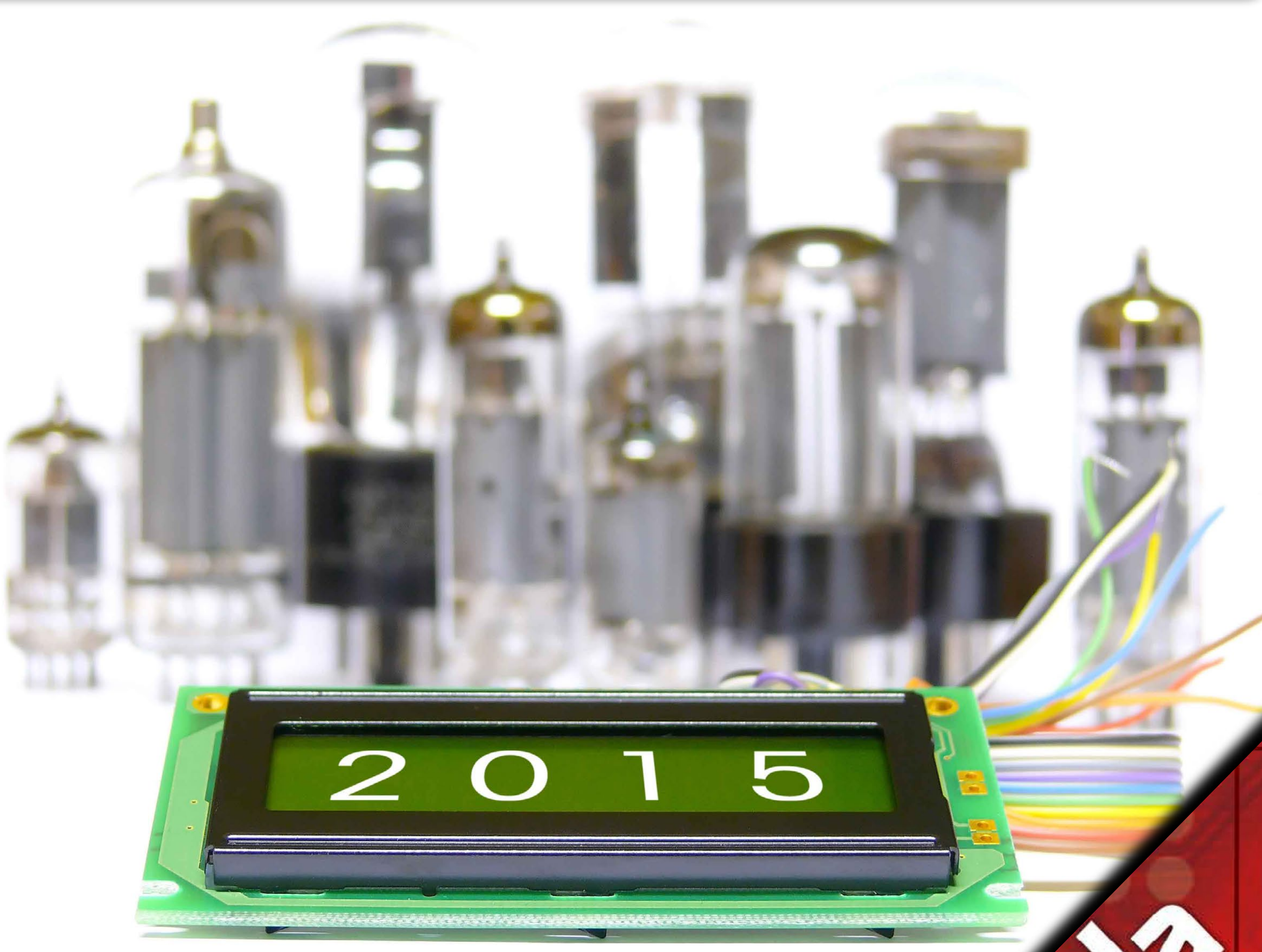
THE pcb design MAGAZINE

January 2015

an I-Connect007 publication

WHAT'S AHEAD FOR 2015?



Past and Future Trends in PCB Design

by David Wiens, page 10

isola
The base for innovation

the pcb list

THE best way to find a PCB fabricator, anywhere.

TTM TECHNOLOGIES - CHIPPEWA FALLS DIV

Chippewa Falls, Wisconsin, U.S.A.

Website | Contact Me | RFQ

TTM Technologies
Time-To-Market Interconnect Solutions

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

The TTM Chippewa Falls facility is located in Chippewa Falls, Wisconsin and is approximately 234,000 sq ft. This facility employs approximately 1,000 production staff and specializes in NPI and volume production of high layer count PCBs.

This facility began as a captive operation for Cray Computer. It was sold to Honeywell and operated as Honeywell ACI until being purchased by TTM Technologies in December of 2002.

Markets:	Communication, Computers, Industrial, Medical, Military/Aerospace
Board Types:	Multilayer
Mfg Volumes:	Medium, Large
Specialities:	Blind/buried vias, Sequential [illegible]mination
Certifications:	Certifications: ISO 9001, ISO 14[illegible]0, UL

Featured Showcase

Click here to see a demo

Why YOU should Showcase:

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

Click to see a partial list of registered OEMs!

www.thepcblist.com

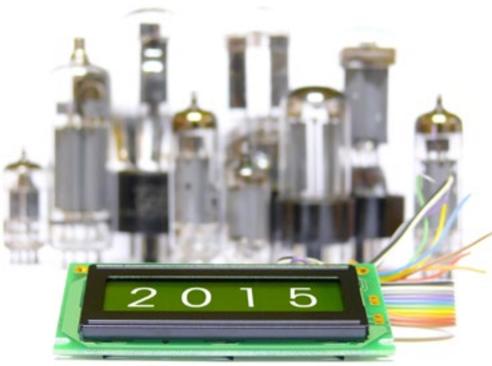

2015 FLEX
Monterey, California
February 23–26, 2015
Same Great Conference in a New Location!
» Best Technical Program by and for the industry
» State-of-the-art conference and exhibition facility
» Two adjacent 4-star hotels to choose from
» Easy access to Silicon Valley and other San Francisco Bay Area locations
» Regional airport served by most major airlines
» Regular shuttles to San Francisco & San Jose Airports
» Ideal location for both networking and relaxing with clients & colleagues
» Iconic California experiences easily available: wine tasting, whale watching, Cannery Row tours, legendary golf courses, breathtaking BigSur coastline
More information at www.2015FLEX.com.
Presented by: FlexTech Alliance
for Displays & Flexible, Printed Electronics

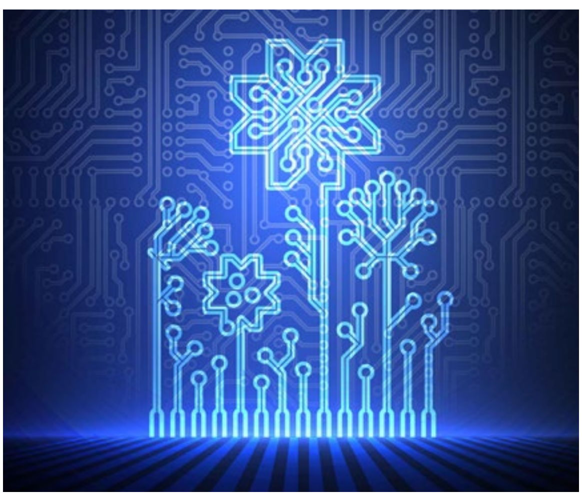
THE pcb design MAGAZINE
January 2015

This Issue: WHAT'S AHEAD FOR 2015?

FEATURED CONTENT

It's a new year, and time to put our party hats away. We're sure to face a variety of opportunities and challenges over the next 12 months. This month, our contributors David Wiens, Mark Thompson, Martyn Gaudion, Sharon Starr, and Abby Monaco peer into their crystal balls and discuss what we can expect in the coming year or so.





FEATURE COLUMNS

isola

Low-loss Laminate Materials for RF/Microwave/Millimeter-wave Designs

For over 100 years, Isola has been driving technology forward while improving end-product design with top-performing laminate and prepregs.

Our legacy continues with global R&D pushing our product line to the forefront in demanding RF/microwave designs.

With 10 manufacturing sites around the world, global quick-turn capability, world-class technical service, measurement & modeling capabilities, and industry leading material development capabilities – Isola is well positioned to be your preferred RF/microwave partner.

The newest members of Isola's growing RF/Microwave laminate family now includes Terra Green™, a halogen-free, very low-loss, thermoset material that is available in a variety of laminate and prepreg offerings. This material is inexpensive to process – improving your company's bottom line, as well as the environment.

Astra®MT is a revolutionary ultra low-loss thermoset laminate replacement for PTFE, suitable for automotive radar, mm-wave and AESA applications.

I-Tera MT is available in core thicknesses from 0.002" to 0.018" and offers 18 different prepregs for optimal design flexibility.

I-Tera®MT RF is available in 20 and 30 mil cores and a 3.38 and 3.45 Dk.

	TerraGreen™	Astra® MT	I-Tera® MT/ I-Tera MT RF	IS680
Tg	200°C	200°C	200°C	200°C
Td	390°C	360°C	360°C	360°C
DK @ 10 GHz	3.45	3.00	3.45	2.80 - 3.45
Df @ 10 GHz	0.0030	0.0017	0.0031	0.0028 - 0.0036
CTE Z-axis (50 to 260°C)	2.90%	2.90%	2.80%	2.90%
T-260 & T-288	>60	>60	>60	>60
Halogen free	Yes	No	No	No
VLP-2 (2 micron Rz copper)	Standard	Standard	Available	Available
Stable Dk and Df over the temperature range	-55°C to +125°C	-40°C to +140°C	-55°C to +125°C	-55°C to +125°C
Optimized Global constructions for Pb-Free Assembly	Yes	Yes	Yes	Yes
Compatible with other Isola products for hybrid designs	Yes	Yes	Yes	For use in double-sided applications
Low PIM < -155 dBc	Yes	Yes	Yes	Yes

NOTE: Dk Df is at one resin %. 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 products will be governed by the terms and conditions of the agreement under which they are sold.

FREE WEBINAR

PCB Material Selection for RF/Microwave/Millimeter-wave Design

This webinar will educate you on how to select the appropriate PCB material for RF, microwave and millimeter-wave applications through careful consideration of cost constraints and system-performance parameters, such as the frequency of operation and bandwidth, the electrical size of board and its critical features, loss and uniformity requirements, temperature range of system operation, and thermal and processing considerations.

www.isola-group.com/RF

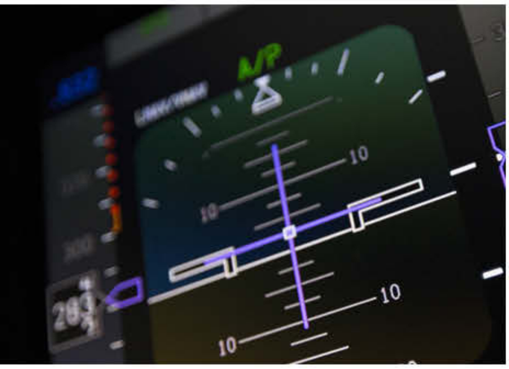
JANUARY 2015
VOLUME 4
NUMBER 1
thepcbdesignmagazine.com

THE pcb design MAGAZINE™

an I-Connect007 publication

THE OPTIMUM MAGAZINE DEDICATED TO PCB DESIGN

CONTENTS



Unsurpassed Quality. Cutting Edge Technology.

AS9100, MIL-PRF-55110, and MIL-P-50884 Certified for Rigid, Flex and Rigid-Flex Printed Circuit Board Fabrication.

DRAGON CIRCUITS

www.dragoncircuits.com

Happy New Year: Now, Back to Work

by Andy Shaughnessy
I-CONNECT007

Welcome back! It's January again, and it's a brisk 10 degrees here in Atlanta. It's so cold here that socialites have abandoned the veranda and taken their mint juleps into the parlor.

Yes, it's time to get back into work mode. We ended the year on a good note, with unemployment down and almost 3 million jobs created in the first 11 months of 2015—the most for any similar period since 1999. There are still too many people working part-time who would like a full-time job, but the trend looks positive.

Show Time

We have quite a busy few months ahead. This month, we'll be covering DesignCon, which takes place January 27–30 in Santa Clara, California. This year, the conference offers a wide variety of PCB sessions, panels and tutorials. DesignCon has come a long way since it focused primarily on high-level IC content that was of little interest to the lowly PCB designers! And even huge EMS companies exhibit at DesignCon now, hoping to appeal directly to

the front-end engineering teams who attend. This show had solid attendance even during the recession; it's not bulletproof, but close to it.

Then, during the week of February 22–26, it's off to San Diego for IPC APEX EXPO and the Design Forum. (And three cheers for IPC moving APEX back to "America's Finest City." It's hard to have a bad day in San Diego.) Appropriately, the Design Forum kicks off the week, with a keynote by Carl Schattke, a PCB design engineer with Tesla Motors. How many PCBs can you find in a Tesla?

I know that a lot of people at APEX will be thinking about Dieter Bergman. This will be the first APEX since his death, and Dieter always opened the Design Forum. It will be bittersweet for many APEX attendees, especially those who worked with him on standards and education for decades. I'm sure attendees will be sharing their favorite Dieter stories, and probably some of his hysterical off-color jokes. We'll miss interviewing him, too; Dieter was always great in front of the camera.

"A lot has changed in China since 2006. But one thing hasn't changed: China still hasn't embraced PCB design. OEMs still send design work to China, but for the most part, they haven't found many designers who can come close to rivaling the designers in the U.S., and the West in general."

Next, it looks like I'll be going to China for CPCA March 17–19. I went to China nine years ago for CPCA, and wound up running into Barry Matties and Steve Gold on opening day, back when the three of us worked at three rival publications. Barry had a video camera set up, and Steve and I asked, "What's with the camera, dude?"

A lot has changed in China since 2006. But one thing hasn't changed: China still hasn't embraced PCB design. OEMs still send design work to China, but for the most part, they haven't found many designers who can come close to rivaling the designers in the U.S., and the West in general. But China has identified PCB design as the one piece of the puzzle they're lacking, and they're working hard to fix that. Don't count them out.

Then, I'll be covering SMTA Atlanta on April 15. It's a great little tabletop show, which draws exhibitors from all over the East Coast. I particularly enjoy the PCB Designers' Roundtable, which is moderated by UPMG's Pete Waddell. We always get a lot of good information (and the occasional horror stories) from the members of the Atlanta design community. Maybe we can play some guitar afterward, and get Matt Kehoe of SIPLACE to play harmonica and spoons.

And, of course, we'll be providing Real Time with... video coverage of industry events for the rest of the year. Stay tuned!

A Shameless Plug

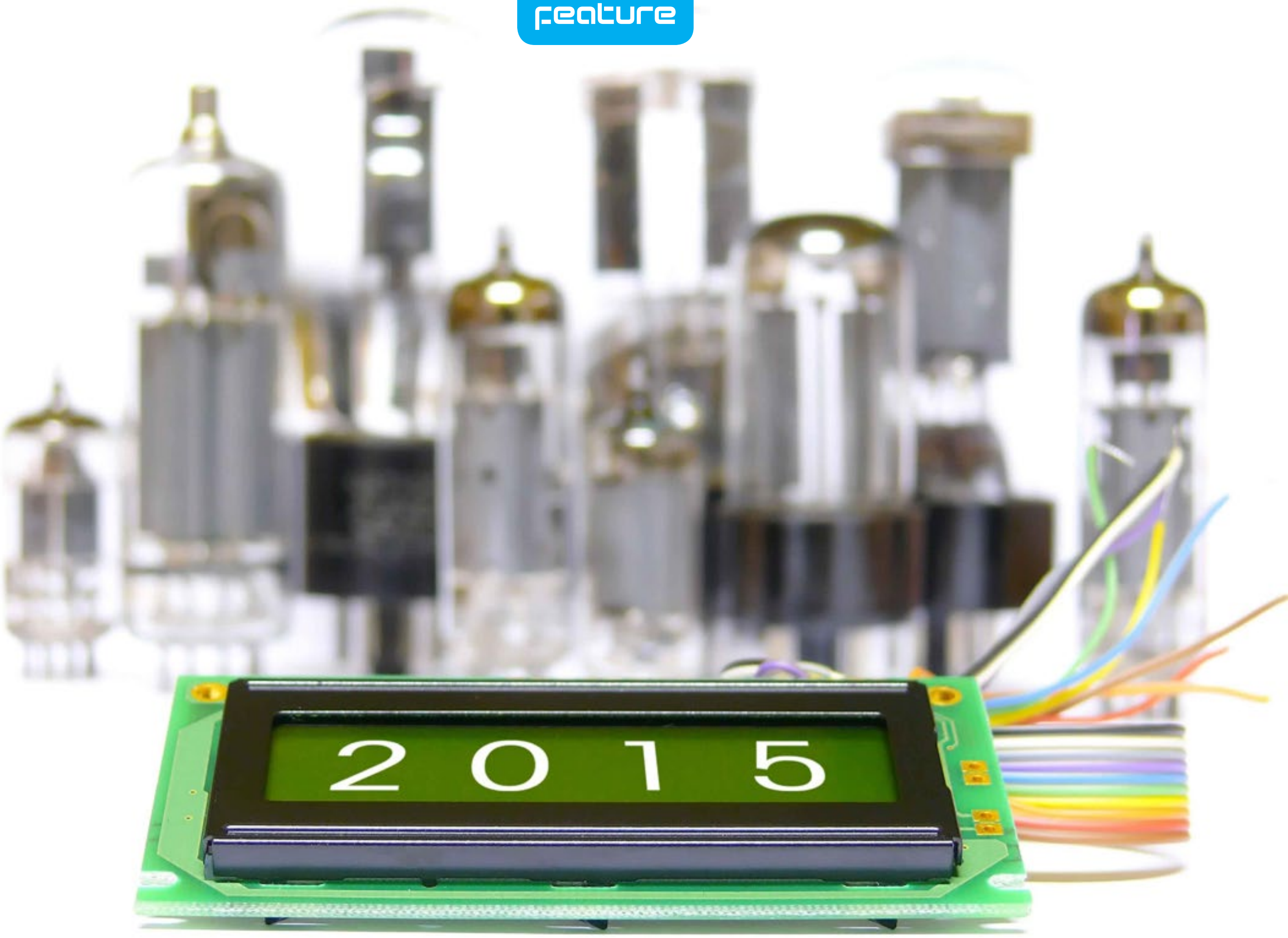
Clyde Coombs and Happy Holden are editing the 7th edition of *The Printed Circuit Handbook,* and they were kind enough to ask me to contribute a chapter on PCB design tools. This book is considered the Bible of the industry, and I'm just lucky to be part of it. The other design contributors are Lee Ritchey, Susie Webb, and Bill Hargis, so I'm in good company.

I've been an editor for so long that I had almost forgotten what it was like "on the other side of the desk," so to speak. I've edited many of Happy's articles over the years, but I spent most of last fall working for him!

The Printed Circuit Handbook is a huge book, with all new content; I can only imagine the logistics behind putting it all together. It will be published later this year by McGraw-Hill.

So, have a happy and prosperous new year, and have fun at DesignCon. **PCBDESIGN**

Andy Shaughnessy is managing editor of *The PCB Design Magazine.* He has been covering PCB design for 15 years. He can be reached by clicking here.



Past and Future Trends in PCB Design

by David Wiens
MENTOR GRAPHICS CORP.

As part of the team that administers the Mentor Graphics Technology Leadership Awards (TLA) for the best PCB designs, I have been able to see and record a great deal about how the designs have evolved over the years. This year, because it's the 25th year of the TLA program, I asked some of our senior judges to relate their early experiences and history. In this article, I'll explore those inputs as well as the data we have collected over the years, and add just a bit of trend forecasting.

A Short History of the PCB World

The judges' feedback painted a clear picture of how PCB design and design tools became what they are today. Most of the judges began their PCB career when boards were single sided, contained no surface mount components, and may have even included vacuum tubes! (Figure 1) Until the late '70s or early '80s, most boards were designed using tape to represent traces and pads, and a Mylar sheet as the PCB. Boards were often designed at a 4:1 scale.

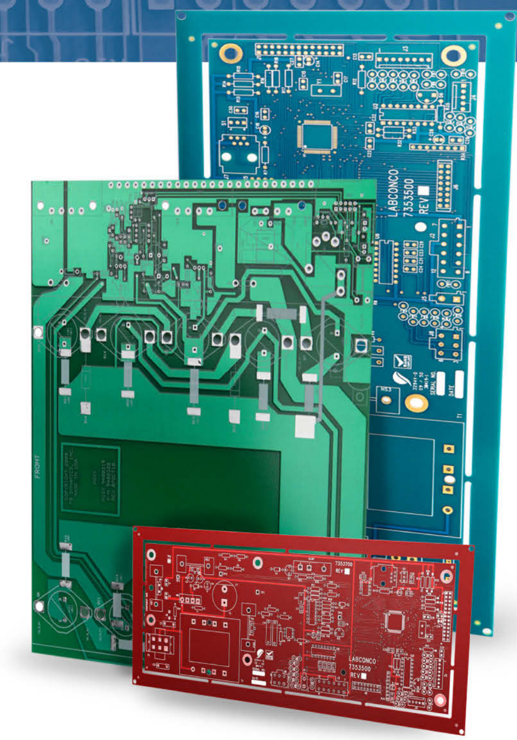
In the '70s, double-sided boards became common, but with no SMT capability, the second side was strictly for copper, not components. (Sometimes, though, prototypes were "fixed" by cutting traces on the back and installing jumpers or, sometimes, resistors or capacitors). Double-sided boards were designed on Mylar using blue tape for one side, and red tape for the other.

Why? In the days of tape and Mylar, the finished tape-out was photographed at a 1:4 scale to produce a 1:1 film negative. The negative was then projected onto the photoresist-coated board and developed to reveal the trace design. For double-sided boards, two types of film were

Quality PCBs from the established industry leader

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

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



Get a quote instantly at Sunstone.com

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

SUNSTONE® CIRCUITS

40 YEARS SUNSTONE 40 Years of Serving You

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

used. Orthochromatic film is blind to red, so it recorded only the blue traces. Similarly, panchromatic film is less sensitive to blue and with filtration it only records the red traces. Thus, the red and blue tapes allowed two negatives to be produced, back and front, while ensuring exact alignment of pads and vias.

There was another popular method of designing PCBs at the time, although not nearly as popular as tape and Mylar, using a material called Rubylith. This was a polyester base covered with a red plastic laminate. The designer used an X-Acto knife to remove the red where traces and pads would be. Then the "negative"

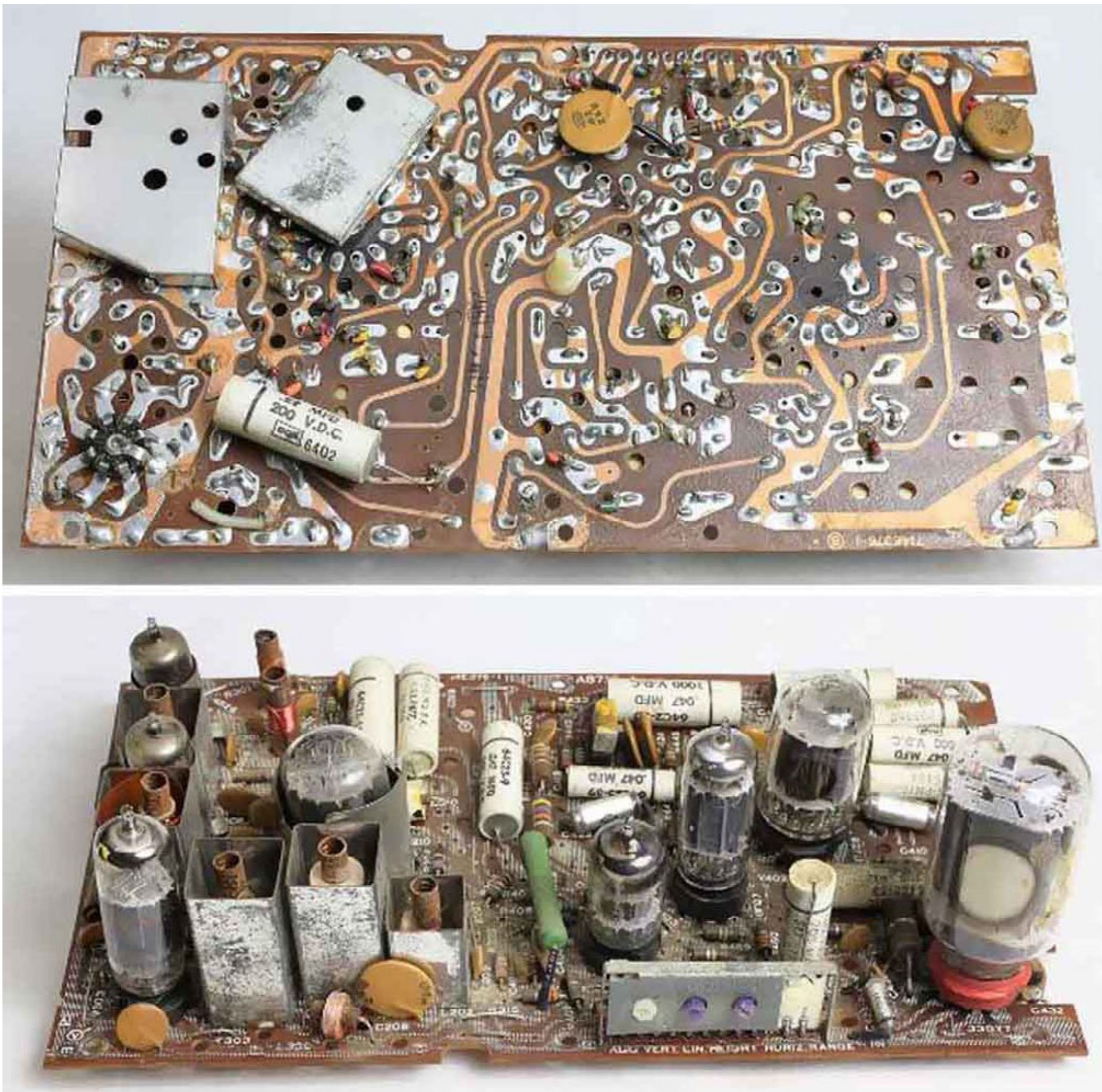
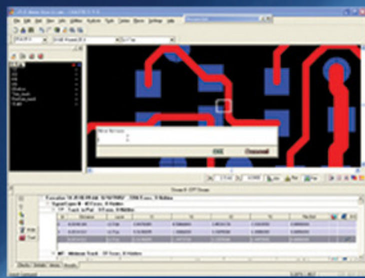


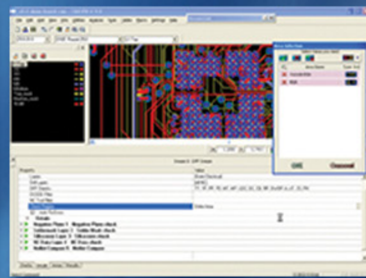
Figure 1: Yes, vacuum tubes and PCB technology overlapped. This is from a 1971 television (Mark Forbes' photo and collection).

Solutions to Ensure PCB Manufacturing Success!

CAM350®

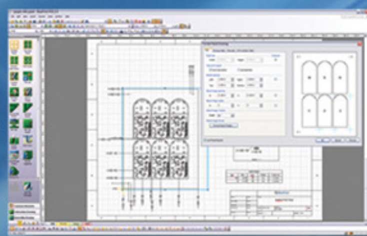
Verify and Optimize PCB Designs for Successful Manufacturing.

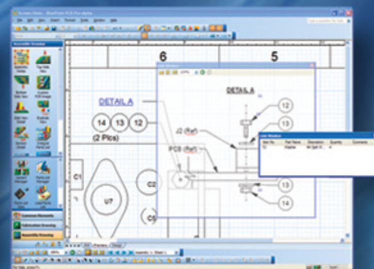




BluePrint·PCB

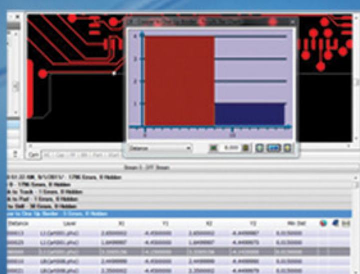
Create comprehensive documentation to drive PCB fabrication, assembly and inspection.

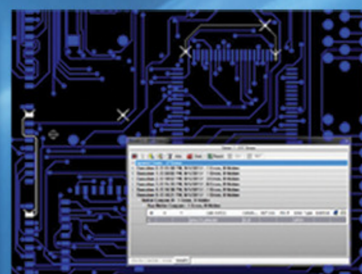




DFMStream™

Verify Design and Manufacturing Rules Any Time During the PCB Design Cycle.





At DownStream Technologies we are committed to delivering comprehensive solutions and workflow process improvements for all your PCB post processing needs.

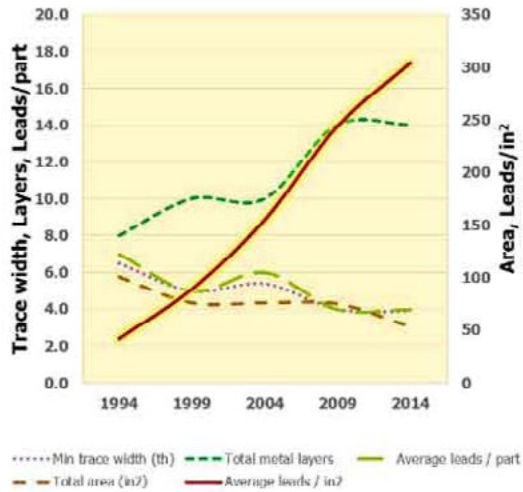
For more information please visit downstreamtech.com or call 800-535-3226.

©2014 DownStream Technologies, LLC. All rights reserved. All trademarks and registered trademarks are the property of their respective owners.

DownStream Technologies

Trends in PCB Complexity

Design Trend	1994	2004	2009	2014
Min trace/spacing (th)	6.5/6.5	5.4/5.4	4/4	3.9/4.2
Total metal layers	8	10	14	14
Total area (in²)	101	76	75	53
# Nets	1465	2952	3411	2109
# Pin-to-pin connections	5190	8813	10960	6228
# Components	649	1981	3400	2608
# Component pins	4214	7760	13505	10122
Leads / part	7	6	5	4
Leads / in²	42	155	244	304



* Data based on Technology Leadership Awards entry averages

Figure 2: This is a 20-year analysis of several key factors in PCB designs. There are several clear trends.

could be used to produce a board on photoresist-coated blanks.

TLA judge Charles Pfeil recalls entering the PCB career path when his father's company hired him part-time in high school to inspect the finished Rubylith designs for scratches.

"Was the fabrication process so exact that a scratch that could only be seen through a magnifier could really affect the result?" Pfeil asked. "I dared not say anything for fear of losing my cushy $1.75 per hour job!"

There was one other "technology" that was fairly rare, yet used with success. TLA judge Gary Ferrari was just out of school and designing gears by using ink on linen. Because he was so good at it, he was moved to PCB design where they laid out the boards by drawing on linen sheets with ink!

CAD systems had been used with some degree of success in the late '70s and '80s (and some existed even in the '60s), but in the late '80s they become nearly universal. Good thing, because SMT had arrived shortly before and design difficulty rose immediately.

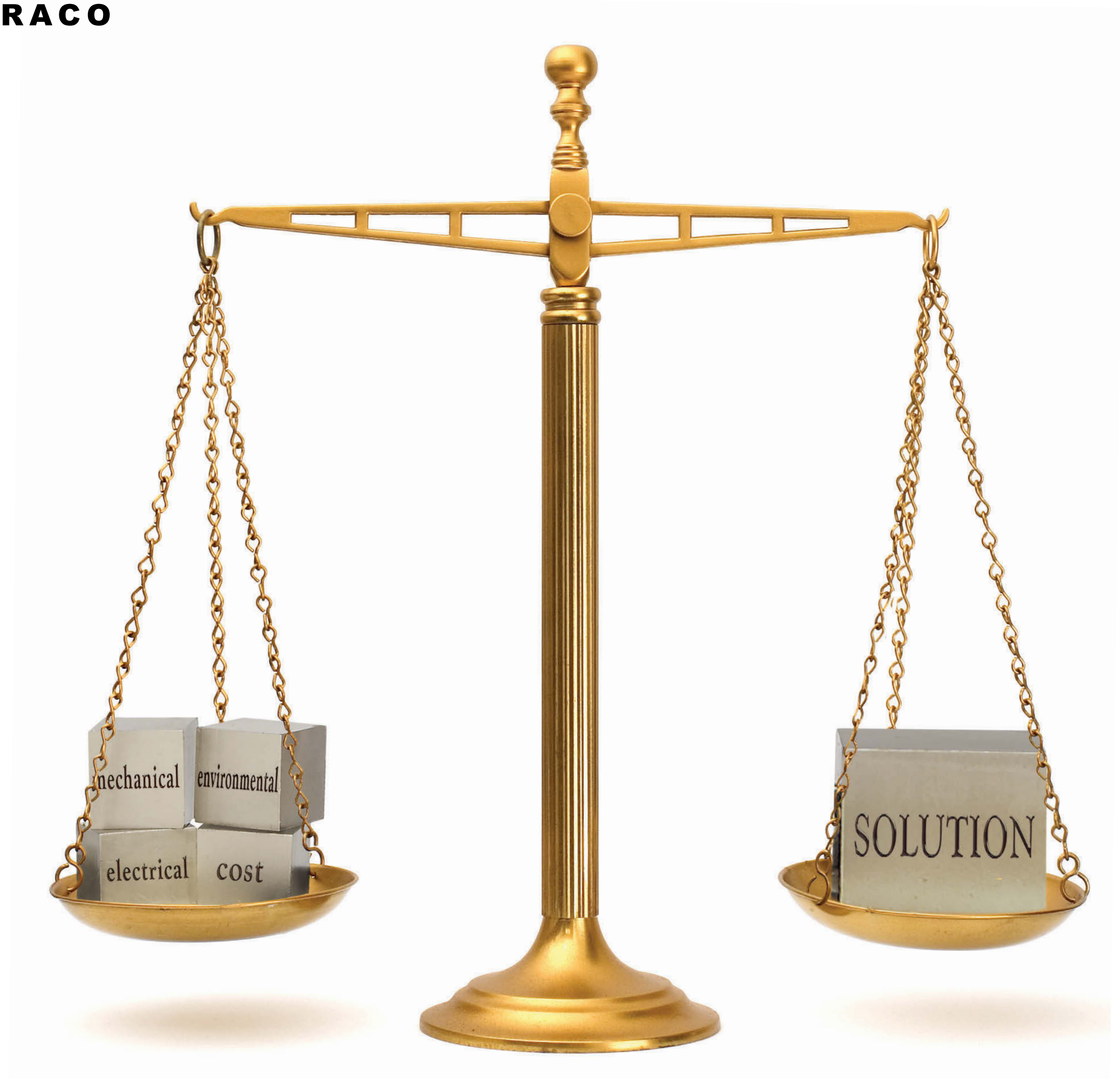
As soon as SMT was introduced, a small explosion of package types began to appear. Another judge, Andy Kowaleski, remembers the introduction of the BGA package, and he made an interesting observation.

"The introduction of the BGA package led to a massive increase in multilayer boards," said Kowalewski. "Once that happened, the rest of the board could be packed with components like never before, so boards got smaller and more functional (usually both)."

That is a thumbnail sketch of the situation up until today, when high-speed signal integrity issues can be more problematic than ever before. High speed generates heat, and that has to be dealt with too. And all the while, boards are more complex and need to be designed in less time than ever before!

We'll take a look at those trends. But first, check out Figure 2, which illustrates the changes we've seen since 1994 in the designs submitted to the TLA program. Probably the most striking thing to notice is the highlighted red line, which represents the number of leads per square inch. It is surprising that this trend can be so steep, but still continue for 20 years uninterrupted. On average, that number went from 42 to 304 leads per square inch over that time. Over that same period, the number of average leads per part has diminished by nearly 50%, MIRACO

Engineering And Providing Balanced Interconnect Solutions



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

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

Partnering With Your Engineering Team To Deliver The Best Solution

...connecting technology

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

meaning even more density than the leads/in^2 number alone indicates.

Interestingly, one trend that appeared in the first decade has leveled out. The minimum trace width decreased by nearly 50% over the first five years of the awards and then has been mostly stable since then. The total number of metal layers had two "growth spurts" and now also seems to have leveled out around 14.

Future Trends

So what does this trip down memory lane tell us about the trends for the future?

- The clearest message, and the one that has continued unabated, is that the boards submitted to the TLA awards have been on a long and fairly steep linear increase in complexity. Not only are there more component leads from fewer components, but the average board area is shrinking at the same time.
- The reduction of the number leads per part, combined with the increasing ratio of passives to active components, points to integration of more and more functionality onto silicon with performance criteria that demand high volumes of resistors and capacitors for signal and power integrity.
- Over the last five years, layer counts have stayed about the same, while area has dropped 29%, and densities have gone up by 25%. You can see the density increase more dramatically over the last 20 years, highlighted in the chart on the right.
- Signal integrity on the submitted designs was extremely important with 92% of the entries employing SI tools during the design.

One thing is certain: PCB designs will not be getting simpler in the future! **PCBDESIGN**

David Wiens is a product marketing manager at Mentor Graphics.

New Process Transforms 2D into 3D Microarchitectures

In the cover feature article of the journal Science, researchers at the University of Illinois at Urbana-Champaign describe a unique process for geometrically transforming 2D micro/nanostructures into extended 3D layouts by exploiting mechanics principles similar to those found in children's "pop-up" books.

Complex, 3D micro/nanostructures are ubiquitous in biology, where they provide essential functions in even the most basic forms of life. Researchers noted that existing methods for forming 3D structures are either highly constrained in the classes of materials that can be used, or in the types of geometries that can be achieved.

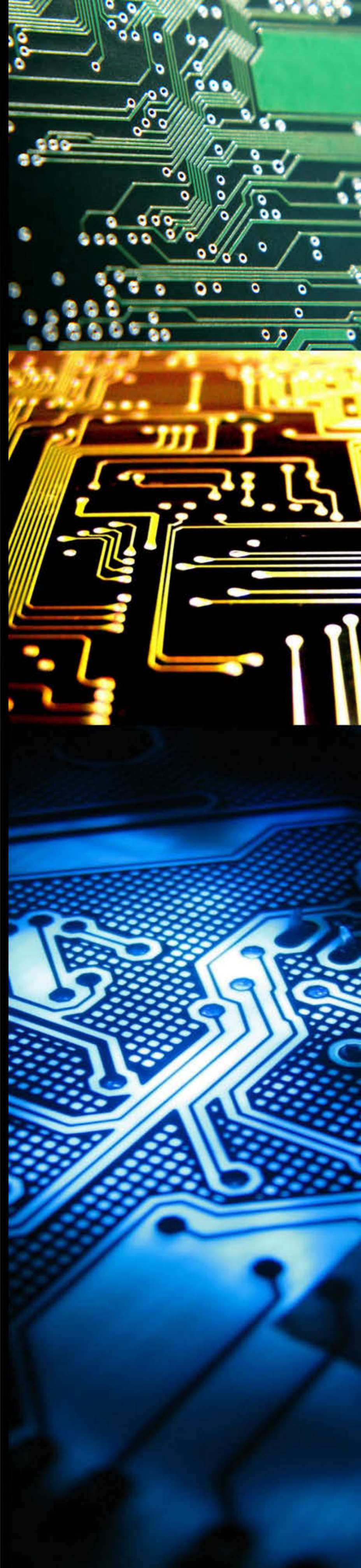
"Conventional 3D printing technologies are fantastic, but none offers the ability to build microstructures that embed high performance semiconductors, such as silicon," explained John Rogers, a Swanlund Chair and professor of materials science and engineering at Illinois. "We have presented a remarkably simple route to 3D that starts with planar precursor structures formed in nearly any type of material, including the most advanced ones used in photonics and electronics. A stretched, soft substrate imparts forces at precisely defined locations across such a structure to initiate controlled buckling processes that induce rapid, large-area extension into the third dimension. The result transforms these planar materials into well-defined, 3D frameworks with broad geometric diversity."

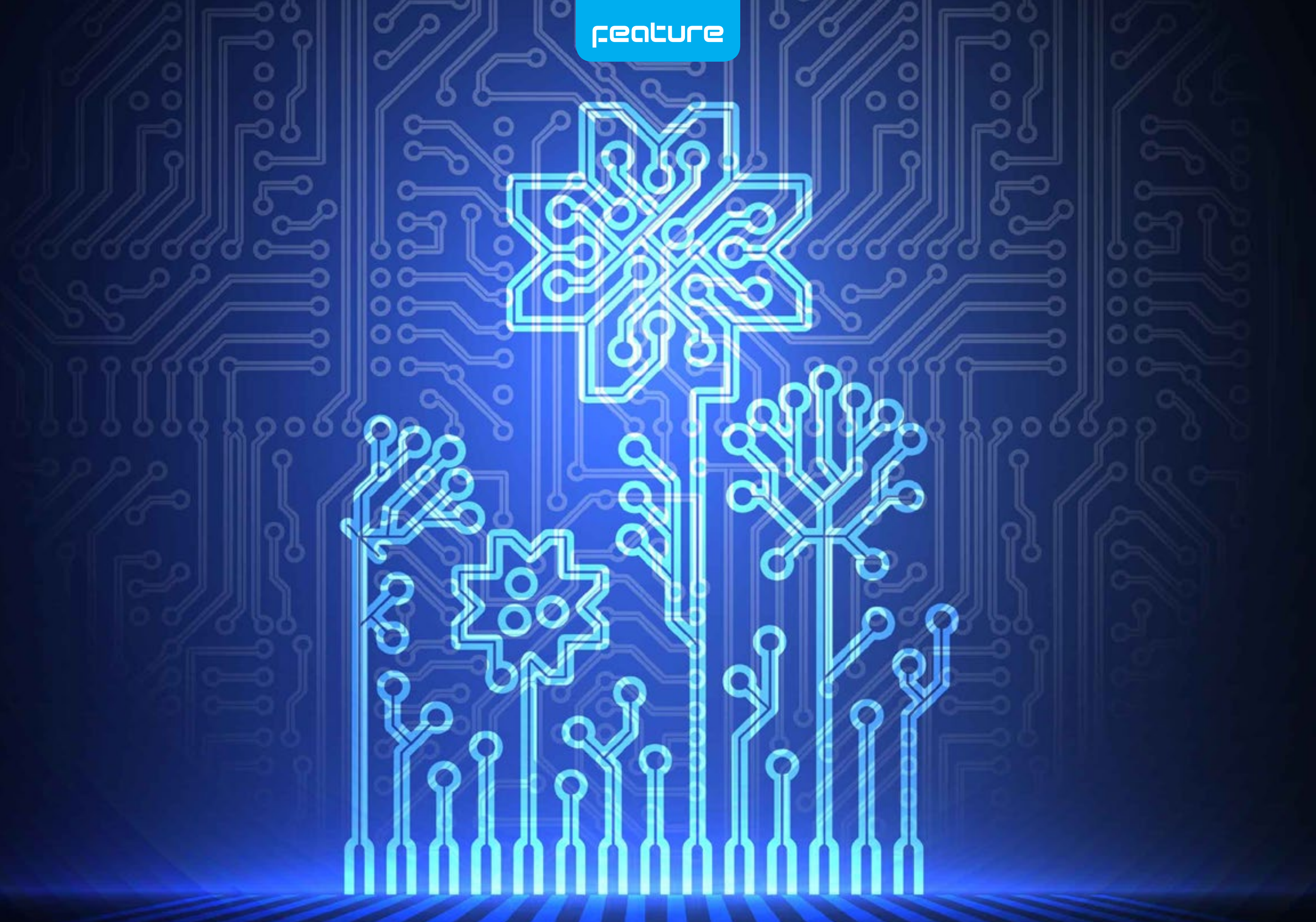
Compatibility with the most advanced materials (e.g. monocrystalline inorganics), fabrication methods (e.g., photolithography) and processing techniques (e.g. etching, deposition) from the semiconductor and photonics industries suggest many possibilities for achieving sophisticated classes of 3D electronic, optoelectronic, and electromagnetic devices.

HIGH-SPEED PCB DESIGN TRAINING

- One-on-one mentoring
- Develop your skills with confidence
- Ensure your next design performs reliably

LEARN MORE





North American PCB Opportunities: Investment is Crucial

by Sharon Starr
IPC

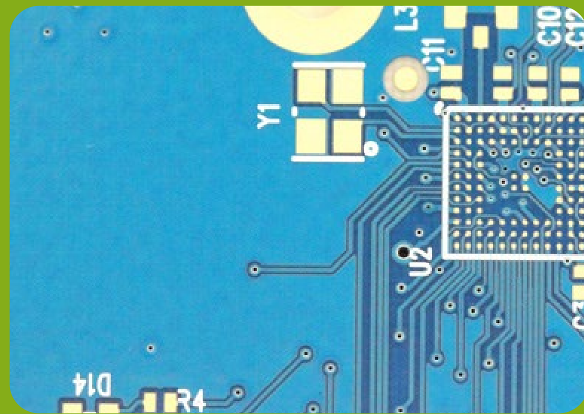
Originally published in The PCB Magazine, December 2014.

North America was the one bright spot in PCB production growth in 2013. Although 0.8% real growth may not seem like a cause for celebration, North America was the only major PCB-producing region with positive real growth in 2013. The estimated value of North American PCB output in 2013 was US$3.05 billion, 5% of the world market, according to IPC's *World PCB Production Report for the Year 2013.*

PCB production in North America is expected to grow at a slow but steady pace over the next three years, averaging 2% growth per year. The North American PCB market, estimated at $3.44 billion, is delivering zero growth so far in 2014, but is expected to resume slow growth in 2015, based on forecasts in IPC's monthly *North American PCB Market Report.*

Some opportunities and positive indicators for the North American PCB market are evident in the industry data IPC published in its *2013–2014 Analysis & Forecast for the PCB Industry in North America*. The industry is projecting growth in PCB sales to the North American military and aerospace market this year which, if it materializes, could push mil/aero ahead of communications to become the largest vertical market for PCBs in North America. Growth is also projected in PCB

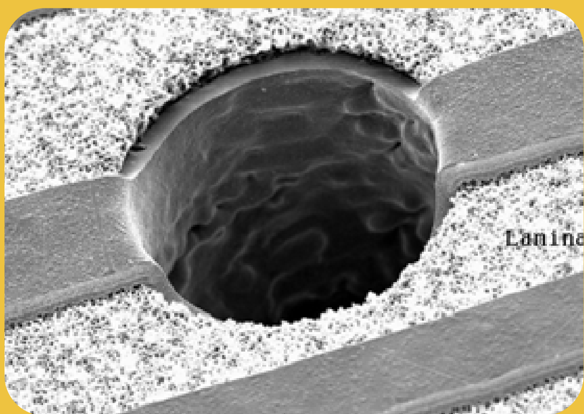
We deliver Advanced PCB Solutions!



Fine line PCB, 2 mil line /space

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

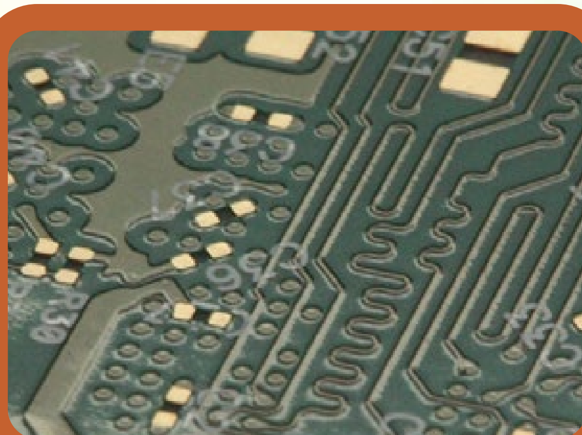
CLICK TO LEARN MORE



Landless Via Technology

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

CLICK TO LEARN MORE



Impedance Tolerance <5%!

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

CLICK TO LEARN MORE

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

CANDOR

CANDOR INDUSTRIES, INC.

ADVANCED PCB SOLUTIONS

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

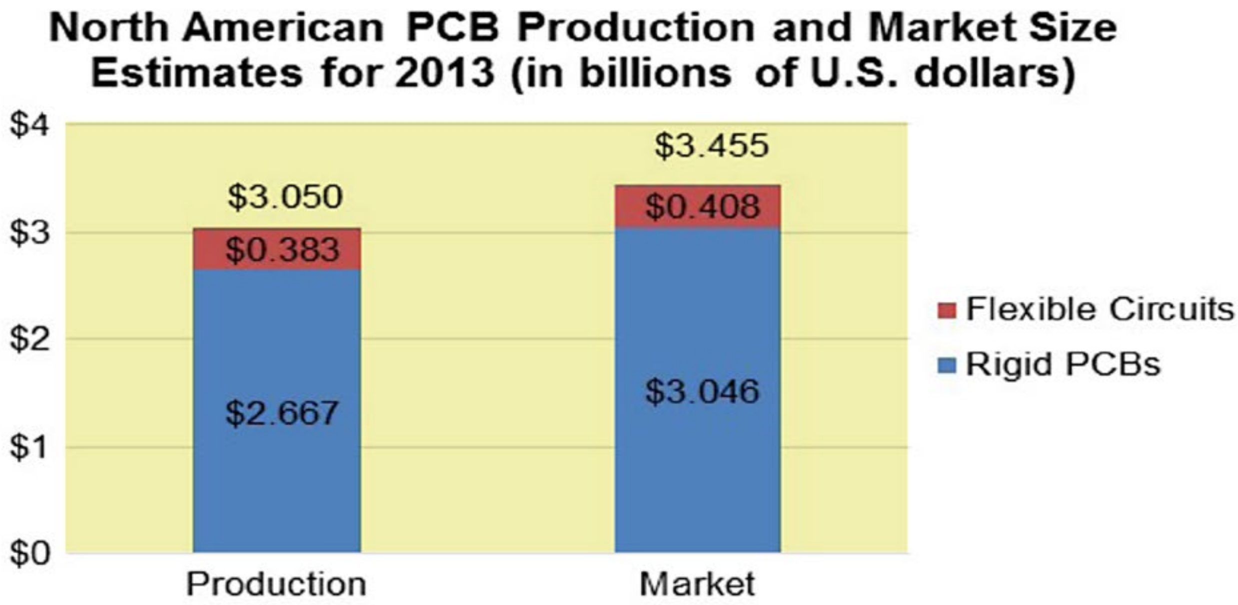


Figure 1.

sales to the industrial and medical electronics markets in 2014.

Even more encouraging is the increase in projected research and development spending among North American PCB fabricators surveyed by IPC in 2014. Research and development spending as a percentage of sales was reported to grow from 3.5% in 2013 to 4.3% this year in the rigid PCB segment of the industry. For the flexible circuit segment, R&D spending growth is equally noteworthy, climbing from 2.2% of sales in 2012 to 2.8% in 2013 and reaching an estimated 3.5% this year.

R&D activity is a crucial driver of demand in electronics through the introduction of new products. Electronics manufacturing is a cyclical industry, with growth patterns that reflect the ups and downs of the world economy. While the patterns look similar, growth in electronics markets has tended to outpace economic growth due to the demand created by new products.

Investment in new equipment and processes has become the North American PCB industry's life blood, enabling the industry to maintain its technological leadership as it restarts its engine of growth. There is evidence that this is occurring. Data IPC collected for its *2014 PCB Technology Trends* study, due to be published by the end of this year, indicates that some North American fabricators are already embedding passive components and a few are embedding active components. Many more anticipate embedding both passives and actives within the next five years, as well as incorporating printed-in-place features.

IPC's market research focuses on the core elements of industry vitality: the market (demand), production and inputs (supply), and the enabling technologies. Labor is one of the crucial inputs that is a current subject of IPC's research. A new study on issues in the North American labor pool for electronics manufacturing will also be published by the end of this year.

For more information about the referenced IPC market research reports, visit www.ipc.org/market-research-reports. **PCBDESIGN**

Sharon Starr is director of market research at IPC.

Association Connecting Electronics Industries

IPC®

DESIGN NEWS

DFX — Design for Excellence

Technical education with a design focus

February 20–26, 2015	**IPC APEX EXPO in San Diego, CA** Premier industry event features a range of design offerings • Professional development courses — three hours of classroom instruction led by SMEs • Design Forum — a day of technical presentations from thought leaders in design • Designer Certification

IPC Designer Certification

CID (Certified Interconnect Designer-Basic) and CID+ (Advanced) are valuable professional credentials, earned in three-day sessions that include classroom instruction and testing.

USA sessions

February 20–22, 2015	CID and CID+	San Diego, CA in conjunction with IPC APEX EXPO

For USA sessions register at +1-800-643-7822.

New edition of CID and CID+ course materials will be available starting January 2015.

If you are not already Certified:
Be sure to include the program in your budget for next year — contact your training center for information about class schedule and fees.

IPC Designers Council Members Only
SUBSCRIBE

An international society for individuals with interest in design.
No fees, only benefits — like discounts on designated IPC documents.

Already in Designers Council? Be sure you can access the IPC technical forum for designers worldwide — subscribe at left.

The Road Ahead: 2015 and Beyond

by Martyn Gaudion
POLAR INSTRUMENTS

Editor Andy Shaughnessy kindly offered me the chance to write a column looking ahead at 2015. Predicting the future—that's a tall order. But as I write this in mid-December, the news feeds trumpet that the USA is opening a new chapter in ties with Cuba, and that the price of oil is at $55 and headed south. Really? Did anyone see that coming? What happened to "peak oil?"

I won't even try to go into why. As one UK news channel put it, it's a case of "Pick your conspiracy theory." Needless to say, predicting the future is a tough call, and the most likely scenario is the one you may least expect.

Having said all that, in our electronic world there does seem to be relentless trends towards increasing electronic content in all we see, hear, touch and do. I predict that trend (with the possible exception of the rumour that some celebrities are planning to go retro, with low or no tech in their lives) of increasing electronics in all we experience will continue.

From the Crystal Ball

Electronica 2014 is still fresh in my mind, and notable from a Polar perspective is the increasing interest in signal integrity tools shown by the automotive sector. At one time a commodity and low-tech arena, electronics is increasingly deployed for active and passive safety and to further extract the maximum energy from the minimum amount of fuel. That may seem less relevant given the plunge in oil prices, but the old caveat that accompanies most investments—"the price of oil may go up just as easily and unexpectedly as it went down"—means the automotive desire for efficiency and the need to give the combustion engine a helping (electronic) hand is likely to continue.

Electronic displays are popping up all over the place, with some replacing analog instrumentation completely with a flat panel display, and I also note that Jaguar are toying with using displays on vehicle A and B pillars, directly connected to external cameras to give the driver a full 360-degree view of the road, making the pillars "virtually transparent." Increasingly high-speed communications such as CAN bus are used to control vehicle subsystems driving automotive electronics designers into the high-speed arena. Signaling speed, low cost and durability in harsh environments all come into play. Not only in passenger automotive, but the intensive spread of control systems is also apparent in low-

Privately Held Global Printed Circuit Board Fabrication

LOCAL SUPPORT

and a *WORLD* of Circuit Board Manufacturing Options

USA GERMANY JORDAN TAIWAN CHINA

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

Download Digital Brochure

Download Our Capabilities

HT GLOBAL CIRCUITS

sales@htglobalcircuits.com | (727) 327-6236 | www.htglobalcircuits.com

er volume and specialist commercial vehicles.

From the PCB industry perspective, a reversal in oil prices must surely feed through into lower raw material and transport costs; the only downside could be if the fall is so sharp as to disrupt the economy in an unexpected way. However historically lower oil prices feed through into growth, and for the environmentally minded the lower oil price also buys more time to look for realistic alternate energy sources in the event that prices flow back up again.

Watching the Signals

Whilst looking ahead from a signal integrity perspective, just as in other areas, there seems to be a desire to model more and measure less. Measurement has not gone away; it's just that better modeling tools give everyone a chance to get it right the first time. Measuring in the multi-gigahertz arena is always going to be more costly than at lower frequencies, but what surprises the fabricators more than the price of the instrumentation is that the accessory costs (cables probes, etc.) become an ever-larger (and more delicate) slice of the investment pie when moving to a higher bandwidth test.

"Whilst looking ahead from a signal integrity perspective, just as in other areas, there seems to be a desire to model more and measure less."

No doubt as volumes increase the situation will improve, but the laws of physics dictate that fine-pitch probing and low-loss cabling are never going to be as low cost to source as tools for the MHz arena, but perhaps this is an area where I would be pleased to have an incorrect forecast. Insertion loss measurement is a challenging arena, not least because the industry has yet to converge on a single method. There are many approaches all with strengths and limitations, and the fabricator and OEM need to choose carefully between test speed and capability, and also how much real estate on the panel is sacrificed for coupon provision.

Materials qualification is a lower volume or "one-time" activity, and may favour more real estate on the coupon, possibly with a four-port VNA measurement post-processed through a technique such as Delta L, or a four-port TDR approach such as SPP. For high-volume test and minimum coupon size and a two-port method minimising equipment investment is the volume test method of choice at present. I recently conducted a comparative study of surface finishes with a variety of passivating coatings from SN to AG to ENEPIG and some other newer chemistries, I hope to be able to share more on this as the year unfolds. The study was easy to set up and the comparative insertion loss measurements rapidly made with the minimum of setup by utilising the SET2DIL method. Expect some enhancements in all methods as the year progresses, and as time passes I am confident the industry will start to converge on a small number of proven methods which will drive down the cost of GHz test for the fabrication industry.

Supply chain management is increasingly relevant from the high-speed perspective; I hear more and more people who are working hard to manage geographically stretched supply chains and need to have engineers communicate with fabricators and buyers to ensure PCB materials are correctly specified. Even brokers who historically have been outright commercial operations are now investing in tools for PCB stackup communication. Some leading brokers have done this for several years, but the need is increasingly becoming mainstream.

My last prediction for the year is that we will see a continuation of the exciting developments in health enhancement and the ability to improve our quality of life. The ability to directly control prosthetic limbs through under-skin implants would have seemed the work of science fiction only a couple of decades ago.

Best wishes for a healthy and successful 2015. **PCBDESIGN**

Martyn Gaudion is CEO Polar Instruments.
To contact him, click here.

INTRODUCING COOLSPAN® TECA

thermally & electrically conductive adhesive

Rogers can help by being your reliable conductive adhesive film source

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

CONTACT YOUR SUPPORT TEAM TODAY

ROGERS CORPORATION

www.rogerscorp.com

MEET YOUR COOLSPAN® TECA FILM SUPPORT TEAM

Leading the way in...

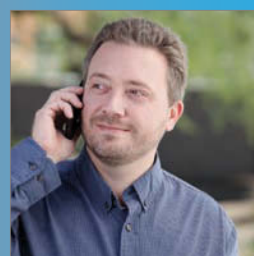
• Support • Service • Knowledge • Reputation

SCAN THE CODE TO GET OUR CONTACT INFO.

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

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

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



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

John Hendricks
Regional Sales Manager
Europe

Kent Yeung
Regional Sales Director
Asia

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

2015: A Look at Moving Trends

by Abby Monaco
INTERCEPT TECHNOLOGY

You know that feeling of excitement after the holidays are over? Getting up with your alarm, digging out clothes that are a little (or a lot) tight after the holiday binge, staring at the bags under your eyes as you brush your teeth, getting into a freezing cold car, and driving in to work for the first time this year? Sure you do.

Well, to quote a song by Death Cab for Cutie, "Well, this is the New Year. And I don't feel any different."

We've all read about the surge in the U.S. job market, the recovery of the real estate market, the jump in the Dow Jones and the NASDAQ, our renewed strength as a world power with our abundance of oil. So, when are all of these positive events going to lead to greater things for our industry?

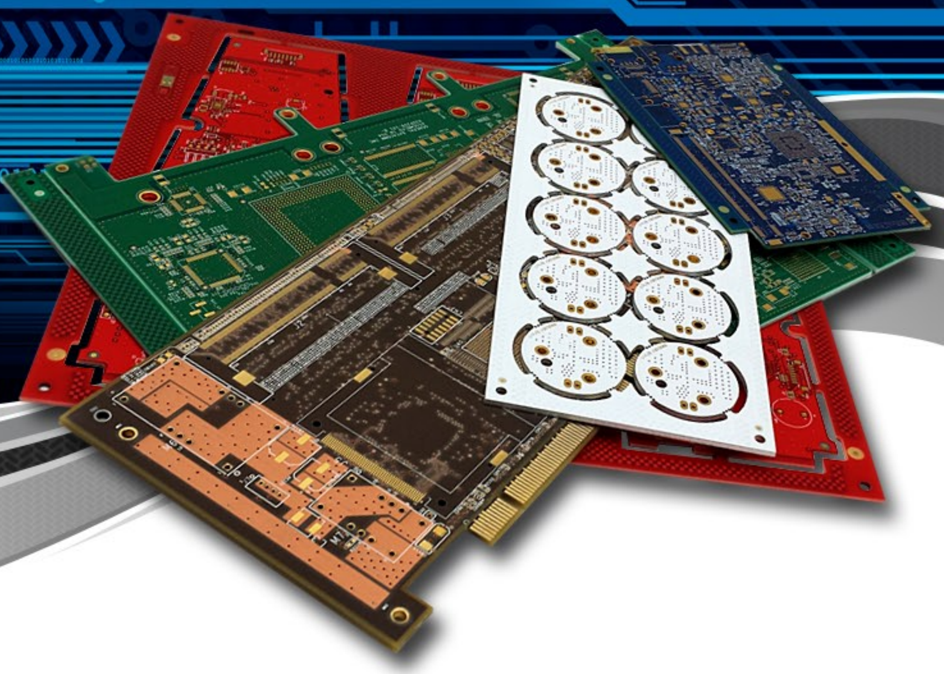
Without going into too much economics, I think we're seeing a bit of a balancing out. A surge in the American markets is met with volatility in the rest of the world, and we are more of a global economy than ever. Americans are no longer as fearful as they were during the recession of years past, but I still don't feel like partying like it's 1999.

From what I've seen here at Intercept, I think it's fair to say that 2014 was a year of building new trends and pouring efforts into researching new technologies, but not exactly getting to capitalize on them—not yet! We've seen some larger customers break new ground with more complex projects, while others have simply been working on new revisions of their tried-and-true staple products. Larger, more cash-rich companies are still forging ahead, and smaller businesses are taking it more carefully.

Regardless of economics, innovation continues in electronics. Sometimes the changing of methods or materials produces even better innovations than would otherwise have come about. Industry continues to mold itself around its economic conditions, and when times are tenuous, it sometimes serves to humanize an otherwise cold machine. Let's take a look at

Figure 1: The trend line of the Global Dow in 2014 shows -1.24% growth.

The Absolute Best Value in High Technology Printed Circuit Boards



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

EAGLE electronics Inc.

MANUFACTURERS OF QUALITY PRINTED CIRCUIT BOARDS

With Eagle, you can expect:

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

click here for a virtual tour of our facility!

www.eagle-elec.com

some of the 2014 trends that are promising to make even more headway into 2015.

Aerospace Exploration

One of the main trends that I found most interesting in 2014 was the increase in the amount of aerospace projects making headlines. The rise of Virgin Galactic and Space X in past years signaled a turning point for the general public's waning enthusiasm for space exploration. The notion of privatized space tourism stirred the entrepreneurial and exploratory spirit of the nation, and has given private space exploration companies momentum to become real contributors.

NASA recently awarded Space X and Boeing contracts to develop, test, and fly space taxis to carry U.S. astronauts into orbit, signaling a growing confidence in newer, smaller vendors to provide leadership. The space exploration arena has always been a tough area for private investors, mostly because of the massive amount of money required to do research and development. I am personally excited and grateful to Elon Musk, founder of Space X, and Sir Richard Branson, founder of Virgin Galactic, for their aggressive moves into the space arena. While 2015 is not likely to see private space tourism take flight, I know we will see great moves toward making it happen.

Figure 2: Unmanned test launch of NASA's Orion spacecraft, Dec. 5, 2014 (courtesy of NASA).

But the excitement of space exploration goes far beyond just the new private companies entering the arena. Don't forget NASA's exploration of Mars with its Curiosity Rover, and the exciting discovery of methane, a possible sign of organic life on Mars. The Rosetta Mission finally saw its day, with the Philae probe making its risky landing on comet 67P. In December was the most exciting news: the first launch of the new Orion space shuttle. I was personally very proud of that, knowing that some of our customers used our software to complete some of the pieces installed on the Orion.

There were also setbacks, such as the Virgin Galactic rocket crash in October of this year, but space exploration is a pioneer effort. Danger lurks everywhere, partly because all of this technology is new, or just barely proven. But with government-funded and private companies now working together to propel us deeper into space, we can be sure that electronics companies associated with space exploration will continue to be major innovators.

3D Printing

Almost every time I picked up a newspaper or trade publication last year, I read about another feat in 3D printing. 3D printing has a huge following because of the possibilities it offers to the individual consumer. It looks as though companies are capitalizing on the slowly recovering retail markets by appealing to the consumer end of the market, but the technology has also become widely explored in the medical fields. 3D printers are already creating successful prosthetics for humans and animals alike, and the R&D into printing organ transplants and reconstructive tissue is well under way.

3D printing could influence the landscape of PCB manufacturing as well, though the technology still seems too new to understand how it will best be put to use. Certainly, the printing of quick prototypes will speed along the design phase, but how it might trickle down to the PCB manufacturing level, or disrupt it, remains to be seen.

A lot of people believe that Amazon has far too much leverage against smaller businesses. Well, imagine if some of these smaller businesses were able to cut costs and offer their products to consumers at lower prices than Amazon by making products available at local stores more readily. Or, perhaps these businesses could offer free direct shipping in exchange for their own cost savings with in-house 3D printing and still make a decent profit. If small businesses can break back into the factory-direct-to-consumer model, this may reduce the power of Amazon's monopolistic structure. It could also signal a resurgence in specialized local stores, if the convenience of purchasing locally becomes more effective than waiting for shipping.

Wearable Tech

From my engineering and software perspective, this set of products honestly has me surprised. But Apple's iTunes also baffled me; I never understood why people would want Apple locking them into music files that can't be transferred. Or why anyone with a Mac would want an iPad. Or why anyone with an iPhone would want an Apple Watch.

Oh, but they do! Forget efficiency; consumer technology is now all about convenience and cool. Apple has achieved an emotional attachment to its technology. After all, what would we do without Siri?

With the smartwatch trend set, we saw Sony come out with its surprisingly affordable SmartWatch, Samsung with its Android-based Gear line, and umpteen other add on options from various vendors. Now the world is holding its breath anticipating the 2015 version of the new Apple Watch. I'm still not so sure why I would want to check my husband's heartbeat on my watch, but I guess there are plenty of people out there who like the idea.

"My view on the fitness watch trend is that it's simply a fad, a new toy for us to buy for gifts because we can't think of anything else. Our smartphones can already do all, or most, of this tracking with the right apps. As for the smartwatch, I'm not sure I like the idea of knowing who texts me, emails me, or calls me the instant I lift my wrist."

Alongside the smartwatch trend, I was surprised to see a less expensive and simpler option flood the market: the fitness watch. Again, I don't feel a need to know how many steps I take in a day, but it seems that a great number of people dig the concept. The popular Fitbit, for example, tracks your steps, distance, calories burned, and active minutes and then synchronizes the information to your smartphone whenever you want to check in and see your progress. It even monitors your sleep patterns.

Do any of you own smartwatches or fitness watches? What will you choose in the coming year? Do you think these products are desirable or just added noise in a glutted technology market?

My view on the fitness watch trend is that it's simply a fad, a new toy for us to buy for gifts because we can't think of anything else. Our smartphones can already do all, or most, of this tracking with the right apps. As for the smartwatch, I'm not sure I like the idea of knowing who texts me, emails me, or calls me the instant I lift my wrist. The kids who can't put their phones down might like it, but as a parent leery of kids addicted to tech, I can assure you I will make no such purchase. How will these wearable tech products evolve or fail moving into 2015?

All in all, I believe that the trends in the consumer, medical, and aerospace segments in

2015: A LOOK AT MOVING TRENDS *continues*

Figure 3: The Fitbit Flex, 2014's cool new wearable device.

2014 are positive harbingers for the PCB industry, offering us insight into 2015 and beyond. When times are tough, we find ways to innovate. I believe that 2014 was a year in which we continued to gain momentum, a year when shelved ideas and projects were dusted off and resumed. My view is that 2015 will be a year of positive growth, maybe not swift and not without bumps along the way, but growth nonetheless.

What do you think about 2015? Do you disagree, or see growth in other areas of our industry that I didn't mention? Tell me what you think. Have a great new year! **PCBDESIGN**

Abby Monaco, CID, is a director of products and marketing for Intercept Technology. She can be reached by clicking here.

---

video interview

IPC Focuses on Training Project

by Real Time with... HKPCA & IPC EXPO 2014

Guest Editor Bob Neves and IPC President Dr. John Mitchell sat down to discuss IPC's shows and its expansion of certification and training initiatives. Mitchell jokes that IPC is "moving into the 20th century" by moving more training online.

CLICK TO VIEW

realtimewith.com

“ Providing *Solutions* to Board Fabrication Challenges”

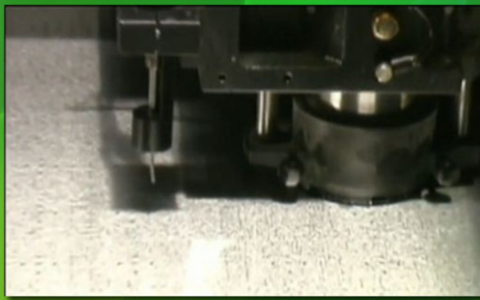
FROM CONCEPT TO COMPLETION

Solutions for Every Complex Situation

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

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

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



REQUEST A QUOTE

WWW.MULTILAYER.COM

ITAR

Mil-PRF-55110

Certified System

AS 9100

Multilayer Technology
3835 Conflans Rd
Irving, TX 75061-3914

(972) 790-0062

What Will 2015 Bring?

by Mark Thompson, CID
PROTOTRON CIRCUITS

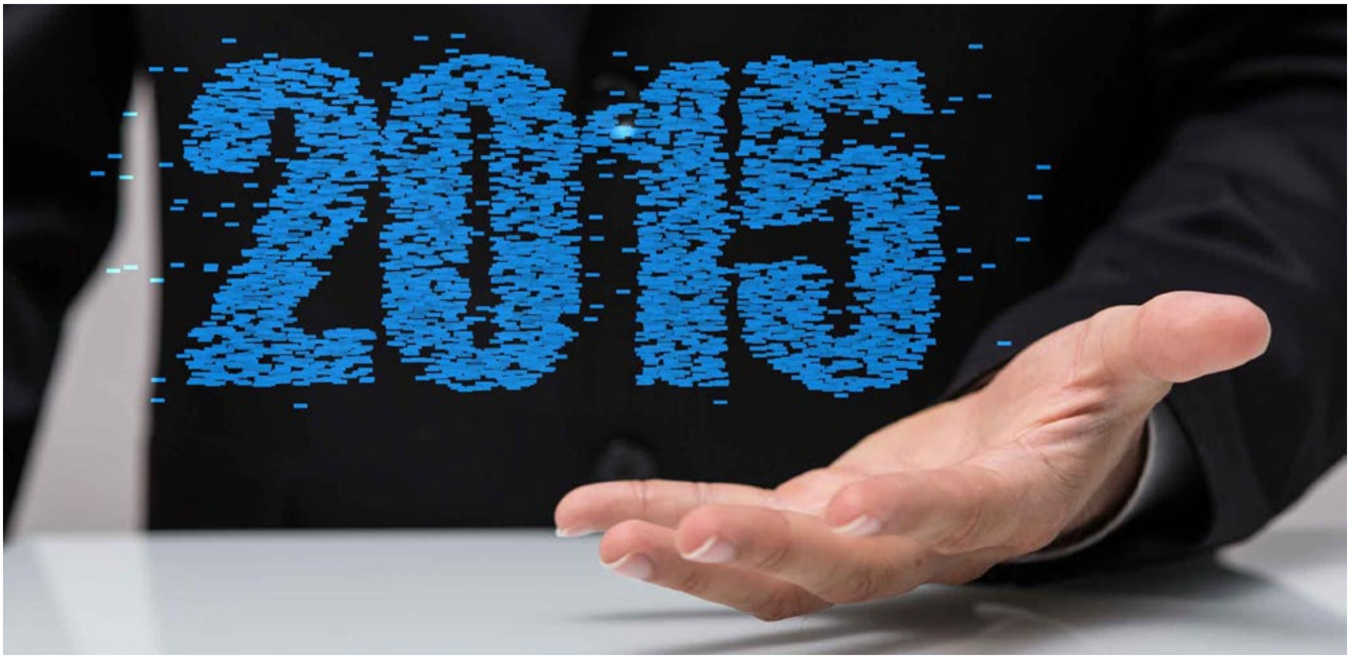
I've been thinking over what 2015 might look like, from my point of view at a PCB fabrication company. Let me first start out with some broad overviews of trends from 2014 that I see continuing.

1. More RF work. Whether it's phased arrays or waveguides, ID tags or implants, it's not just cell phones and tablets that require this technology; we're seeing more young people gravitating towards careers in RF design and applications. From a fabrication standpoint, I see a lot more people are asking the right questions:

- How do I ensure the highest degree of positional accuracy for a given feature?
- How do I make sure a fabricator will not trim back any RF launches I may need in an effort to avoid rout burring?

By its very nature, RF is different in many ways at the fabrication level. For instance, as fabricators we may be more concerned with space or air-gap issues based on etching, but in an RF world, geometry is everything, in both metal and non-metal areas. All calculations and estimates are based on very specific metal and anti-metal geometries, and they need to be as perfect as they can be. Material substitutions can require an entire rethink of an RF design. Small variations like surface topography post-via fill /planarization can create unwanted variations as well.

This applies all the way down to little things that everyone is starting to embrace, like specifying a starting or finished copper weight on drawings and "read-me" notes. Typically, the word "weight" means a STARTING copper weight prior to final plate-up for through-hole continuity. The use of the word "finished" means copper foil AND the associated electroplated final plate-up. This needs to be clearly understood. Many times customers will run simulations and calculations based on 1 oz finish after plate. Please note: Whereas many offshore fabricators have no problem doing this, remember that IPC



INTRODUCING THE REVISION TRACKING HEADACHE AVOIDING FILE MANAGING KNOW WHO DID WHAT WHEN OrCAD EDM

Data Management Made Easy

Let OrCAD® EDM handle your data management problems so you are free to focus on PCB design. OrCAD EDM puts revision control, check-in & check-out support, page management, and team configuration capabilities all inside the OrCAD Capture interface. This allows you to focus on creating new and differentiated products instead of trying to figure out which "version 7" of the latest design is actually the latest.

Learn how OrCAD EDM solves the PCB data management problem. View Demo ›

OrCAD™
CADENCE PCB SOLUTIONS

EMA Design Automation®

dictates a minimum of .8 to 1 mil in the barrels of plated copper for continuity. So, here in the U.S., even starting on quarter-ounce outers will result in 1.5 oz. finish after plate typically. If you have numerous impedances on the surface, having to redo them all because this was not considered will result in having to redo all the calculations, which takes more time. The old adage, "Time is money" is certainly true in the PCB fabrication business, especially with RF parts.

2. More hybrid analog digital parts. Why make a digital part that may require a series of smaller piggybacked or surface mounted parts when you can get the best of both worlds (depending on the application) with a part that is a hybrid of analog and digital type materials? A combination of high-speed analog RF materials where surface CPWGs can be maintained at a reasonable geometry and digital signaling can be kept to the inside in the form of DDR, SATA or LVDS lines.

3. More surface finishes for lead-free assemblies. Wait and see. The surface finish segment is likely to keep growing with demand for lead-free PCBs.

What This Means to Designers

These first three items alone should be enough to convince you that you should take some time to discuss your specific needs with any potential fabricator partner. A simple 15-minute conversation can save everyone a lot of grief.

A recent example:

One of our longtime customers recently started adding edge slots and cutouts as separate entities instead of with the part profile or outline. After discussing this with the customer, we found the reason. Their thought process was that they understood slot positional accuracy to be around +/-.003", whereas the rout positional accuracy could be as much as +/-.005". What they really wanted was edge features relative to inboard features with an accuracy of about +/-.003".

We fabricators can simply say, "Make sure these additional edge-routed positional features are done at primary drill when all the other holes are done." This ensures a better positional relationship than two separate entities. Additionally, we came to find out this was a suggestion from another fabricator! Some fabricators propose solutions that may not work for all fabricators. Again, having a short conversation can sometimes lead to a much greater understanding of the issue and much quicker solutions to the problem.

> "Some fabricators propose solutions that may not work for all fabricators. Again, having a short conversation can sometimes lead to a much greater understanding of the issue and much quicker solutions to the problem."

I also see many customers finally embracing simple things like differentiating impedance line sizes by a tenth or even a hundredth of a mil; this identifies them but keeps them close to the original intended and calculated line size. This is considerably faster than attempting to identify each impedance scenario by the net names they are associated with or their component locations.

Other Predictions for 2015

I predict the continued use of holes .008" and smaller based on board geometries and available real estate. I see more usage of .006" vias to help reduce associated pad sizes and still be able to ingress and egress out of fine-pitch parts. More folks are specifying these as +.000"- the hole size and more folks are either silver-filling them (if they need the extra conductivity for, say, heat considerations) or epoxy-filling them just so they are flat at assembly.

I also see more use of true .003"/.003" spaces and traces on surface layers for the same reasons. As the pitch between the BGA balls and vias shrinks, so too will the associated trace widths. We are seeing a lot of true .003"/.003" for .5 and .4 mm pitch devices. Bear in mind that in a fabrication environment, these will start on no more than quarter-ounce copper foils. I harken back to previous columns about the general rule of thumb for etch compensations based on starting copper weights. For every half ounce of copper, we do a half mil of compensation, so that when all is said and done after etch the trace is as the customer desires. For true .003"/.003", if we attempted to start with .5 oz., 1 oz., or 2 oz. copper, we would NOT be able to impose an etch compensation and still be able to process the resulting space. This is one you will want to talk with your fabricator about.

I also see many more alternative surface finishes being used this coming year. For the past five years, the surface finish of choice has been electroless nickel immersion gold (ENIG). I see more of that, but I also see more electroless nickel electroless palladium immersion gold (ENEPIG) being used for telecommunications devices, tablets, cell phones, etc.

Lastly, I foresee more interaction with the board houses. As the design complexity grows there are more challenges for fabricators. And I imagine we'll have many more new customers with new design engineers who are not dissuaded by the speed bumps of the past. I see this as an opportunity for the U.S. fabricators to help educate the customer about how we operate and what we can do for future board fabrication.

Not all solutions require a new process or new technology. Many times it is simply a matter of understanding your customers' desires so that you become a part of their team and volunteer solutions.

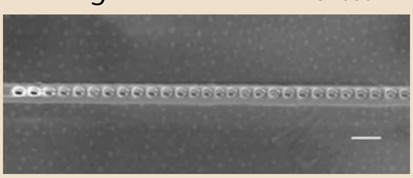
Many times it is as easy as material and surface finish selection based on their applications; other times it may require process changes. Either way, the extra few moments you spend understanding your customers' needs will pay off in time and money.

As always, thanks for your time. I appreciate any feedback. Please feel free to contact me. **PCBDESIGN**

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

Diamond Defect May Lead to a Qubit Computer

In the race to design a universal quantum computer, a special kind of diamond defect called a nitrogen vacancy (NV) center could play a big role. Now a team of researchers has taken a major step forward in effectively enhancing the fluorescent light emission of diamond nitrogen vacancy centers—a key step to using the atom-sized defects in future quantum computers. The technique hinges on the very precise positioning of NV centers within a structure called a photonic cavity that can boost the light signal from the defect.



Photonic cavities best enhance the signal of NV centers located in a "hot spot" where the cavities' resonant fields are strongest, but making sure an atom-sized defect's location matches up with this spot is extremely tricky.

The team took an important first step toward this goal by controlling the depth of the diamond defects using a technique called delta doping. The technique confines the possible location of NV centers to a layer approximately 6 nanometers thick sandwiched inside a diamond membrane approximately 200 nanometers thick. The researchers then etched holes into the membrane to create the photonic cavities.

The team believes they can further enhance the emission by also controlling the position of the defects in the horizontal plane and are currently working on possible ways to achieve full 3-D control.

PCB007
News Highlights

TTM, Viasystems Receive Second Request from FTC

TTM Technologies, Inc. and Viasystems Group Inc. announced they have each received a second request from the United States Federal Trade Commission for additional information associated with TTM's proposed acquisition of Viasystems.

Dragon Circuits Battles Industrial "Planned Obsolescence"

The company is announcing an ongoing effort to combat planned obsolescence. Vice President Rajan Babaria says, "I recently discovered complaints about an Asian-made cable (for a gimbal) that was designed to break easily, ensuring customers would have to repurchase after a just a few uses. Our team discussed it and had it redesigned and built with better, more durable material."

All Flex Invests in Capital Equipment

All Flex has improved its component inspection capability with the purchase of an inline cleaner and an automated optical inspection (AOI) system to handle flexible circuits with high volume components. The AOI has the latest optics technology that rapidly scans for placement accuracy and the inline cleaner has an advanced system that helps deliver defect free parts.

N.A. PCB Book-to-Bill Ratio Recovers in October

"Unusually strong growth in North American PCB orders in October is good news for an industry that has experienced declining sales for more than three years," said Sharon Starr, IPC's director of market research. "The impact can be seen in the book-to-bill ratio, which made a strong comeback this month, indicating that sales may begin to rebound by the end of the year."

Infineon Acquires 9.4% Stake in Schweizer Electronic

By investing into Schweizer, Infineon emphasizes its intention to jointly develop technologies for the integration of power semiconductors into PCBs and to tap the chip embedding market for high-power automotive and industrial applications.

Cicor Wins DeviceMed Award

The DeviceMed Awards were presented at the Compamed in Düsseldorf, the leading international trade fair for medical technology industry suppliers. Cicor came out on top in the Outsourcing Partner category.

South Korean FPCB Firms See Sharp Fall in Sales

In 2014, price decline continued to deteriorate, which led to a sharp fall in revenue of South Korean vendors. The implications of high-speed capacity expansion are high fixed costs, which would thus result in a plunge in profit margin.

Saturn Electronics Now UL Certified for Aismalibar IMS

The certification allows Saturn to UL stamp their products using the entire line of Cobritherm materials which includes ALCUP-G 1.3w, ALCUP 1.8w, HTC 2.2w, and HTC 3.2w. The company can now offer a high-quality European alternative the small group of IMS materials currently available in the North American market.

Candor Invests in Equipment from atg, Olec, Pluritec

Canadian high-technology PCB manufacturer Candor Industries announces it has improved overall production, speed, and capability through recent capital investments from atg, Olec, and Pluritec.

Aspocomp to Cease Operations at Teuva Plant

It was decided that a total of 34 personnel, consisting of 28 non-salaried and six salaried employees, will be made redundant and production at the Teuva plant will be closed down.

Onshore. Genuine. ITAR Registered.

Accept no substitutes.

"Made in America" isn't just a slogan for us. We're proud to be an **onshore** company and, along with our partners at eSurface, we are committed to being an ITAR-registered manufacturing facility.

Using only genuine components is just one of the reasons why we proudly stand behind the products we produce for our military and medical customers.

Murrietta Circuits is proud to be an eSurface Licensed Manufacturer

Click to learn more

Registered ITAR International Traffic in Arms Regulations

Murrietta Circuits

The Integrated Solution

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

column

THE TOWN CRIER

True MCAD-ECAD Architecture: A Common-Sense Approach

by Daniel J. Smith
RAYTHEON

In the fall of 1998, I authored a white paper about an entire design project that would be controlled by a single database, and all hardware and software design tools would "check in and check out," only the design data that each engineering discipline needed. Within this universal database, all of this data fell into two categories: a) common data, and b) proprietary data (trade secrets and patents).

I actually approached the CEO of the company I was working for several years ago (not my current employer) with this idea. Although he understood its merits, he did not attempt to fund this next-generation design environment or request that I submit a patent to protect it. Instead, he allowed me to introduce this concept to the body of the "corporate think tank" over several months. Some feedback from forward-thinking people like me was very positive, but for the most part, I was greeted by skeptics who preferred to be naysayers. You know: "Too expensive, too this, and too that."

At DAC 2000 in New Orleans, I came upon a spritely elderly lady named Hilary Kahn, who introduced me to what she and her students had done at the University of Manchester with the EDIF 4.0 standard. Perhaps there was a pre-Harry Potter spell that she cast onto my inquisitive mind, but I actually saw mechanical and electrical tools (although they were the lesser EDA tools of the day) interact with the same design data for their specific engineering needs. This integration was not at the scale of my white paper, but I believed in the goodness

ventec

A HIGHER DEGREE OF THERMAL CONDUCTIVITY, WHERE AND WHEN YOU NEED IT.

	Thermal Conductivity	Ceramic Filled	Fully Formable	Non-Glass Reinforced	Halogen Free
VT-44A	1.0W/m.K.	✓			
VT-4A1	1.6W/m.K.	✓			
VT-4A2	2.2W/m.K.	✓			
VT-4B1	1.0W/m.K.	✓	✓	✓	✓
VT-4B3	3.0W/m.K.	✓	✓	✓	✓
VT-4B5	4.2W/m.K.	✓	✓	✓	✓
VT-4B7	6.5W/m.K.	✓	✓	✓	✓

VIEW THERMAL MANAGEMENT PRODUCTS

Wherever technology takes you, we deliver.

Ventec Europe
www.ventec-europe.com

Ventec USA
www.ventec-usa.com

Ventec International Group
www.venteclaminates.com

of a more integrated design environment than what was currently commercially available.

When I approached a major EDA tool vendor at that show about what they were working on to integrate this standard, I was greeted with a stern, "We will never consider it, because we cannot expose our proprietary IP!" Although I understood the IP argument made complete business sense, I still believed at some point in the future that the engineering masses, the merging EDA companies, and the power of computers and networks would be at a state where all of these current barriers would be gone.

In the 2000s, all of these barriers did erode. Even the claim that EDA proprietary IP was sacred is no longer valid, as major companies have switched from one EDA vendor to another on multiple tools. Companies have recreated their IP on to their new EDA systems, and have now gone forward designing products with their new EDA vendors' tools and their IP.

But despite all of this progress, a true MCAD-ECAD collaborative integrated design process tool environment has never materialized. In the simplest Dr. Phil terminology, EDA vendors still don't get it.

I have tried to use some of these tools. I've sat down with these companies' integration experts, and I've offered them insights of a design process system architect. Later, after they tell me that they have listened, I've gone back to their developers who now want to show me their new roadmaps. I still leave these meetings looking at a giant Error 404 expression on their faces. I am not alone in my beliefs, as other senior industry experts have privately confided to me that they feel and see the same non-direction.

So, rather than talking to more experts, going into more technology meetings, or reading press clippings on these new collaborations just to be disappointed yet again, I am willing to offer the following solution on behalf of all engineers who have been waiting for a true MCAD-ECAD tool over these years. The goal of the following roadmap is to create a complete bi-directional, fully integrated tool within the next five years. By 2020, a true cost-affordable MCAD-ECAD tool needs to be available to every project engineering team.

I can already hear the naysayers of the existing EDA tool vendors raising their voices. But, for those of you who have not ever watched a single episode of Star Trek, remember how the Starship Enterprise and all of its technology came about: They just did it.

> "I can already hear the naysayers of the existing EDA tool vendors raising their voices. But, for those of you who have not ever watched a single episode of Star Trek, remember how the Starship Enterprise and all of its technology came about: They just did it."

So, here is my 10-step strategic roadmap to create this MCAD-ECAD collaboration environment tool:

1) There needs to be a declaration by every EDA tool company within a month explaining whether they are in this initiative or out. It's that simple. Companies' strategic business leaders, the CEOs and CFOs, need to know who is willing to join this coalition, and who still thinks that they can do this independently (or isn't even interested in MCAD-ECAD). PS: You must remain committed throughout its entirety, and not back out because you and your company may not benefit directly. This initiative is for the majority of the engineering user community and NOT your egos.

2) There needs to be an independent leader who will drive the development of a true MCAD-ECAD standard built on a consensus for commonality. "Independent" means a person who is not tied to an EDA vendor in any way, shape, or form. The EDA vendors participating in this standard creation will be allowed to provide many of their SMEs (subject matter

experts), but they are to only be contributors. This industry has enough legacy examples of someone within a certain company asserting their influence so that they specifically come out on top.

3) This new MCAD-ECAD standard will contain a mapping of all of the properties that exist in every EDA design tool. There are enough existing mapping standards to provide reference. Study them all, extract the best information (including corner use-cases), and structure them in a data format that all can see and understand without requiring a master's degree in computer science or database management. KISS (keep it succinctly simple).

4) Once this standard has been finalized, a single MCAD-ECAD collaboration tool should be made by a new EDA company that will submit a business proposal for creating this tool, and it will be funded by all of the MCAD-ECAD EDA tool participants. That means all current EDA tool vendors can stay working on exactly what they are specialists in, and someone else will take on this responsibility of "getting it right the first time."

5) Both the EDA tool vendor and the user have the right to label any property as IP to be shared in this collaborative environment. Be frugal with this label, as it should be very well assumed based on today's network shortcomings that too much protection attracts the wrong kind of people to exploit hidden information. Anything that is not labeled IP is made available for this collaborative specification.

6) This collaborative environment should be able to take all of the available design data and send it bi-directionally between all tools seamlessly.

7) Create outputs to all existing formats for all downstream vendors of the MCAD-ECAD design process that are easy to setup and output as a "one-button solution."

> When the tool is created and released, the EDA vendors who funded this project will get a percentage back of the sales of this new tool from the new MCAD-ECAD company. This can be downgraded over time, so I leave these specifics to the business experts.

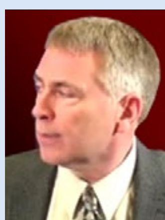
8) When the tool is created and released, the EDA vendors who funded this project will get a percentage back of the sales of this new tool from the new MCAD-ECAD company. This can be downgraded over time, so I leave these specifics to the business experts.

9) This MCAD-ECAD company cannot be purchased by any of the current EDA tool vendors for a period of at least 10 years. If any EDA tool company wishes to purchase this new MCAD-ECAD company for their own portfolio after this deadline, the purchasing company must allow for a five-year grace period where the standard MUST remain open and not proprietary for at least five years.

10) If and when No. 9 were to happen, this would give all of the other EDA vendors a chance to decide on what to do going forward.

I am 100% on the side of the EDA tool users who really want a quality seamless MCAD-ECAD no-nonsense tool solution by 2020. Please e-mail your feedback to Editor Andy Shaughnessy, who has agreed to post responses to this column.

Thank you all for your time, as always. **PCBDESIGN**



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

column

BEYOND DESIGN

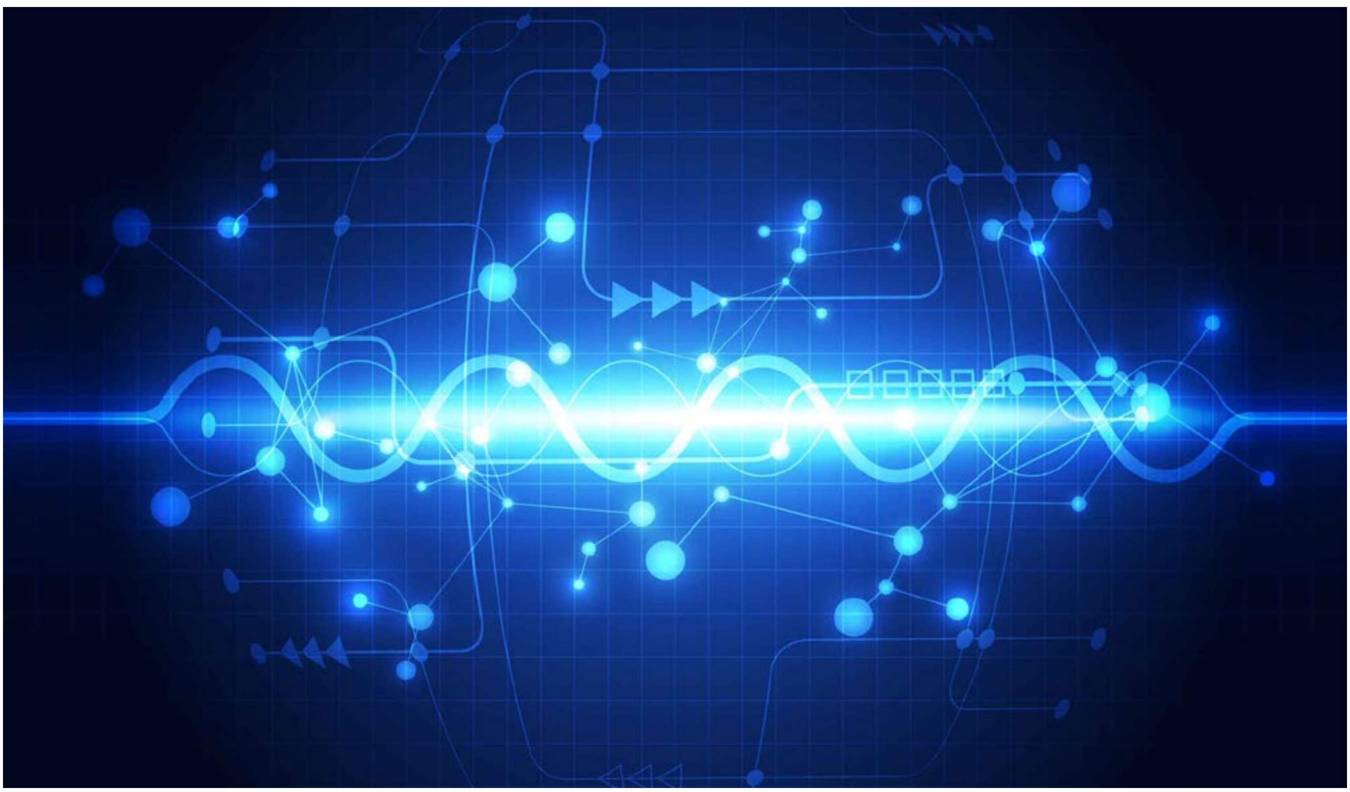
Electromagnetic Susceptibility

by Barry Olney
IN-CIRCUIT DESIGN PTY LTD

As PCB designers, we are concerned with electromagnetic emissions, as every product we design must pass the FCC/CISPR compliancy, but what about susceptibility to external sources? Noise sources range from medium to high frequency, RF and microwave radiation, and can be generated by nearly any electrical appliance or device. Interference tends to be more troublesome with older technologies, but is also prevalent in modern-day digital systems. There are ways to combat this noise such as in Wi-Fi, where error-correction techniques can be used. Spread-spectrum and frequency-hopping techniques can also be employed with both analogue and digital signaling to improve resistance to interference.

On the other hand, cosmic radiation (rays) is all around us. Subatomic particles—mostly protons and helium nuclei—of extremely high frequency are constantly streaming from distant galaxies and are of such short wavelength that they pass right through everything around us. These particles can wreak havoc in electronic systems in a number of ways. One of the most common is called “a single event upset,” in which cosmic rays ionize atoms in a semiconductor, releasing a burst of electrons that can flip a digital bit from, say, a 1 to 0. This is known as a soft error, but can still blue-screen a computer. A hard error or “a single event burnout” is more serious in which components are damaged or destroyed by a sudden short-circuit caused by the burst of electrons.

In order to cram more processing power into integrated circuits, the last decade has seen the size of transistors within ICs shrink from 180nm to less than 20nm. According to a study conducted by Oracle, this comes at a price:



How can you find the best PCB Partner? We can tell you. We wrote the book on it.

Since 1985, U.S. Circuit has been a premier supplier of both commercial and military Printed Circuit Boards in the United States. We know what it takes to be a good PCB partner. And we want to share what we've learned with you.

Choosing a PCB partner isn't hard—if you know what to look for. We've broken it down into "The 5 Commandments" and you can download it for free.

We are proud to be the Circuit Board Manufacturer of more than 400 growing companies, and even prouder to be a good partner for all our customers!

Click to Download

U.S. CIRCUIT

U.S. Circuit, Incorporated
2071 Wineridge Place
Escondido, CA 92029
+1 (760) 489-1413
www.uscircuit.com

ISO 9001:2008 REGISTERED

ITAR

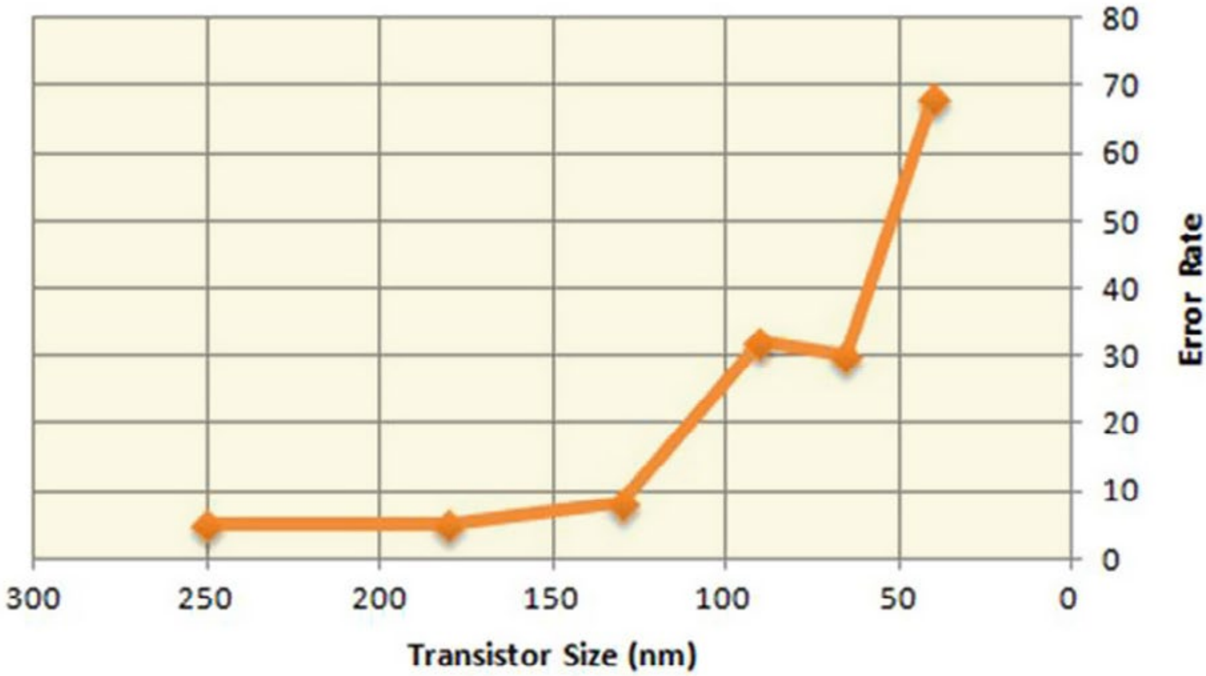


Figure 1: Transistor size vs. microprocessor error rate (source: Oracle).

Cosmic rays create eight times more soft errors in ICs with 40nm transistors than those with 130nm. As transistors shrink, the amount of charge required to incite the circuit is reduced. This trend is also evident in the drive for more efficient chips that run at low voltages; circuits running at 0.5V have twice the rate of soft errors as those running at 0.7V.

Not surprisingly, the space industry began designing its craft with these problems in mind way back in the Apollo mission days. Thin sheets of gold foil, less than 0.15mm thick, were used in space programs as a radiation shield. The lunar modules of the Apollo flights were shrouded in foil. Unfortunately, this thin gold foil offers no obstruction to the extremely short wavelength of cosmic rays. However, it may have offered the astronauts some protection from solar radiation, but provided little shielding for sensitive electronics. More recently, materials that have high hydrogen content, such as polyethylene, have been used to reduce radiation to a greater extent than metals, such as aluminum. Hydrogen atoms are good at absorbing and dispersing radiation. Demron, a material said to have radiation protection similar to that of lead shielding while being lightweight and flexible, is also being trialed. Spacecraft de-

Figure 2: Gold foil acts as a radiation shield on the Apollo lunar module (courtesy of World Gold Council).

Figure 3: RAD HARD device (courtesy of Intersil).

signers must be able to shape shielding materials to make various parts of the spacecraft. The material must protect the crew from radiation, and it must also deflect dangerous micrometeoroids.

Fortunately, back down on Earth we are much safer from cosmic rays. Earth's atmosphere offers the same protection as a layer of concrete four meters thick. Incoming particles strike atoms in the atmosphere, producing an avalanche of protons and neutrons that in turn create a shower of millions of fragments—gamma rays, muons, neutrinos, electrons and other particles—that rain down over several square kilometers. Those reaching sea level are thought to be too weak to have any significant impact on electronic systems.

Cosmic radiation intensity can also increase during periods of strong solar activity (flares), where masses of electrons, created by thunderclouds, can produce bursts of radiation known as terrestrial gamma ray flashes. These flashes can pack more of a punch than cosmic rays and can dramatically affect aircraft electronics.

The electronics industry has responded by developing a number of ways to combat the cosmic ray threat. The most common defense against soft errors is error-correcting software, typically involving check bits sent along with each packet of data, which is used to confirm that the contents have not been corrupted. In addition, multiple copies of data are stored in different locations in memory. In particular, vulnerable or critical systems like those on satellites or aircraft have their complete hardware systems built in triplicate.

Environments with high levels of ionizing radiation create extraordinary design challenges. A single charged particle can knock thousands of electrons loose, causing electronic noise and signal spikes. This is a particularly serious problem in the design of satellites, spacecraft, military aircraft, nuclear power stations, and nuclear weapons. In order to ensure the proper operation of such systems, manufacturers of integrated circuits and sensors, intended for the military or aerospace markets, employ various methods of radiation hardening. Radiation-hardened (RAD HARD) components are based on their non-hardened equivalents, with some design and manufacturing variations to reduce the susceptibility to radiation damage.

Radiation-hardening techniques:

- Hardened chips are often manufactured on insulating substrates instead of the usual semiconductor wafers. Silicon on insulator (SOI) and silicon on sapphire (SOS) are commonly used. While normal commercial-grade chips can withstand between 5 and 10 krad, space-grade SOI and SOS chips can survive doses many orders of magnitude greater.
- Bipolar integrated circuits generally have higher radiation tolerance than CMOS circuits. The low-power Schottky (LS) 5400 series can withstand 1,000 krad, and many ECL devices can withstand 10,000 krad.
- Magnetoresistive RAM, or MRAM, is considered a likely candidate to provide radiation hardened, rewritable, non-volatile conductor memory. Physical principles and early tests suggest that MRAM is not susceptible to ionization-induced data loss.
- Shielding the package against radioactivity, to reduce exposure of the bare device.
- Capacitor-based DRAM is often replaced by more rugged (but larger, and more expensive) SRAM.
- Shielding the chips themselves by use of depleted boron in the borophosphosilicate glass passivation layer protecting the chips, as boron-10 readily captures neutrons.

So, it seems that PCBs cannot be totally shielded against the impact of cosmic rays. However, we can take precautions to avoid interference from shorter-wavelength radiation that is more prevalent in our environment.

1. Route high-speed signals between the planes, fanout close to the driver (200mil) dropping to an inner plane and route back up to the load again with a short fanout. This will help shield the sensitive signals.
2. Keep critical signals away (200mil) from the board edges.
3. Avoid short stubs as these are more compatible with shorter wavelengths.
4. Use differential pairs for high-speed signal routing as their equal and opposite polarity rejects common-mode noise.
5. Use higher operating voltage components with a larger die size where possible.

Embedding signals between the planes reduces susceptibility to radiation, as well as providing ESD protection. In doing so, not only do we prevent noise from being radiated but we also reduce the possibility of being affected by an external source.

So next time your PC blue screens, your cruise control locks up or your 747 suddenly decides to do a death dive—it may just be a random glitch from cosmic rays or a solar flare.

Points to Remember

- Noise sources range from low frequency to high frequency, RF and microwave radiation and can be generated by nearly any electrical appliance or device.
- Cosmic rays can wreak havoc in electronic systems in a number of ways.
  – One of the most common is called "a single event upset," in which cosmic rays ionize atoms in a semiconductor, releasing a burst of electrons.
  – A hard error or "a single event burnout" is more serious in which components are damaged or destroyed by a sudden short-circuit caused by the burst of electrons.
- Cosmic rays create eight times more soft errors in ICs with 40nm transistors than those with 130nm.
- Circuits running at 0.5V have twice the rate of soft errors as those running at 0.7V.
- Materials that have high hydrogen content, such as polyethylene, have been used to reduce radiation to a greater extent than metals, such as aluminum.
- Demron, a material said to have radiation protection similar to that of lead shielding,

while being lightweight and flexible, is being trialed.

- Earth's atmosphere offers the same protection as a concrete layer four meters thick.
- The most common defense against soft errors is error-correcting software. In addition, multiple copies of data is stored in different locations in memory. Critical systems, like those on satellites or aircraft, complete hardware systems are built in triplicate.
- Manufacturers of integrated circuits and sensors, intended for the military or aerospace markets, employ various methods of radiation hardening.
- Printed circuit boards cannot be totally shielded against the impact of cosmic rays; however, we can take precautions to avoid interference from shorter wavelength radiation.
- Embedding signals between the planes reduces susceptibility to radiation, as well as providing ESD protection.

For information on the ICD Stackup and PDN Planner, visit www.icd.com.au. **PCBDESIGN**

References

1. Barry Olney: Beyond Design: Embedded Signal Routing
2. David Hambling: "Burnout," New Scientist, September 2014
3. Cygo.com: "Space Radiation Shielding"
4. Nasa.gov: Radiation shielding research
5. Wikipedia: Radiation hardening

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

video interview

European Industry Leaders Discuss China Partnership

by Real Time with...
HKPCA & IPC EXPO 2014

Rex Rozario, executive chairman of Graphic PLC, and Giovanni Tridenti, CEO of SOMACIS SpA, join guest editor Bob Neves to discuss the success of Dongguan SOMACIS Graphic PCB Co. Ltd. (DSG) in southern China. After nearly 10 years of cooperation and following the market, efficiency is still a priority and the company recently invested $10 million on new equipment.

realtimewith.com

Mil/Aero007 News Highlights

ACE Recertified as Northrop Grumman Preferred Supplier

ACE has held this prestigious certification for the last five years and is extremely honored to have again been certified by NGC's Supplier Quality Team.

Eltek Suffers from Defense Sector Decline

Yitzhak Nissan, chairman of the Board and CEO, commented, "Eltek's revenues in the third quarter were still weak. We experienced a decline in the demand by the domestic defense sector for printed circuit boards manufactured locally. This decline was not sufficiently compensated by sales to the domestic civilian market. In addition, there was a reduction in orders received from the United States."

Murrietta Recognized as Raytheon Three Star Supplier

After receiving their second 5 Star award earlier this year, another Raytheon division, Space and Airborne Systems, awarded Murrietta their 3 Star Supplier Excellence Award. The annual award was instituted to recognize suppliers who have provided excellent service and exceeded customer requirements.

Invotec Enjoys Strong Demand from Space Sector

In addition to NADCAP and AS9100 Rev C accreditation, Invotec is one of the very few PCB manufacturers to have been awarded two separate ESA approvals: one for sequential rigid circuits and one for sequential flex-rigid circuits.

Multicircuits Earns UL Cert for Bergquist Metal-clad Materials

Mike Thiel, director of Operations at Multicircuits, announces his company achieved UL Certification for a number of Bergquist thermal-clad materials including high temperature (HT) and multi-purpose (MP).

PEC Debuts Contest to Spur Manufacturing Innovation

At a meeting of the President's Export Council (PEC), President Obama will announce nearly $400 million to help improve the competitiveness of American businesses and workers by spurring new manufacturing innovations and giving America workers additional opportunities to improve and expand their skill sets for middle-class jobs.

Global Business Aviation Reaches Cruising Altitude

Global business aviation activity saw modest gains in Q3 2014, with 2% growth quarter-over-quarter and 2.2% growth year-over-year, according to JSSI's most recent Business Aviation Index.

Security, Economic Concerns Fuel Military Radar Market

The worldwide market for military radar was worth US $6,900 million in 2012 and will reach US $8,440 million in 2019, registering a 2.9% CAGR during the forecast period.

Military Radar Market Continues on Growth Trajectory

"The total number of radar shipments is forecast to grow at a CAGR of 4.1% through 2023, to reach 1393 units," notes Eric Higham, North American director for ADS.

2014-2024 Electronic Warfare Market Forecast

The global electronic warfare (EW) market is currently going through a period of growth, as many countries are realising its benefits and attempting to expand their capabilities therein.

www.5pcb.com

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

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

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

Create free account

DME

D.M. Electronic International Co., Ltd.

column

DESIGN FOR MANUFACTURING

Tips Every Designer Should Know

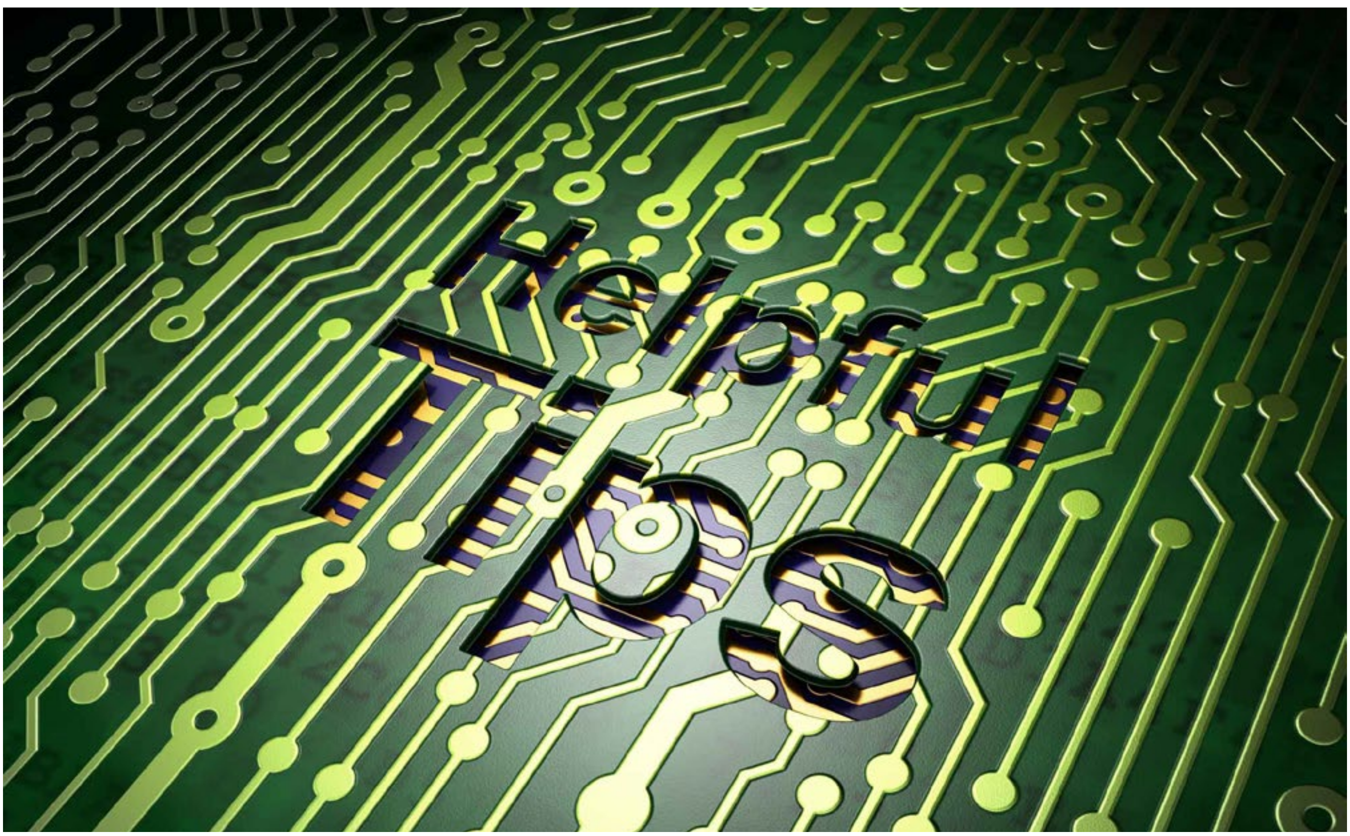
by Amit Bahl
SIERRA CIRCUITS

As the chief PCB designer resident within a company that manufactures circuit boards, Greg Albers has unusual insight about how fabrication and assembly processes ought to influence design. In his position at Sierra Circuits, he's often asked by customers to recast designs that would be tricky to build as submitted, in addition to taking projects from schematics forward to achieve optimum yield.

Albers' career spans 30 years, including stints within several major OEMS—among them a network card supplier and a manufacturer of computers and smartphones—before joining the company. He's designed thousands of boards. I asked him to pass along some basic PCB design tips, simple reminders based on the path he treads to ensure the best outcome from a manufacturing perspective.

A Few Words to the Wise

First of all, Albers emphasizes, engineers and designers should understand the unique relationship between the fabrication process, design rules, and the costs of different design approaches. The easiest and definitely the cheapest way to accomplish that is through consultation with the intended manufacturer at the stackup stage. Although practically any design can be built, determining up-front the architecture that will result in the best yield should be the objective, certainly for volume production. Designers who take into account ahead of time how boards are built can save a good deal of money, and that economy extends beyond just fabrication charges, including fast manufacturing turnaround at the front-end and heightened product reliability in the field.



APEX EXPO IPC

Visit us in San Diego
Feb. 24-26, 2015
Booth # 3019

Advancing the applications of conformal coatings

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

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

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

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

Tel: 888-501-9203

www.electrolube.com

ELECTROLUBE
THE SOLUTIONS PEOPLE

Electronic & General Purpose Cleaning	Conformal Coatings	Encapsulation Resins	Thermal Management Solutions	Contact Lubricants	Maintenance & Service Aids

Footprint Development: The Foundation of PCB Design

Needless to say, every project starts with a schematic, a mechanical description of the board, a stackup, and a complete bill of materials, or BOM. The bedrock of PCB design, the foundation, is footprint development, Albers advises. He refers to the data sheet for virtually every part on every BOM to leave no doubt about exact package descriptions and create proper footprints. In fact, footprint development is the most time-consuming aspect of nearly every project.

"As a rule, the spacing I use between parts is very large," Albers points out. "When I build a footprint, I make a reference line for the maximum size of the part, extend outside that by 15 mils for the silkscreen, then 5 mils more for an assembly line, and then a minimum of 15 mils beyond that for a package keep-out. That's 35 mils between the body of one part to the edge of the next part footprint, which has the same 35-mil spacing, so that's 70 mils in total from the body of one part to another."

"Discretes ordinarily are less than that," he continued. "For a connector that I know you have to get your fingers around, I may enlarge the keep-out boundary by 40 mils. If it's a right-angle connector, the keep-out accounts for clearance needed for the cable at the edge of the board, and so forth: Each part has its own keep-out value. Keeping the boundaries large certainly helps via placement, eases routing, and is better for the silkscreen, but it leaves enough latitude for me to violate the rule when necessary without jeopardizing assembly, if I need to place two parts 10 mils closer."

Via size, spacing, and location are the most challenging tasks in design, not routing or part placement. Through-hole vias impact where you can put parts on the opposite side of the board; of course, that's a principal advantage of using blind and buried vias. There's a point at which an HDI approach becomes unavoidable (generally the result of tight BGA pitch) or more economical in terms of yielding more boards per panel to counteract the increased processing cost for multiple laminations. A board designer who understands fabrication can make a reasoned analysis.

"If a board will be used merely on a benchtop as a test or development tool and only four or five are needed, then the size is irrelevant for manufacture, provided it fits on a panel, coupon width included. A design that will be produced in any quantity, however, had better take into account how it can be replicated per panel with minimum waste."

Fab Complexity vs. Yield

The balance between fabrication complexity and yield per panel is not the only economic concern. Sometimes PCB designers need to push back, forcefully, against decisions by their engineering department or procurement. Albers recalls working at a computer company and encountering a BOM that called for a large through-hole capacitor for which there was a surface-mount substitute that would consume much less territory. The capacitor in the call-out cost $0.17 and the substitute cost $2.40 each. Procurement was adamant his group had to use the $0.17 capacitor, not the substitute. So much room was required for the part that just one board would fit on a panel, instead of two if the substitute had been acceptable. It cost the company $7.40 more per board to use the cheaper part instead of the substitute.

If a board will be used merely on a benchtop as a test or development tool and only four or five are needed, then the size is irrelevant for manufacture, provided it fits on a panel, coupon width included. A design that will be produced in any quantity, however, had better take into account how it can be replicated per panel with minimum waste. If you contemplate using large panels for economy, you need to consider minimum hole and pad sizes because the tolerance across the panel may not support using 6-mil drills and small annular rings, for example. In any event, unless a board involves a BGA with a very tight pitch or one with a great many connections, there's seldom any need for a drill size smaller than 8 mils.

Many designers don't realize the soldermask is what determines design rules, not copper-to-copper or pad-to-pad minimums, Albers points out.

"The smallest spoke of a solder mask is 4 mils, period. You can't go below that whether or not you use standard LPI soldermask or laser-defined soldermask or there will be registration issues," says Albers. "You can decide to use a laser-defined solder mask to provide the better precision than LPI, which is unavoidable if there's a BGA with tight spacing, but that costs more. Getting the most for your money should be the objective when you need production quantities. However, unless you really know fabrication, collaborate with your manufacturer before you design."

Albers notes that engineers often pick BGAs with the finest pitch available, figuring the smallest packages will conserve board territory. However, the breakout for larger packages in many cases will actually result in less area consumed. Package selection for complex devices ought to involve both the circuit designer and the PCB layout designer.

"As a rule of thumb, to estimate the number of layers a board will include, count the number of ball rows in the most complex BGA. You'll need a power plane per supply value for the part and associated ground planes. That package guides the project," Albers says.

Albers places and routes the power section of designs first, keeping most of the active devices on one side of the board and discretes on the other with assembly in mind. He can almost always accomplish a design without turning to fine trace and space widths.

Albers' advice in a nutshell? Think big. **PCBDESIGN**

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

Sensors Enable Further Flexing of Flex Displays

Crocus Technology's new magnetic logic unit-based (MLU) solution can detect the position and shape of flexible two dimensional surfaces. Wearable devices, curved panel displays, flexible solar panels, and, in the future, mobile phones will integrate flexible shape sensor foils.

Crocus' magnetic sensors aim to provide an efficient solution for shape sensing in flexible surfaces and foils to overcome deficiencies occurring in other solutions, such as piezoelectric sensors. Unlike other solutions, Crocus' MLU sensors exhibit high sensitivity and directional capabilities. This means that only a minimal number of MLU sensors need to be embedded in flexible shape sensor foils. In its prototype, Crocus only uses 0.25 sensors per square centimeter, making its solution extremely cost-effective.

In addition, Crocus' MLU sensors offer advantages in low power consumption and high-speed detection. They provide strong signals without active components. Crocus' 20cm x 20cm prototype consumes less than 10mA during the sensing cycle that lasts less than 1ms.

"Crocus has created a new IP-based on magnetic sensors for flexible surface position detection. This enables equipment makers to gain in the added performance of flexible shape devices, while reducing costs," said Bertrand Cambou, chairman and CEO of Crocus Technology.

As flexible displays are light, thin and unbreakable, they are expected to replace conventional displays. According to Crocus' press release the market for flexible displays is expected to reach USD $3.89 billion by 2020.

TOP TEN

News Highlights from PCBDesign007 this Month

1 Mentor: PCB Technology Leadership Award Winners

Mentor Graphics has announced the winners of its 25th annual PCB Technology Leadership Awards. Started in 1988, this program is the longest running competition of its kind in the electronic design automation (EDA) industry.

2 Top 10 Most-read PCB Design News Stories of 2014

The past year brought plenty of interesting design news items to our readers around the world. As part of our annual industry review, the editors at I-Connect007 have pulled together a list of the 10 most-read design stories from the pages of PCB-Design007. Join us for a look back at the top news of 2014.

3 Zuken's E3.series Adopted by Piaggio

The Piaggio Group, which owns several prestigious scooter and motorcycle brands including Piaggio, Vespa, Aprilia and Moto Guzzi, faced changing requirements to their electrical design process. Global expansion brought an increase in the number of product specifications for different markets, so they needed to streamline their design processes and reduce their time-to-market to accommodate the increased scale of operations.

4 EDA Industry's Revenue Up 6.3% in Q2

"Double-digit increases in PCB/MCM and Semiconductor IP led the way to an overall increase in industry revenue for Q2," said Walden C. Rhines, board sponsor for the EDAC MSS and chairman and CEO of Mentor Graphics. "Geographically, all regions except Japan had solid revenue increases."

5 Altium Designer Updates Library Expert

Major enhancements have been to the Library Expert's output to Altium Designer. This latest update of the Library Expert (v2015) offers users a very streamlined method of creating high quality IPC-7351 compliant Footprints, 3D STEP models, and Footprint Attributes for Description and Height, simultaneously, in Altium Designer.

6 Cadence Earns Commendation from City of San Jose

San Jose Mayor Chuck Reed and the San Jose City Council have recognized the company as one of the 25 industry-driving companies generating the most growth and jobs in the city. "The people of San Jose and Silicon Valley have changed the world, and it's thanks to industry-driving companies like Cadence that our city continues to be the capital of Silicon Valley, the world's center of innovation," Mayor Reed said.

7 IEEE to Adopt the HDBaseT Standard

The HDBaseT Specifications 1.1.0 and 2.0 have been approved by the IEEE Standards Association (IEEE-SA) Standards Board for adoption as part of the organization's standard portfolio. HDBaseT will complement IEEE-SA's portfolio of market-proven communications and technology standards.

8 Zuken Wins Award for EV Research Collaboration

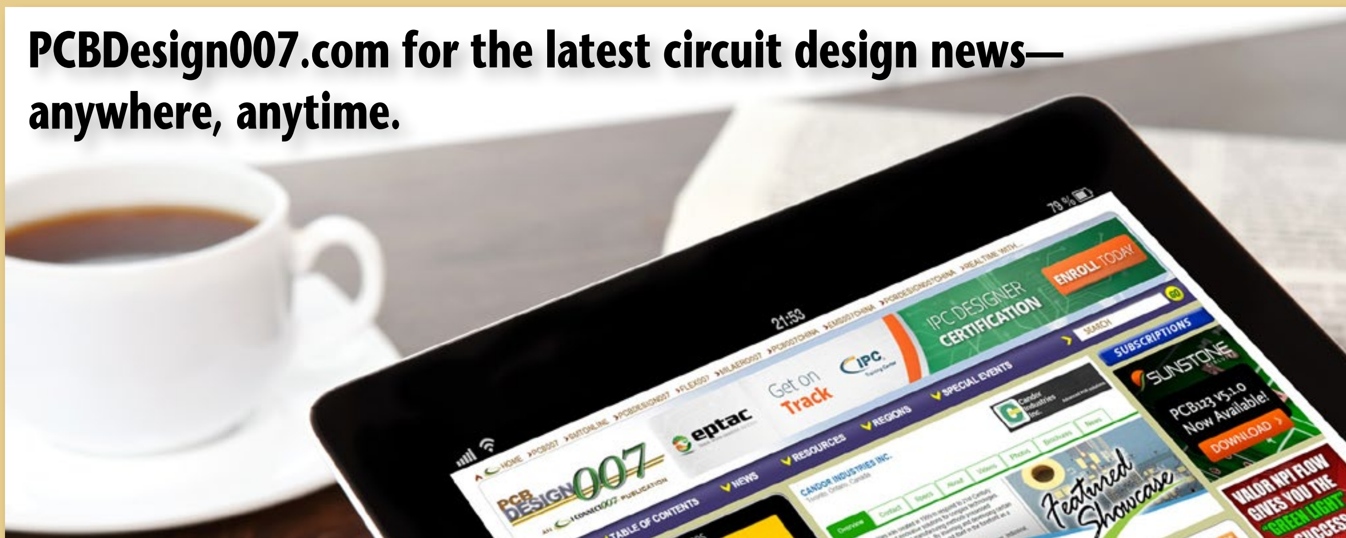
Zuken plays an important role in this research by advancing new technology concepts for EMC-related modelling of power and ground systems for control unit structures and for incorporating power-ground related noise in signal integrity simulation. EMC issues in electric vehicles can become critical due to the high currents and voltages required by electric vehicle engines.

9 INLYNK Teams With Octopart and Altium for Optymo

INLYNK has released version 4.3 of its flagship product. Optymo is a sophisticated BOM and parts management solution that helps organizations to better manage their design and production steps. This new version is the result of a very successful collaboration between Altium, Octopart and INLYNK.

10 Top 10 Most-read PCB Design Magazine Columns

It's been a wild year in the PCB design community. Naturally, the top columns in The PCB Design Magazine during 2014 cover a maze of topics, from high-speed design to design for manufacture. So, without further ado, here are the Top 10 columns from The PCB Design Magazine of the past year.



events

For the IPC Calendar of Events, click here.

For the SMTA Calendar of Events, click here.

For a complete listing, check out *The PCB Design Magazine's* event calendar.

2015 European 3D TSV Summit Enabling Smarter Systems
January 19–21, 2015
Grenoble, France

SEMI Arizona Tech Talk: INanoBio
January 23, 2015
Chandler, Arizona, USA

DesignCon 2015
January 27-30, 2015
Santa Clara, California, USA

IPC Conference on Assembly and Reliability
January 27, 2015
Ho Chi Minh City, Vietnam

Rocky Mountain Expo & Tech Forum
January 28, 2015
Denver, Colorado, USA

IPC Conference on Assembly and Reliability
January 29, 2015
Bangkok, Thailand

SMTA Pan Pacific 2015
February 2–5, 2015
Kauai, Hawaii, USA

SEMICON Korea 2015
February 4–6, 2015
Seoul, Korea

MEDIX 2015
February 4–6, 2015
Osaka, Japan

LED Korea 2015
February 4–6, 2015
Seoul, Korea

2015 Flex Conference
February 23–26, 2015
Monterey, California, USA

IPC APEX EXPO 2015
February 24–26, 2015
San Diego, California, USA

CPCA Show 2015
March 17-19, 2015
Shanghai, China

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

PUBLISHER: **RAY RASMUSSEN**
(916) 337-4402; ray@iconnect007.com

SALES MANAGER: **BARB HOCKADAY**
(916) 608-0660; barb@iconnect007.com

MARKETING SERVICES: **TOBEY MARSICOVETERE**
(916) 266-9160; tobey@iconnect007.com

EDITORIAL:
GROUP EDITORIAL DIRECTOR: **RAY RASMUSSEN**
(916) 337-4402; ray@iconnect007.com

MANAGING EDITOR: **ANDY SHAUGHNESSY**
(404) 806-0508; andy@iconnect007.com

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

MAGAZINE PRODUCTION CREW:
PRODUCTION MANAGER: **MIKE RADOGNA**
mike@iconnect007.com

MAGAZINE LAYOUT: **RON MEOGROSSI**

AD DESIGN: **SHELLY STEIN, MIKE RADOGNA**

INNOVATIVE TECHNOLOGY: **BRYSON MATTIES**

COVER: **RON MEOGROSSI**

THE pcb design MAGAZINE

The PCB Design Magazine® is published by BR Publishing, Inc., PO Box 50, Seaside, OR 97138
©2015 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.
January 2015, Volume 4, Number 1 • The PCB Design Magazine© is published monthly, by BR Publishing, Inc.

ADVERTISER INDEX

Coming Soon to *The PCB Design Magazine:*

February:
High-Speed Design Tips

March:
Design for Manufacturing

April:
Surface Finishes