mikel Williams' Government Relations Update

by Real Time with...IPC APEX EXPO 2012



DDi Corporation CEO Mikel Williams joins Editor Andy Shaughnessy to discuss his work as Chairman of the IPC Government Relations Committee. He recently spoke on a panel in Washington, D.C. about the importance of using American companies to manufacture military weaponry.



realtimewith.com





Most-Read PCBDesign007 News Highlights This Month



Mentor Graphics Among DesignVision Winners

The 2012 DesignVision Award winners include Valydate Inc., Mentor Graphics, Altium, Molex, Memoir Systems, Akros Silicon, Apache Design and Oscium. The DesignCon technical committee, including 130 engineers and industry editors, evaluated products from hundreds of exhibitors to choose the winners.

Sunstone Circuits: 40 Years of Serving Design Engineers

"For 40 years, Sunstone Circuits has been committed to meeting the needs of our customers by providing the best and most efficient PCB design experience from quote to delivery," said Sunstone's CEO, Terry Heilman.

SiSoft, Ericsson Present Case Study of Virtual Prototype Analysis

"Virtual prototype analysis provides substantial amounts of data," noted Donald Telian of SiGuys. Key performance metrics can be mined to identify important system-level trends.

Zuken Releases E3.WireWorks Unified

Zuken has released a new version of its E3.WireWorks software, E3.WireWorks Unified, which is now a full part of the company's complete E3.series solution. E3.WireWorks Unified now uses native E3.series files and provides users with access to modules that were previously only available in the E3.series.

Intercept Launches Pantheon 7

The new Pantheon includes a completely redesigned user interface with significant performance and productivity increases. The last 15 years of Pantheon's development as an advanced PCB/hybrid/RF design application has converged with a modern user flow to provide major productivity increases for all engineers and designers.

IPC Stresses PCB Design Export Controls in D.C.

"Reform of the current export control system is long overdue, and IPC supports reform and the opening of foreign markets to our manufacturing companies. The current system is complex, bureaucratic, and does not adequately protect our national security, nor facilitate the export opportunities we need to grow our economy," testified Mikel Williams, President and CEO of DDi Corporation.

Designers Forum Q&A With IPC's Anne Marie Mulvihill

On February 27, the Designers Forum will mark the beginning of IPC APEX EXPO at the San Diego Convention Center. I-Connect's Andy Shaugnessy recently caught up with Anne Marie Mulvihill, IPC's manager of professional development and the Designers Council, to get the inside scoop on this year's Designers Forum.

Omni PCB to Hold PCB Coffee Talk with Paul Reid

Omni PCB, LLC, a nationwide manufacturer's rep firm specializing in PCBs, announced the March 7 session of PCB Coffee Talk. This series of 20 minute Webinar "chats" with industry experts provides a fun, fresh way to learn more about PCB basics.

AWR Receives U.S. Patent for Via Creation

AWR Corporation has been issued a U.S. Patent by the United States Patent and Trademark Office for "automatic creation of vias in electrical circuit design." The patent addresses a methodology of creating and detecting crossings of conductive traces on different layers of an integrated circuit or PCB.

Agilent's Genesys 2012 Enhances RF System Design

Agilent Technologies has released the latest version of its industry-leading, low-cost, high-performance software for RF and microwave board design, Genesys 2012. This release delivers significant improvements to the planar EM simulator Momentum GXF, including a quadrangle mesher to efficiently mesh a planar layout with the fewest number of unknowns.

Prototron Circuits
Quality printed circuit boards

Here's what one of our customers had to say about Prototron...

X2IMPACT

I've been using Prototron for about 12 years now. They have always provided great quality and are really easy to work with. I ran at least 25 separate board orders through their plant last year and didn't have problems with any of them. Their quotation department is fast and thorough, and their technical guys in CAD/CAM are the best I've ever worked with. In one instance, they had a production problem on a fast-turn board which resulted in a slip of a day. But I was informed as soon as they knew there was a problem and they put the re-spin in the front of their queue to get good boards to me as quickly as possible. They are the only board house I use. I can't say enough good things about them.

Chris Doughty
X2IMPACT
Seattle, WA

...and of course the customer is always right!



Standards Boost Business for an Innovation Nation

by S. Joe Bhatia
AMERICAN NATIONAL STANDARDS INSTITUTE (ANSI)

SUMMARY — Today, Americans in every sector are focused on two critical efforts: creating jobs and building a lasting economy for the future. As we set a course for sustainable economic growth, a renaissance in American manufacturing and innovation and continued U.S. competitiveness are absolutely critical.

As has always been the case, America's competitiveness and prosperity are directly tied to our ability to translate ideas into new products, services and jobs. From computers and information technology to the robotics industry, vast opportunities for U.S. innovation and economic growth are arising from new and emerging sectors. Further, standardization has the power to help speed those developments and translate them into valuable gains in the global market.

An Innovation Nation

Particularly in high-tech industries, standardization can help U.S. business shape enormous growth and drive innovation. Standards and conformance programs are essential to helping U.S. businesses tap into these technologies and bring them to market, resulting in increased export opportunities for U.S. companies abroad, which in turn helps create more jobs in America.

That is why the American National Standards Institute (ANSI), together with partners from industry and the standardization community, have launched Standards Boost Business (SBB), designed to help U.S. companies across all industry sectors tap into the power of standardization to build business performance and gain a powerful trade advantage in the global market.

One of the most powerful examples of innovation and standards intersecting is the electronics industry. The rapid pace of innovation in electronics technology creates an ever-growing demand for newer and more advanced products and applications. Technological revolutions in personal computing devices and telecommunications continue to change the way we live and conduct business.

From design and manufacturing to distribution and marketing, electronics — like all prod-

"According to the White House, the manufacturing sector is recovering faster than the rest of the economy."

ucts and services—are affected at some point by standardization. Standards have the potential to accelerate the idea-to-market cycle, increase the number of U.S. products and services, and save companies millions of dollars. In short, standards have the power to turbo-charge innovation and fuel business growth.

Increased Exports = More Jobs at Home

In many respects, the manufacturing industries can be seen as a barometer of the U.S. economy. U.S. manufacturing is responsible for nearly two-thirds of the nation's exports[1] of goods and 11% of the gross domestic product[2]. Manufacturing also supports 18.6 million jobs in the United States[3], including roughly 65% of U.S. scientists and engineers[4].

According to the White House, the manufacturing sector is recovering faster than the rest of the economy. Over the last two years, the economy has added 334,000 manufacturing jobs, making it the strongest two-year period of manufacturing growth since the 1990s.[5]

The United States is still the world's largest manufacturing economy, accounting for 21% of all global manufactured products[6]. However, our global competitors are not far behind, and they are closing the gap: China is second at 15%, and Japan is not far behind at 12%.

To Grow Our Economy We Must Grow Our Exports

With 95% of the world's consumers outside the United States, the greatest opportunities for growth are in other parts of the world, and it is imperative that U.S. companies find new customers in these markets. Ninety-seven percent of U.S. companies that export are small and mid-sized businesses, and yet the majority of these companies export to only one or two countries, leaving vast opportunities uncovered.

Under the National Export Initiative (NEI), President Barack Obama set the goal of doubling U.S. exports in five years, an increase that will support two million American jobs. To attain this goal, exports will need to grow from $1.57 trillion in 2009 to $3.14 trillion by 2015. So far, we're on track to meet that goal ahead of schedule. As of October 2011, American exports totaled $2 trillion—an increase of almost 32% over 2009[7].

To maintain the nation's success as the world's leading manufacturer, the United States must capitalize on technological and market opportunities, accelerate the idea-to-market cycle, and increase the number of U.S. innovations and products in foreign markets. As the technical underpinning of many products and services, standards play a critical role in removing barriers to trade, enforcing free trade agreements, and expanding foreign markets for U.S. goods and services. This opens export opportunities for U.S. companies in global markets, which in turn creates jobs here at home. It's time to turn the tide and take toward a 21st century manufacturing economy and create high-tech, good-paying jobs here in the United States.

Tap into the Power of Standardization

The U.S. Department of Commerce estimates that more than 80% of global commodity trade is impacted by standardization, which amounts to more than $13 trillion each year. These numbers highlight a critical fact: Companies that participate in standardization gain an important competitive advantage in the global marketplace. Time and again we have seen that those companies that leverage standardization are the ones to capture opportunities in the domestic and international markets. In today's economic climate, companies need every advantage they can get.

Participation in standards development provides insiders with knowledge and early access to information on emerging issues, and helps



reduce redundancy, minimize errors and shorten time to market.

Here are three important steps you can take today to make sure that your company reaps the rewards that standards and conformance provide:

1. Participate in standards development activities, both domestic and international. This allows your company to exert influence on technical content, and align your products and services with market demand. By participating in standards development activities, you have an opportunity to directly influence the requirements and guidelines for your product.

2. Rely on standards to design your products and services, and turn to recognized conformity assessment systems to test, inspect, certify and accredit them. Not only will this help ensure quality and customer satisfaction, it will also facilitate cross-border interoperability, creating market access for components and products you manufacture in all countries in which you want to sell.

3. Treat standardization as a strategic business tool. Institute a corporate policy that supports standards development and use it alongside quality, safety, intellectual property and environmental performance. It is equally important to the long-term health of your business.



WE HAVE GLASS

Did you know that C.A.Picard is the largest supplier of high-quality glass in the world?

We provide high-strength, high-quality, inspected, graded, heat tempered glass matched to your application.

LEARN MORE

C.A.PICARD® INTERNATIONAL

CONTACT GARY WEIDNER: GWEIDNER@CAPICARD.COM • 1 (269) 962-2231



What it comes down to is this: Those who understand how to effectively influence and address standardization and compliance issues have the greatest success in the global marketplace. If your experts are active participants in standards development, they will be the ones to help shape the requirements and our preferred positions that the U.S. brings to these international forums. They will be the ones to influence the global standards that cross borders and expand markets worldwide. Domestically, the opportunities are just as great.

The U.S. government recognizes the advantages of flexible, responsive, timely, and reliable standards and conformance systems. And through the National Technology Transfer and Advancement Act (NTTAA), the government has affirmed its commitment to using standards developed by private-sector standards bodies as a means to carry out their policy objectives or activities whenever possible.

Another thing to keep in mind is that voluntary standards are often later adopted into legislation. Consider the advantage this affords to those companies that participated in shaping those standards: Not only are they ahead of the game and aligned with regulation, but they are actually influencing the way their entire industry must do business. This is a huge competitive edge.

Take Your Seat at the Table

As an executive, you really have two choices: position your organization to take a seat at the table and be part of the process, or let your competitors—both foreign and domestic—dictate the way you do business. Remember, standards and conformance are critical business tools that must be managed alongside all of your organization's other policies.

With job creation and the economy at stake, we cannot afford to wait. No challenge is more urgent. The time to act is now.

Tap into the power of standardization today and forge a path for sustainable economic growth for the nation; spearhead the next generation of high-tech manufacturing, create jobs and open global markets for American exports. Visit www.standardsboostbusiness.org to get involved in standardization today. **PCB**

References

[1]International Trade Administration press release, "Data Confirms Manufacturing Exports Support and Sustain American Jobs," March 8, 2010. www.trade.gov

[2, 3, 6] National Association of Manufacturers, "Facts About Manufacturing," www.nam.org

[4] Jon Gertner, "Does America Need Manufacturing?" New York Times Magazine, August 24, 2011, www.nytimes.com

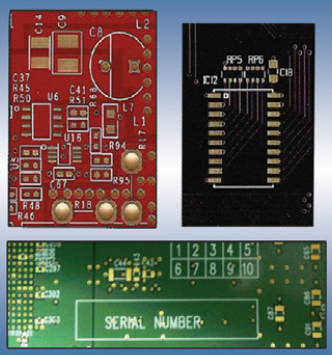
[5, 7] www.whitehouse.gov



S. Joe Bhatia began his tenure as president and chief executive officer of the American National Standards Institute (ANSI) on January 1, 2006. Prior to joining ANSI, he held the position of executive vice president and chief operating officer of the international group at Underwriters Laboratories Inc. (UL). During his 30-plus-year tenure with the organization, Bhatia assumed positions of progressive leadership in global business operations.

YOUR ULTIMATE SOLUTION for Digital Imaging

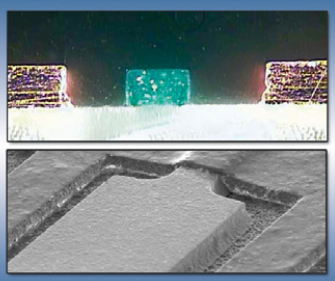
IJR™-4000™ MW300 INKJET LEGEND



Inkjet developed by TAIYO Korea

- The Only Inkjet that will Adhere to Cured and Semi-cured Solder Mask
- Designed for Inkjetting Equipment
- Fast UV Curing
- Bright White (High Opacity and Non-Yellowing)
- High Cured Hardness
- Halogen-free
- RoHS & REACH Compliant
- Excellent Chemical Resistance
- Compatible with Lead-Free Processing

PSR-4000® LDI SERIES



Laser Direct Image Solder Resist developed by TAIYO America

- Fine Feature Resolution
- LDI or Conventional Exposure
- Excellent Small Hole Clearing
- Rich, Green Color
- High Productivity – As Low as 50 mJ/cm²
- Compatible with ENIG, Silver & Tin Final Finishes
- Low Halogen, Lead-Free Assembly Compatible

PSR®-9000™ LDI

For Flex Circuit Applications Available in Amber & Green
Same Excellent Properties as PSR®-9000™ FXT

Recommended Reading:
"Why Inkjet for PCB has become a Reality" by Guy Alon and Ron Ellenbogen of Orbotech

CLICK TO READ



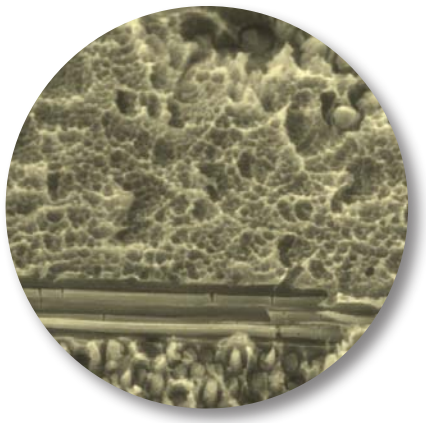
TAIYO AMERICA™
YOUR ULTIMATE SOLUTION

email: info@taiyo-america.com
Phone: +1 (775) 885-9959 USA

WWW.TAIYO-AMERICA.COM

TROUBLE IN YOUR TANK

It's Not Your Father's FR-4 Anymore



by Michael Carano
OMG ELECTRONIC CHEMICALS

SUMMARY — As PCB fabrication has shifted to the use of a myriad of higher-performance materials, many are finding a significant place in the electronic interconnect industry. In this month's column, concerns with improving desmear and overall resin removal on these higher performance materials is examined.

Introduction

There is no question that the materials used in PCB fabrication have shifted from that comfortable center of gravity (FR-4 T_g, 135°-140°C) to a myriad of higher-performance materials. While conventional FR-4 remains in widespread use, these higher performance materials are finding a significant place in the electronic interconnect industry. Also along for the ride are the challenges of preparing these higher performance materials to be desmeared and subsequently metalized. As one moves toward these higher performance materials, desmear and overall resin removal becomes more difficult. Exactly what are the concerns with improving desmear on these higher performance materials?

Corny title aside, it is the truth: One only needs to glance at the IPC and iNEMI Technology Roadmaps to recognize the standard FR-4 is quickly giving way to alternative materials. Whether the fabricators are building large format back panels with high-speed requirements or enhancing the long-term reliability of the circuit with PPO, PPE and epoxy-Novolak® type resins, the desmear and metalization game has changed. It would be prudent to review the desmear process and understand the limitations to permanganate desmear as one moves into these higher performance material sets.

Desmear Process

When beginning any work with higher T_g resin materials, conventional wisdom must be put aside. First, evaluate the current process by examining conventional permanganate parameters, including a solvent-based sensitizer and a high pH potassium permanganate etch. Weight loss testing is used to measure effectiveness. A weight loss of approximately 0.3 mg/cm² is considered representative of sufficient desmear of most drilled holes. FR-4 responds in typical fashion and no special steps were required. Some higher T_g materials, on the other hand, exhibited low weight loss; the high cross-linkage chemistry of the dielectric resin required some process refinement to reach an acceptable level of weight loss. Other resin types are less chemically resistant and can be considered equivalent to FR-4 for process compatibility, although they also have lower T_gs, which may not match up with the performance of the substrate material. Table 1 shows the parameters used to desmear the FR-4 material.

		0.3-0.4 mg/cm² wt loss
Hole Swell	Commercially available or butyl caustic mix	2 minutes/175° C
Permanganate	80 g/l $KmnO_4$ 45 g/l NaOH	10 minutes/175° C
Neutralizer	Commercially Available	5 minutes/115° C

Table 1: FR-4

MEPTEC PRESENTS

THE HEAT IS ON 2012

Revolutionary and Evolutionary Innovations in Thermal Management

Monday, March 19, 2012 • Doubletree Hotel • San Jose, CA

Topics will include:

- Trends and Challenges in Thermal Management
- What's New in Thermal Devices
- Thermal Management Issues for 3D Packages
- Thermal Management Issues for Data Centers
- New Thermal Materials for Tablets, Handhelds, etc.
- Energy Efficiency in Thermal Management
- Advanced Research in Thermal Management

REGISTER ONLINE TODAY AT WWW.MEPTEC.ORG

IEEE Semiconductor Thermal Measurement and Management Symposium is an international forum dedicated to the thermal design and characterization of electronic components and systems. The symposium fosters the exchange of knowledge between practitioners and leading experts from industry, as well as the exchange of information on the latest academic and industrial advances in electronics thermal management.

New for 2012!

The SEMI-THERM organizers are pleased for present a Data Center Track with 13 papers and 2 short courses targeted at thermal issues ranging from the component level through the system level all the way up to the complete center level. A panel discussion will be included, bringing together leading members of the data center industry to discuss key industry issues. Collectively, this comprehensive overview of thermal issues at the various levels is a must for participants in the data center industry and for those that wish to learn more about this fast growing industry.

- Pre-conference ***Short-Courses*** from world-class thermal experts.
- ***Over 55 peer-reviewed papers*** presented by the industry's best.
- ***Tracks include*** 3D Packaging, Liquid and Air Cooling, Multi-disciplinary Thermal Management, Thermal Management in Multi-Core Architectures and more...
- ***New!*** Special ***Data Center Cooling*** track with short-course and paper presentations.
- SEMI-THERM Exhibition with ***over 40 vendors plus workshops.***



MEPTEC will again join SEMI-THERM with their **"Heat Is On" Symposium** to be held on Monday, March 19th titled "Revolutionary and Evolutionary Innovations in Thermal Management".

The 28th Annual Thermal Measurement, Modeling & Management Symposium

SEMI-THERM® TWENTY EIGHT 28

March 18-22, 2012
DoubleTree Hotel
San Jose, CA USA

Our Sponsors

IEEE CPMT NIST National Institute of Standards and Technology

SEMI-THERM®

3587 Kifer Road, Santa Clara, CA, USA
Phone: +1 408-642-5170 Fax: +1 408-841-7597

REGISTER ONLINE AT WWW.SEMI-THERM.ORG

In order to resolve the issue with the more chemically resistant resin systems, such as FR-5, BT, and FR-4 materials in the 170°-180°C T_g range, a new approach to sensitizer chemistry was developed. This patented system was engineered through a careful selection of biodegradable solvents with a specific range of solubility parameters. By carefully matching the solubility parameters of the solvent(s) to the resin system, optimum penetration was achieved. Through optimum solvent/resin interaction, the permanganate was more efficient at micro-roughening the topography and removing sufficient amount of resin. Standard, commercially available systems did not meet the strict criteria of topography enhancement. Thus, the new system had to be developed for resin removal on high-performance resins to be successful. It is quite evident that the modified solvent swell system is necessary for adequate texturing and resin removal for some of the more chemically resistant materials. Table 2 also shows the degree of resin weight loss utilizing the standard solvent system and the modified process.

The SEMs below depict the differences on high T_g material with the butyl/caustic conditioner (Figure 1) versus the modified solvent system (Figure 2).

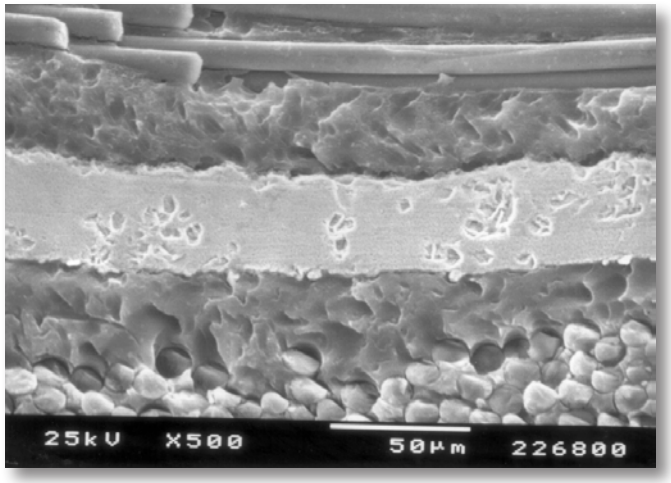


Figure 1: High T_g (170°C) resin with standard butyl/caustic conditioner and permanganate desmear.

A less than optimum topography on any resin system as shown in Figure 1 presents problems with respect to the subsequent metalization process. Smooth topography has less surface area to attract either the catalyst (from the electroless copper process) or the conduc-

		0.1-0.12 mg/cm² wt loss
Hole Swell	Commercially available or butyl/caustic system	2-3 minutes/175° C
Permanganate	80 g/l $KMnO_4$ 45 g/l NaOH	10 minutes/175° C
Neutralizer	Commercially Available	5 minutes/115° C

Table 2: High T_g FR-4, 170°C T_g Standard Solvent System
Note: Weight loss with butyl/caustic system on high T_g material less than optimum.

		0.25-.32 mg/cm² wt loss
Hole Swell	Modified Solvent System	5 minutes/160° C
Permanganate	80 g/l $KmnO_4$ 45 g/l NaOH	15 minutes/175° C
Neutralizer	Commercially Available	5 minutes/115° C

Table 3: Modified Solvent System – High T_g FR-4
Note: The modified solvent system improved penetration on the high T_g material, enabling the permanganate to increase the degree of resin removal and sufficiently roughen the resin to enhance the subsequent adhesion of the metalization layer.

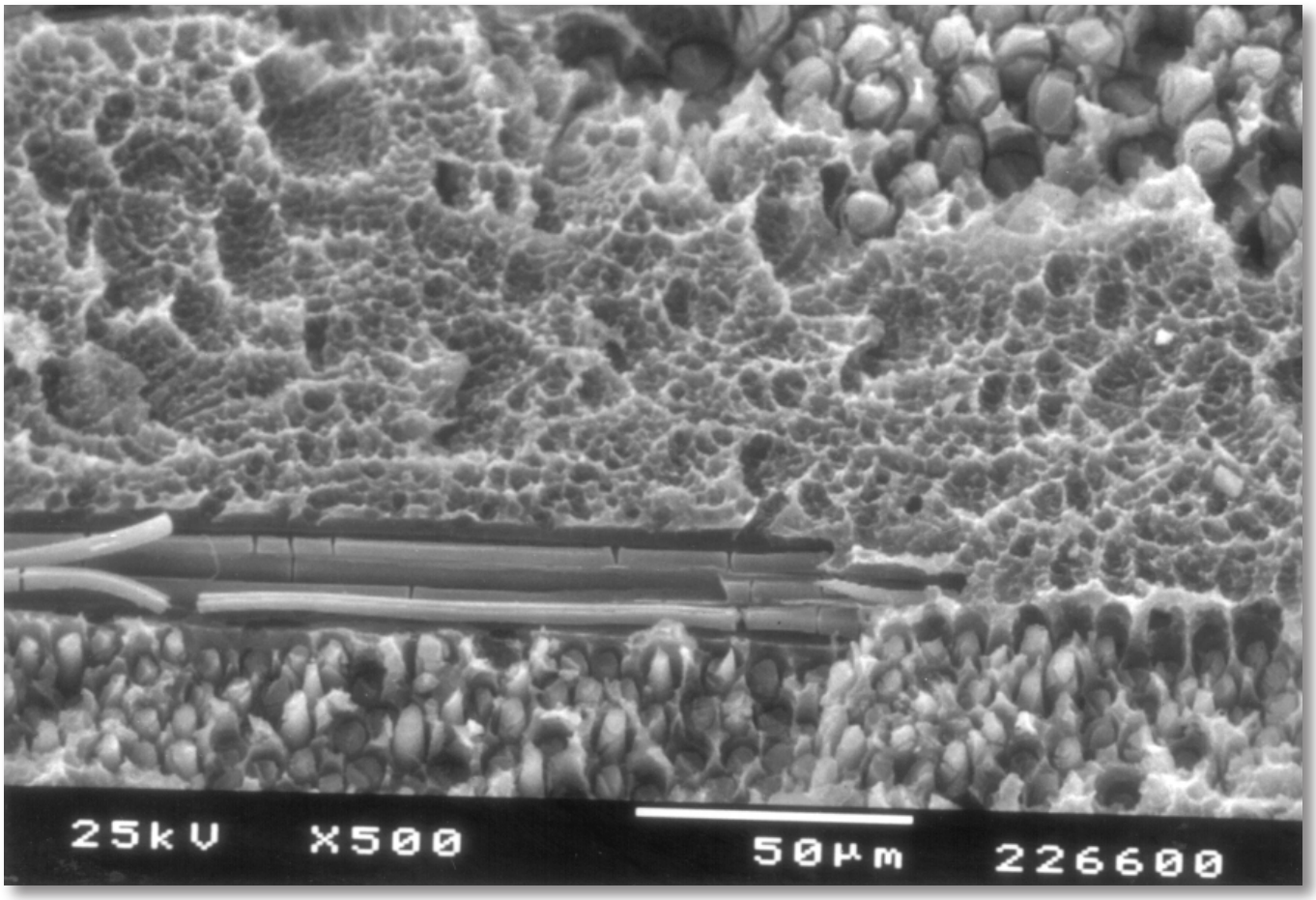


Figure 2: High T_g (170°C) resin with modified solvent system and permanganate desmear.

tive coatings used in the direct metalization processes.

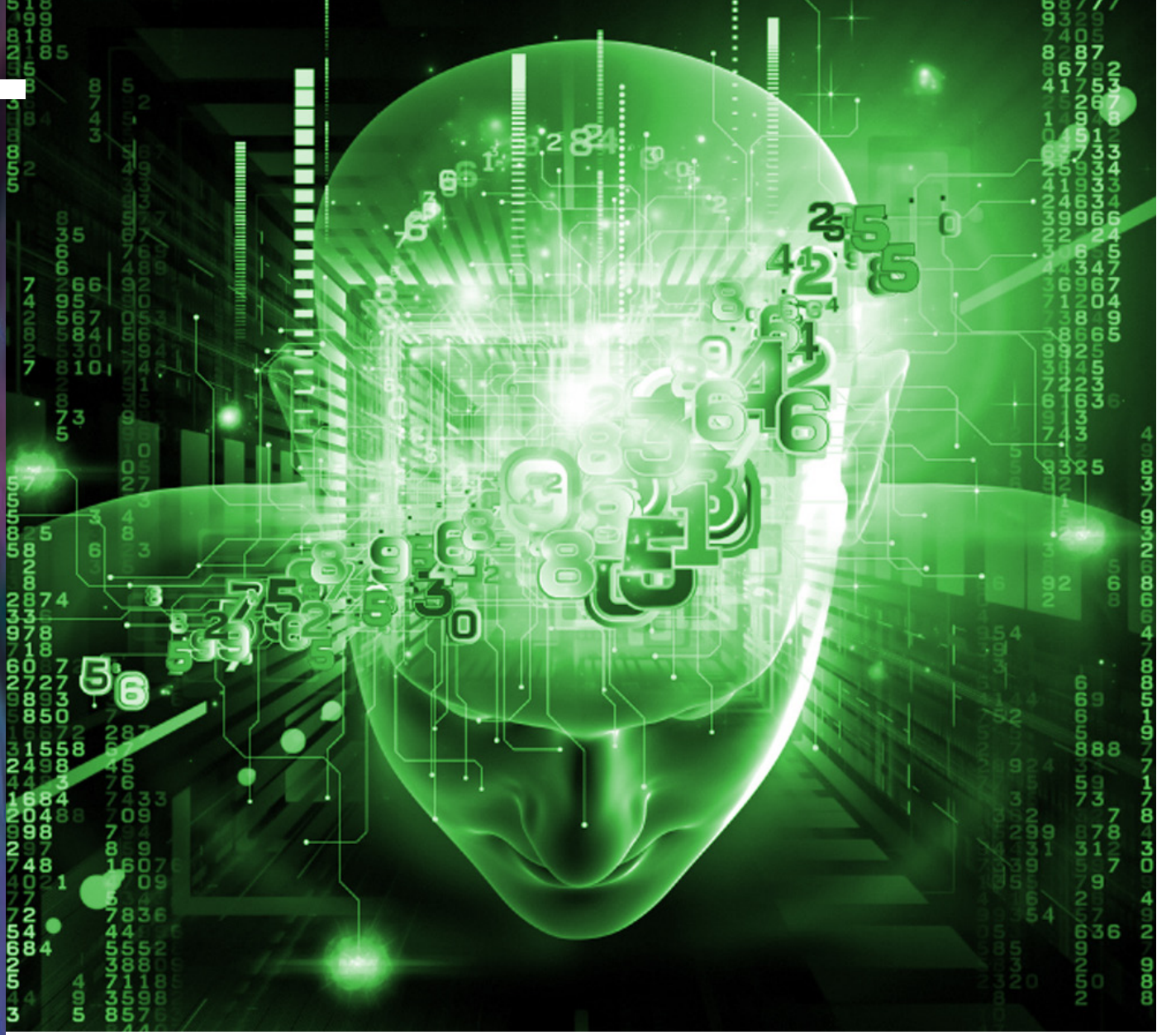
High T_g resins are more chemically resistant to certain desmear systems. The solvent conditioner must have the ability to penetrate the polymer resin matrix and weaken those polymer-polymer bonds. This interaction is sometimes referred to as a swelling action. Once weakened, (or swelled) the bonds are more easily attacked (oxidized) by the alkaline permanganate solution. The oxidation that takes place is responsible for the honeycombed appearance of the resin. However, with less solvent interaction and penetration with the resin, the topography will be less pronounced. Bear in mind that the modified solvent systems do not dissolve the resin. Through the swelling action, the solvent actually prevents further penetration of the solvent.

It stands to reason that the higher T_g materials exhibit lower resin removal rates. However, experience tells us that resin removal is less an issue than actually achieving some degree of topography. The topography is needed to enhance the adsorption and adhesion of either the electroless copper catalyst or in the case of direct metalization, the conductive material required to enable electrodeposition.

In a future column of *Trouble in Your Tank,* I will cover desmear and etchback with different materials. **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.



INDUSTRY STANDARDS

The Glue That Holds Industries Together

by Joseph Fjelstad
VERDANT ELECTRONICS

SUMMARY — Without standards, it is safe to say that collectively, we might still be languishing in the 16th century, where craftsmen made individual products of their own design, each one unique from all others and priced in a similar manner. In today's world, the value of standards should not be underestimated.

Standards are a key element of virtually every industry. They underpin, support and enable the cost-effective production of products of every sort, from cloth to computers. Without standards, it is safe to say that collectively, we might still be languishing in the 16th century, where craftsmen made individual products of their own design, each one unique and different from all others and priced in a similar manner. Thus, to say that standards are important is to grossly understate their value to both industry and mankind.

The roots of standards reach far back, arguably to the invention of language itself, wherein users were required to agree on the meanings of sounds in order to effect reliable communications. Standards around communications continue to this very day and are used in the realm of information transfer protocols for everything from voice to data.

Perhaps the most widely circulated and entertaining example of standards is that which surrounds the selection of the standard gage for train tracks, which according to the example to follow, traces the origin back to two Roman chariot horses' rear ends. Whether or not it is true, it underscores one aspect of standards and that is that once they are set, they are very hard to change. For example, the U.S. is the only nation in the world that has declared itself wedded "'til death do us part" to the English system of units and measures, a system that is, at its core, arbitrary nonsense. Yet, while the rest of the world works in the wonderfully logical and easy to navigate realm of the metric system, we in the U.S. continue to condemn our children to suffer the folly of a road wrongly taken.

That digression aside, the message contained within it is important because standards are in fact a double-edged sword that can make things better, much better, worse, or much worse. One of the problem areas for standards is in the realm of what is called de facto standards. These are the seemingly innocuous adaptations that industries (or societies) make to facilitate some aspect of life or trade. Some of them are, as suggested, relatively harmless, while others can be quite troublesome when it comes to making progress in industry. When one holds tightly to the status quo, all manner of problems can ensue. A delightful late 19th century poem by New Englander Scott Walter Foss (1858-1911) illustrates this point. The poem, "The Calf Path," follows the history of a road built on a weaving path taken by a calf as it wandered home three hundred years before. One applicable stanza is this:

"For men are prone to go it blind,
Along the calf-paths of the mind;
And work away from sun to sun,
To do what other men have done."



The warning is that we can become stuck in our thinking if we pay too much reverence to what others have done. What we think and do should be tested against new information on a regular basis, and those things that we believe to be true because they were handed down by our predecessors must be tested as well. Industry should be dynamic if progress is to be assured.

That said, standards as mentioned at the outset are very important and they always will be. Standards truly are the glue that binds the often disparate elements of all industries. However, standards need to be every bit as dynamic as the industries they serve if they are to help facilitate the purpose of industrial growth and development for which they were originally developed.

Click here to read "The Calf Path" in its entirety. **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 with more than 250 U.S. and international patents issued or pending. He is the author of *Flexible Circuit Technology* and author, co-author or editor of several other books and more than 300 technical papers, articles and columns.

ARTICLE

REGISTERING MULTILAYERS FOR LAMINATION:

The Challenge of Registering New Materials on High-Density Interconnection PCBs

by Victor Lazaro
CHEMPLATE MATERIALS, S.L.
and Paul R. Waldner
MULTILINE INTERNATIONAL EUROPE

SUMMARY — Images on multilayers present many dimensions of problems to solve besides the conventional issues of imaging onto a PCB. Proper registration of inner layers for the lamination process involves material and process expertise, experience and the right equipment.

Many people think of imaging as the first step in the manufacture of a PCB. Imaging on the PCB raw substrate is, however, the culmination of much planning; it is also the realization of only one step in the design of a product, and one aspect of information processed into reality. The connections made (or avoided) on a PCB are the physical embodiments of an electronic engineer's scribbles on a schematic. The designer of the product must bring together many images in order to complete his circuit design. Obviously, if the board is multilayered there are as many images as layers. Then, there are the two solder mask images, a legend, a layer of solderable material added to the pads (a kind of pseudo-image), not to mention the needles of the electrical test machine and the solder paste mask prior to assembly.

One can't forget the holes which combine the layers into a working circuit. A modern circuit board contains multilayered holes. Consider that there are through holes, blind holes and buried holes. When considering buried holes, there are those made by drilling through the inner layer and those made by drilling through a laminated assembly, which is then laminated again into a sequentially pressed multilayer. When considering blind holes, there are those made through sequential lamination, those formed by laser from the surface down and those mechanically drilled — there are even holes created by plasma (dry) etching. All holes are like layers of image that need to be brought into the proper relationship to the images of a circuit to make it work correctly.

The Importance of Registration

When we discuss imaging in PCB production there is nothing more important than registration. The process of registration is no less than the orchestration of all images, features and processes such that each occupies the proper relationship in space with all the other images, features and processes of the PCB. This orchestration begins in the design phase with the design of a data set synchronized to a given Cartesian (X, Y, Z) co-ordinate set of dimensions.

> ***"The practice of using diazo film is slowly fading away, but among small prototype companies in the West and for many under-financed companies in Asia, diazo film is still used despite its disadvantages."***

From data, the image is fed to CAM software where the data is elongated, or foreshortened and fed as an instruction set to plotters, LDI machines, silk-screening machines, electrical test machines, AOI machines, ink-jet machines, drilling and routing machines, registration punches and any other machines responsible for putting features into or onto substrates at specific locations in space, relative to other features of the PCB.

The registration system is there to follow the images through all their changes. Through wet processes, temperature and humidity changes and scores of processes, the registration system makes certain that the holes meet the pads correctly; that the solder-mask openings are where they are supposed to be; that the conductive part of the solderable surface finish doesn't build unwanted connections between pads and tracks, and the legend designates the location of a component correctly.

In short, the registration system makes all the images usable or not.

The early days of PCB manufacture involved much eye alignment of images to other images or circuit features. The use of such alignment is still practiced today to make highly accurate and professional circuits. The practice of using diazo film is slowly fading away, but among small prototype companies in the West and for many under-financed companies in Asia, diazo film is still used despite its disadvantages. The distortions to the images introduced by the very production of the diazo image has made the practice of aligning diazo images to circuit features unworkable for most modern factories.

Among the earliest professional registration systems was a system developed by Multiline Technology in the late 1970s and involved the use of a machine for punching slots in artworks and a punch machine (positioned in the yellow room between the dry film laminator and the exposure process), which placed four slots into the inner layers. The slots in the films were then matched to the slots in the laminate during the exposure process using pins set into or upon the glass of the exposure machine. As the film had been punched to match the drilled holes of an especially created *drilled master panel,* it was assumed that any panels containing slots in the same positions as the film would align to the drilled image that came much later in the process. After imaging, developing, etching, stripping and black oxidation, the inner layers were pinned together for the lamination of the layers into a multilayer panel with the use of the same slots that had been used to register the images to one another. Later, the same slots were used to align the panels on the drilling machines with the use of so-called *hard tooling* or *soft tooling* plates. The drilled holes were, in this way, aligned to the image in the artwork punching process. The relationship between drilled holes and the four-slot tooling holes was created early on and carried through the inner layer process.

The problems of using such a system became apparent as layers became thinner and the personality of every layer could express itself more freely. The material was hardly ever homogeneous. The materials absorbed moisture, were heated and cooled, were subjected to mechanical pressures and stresses which caused localized anomalies, which in turn created localized mechanical differences. Nothing could be counted on to be where it was supposed to

be after going through the complete process of inner layer manufacture.

Manual and automated exposure processes had difficulties working with the four-slot pinning system because of its relative inflexibility to various panel sizes on the same piece of glass. As a result, a series of other pinning solutions such as edge-tooling, T-tooling and L-tooling became popular pinning systems for exposure. Today, most manufacturers with the financial resources to do so align images for exposure using some kind of automatic optical alignment means.

Automatic optical aligning came to the printed circuit industry in the form of the post-etch punch (and the automatic artwork punch) in 1983. The post-etch punch automatically positioned a panel with respect to a four-slot punch using closed circuit video technology, measured the relationship between two targets on the panel and then punched the panel very accurately. Post-etch punching works as a multilayer registration process because it holds the panels flat and mimics the lamination process before punching the panels. During the process of punching, the panel is held in position and has a punch inserted in much the same way as a pin will be inserted later for the lamination process. The process is successful because it reproduces the very act of lamination. Every panel was inspected, measured, aligned and punched with speed, precision, accuracy and careful handling. This leap in technology yielded the following benefits:

“The process is successful because it reproduces the very act of lamination. Every panel was inspected, measured, aligned and punched with speed, precision, accuracy and careful handling.”

- The process freed the registration process from reliance on the eye-hand coordination of workers
- It allowed the creation of process data on the size of images, up to punching process and through all the wet processes. For the first time one could know how every single panel had reacted to the production processes preceding punching
- All process tolerances were automatically split to the middle of the panel and the four slots preserved this split into the lamination process
- Accuracies of inner layer registration to one another improved by one order of magnitude. Previously, layer-to-layer accuracies of 2, 3 and 4 mils (up to 100 microns) were possible; today, punches are specified for layer-to-layer accuracies of 10 microns
- The exposure process became quicker as the layers needed only to be aligned top-to-bottom and no more pinning of layers was required for exposure. The introduction of optically aligned artworks for top-to-bottom alignment made the inner-layer exposure process faster, more reliable, cleaner and cheaper
- Inner layer productivity soared. Today, there are factories that are capable of producing up to 50,000 highly accurate inner layer panels per day using automatic exposure and post etch punching!

Most PCB professionals today agree that registration of multilayers is best done by post-etch punching and pin lamination. Regarding pin lamination (pin-lam), most people understand a process that involves the use of lamination tools (press-plates, caul plates or lamination plates) with holes (most often contained in bushings) that are either round, oval, slotted or slot-shaped (flatted round) separator plates that also contain similar holes and pins held in place by the lamination plates and are used to align the various layers of the to-be-laminated panels. The most popular pin-lam system is the four-slot “multiline” system which guarantees a good center-zero lamination of the panels. The benefits of four-slot pin-lam are well known, but a couple of disadvantages keep the industry on the lookout for a possible alternative. The disadvantages can be summarized in the following two statements:

1. Pin-lam (and particularly four-slot pin-lam!) is a process which allows prepreg epoxy to migrate up, down and out of the pin-holes to

Printed Electronics EUROPE2012

co-located with:

PHOTOVOLTAICS EUROPE 2012

IDTechEx

PART OF A WORLD SERIES OF EVENTS

PRINTED ELECTRONICS APPLICATIONS.

3 – 4 APRIL 2012 / BERLIN, GERMANY.

This event uniquely features organisations discussing their needs and experiences with printed electronics.



25% ATTENDEE EARLY BIRD DISCOUNT

Quote: **PEE-PCB**

WORLD'S LEADING USERS PRESENTING:

Abbott · MARS · METRO GROUP · P&G · Schneider Electric · NOKIA · Fulton · PHILIPS · oxylane · Panasonic ideas for life · SAMSUNG · United Technologies Research Center

and many more

www.PrintedElectronicsEurope.com

cause possible contamination to the outer-layer copper surface, which can then cause imaging defects during outer-layer imaging. In any case, the prepreg flow causes the necessity of expensive and difficult maintenance of the plates and process.

2. Four-slot pin-lam uses more panel area than some manufacturers would like to use because of its requirement of having a slot in the middle of each edge of the panel.

An Introduction to State-of-the-Art Multilayer Registration and Lamination

While most people concede that the best registration is achieved using post-etch punching and four-slot pin-lam, mass-lamination (mass-lam) techniques have become popular for manufacturers of multilayer PCBs who are not seeking the most accurate of layer-to-layer registration systems because of the relatively economical panel utilization and the relatively pristine outer-layer surface offered by mass-lam.

Accepting that there are advantages and disadvantages in each of the two main methods of aligning inner layers into multilayers, the difference is basic. Pin-lam requires tooling pins (round or slot) during the lamination process, whereas, in mass-lam, the pins are not present during lamination because the layers were registered and brought into alignment previously, and maintain alignment through some kind of attachment means.

i. **Pin-lam:** Tooling pins are present during lamination.
ii. **Mass-lam:** The registered multilayer is laminated with no pins during lamination.

Our goal now is to describe the different mass-lam methods presently used in the industry in order to compare them and show the advantages and disadvantages of each. In this way we will be able to offer a roadmap to the best performance for today's high-tech materials and high-density PCBs.

Mass-Lam Registration Systems

In most mass-lam methods, the inner layers of a multilayer PCB are aligned using a high precision template with tooling pins as in the pin-lam method. The templates can use two or three round pins, as well three- or four-slot pins. If the inner layers were properly image scaled, drilled or punched with an accurate image compensation system, the registration template tooling is highly accurate, and the operator is careful during pinning of the inner layers, then it is possible to achieve, before lamination, tight alignment tolerances (within a range of 15µm) from one layer to another, as shown in Figure 1.

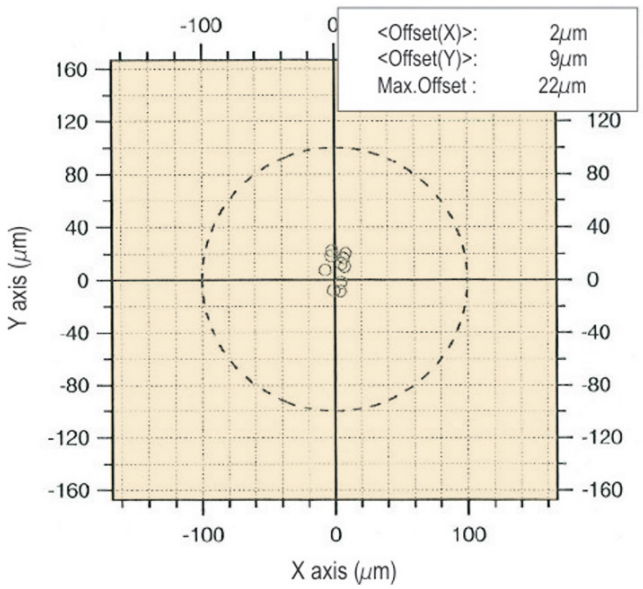


Figure 1: Displacement of L3 to L17 in a 20-layer PCB.

Let's consider, for the purpose of this system comparison, this highly accurate alignment result of around 15µm from layer-to-layer as a constant given. Our consideration of various mass-lam methods will then focus on how the mass-lam attachment means hold the registration through lamination that was achieved at the beginning of the alignment process. We will consider three popular and usable mass-lam systems:

i. Riveting (metal or plastic)
ii. Thermal bonding
iii. InduBond®

Riveting Mass-Lam

This mechanical system is the oldest mass-lam system we will consider. After many years it

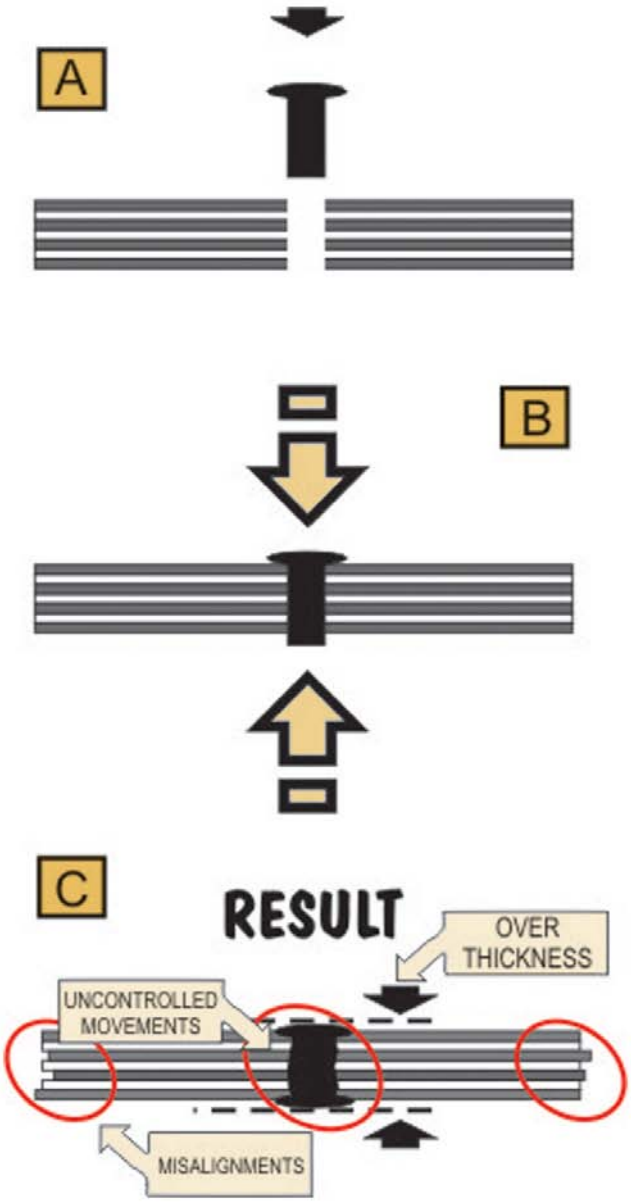


Figure 2: Riveting process and results (a, b, c).

is still used in numerous companies in spite of its inherent limitations. The most popular rivets are made of brass but they can also be made of high-temperature plastic. Basically, the process of riveting begins by aligning the inner layers to one another on a tooling template. Then the layers are drilled as a package (generally a total of four holes or more) along both long edges of the panel, which most often corresponds to the warp direction, including the prepreg insulation layers (Figure 2, a). The rivets are then inserted into the drilled holes and compressed using a high-pressure riveting tool (Figure 2, b). The uncontrolled mechanical expansion of the rivets caused by the force of the riveting tool produces internal stresses and movements. The result is misalignment of the individual inner layers one to another within the stack, especially when the inner layers are thin (Figure 2, c).

Thermal Bonding Mass-Lam

This bonding system principle relies on the prepreg layers bonding to the inner layers by means of a resin polymerization effect. To polymerize the prepreg resin, the thermal bonding system uses high-temperature electric heaters similar to soldering irons, which transfer heat from the outside to the cores of the stack. The heaters are elevated to a temperature of approximately 300°C (572°F). Between three and eight bonding points are typically used on each long edge of the panel (Figure 3, a).

An inherent drawback with the design is the inefficient heat transfer through the substrate material to the internal layers in order to initiate polymerization of the prepreg. Due to space restraints, the bonding heads must be as small as possible (as small as 4mm diameter). The system needs to apply high pressure to overcome the thermal insulation properties of the laminates. This excessive bonding pressure causes deformation around the bonding area (Figure 3, b).

At the start of the bonding cycle, the outer surfaces of the bonding points are subject to extreme temperatures (of around 300°C) while the center of the multilayer remains relatively cold. To assure good polymerization it is required to compromise in order to come to a balance among cycle time, pressure and temperature settings. Obviously, this attempt at balancing of parameters becomes much more critical in high-layer count PCBs (>10L). Two possible negative scenarios can result:

1. Once the center of the thickness achieves the right polymerization temperature, the outer layers (top and bottom surfaces) in touch with the bonding heads can become too hot and possibly burn, causing poor adhesion as a result of carbonization of the epoxy resin.

2. Conversely, the surfaces in contact with

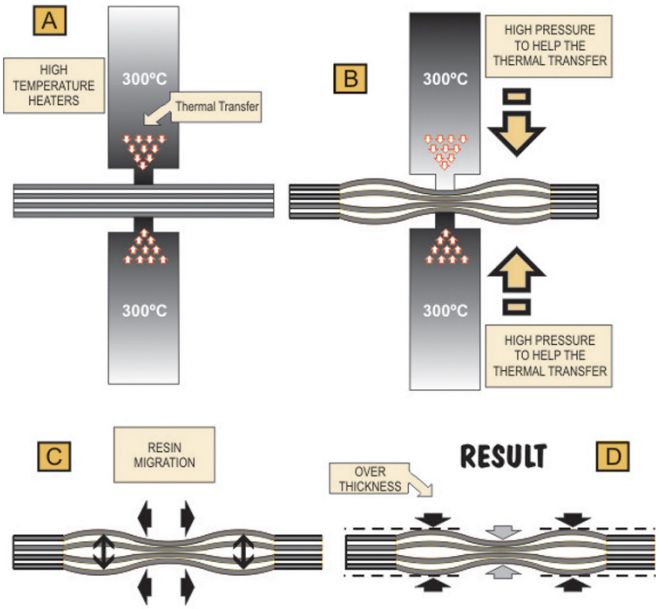


Figure 3: Thermal bonding process and results (a, b, c, d).

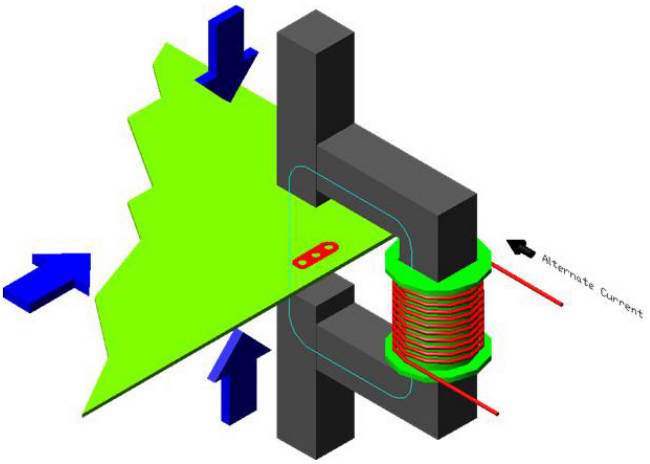


Figure 4: InduBond head draw, function principle.

the heads are at the right temperature while the center of the thickness is too cool, resulting in poor polymerization of the prepreg resin.

The combination of high temperature and compression applied by the small surface area of the heaters can create a crater/volcano effect, causing resin migration away from the bonding points (Figure 3: c, d).

InduBond® System

The InduBond is a registered trademark of a patented technology (US7009157 B2) that utilizes inductive bonding heads that heat up etched copper patterns (heater circuits) on both sides of each layer of the multilayer assembly stack. These etched copper pads are heated by means of a high-frequency magnetic field that passes across the panel thickness producing high-energy eddy currents. Each copper pattern is penetrated by the same magnitude of the magnetic field. The result is that all copper patterns on each of the layers are heated with the same level of energy and polymerization of the resin is uniform throughout the stack. No matter the thickness, the heat is uniformly generated throughout the panel build-up (Figure 4).

The core of the inductive bonding head is made with a specially selected magnetic material (ferrite). The generated high-frequency magnetic field only creates heat through induction of magnetically excitable materials lying perpendicular to the magnetic field, so the inductive bonding heads remain cold at ambient temperature of the environment (Figure 5, a, b).

This technology produces uniform temperature in all thicknesses because each copper pattern heats the resin of every prepreg above and below the inner layers to cure the prepreg in the area of the heater circuit (Figure 5, c). The result is that the inductive bonding point remains a flat bonding point, maintaining the same thickness as the complete multilayer circuit after pressing.

The created bonding points are robust and flat. They are capable of withstanding the expansion and contraction of the materials during the hot-press cycle. Each bonded point provides maximum assurance to the linear movement of all layers in a multilayer stack. An added benefit is minimal internal tensions (stress) that cause warping or other deformations of the panel during pressing and possible misalignments between inner layers (Figure 5, d).

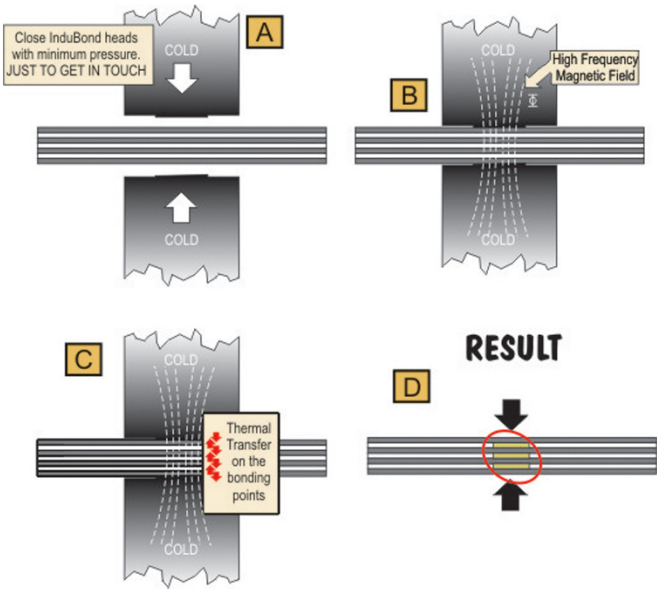


Figure 5: Inductive bonding (InduBond®) process and results (a, b, c, d).

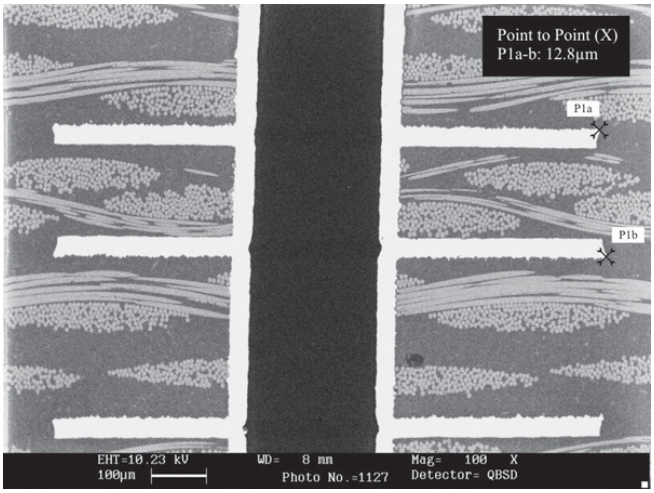


Figure 6: Layer-to-layer alignment on a random sample of 28L PCB on InduBond® process.

Basically, all the mass-lam methods attempt to hold the registration of layers achieved in the laying up on a precision template without the requirement to use pins in the press. The goal is to achieve the same registration results achieved by a four-slot pin-lam system without the disadvantages of the pin-lam system. All of the systems discussed here have their pros and cons, but the one that delivers similar registration results to pin-lam while avoiding pin-lam disadvantages is the InduBond® technology (Figure 6). Note the following summarized results of InduBond® processing:

- Tight alignment tolerances similar to pin-lam as a result of using the same tooling for pinning.
- Good registration and flat panel during lamination process similar to pin-lam.
- Robust, inductive bonding points that hold the registration during handling and pressing, similar to pin-lam.
- Robust, inductive bonding points that let the different mechanical expansion of each layer in their natural axis as pin-lam.

CCD Camera Alignment (Pinless Mass-Lam)

The pinless mass-lam concept is a very attractive system for manufacturers of PCBs who wish to implement a mass-lam concept because it eliminates the need for layers to be punched or drilled for alignment purposes. The process aligns every inner layer by means of the image, using CCD cameras to measure, compensate any error and align the layer. Once the system aligns all the layers to one another, a system to mechanically maintain the alignment is still required. It is possible to use any of the mass-lam attachment methods to maintain alignment to and through the lamination press. An important consideration is whether the pinless mass-lam process truly delivers pin-lam-like registration results with all the advantages of mass-lam.

The existing pinless mass-lam technology is based on a bottom vacuum plate that holds the first inner layer (L1) of the stack. A 2- or 4-CCD camera system measure the fiducials' distance position of this first layer and maintains this positional relationship in memory. The first layer (L1) is the reference layer for the rest. The successive layers (L1+n) are then placed on the alignment unit where it is picked up by a top vacuum plate, which is typically smaller than

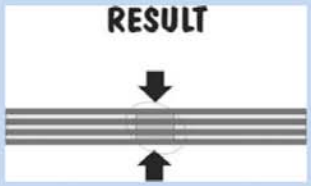
	Advantages	Disadvantages
Riveting	•Relatively fast operation. 	•Relies on a non-uniform mechanical expansion of each rivet, which increases the risk of inaccurate alignment between the layers. •The Holes for rivet placing are made after laid up, this process can result in copper or substrate debris being trapped inside the stack. •The rivet is a consumable item and different lengths are required for each panel thickness. •Longer rivets increase the risk of misalignment for high layer count multi layers. •The edges of the rivet heads (top and bottom) can cut and break thin material outer layers causing them to move independently of the inner layers. •The rivets over thickness can cause lamination press plates damages and not surface pressure balance.
Thermal Bonding	•Simple machine and less costly compared to other systems. •No consumables are required like the rivets. 	•Uncontrolled process for different thicknesses. •The over thickness points will not permit good pressure balance transfer to the surfaces of the stack during the press cycle. •The generated over-thickness around all bonding points can damage the lamination press plates on the press. •Bonding points can break during lamination due to over pressure on theraised areas of the bonding points, resulting in layers shiftment misalignment . •The non lineal or controlled polymerization in the vicinity of the bonding points cannot guarantee uniform movement of each inner layer during the press cycle. •Very unreliable system for thick panels due to the uncontroling polymerization process •The thicker the multilayer stack the deeper the crater/volcano effect is. •High energy cost, all heater are allways switched on at 300ºC (572ºF).
InduBond ®	•No thickness limit for high quality bonding. (The mechanical limit is about 30mm, however the template pin height is typically 10mm). •Flat finishing surface. No over thickness points provide cost savings due to increased life expectancy of separator press plates (if the stacks are flat, no press plates damage will result) •Flat finishing surface. No over thickness points provideMAXIMUM pressure balance transference to all layer surfaces within the stack during the press cycle. (Lineal resin flow, reduces internal stresses of the material helping to minimize banding effects). •Highly stable bonding points maintain their integrity during the press cycle. (No misalignment caused by fractures or breakage). •Polymerization control provides flat and robust bonding points that accommodate the different movements of the inner layers during press cycle without losing the registration. (reliable process and therefore less scrap). •The induction method allows 2 or 3 multilayer assemblies to be bonded simultaneously. The assemblies can be registered in the same pin tool. •Easy set up by touch screen for different multilayer constructions or different materials. Similar to a lamination press profile set up •Energy savings, the InduBond® heads (total 300W/each) are always off until the bonding cycle is initiated (standard bonding cycle from 30 to 45 seconds). •No need consumables like rivets and is very clean process. •Template tooling can be customize, 4 slot or 2 round pins are available.	•The machine is a little more expensive, but easy investment recover by the money saved by increased press plate longevity, energy savings or toolings maintenance. •The cycle time for a single assembly is a little bit longer that the rivet system. Tipically about 60" for a standard 10 layers PCB. 

Table 1: Mass-lam systems comparison of pros and cons.

the inner layer. The CCD camera alignment system measures the fiducial targets of the new layer (e.g., L2) and compares the relative position of targets to the memorized data from the reference fiducial positions of L1. The unit calculates the best fit position that guaranties the best alignment between L1 and L2 and the motorized top plate moves according to the calculated results. Once the top plate achieves the best-fit position, the top plate moves the layer down into contact with L1, maintaining a holding pressure while a side mechanical clamping system clamps (L2) against the reference layer (L1). The process repeats for each inner layer to be added (Figure 7).

In theory, the concept seems wonderful but it is well known how fragile and mechanically unstable the very thin cores of high-layer count multilayers are. It is extremely important to assure that the thin cores remain flat while being measured, aligned and registered to the stack using rivets, heat or inductive bonding. The lack of flat handling of the layers, combined with the repeated mechanical disruptions of clamping and unclamping the layers, disturb the registration process. Errors and additional tolerances can be attributed as follows:

1. The equipment is measuring the targets while the new inner layer is not completely flat. The thinner the inner layer, the bigger the error of measuring can be (Figure 8, a).

2. Once the system considers that the alignment between the reference layer and the new

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

one is good, a lateral clamping system moves in to clamp both long edges of the panel in order to fix the position of the layer with respect to the already aligned layers. During the clamping process, the thin cores are not pressed completely flat. This lack of flatness, along with the movement of the clamps in and out, produce uncontrollable and random small movements of the cores (Figures 8, b,c).

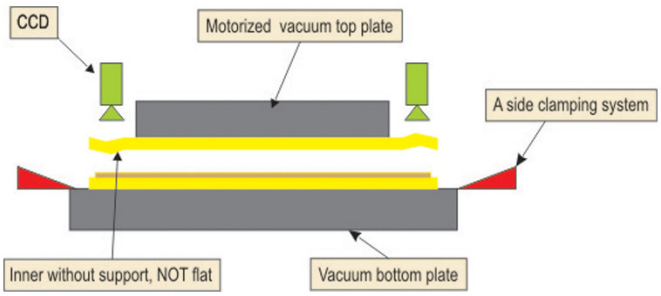
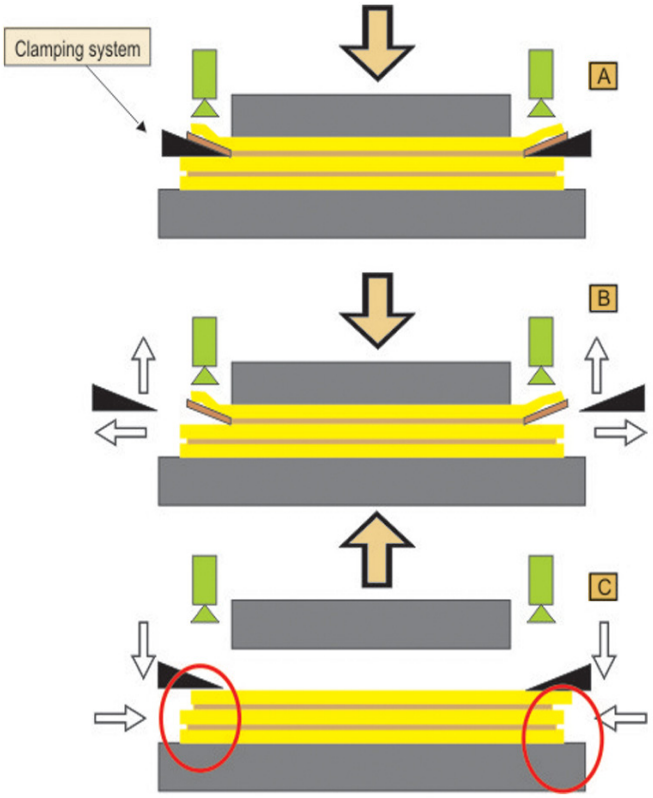


Figure 7: Pinless functioning block diagram.



Figures 8: Pinless functioning, clamping sequence (a, b, c).

3. This operation of clamping and unclamping the stack is being done with the addition of every new layer. It is easy to understand that thin cores without enough internal rigidity, working under these conditions, will succumb to random movements while the wish of the industry is to have an accurate and, moreover, repeatable results to achieve a reliable and controllable process.

4. This sequential laying up, waiting for alignment, unclamping and clamping needs time to proceed through all the routines and steps. Productivity of such a system is low when compared to punching and drilling layers and pinning them to a template in a typical mass-lam process.

As per our tests and analyses we have made in cooperation with several PCB companies, we conclude that the pinless systems of today, in addition to low productivity, can only achieve alignments of layer-to-layer in the range of 60µm when the equipment is handling thin cores. **PCB**



Victor Lazaro is the inventor of the inductive bonding technology (InduBond®) applied to the inner layer registration process as well as the procedure for bonding by induction the constituent layers of a multilayer PCB. He has worked at Chemplate Materials, S.L. since 2001. Contact Lazaro at v.lazaro@chemplate.com.



Paul R. Waldner has spent most of his professional life in Germany and considers himself a proud citizen of the world of printed circuits, which he entered via a summer job as a circuit designer in 1974. Later responsibilities included the development of several medical devices and the development of processes and equipment used in the production of advanced printed circuits. Waldner is the owner and managing director of Multiline International Europe and may be reached at Paul.Waldner@mie.de.

Association Connecting Electronics Industries

IPC



IPC Flexible Circuits Conference

June 12–14 • Irvine, California | www.ipc.org/flex-conference

Explore the amazing potential of **flexible circuits technology:** to manage high temperatures, reduce weight and space, save on assembly.

Learn more about methodologies with **half-day workshop sessions** on **June 12.**

Presentations June 13–14 from an outstanding roster of speakers will provide practical information on a range of technical topics, along with the business outlook for this sector. **A special focus on printed electronics is included.**

Network in a professional environment during the afternoon luncheon and evening reception.

REGISTER NOW

AGENDA HIGHLIGHTS

EXHIBITOR OPPORTUNITIES

Hospitality Sponsor

Polar

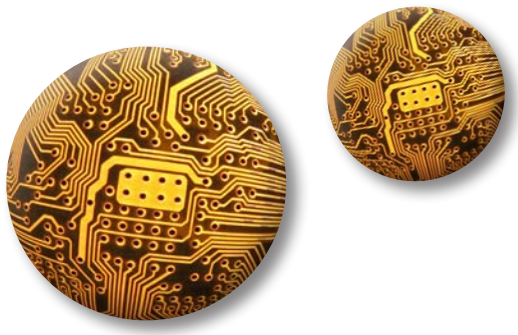
Media Sponsors

PCB007 the real-time online magazine

MILAERO007

COLUMN

Advances in Copper Plating, Part B



By Karl Dietz
KARL DIETZ CONSULTING, LLC

Summary — Continuing on from last month's Tech Talk column (Advances in Copper Plating, Part A), this second column in a two-part series on copper plating for electronic applications examines conveyorized plating modules and pulse plating.

Acid copper (electroplating) for electronic applications has seen many advances over the decades which include the introduction of the high through bath with lower copper and higher acid concentration, the development of proprietary organic additives for improved leveling, grain refining, and ductility, improved bath control methods such as CVS and many more. Last month's Tech Talk column (Advances in Copper Plating, Part A) began a two-part series dealing with more recent innovations and evolutionary improvements in copper plating by introducing the topics of inert anodes and plating "shut" blind vias and through-holes. The order in which the topics are presented is somewhat arbitrary and not meant to suggest order of importance.

Conveyorized Plating Modules

There are a number of variables to be controlled to minimize plating non-uniformity:

- Improve primary current distribution (shielding, anode/cathode sizing)
- Controlling throwing power (plating additives, current density, Cu/acid ratio)
- Anode size control (use of inert anodes)
- Bath movement (use of eductors), and
- Uniform plating cell environment (use of conveyorized platers)

The latter addresses plating non-uniformity due to the position of a particular board location relative to the plating cell environment or cell geometry in plating baths with stationary plating racks. Plating differences due to location can be smoothed out in the more uniform plating cell environment of so called continuous or conveyorized platers. There are continuous plating lines that convey the boards horizontally; others convey the boards vertically. Such systems had



Figure 1: Vertical continuous plating (VCP) equipment for panel and pattern plating.



Figure 2: Horizontal plater example: Atotech's horizontal pulse plating system.

SUBSCRIBE
to our PUBLICATIONS

Get the industry's top stories and most original content in an easy-to-read, easy-to-navigate format. Our Daily Newsletter, the offering from PCB007 and SMT Online, provides daily updates of the industry's late-breaking news. Inside Design covers PCB design and design engineering each week, while Flex007 provides the industry's only flex-focused content, including news on the growing printed electronics technology. Mil/Aero Weekly brings you relevant content for the Military and Aerospace Electronics Industries.

ARTICLES

R&D news technology

CLICK TO PREVIEW THE I·CONNECT007 Daily News

MIL AERO007 CLICK TO PREVIEW

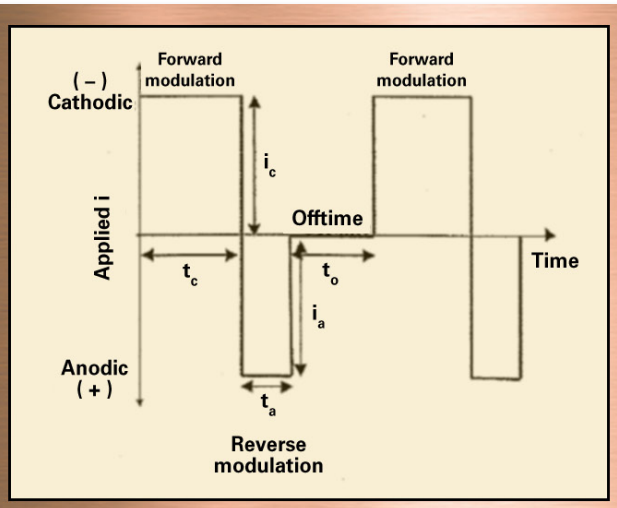
CLICK TO PREVIEW INSIDEDESIGN

FLEX007 CLICK TO PREVIEW

SUBSCRIBE NOW!

I·CONNECT007

CONNECTING THE GLOBAL ELECTRONICS SUPPLY CHAIN



T_o = off-time
T_c = cathodic on-time
T_a = anodic on-time
I_c = cathodic current
I_a = anodic current
Cathodic "duty cycle" =
t_c/total pulse time

Figure 3: Generic PPR Waveform.

to develop sophisticated transport mechanisms (clamp and release), especially for thin boards. Vertical continuous platers have become more popular in recent years for plating more complex boards such as HDI boards.

Figure 1 shows such a vertical continuous plater (Example: VCP from Process Automation International Ltd. (PAL), Hong Kong, in alliance with Marunaka Kogyo Co. Ltd., Japan, and Applied Equipment Ltd., Taiwan.

Proponents of this type point out the following advantages vs. horizontal conveyorized platers: better side-to-side uniformity, less air entrapment in microvias, lower cost, and smaller footprint.

Figure 2 shows a popular horizontal plating module (source: Atotech).

Pulse Plating

Pulse plating (specifically, periodic reverse pulse plating) has started to make inroads into acid copper plating about twenty years ago, but has really caught on in recent years as rectifiers became more affordable and reliable and as microvia and through-hole plating has become more challenging due to smaller via diameters and higher aspect ratios. Figure 3 shows a typical pulse plating cycle with its different segments and gives definitions of the different current flows and time segments of the cycle.

It is interesting to note that the duty cycle, i.e., the actual plating time cycle is interrupted by reverse plating (or deplating) and off-time, which altogether would suggest that plating would be slower compared to conventional DC plating in which all the time is dedicated to plating (or duty cycle). However, pulse plating typically results in faster plating because it interferes with the formation of a polarized layer on the surface of the board, thus helping the faster replenishment of copper ions. The following are the key features of pulse plating when compared to DC plating:

- Better surface uniformity
- Better throwing power (more uniform through-hole/microvia plating)
- Faster plating (even though there are short time intervals in each pulse cycle when no plating happens; the pulse changes the diffusion layer and enhances copper ion replenishment at the board surface)
- Organic additive systems had to be tailored to pulse plating
- Rectifiers are more expensive than DC rectifiers
- Cell geometry is very important **PCB**



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

Most-Read Supplier/New Product Highlights This Month



Rogers Plans N.A. Launch of MCL-FX-2 Laminates

Visitors to the Rogers booth at IPC APEX EXPO can learn about the company's plans for the North American quick-turn launch of MCL-FX-2 laminates. These high-performance circuit materials are manufactured with prepregs supplied from Hitachi Chemical Co., Ltd. and are ideal for high-speed digital applications operating at speeds greater than 1 Gb/s.

Rainbow Technology Launches Alternative to LDI Mfg

The Rainbow Process represents a breakthrough for the electronics industry by incorporating coating, imaging, and developing of PCBs in one compact, automated unit, which promises to make board production faster, easier and more profitable.

Park Electrochemical to Shut Down PACM Facility

Park Electrochemical's PACM facility will be closing its operations at the end of April 2012 after the completion of the transfer of its aerospace composite materials manufacturing activities to the Park Aerospace Technologies facility in Newton, Kansas.

Endicott Interconnect Appoints Eltek Sales Partner in Israel

Endicott Interconnect Technologies has extended its sales network with the appointment of Eltek as its sales partner for Israel, thereby strengthening the company's infrastructure and presence in the region.

Ventec Europe Invests in New Equipment

Ventec-UK has taken to delivery of a TA Q20 DSC (Differential Scanning Calorimeter). This equipment allows the company to conduct degree of cure and glass transition temperature (T_g) measurements to verify customer lamination processes.

Spirit Circuits Invests in Viking's CMM

Spirit Circuits, Waterlooville, UK has ordered from Viking Test Ltd. an automated Coordinate Measurement Machine (CMM).

Frontline's InSolver Enables Accurate Impedance Modeling

Powered by Mentor Graphics HyperLynx technology, Frontline's InSolver is a cost-effective PCB impedance field solver that enables fast and accurate stackup and impedance modeling for a wide range of PCB structures.

Polar Unveils New Atlas Si - GHz PCB Test

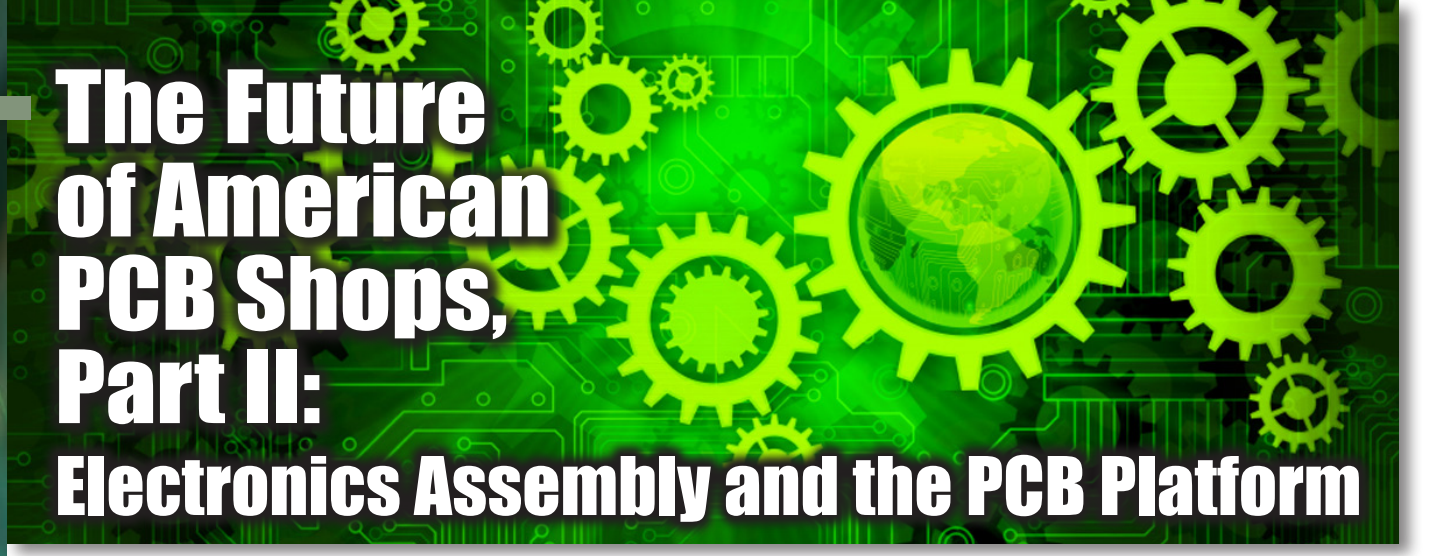
Polar Atlas software enables PCB fabricators to test PCB transmission line losses in a tightly controlled production environment. Atlas employs mathematical processing techniques to extract frequency dependent loss characteristics when used in conjunction with suitably designed test coupons and Atlas precision GHz probe assemblies.

Dow Electronic Materials Sales Down in Q4

Sales in Dow Electronic Materials decreased modestly in the quarter, driven by lower demand in Interconnect Technologies and Semiconductor Technologies.

GCT Unveils New Diamond-Coated Tools

GCT GmbH in Weingarten introduces two new diamond-coated tools: End-mill-type 1328 with two flutes and centre cutting and chamfering cutter-type 138x with single flute.



The Future of American PCB Shops, Part II: Electronics Assembly and the PCB Platform

by Harvey Miller

SUMMARY — The title of this article is borrowed from a TechNet thread, and in its original form, asked the question, "What is the future of electronics assembly?" For the purposes of this article, I'm amending the question to include "… and their printed circuit board platform?"

Before we can answer the two-pronged query summarized above, and predict the futures of both electronics assembly and the PCB platform, it is necessary to ask and answer a few other compelling questions about the surface mount technology that dominates today's electronic assembly. According to the 2011 ITRS Roadmap, IC pin density will increase as much as 15% by 2014. Therefore, can SMT accommodate that increase?

The answer is maybe, but with ample rework and problematic yields. Between 2014 and 2020, pin density will increase as much as 40%, according to the ITRS Roadmap. In this case, the question is whether solder paste can be stenciled and reflowed on < 0.15 mm centers. Here, the answer is yes, but something has to give, and it is probably going to be solder.

To be clear, we are talking about mainstream applications. After all, through-hole technology continues, though long displaced by SMT in mainstream use. Alas, SMT will go on as well, after it loses dominance. Finally, even solder and solder paste will find some usage.

However, what prompted that disquieting TechNet question? I think that many people on today's front lines of electronic assembly, the IPC TechNet thread-posters, are troubled about SMT limits and yield-threatening problems in today's electronic production environment.

Anyone who peruses TechNet threads written by these courageous front-line fighters must be impressed with the titles that continually point to the problems of solder paste, both eutectic and lead free, regardless of its composition. Recently, a TechNetter asked the uncomfortable question that gave us our title; it was uncomfortable because no one likes to contemplate a paradigm shift that replaces the way he or she makes a living.

No other SMT issue threatens its future dominance so much as the density limitations of solder joints. As the ITRS Roadmap clearly shows, stenciling and reflowing quarter-mil separated solder joint centers today is just the horrendous beginning of tomorrow's assembly problems.

Let's pause here for a quick walk down the electronic industry's memory lane, in order to improve our vision of its future.

There is No Moore's Law for Printed Circuits (No, it's Not HDI)

As most know, this law is named after Intel co-founder Gordon E. Moore, who described the trend in 1965, which was just six years after Robert Noyce filed the first patent for a monolithic integrated circuit. It states that transistor count grows exponentially, doubling every two years on the IC chip. For 60 years, this empirical law has driven electronic hardware to more than a trillion dollars of annual production value: the world's largest industry.

Also for 60 years, system interconnection has been in a perpetual struggle to catch up to ICs in density. Forget equivalent cost reduction! According to www.icknowledge.com, it's a 35% per year price reduction for ICs, yet IC percentage of electronic equipment value, measured as

"There is a powerful movement afoot in the IC packaging world to resurrect KGD, driven by 3D die stacking needs."

a percentage of revenue is about 25%, or more than $300B ICs in $1.2 trillion of electronic hardware.

Back to system interconnection's struggle to catch up to IC density. One TechNet poster mentioned Joe Fjelstad's Occam process. Could this be the ghost lingering in the background? Here's how it might fit into the puzzle.

First, with embedded components, using proven technology from 3M, Oak-Mitsui, Ticer, DuPont, Dow, Ohmega, etc., most solder joints would be replaced by plated vias, down to the lands. On the other hand, Fjelstad proposes a more general solution that would include all leaded, fully tested active and passive components as candidates. Others are looking at bare die for the active components, or known good die (KGD). There is a powerful movement afoot in the IC packaging world to resurrect KGD, driven by 3D die stacking needs. How opportune for the next generation of all electronic assembly! The financial benefit for better metrology will be forthcoming.

The following excerpt is from a MEPTEC sponsored presentation, November 10, 2011, by leading IC technologists.

"Joe Fjelstad's Occam process would go further, separating the component mounting substrate from the interconnection substrate. All or most component connections would be made in the now underutilized third dimension with plated copper. Think of the enhanced freedom—no circuit lines constraining component placement density. That is just the beginning—reduced crosstalk and potentially much greater density, simpler, easier circuit fabrication and assembly. High-speed design would be vastly simplified. Conductive anodic filament leakage problems would vanish, along with glass because components, free of interconnect constraints, would be mounted on anodized aluminum. Thermal dissipation problems would be gone."

The implications are that fabrication and assembly would be co-located, shortening the supply chain. Pick-and-place equipment would still be needed, but paste printers and reflow ovens would go away—in the mainstream, along with solder paste. Or, stencil printers might be used to deposit adhesive, to replace the role of solder. Laser-drilled vias and all copper interconnects would meet the density challenge.

Most assembly engineers and other personnel would be needed in the merged fab-assembly operations. It all offers new opportunities for fabricators, assemblers, and for the U.S., more efficient, productive manufacturing. Intel once foreshadowed this scenario with the BBUL package, and then shelved it. The time is coming to resurrect it.

Since its inception in 1992, a basic premise of the ITRS Roadmap has been that continued scaling of electronics would further reduce the cost per function (~25–29% per year) and promote market growth for integrated circuits, which historically have averaged ~17% per year, but whose growth has slowed in more recent history). Thus, the Roadmap has been put together in the spirit of a challenge, essentially asking, "What technical capabilities need to be developed for the industry to stay on Moore's Law and the other trends?"

The objective of the ITRS Roadmap is to ensure cost-effective advancements in the performance of the integrated circuit and the products that employ such devices, thereby continuing the health and success of this industry. **PCB**

References:

1. "Update: Intel to build fab for 14-nm chips," Feb 18, 2011, http://www.eetimes.com/
2. ITRS, http://www.itrs.net/



Harvey Miller is an I-Connect007 Guest Editor and has been watching the printed circuit industry for more than 30 years as an economist at the University of Michigan, analyst, and database creator. He began his electronics career as a components engineer for computer and telecom OEMs, Burroughs and GTE among them. At present he's putting it all together, generating powerful marketing database tools for the global PCB industry at www.FabfileOnline.com. To contact Miller, click here.

CPCA 2012

The 21st China International Electronic Circuits Exhibition

China Printed Circuit Association and Shanghai Ying Zhan Business Service Co., Ltd present the 21st China International Electronic Circuits Exhibition, to be held March 13-15, 2012 at Shanghai World Expo Exhibition Center.

Covering an area of 25,000 square meters, the show is expecting more than 500 exhibitors from Asia, North America, Europe, Israel, India, Malaysia and more. More than 50,000 professional visitors and buyer groups are expected to attend.

During the exhibition, the World Electronic Circuits Council common (WECC) will present the "International Forum on Information Technology PCB," which will include international experts delivering speeches and discussing new trends in technology.

Also on the schedule is the 2012 International Water Treatment & Cleanrooms Exhibition, providing comprehensive and professional environmental water treatment and clean technology solutions to PCB manufacturers. Other advanced technology and products to be exhibited at the show include PCB manufacturing equipment, raw materials and chemicals and electronic assembly equipment.

Exhibiting and Visiting Hours
9 a.m. - 4:30 p.m., March 13-14
9 a.m. - 4 p.m., March 15

For more information about the show:
Shanghai Ying Zhan Business Service Co., Ltd.
Unit D, 23/F, Hui-Jia Building, No. 41, Cao Xi Road (N) Shanghai 200030, China
Tel: +86-21-54900077
Fax: +86-21-54904537

E-mail:
cpcashow@ying-zhan.com
yinzhanc@online.sh.cn

Website:
http://www.ying-zhan.com

KEYNOTE SPEECHES

Global Electronics and PCB Industry Growth Outlook

Shiuh-Kao Chiang, Prismark Partners LLC

Summary:

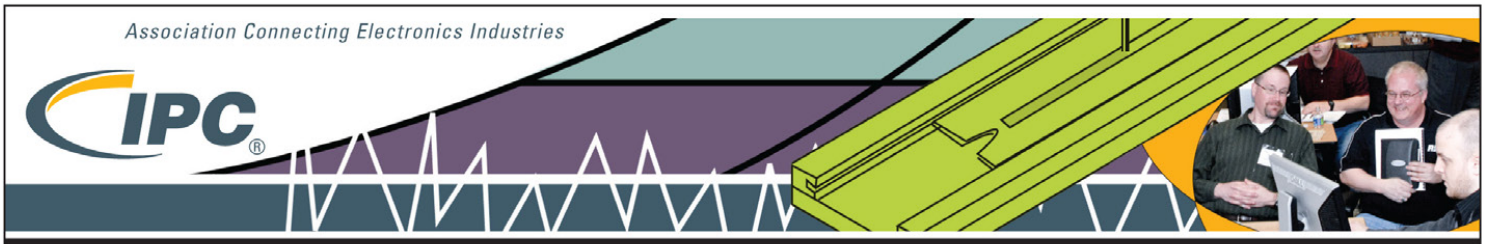
The global electronics and PCB industries have gone through dramatic changes over the past decade. Most electronics assembly and PCB production have migrated to Asia, especially China, because of continued price pressure, intensified market competition, and relatively low technology entry barriers. Today, the PCB value share in the electronics market has reached the bottom. Rising material prices and steadily increasing labor costs have squeezed the margins and forced the PCB industry to re-examine its business practices. Can the PCB industry continue to absorb selling price deflation without hurting future growth? Will China continue to dominate the manufacturing landscape even with double-digit growth of labor costs? The presentation will answer these questions, as well as discuss how the emergence of cloud computing and intelligent portable devices is likely to further change the PCB industry product mix and technology needs in the next few years.



Shiuh-Kao Chiang
Managing Partner
Prismark Partners, LLC.

Association Connecting Electronics Industries

IPC®



TEST AND INSPECTION CONFERENCE

May 15–17 • Orange County, California

Process Improvement — Improved Yields — Cost Savings

Electronics product and manufacturing trends have continued to challenge both the users and providers of test and inspection technologies. Attend the IPC TEST & INSPECTION CONFERENCE and learn new technologies to improve yields that translate to cost savings.

REGISTER

AGENDA HIGHLIGHTS **EXHIBITOR OPPORTUNITIES**

Media Sponsors

SMT® MAGAZINE

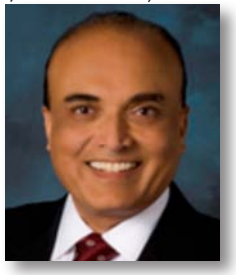
AN I·CONNECT007 PUBLICATION

Flex as an Enabler for 3D Packaging Solutions

Jay Desai, MFLEX

Summary:

Recent advances in electronics have led to the cost-effective miniaturization of formerly large-scale systems, contributing to an incredible boom in mobile technologies. The latest consumer products blend previously unrelated and separate features into a single device. In this environment, OEMs are challenged with fitting an increasing number of components and features into progressively smaller form factors. The best way for OEMs to incorporate these additional features into smaller spaces is through the use of flex circuits as a three-dimensional packaging solution. The presentation will also highlight how flex circuits can be applied in other markets to make devices thinner, smaller and lighter. Traditionally, flex was utilized in devices featuring moving parts; however, flex can now be used to achieve device miniaturization and provide advantages including increased portability, functionality and usability.



Jay Desai
Director of Marketing
MFLEX

Developments in Printed Circuit Technology, Roadmaps and Standards

David W. Bergman, IPC

Introduction

Recent releases of several industry documents are providing a fascinating insight on both near and longer term trends in PCB technology. The first is a release of IPC's 2011 PCB Technology Trend Study, comprised of survey data from some of the world's leading PCB manufacturers. Desai will introduce this study and identify some unique industry trends. The next insight comes from two recently released industry roadmaps: IPC's International Technology Roadmap for Electronic Interconnections, and the Organic Electronics Association's Roadmap for Organic and Printed Electronics. Both roadmaps contain some unique perspectives on the future of printed circuit technology, to be highlighted in this presentation. Finally, the presentation will identify several new test methods and standards efforts that are being undertaken to help aid the implementation of advanced technologies.



David Bergman
Vice President of
International Relations, IPC

Introduction to Printed Electronics

Prof. Dr.-Ing. Werner Jillek, Georg-Simon-Ohm University of Applied Sciences

Summary:

This presentation introduces printing techniques in PCB production, including screen printing of thick film hybrids, manufacturing of devices with conductive polymers, and composition of inks containing metallic nanoparticles. It also introduces inkjet printing of conductor lines on PCBs, and conductivity as a function of sintering methods and parameters. Additional topics include various patterning techniques with respect to conductor line width and thickness, embedding of components, reliability tests of printed electronics, demonstrators for functional inkjet and aerosol printing. Examples for first applications of printed electronics and future prospects of printed electronics versus, or in combination with PCBs manufactured in standard technologies are introduced as well.



Prof. Dr.-Ing. Werner Jillek
Georg-Simon-Ohm
University of Applied Sciences

RealTime™ with... IPC

EXCLUSIVE EVENT COVERAGE





With the help of our generous sponsors, we gave away FREE iPads to numerous lucky attendees this year on the show floor.

I·CONNECT007
CONNECTING THE GLOBAL ELECTRONICS SUPPLY CHAIN

APEX EXPO IPC

We would like to thank all of our *Real Time with...IPC* sponsors.

SMT MAGAZINE

PCB007

PCB007 PRESENTS the pcb magazine

PCB DESIGN007



TOP TEN

Most-Read News Highlights from PCB007 this Month

① IPC Releases PCB Industry Results for December 2011

"North American PCB sales in December increased over the prior month and the book-to-bill ratio strengthened, reaching parity for the first time since last summer," said Sharon Starr, IPC Market Research Director. "North American PCB sales ended the year down 1.9% over 2010, but current indicators point to a resumption of modest growth in 2012."

② IPC-2581: Consortium Leads the Way

Founded by a variety of suppliers across the PCB design/supply chain, the IPC-2581 Consortium made great strides in 2011 to promote, enable, facilitate, and drive the adoption of IPC-2581 as the standard data transfer format in the PCB industry.

③ TTM's Q4 Results Meet Expectations, FY 2011 Sales Rise

"Our fourth quarter financial results were in line with our outlook with revenue towards the high end of our guidance range. Performance in each of our end markets was generally consistent with expectations," said President and CEO Kent Alder.

④ AT&S Delivers First Pacemaker PCBs

AT&S' Medical Segment unit achieved a notable strategic coup in January, with delivery of the first pacemakers to enter into series production featuring highly complex AT&S printed circuit boards.

5 German PCB Market Firms-Up Increasing Orders

During the first 10 months of 2011, total revenues went up 8% compared with the same period in the previous year, despite a 16% decline in October revenues on-year, according to ZVEI PCB and Electronic Systems.

6 Viasystems' PCB Segment Posts 14% Sales Growth in Q4

"In addition to a relatively stable cost environment in our fourth quarter, the combination of a couple of favorable factors resulted in a record earnings quarter for Viasystems," noted CEO David M. Sindelar.

7 Ibiden 3QFY2011 Net Income Plunges 85.8%

Japan-based PCB company Ibiden Co. Ltd. has reported net sales of 214.74 billion yen ($2.79 billion at $1=76.8398 yen) for its third quarter ended December 31, 2011, down 3.4% from the same period in its fiscal 2010.

8 European Circuits to Expand Manufacturing Area

European Circuits has announced that its manufacturing expansion plans have now been approved and are in place for the company's first phase of expansion.

9 FTG Sales Up 9% in 2011

"FTG closed 2011 with another strong quarter making the full year very successful for the company. We achieved our objectives for the year, including growing the company, returning to profitability, and strengthening the balance sheet…" stated Brad Bourne, president and CEO.

10 Cicor Concentrates PCB Production in Switzerland

Cicor has confirmed the concentration of its PCB production sites in Switzerland and the decision to tighten-up operating processes at the conclusion of the consultation procedure.



For the Latest PCB News and Information Visit: PCB007.com

Was it Good For You?



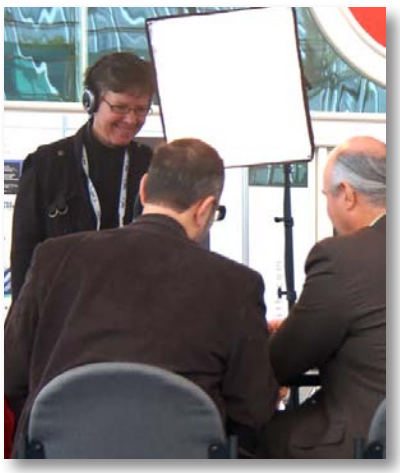
by Barry Matties
I-CONNECT007

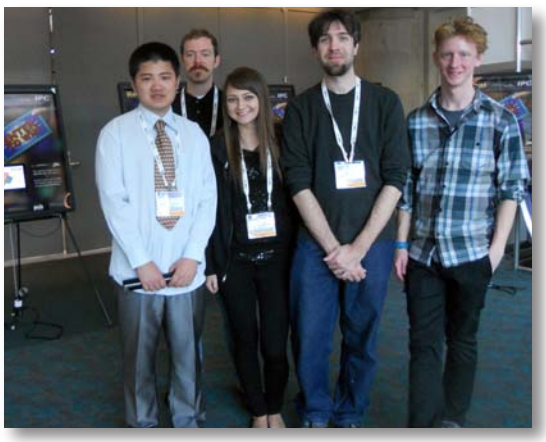
SUMMARY — Now that the show is over, each company has its own assessment of the quality of the event. Here are a few thoughts on the show, including having to say goodbye to an old friend.

The show is over and it was good for several reasons, starting with the venue. It's great to be back in San Diego, though the weather felt more like Oregon for a day or two. The exhibit hall was split between two levels, with our booth and Real Time with...IPC studio on the second level. Throughout the three-day show, a never-ending stream of interviews were conducted on our stage. All told, we produced well over 150 video interviews ranging in topics from the economy to the latest in technology.

On the lower level is where the majority of exhibitors were located, many with larger equipment than what could be found upstairs. It was nice to see the size of the teams manning booths, a noticeable increase from recent past years. All of this is a reflection of the overall upbeat attitude that was present, and that one could feel, just about everywhere. The energy was a welcome and refreshing change from recent years.

To kick off the show, William Shatner performed an early morning keynote presentation. He did not disappoint, even though he looked very tired, having arrived in San Diego that morning at 2 a.m. from New York, where he has been performing his one man show on Broadway. During his keynote, it felt like we were getting some bits and pieces from his show along with some personal insight into who he really is. At nearly 81 years old he has a lifetime of experiences to share, including Star Trek stories, along with his take on God, religion, death and success. After his presentation, the ribbon was cut and the exhibit halls were open for business.





The I-Connect007 Real Time with...IPC video crew.





"The energy was a welcome and refreshing change from recent years."

Just like every trade show I have covered, the common question is, how is the crowd? Some said good, others said not what they expected, but overall, I think most were satisfied with the attendance. One thing is for sure, I talked with many attendees, from all across America and beyond, that were there to make deals. It's nice to see the confidence in the market, and with it, a sense that the recovery is real. Of course, only time will really tell.

Another benefit of the show is seeing old friends. One friend that I had a chance to catch up with is Richard Bain, whom I have known for many years. I met Richard when we launched CircuiTree Magazine in 1987. He was working with his brother, Marv Bain, at Circuit Image Systems (CIS). The news from Richard was not good; Marv has recently passed away.

I'll never forget the incredible support Marv gave us when we were first getting started. He was the president of the CIS and had an amazing understanding of what it took to build a great organization and how to take care of his customers, employees, and suppliers. But even more than that, he really cared about people. I recall many times when I would pull into the parking lot at the CIS headquarters in Orange, California, and before I was a few feet away from my car, Marv would be out the front door of the building to greet me with a huge smile on his face and a very welcoming handshake. He gave us confidence that we, too, could succeed in business. It's people like Marv who have helped so many people in our industry. Of course, the industry looks substantially different than it did in 1987.

One very noticeable difference is with our publications. Of course, we sold CircuiTree in 1999 and started publishing an electronic publication, PCB007, in 2005. Marv was one of the first to encourage us to continue to live our dreams. I, like so many others, will miss him.

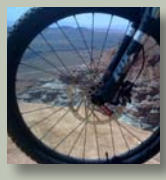
One other thing to note about the trade show is the exceptional job the team at IPC did, particularly during their current leadership transition period. The event was well organized and supported with a warm and friendly staff. To get a good sense of what went on at the show be sure to check out our video coverage at www.iconnect007.com.

Please keep in mind that our coverage is made possible through the support of our sponsors. Each sponsor really understands the value of supporting us, so that we can bring you the best information possible. And finally, I thank our team for doing such an exceptional job. We had five camera crews, editors, customer support, an IT crew, and support teams working to bring you exceptional coverage. With more than 20 people from our company on the show floor, and the support of our guest editors, we covered a lot of ground.

For those of you who knew Marv, here is the obituary, forwarded to me by his family, who asked that we include it in our publications. **PCB**

Marvin J. Bain, 66, passed away on Thursday, February 23, 2012. He is survived by wife, Patricia; mother-in-law, Inza McConnell; sons Tom (Jennifer), Matt (Shannon), John (Azya); daughter Cathy (Eric Garcia); seven grandchildren; brothers Edward, Richard, and sister Angela. Attended St. Boniface Elementary School, Mater Dei High School, and graduated from the University of Southern California. Business executive and successful entrepreneur. Lived by the mantra "FIGHT ON."

Donations in Marvin's name may be sent to St. Josephs Cancer Center, Orange, CA.



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**

MEDTEC Europe
March 13-15, 2012
Messe Stuttgart, Stuttgart, Deutschland

NEW:South Africa 2012
March 13-14, 2012
Sandton Convention Centre Johannesburg, South Africa

CPCA Show 2012
March 13-15, 2012
Shanghai World Expo Exhibition Center, Shanghai China

5th Annual EMEA Summit
March 18-19, 2012
Abu Dhabi, U.A.E.

ISQED
March 19-21, 2011
Santa Clara, California

The Heat is On — Revolutionary and Evolutionary Innovations in Thermal Management
March 19, 2012
Doubletree Hotel, San Jose, California, USA

Indiana Expo & Tech Forum
March 20, 2012
Carmel, Indiana

Conference on Electronics Assembly: Soldering, Assembly & Inspection
March 20-22, 2012
Budapest, Hungary

Avionics Europe 2012
March 21-22, 2012
Munich, Germany

Smart Systems Integration
March 21-22, 2012
Zurich, Switzerland

ELECTRIC VEHICLES LAND, SEA & AIR USA 2012
March 27-28, 2012
San Jose, California, USA

Printed Electronics Europe 2012
April 3-4, 2012
Berlin, Germany



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: **SHELLY STEIN, BRYSON MATTIES**

PCB007 Presents
the pcb magazine

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

©2012 BR Publishing, Inc. does not assume and hereby disclaims any liability to any person for loss or damage caused by errors or omissions in the material contained within this publication, regardless of whether such errors or omissions are caused accidentally, from negligence or any other cause.

March 2012, Volume 2, Number 3 • The PCB Magazine© is published monthly, by BR Publishing, Inc.

ADVERTISER INDEX

atg Luther & Maelzer GmbH..... 25
Atotech..... 2
Burkle North America..... 47
CA Picard..... 79
Electra Polymers..... 37
Fein-Line Assoc..... 97
ID Tech Ex..... 91
Integral Technology..... 27
I-Connect007..... 3, 101, 109
IPC..... 99, 107
Isola..... 63
Meptec..... 83
Microcraft..... 67
OEM Press Systems Inc..... 71
Ohmega Technologies, Inc..... 9
OMG Electronic Chemicals..... 15
Prototron Circuits..... 75
Rogers Corp..... 11
Schmidt Group | Gebr. Schmid GmbH..... 49
Semblant..... 5
SiSoft..... 19
Taiyo-America..... 81
Technica USA..... 57
Uyemura Intl. Corp..... 33
Ventec..... 43
Viking Test..... 7

Next Issue: Imaging Processes

Since Paul Eisler patented the first PCB sixty years ago, the term "printed" has taken on a new context in present-day PCB technology. What methods are used for creating the patterns that end up as complex electrical interconnections?

The April issue of The PCB Magazine will explore a variety of imaging process issues, such as the pros and cons of photolithographic and digital techniques, and how successive images are aligned and kept in register. Look for feature articles by Maskless Lithography, Rainbow Technology Systems, Agfa Materials, Karl Dietz and more.

Don't miss our regular columns, 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.