

THE **pcb** **design** MAGAZINE

November 2015

an IConnect007 publication

SnapEDA: The Female-Owned Startup Revolutionizing CAD Data [p.28](#)

EMA: Helping Technologists Manage Disparate Data [p.36](#)

Downstream Takes on Data Documentation Management [p.46](#)

MANAGING YOUR DESIGN DATA



**GOOD IN,
GOOD OUT:**

*Bay Area Circuits Discusses
Data Strategies*

isola
The base for innovation

PCB BUYERS - compare nearly 1900 manufacturers now at *The PCB List.*



Quick Search



Advanced Search



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

www.thepcblist.com



BOARD LEVEL SIMULATION SPECIALISTS

ICD Stackup Planner - offers engineers/PCB designers unprecedented simulation speed, ease of use and accuracy at an affordable price

- 2D (BEM) field solver precision
- Characteristic impedance, edge-coupled & broadside-coupled differential impedance
- Unique field solver computation of multiple differential technologies per stackup
- Heads-up impedance plots of signal and dielectric layers
- User defined dielectric materials library - over 16,250 materials up to 100GHz

ICD PDN Planner - analyze multiple power supplies to maintain low impedance over entire frequency range dramatically improving product performance

- Fast AC impedance analysis with plane resonance
- Definition of plane size/shape, dielectric constant & plane separation for each on-board power supply
- Extraction of plane data from the integrated Stackup Planner
- Definition of voltage regulator, bypass/decoupling capacitors, mounting loop inductance
- Frequency range up to 100GHz
- Extensive Capacitor Library – over 5,250 capacitors derived from SPICE models

THIS MONTH'S FEATURE ARTICLES

Managing Your Design Data

Designers like you are more productive than ever before, but you're also juggling more data, in more formats, than you ever could have imagined. This month, we discussed design data management in interviews with Stephen Garcia and Brian Paper of Bay Area Circuits, Natasha Baker of SnapEDA, Manny Marcano of EMA Design Automation, and Mark Gallant of DownStream Technologies.

12 Good in, Good out: Bay Area Circuits Discusses Data Strategies

*Interview with Stephen Garcia
and Brian Paper*



28 SnapEDA: The Female-Owned Startup Revolutionizing CAD Data

Interview with Natasha Baker



36 EMA: Helping Technologists Manage Disparate Data

Interview with Manny Marcano



46 Downstream Takes on Data Documentation Management

Interview with Mark Gallant





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.

Visit us at
PRODUCTRONICA
2015
Nov 10-13
Hall B1, Booth 471

www.isola-group.com/RF

TABLE OF CONTENTS

ARTICLE

66 **The Gerber Guide, Chapter 4**
by Karel Tavernier

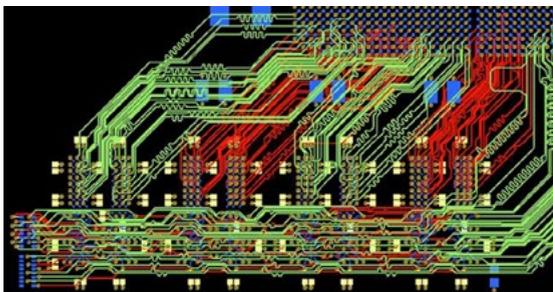


COLUMNS

8 **Are You Drowning in Data?**
by Andy Shaughnessy



54 **Why Autorouters Don't Work: The Mindset!**
by Barry Olney



DEPARTMENTS

74 **Events Calendar**

75 **Advertiser Index & Masthead**

SHORTS

26 **Interim CEO Jeff McCreary on Changes at Isola**



34 **A New Slant on Semiconductor Characterization**

52 **Supercomputer with Turbocharger**

62 **Back to the Future: Serviceable Spacecraft Make a Comeback**



HIGHLIGHTS

44 **PCB007**

64 **MilAero007**

72 **PCBDesign007**



Automotive electronics stay cool with our power electronics PCBs



Solutions in power electronics play a key role in core topics of our times: They enable us to master the future challenges of electro mobility, energy creation and energy distribution.

In order to reduce fuel consumption as well as CO₂ emissions and to increase safety and comfort, it is increasingly necessary to switch higher currents and to dissipate heat.

Amongst all PCB manufactures, SCHWEIZER offers the most comprehensive range of solutions to tackle these challenges. Already today, the automotive industry applies SCHWEIZER PCBs which can operate up to 1,200 Ampere.

Schweizer Electronic AG
Einsteinstraße 10
78713 Schramberg
Germany

www.schweizer.ag

Are You Drowning in Data?

by Andy Shaughnessy

I-CONNECT007

Data management was so much simpler during the days of Mylar and Bishop Graphics tape. Data was handwritten. All you had to do was keep track of your paperwork and you were golden.

Now, you're all much more productive, but you have data coming out of your ears; slowly but surely, incrementally, data has become much more complicated. Now, design teams grapple with schematic symbols, CAD components, footprints, BOM, netlists, simulation models, 3D models, and user-generated and third-party tool-generated data. Not to mention files for Gerbers, ODB++, or IPC-2581, all the way through final data handoff to CAM.

And still, more than 30 years after the advent of EDA tools, not much has changed. How do designers and engineers wind up managing all of this data? With kluged-together processes and software tools, and the occasional handwritten notes.

It all sounds like a headache waiting to happen.

We recently surveyed subscribers about their data and data management processes, and many of you reported that keeping data organized is a constant challenge. Figure 1 shows the breakdown of data transfer formats. Gerber is still far and away the No. 1, but over half of respondents used ODB++, meaning that many of you output both sets of files. IDF, AutoCAD and STEP are also popular.

Figure 2 is illustrative of what we've heard for years: Designers want to hear feedback from their fabricators, assembly partners, customers, and electrical engineers. Among the "other" answers, popular responses were feedback from QC

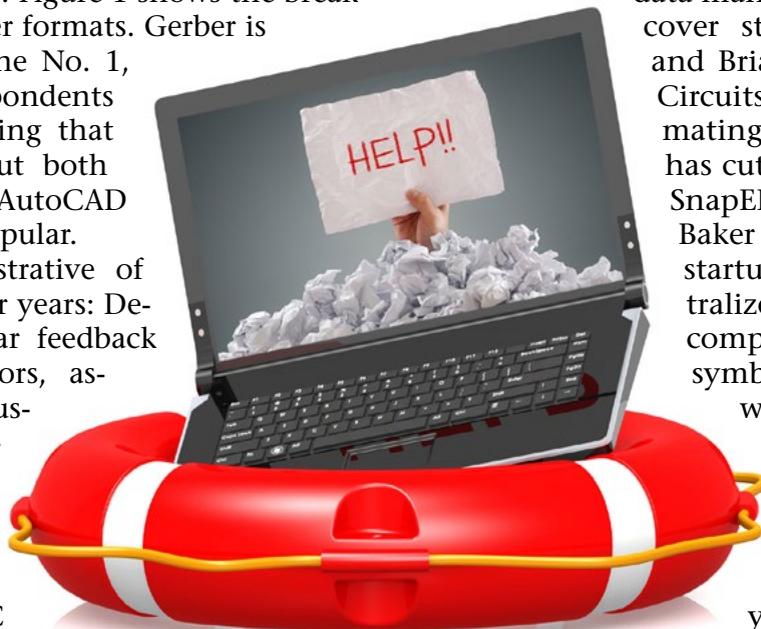
and EDA tool vendors.

When asked about the biggest challenges they faced involving data, respondents ranked accuracy of data No. 1, followed by data management and timeliness of data. Among some of the "other" answers were "verification of data provided to vendors" and "going through all the steps to make sure ODB files generate correctly and completely."

We also found that about a third of respondents don't use any statistical data, but almost half of respondents said they used limited stat data, and 7% would like to use statistical data, but have no idea where to begin.

Asked what data they would like to have, but have trouble acquiring, respondents listed material properties, available thickness of laminates and prepregs, PCB vendor data for stackups and line widths, exporting Gerber files to a different format, history of material information, and 3D STEP models.

All of this resembles herding cats, doesn't it? To help shine some light on this issue, this month, we have a variety of interviews with technologists who are in the middle of the data management fight. In our cover story, Stephen Garcia and Brian Paper of Bay Area Circuits discuss how automating their data systems has cut overall process time. SnapEDA founder Natasha Baker explains how her startup will serve as a centralized location for CAD components, schematic symbols, PCB footprints—whatever an engineer needs. (When was the last time we saw an EDA startup, much less one founded by a young woman?)



Pulsonix

Advanced Schematic Capture and PCB Design Software

A refreshing approach to simplifying Schematic Capture and PCB Design

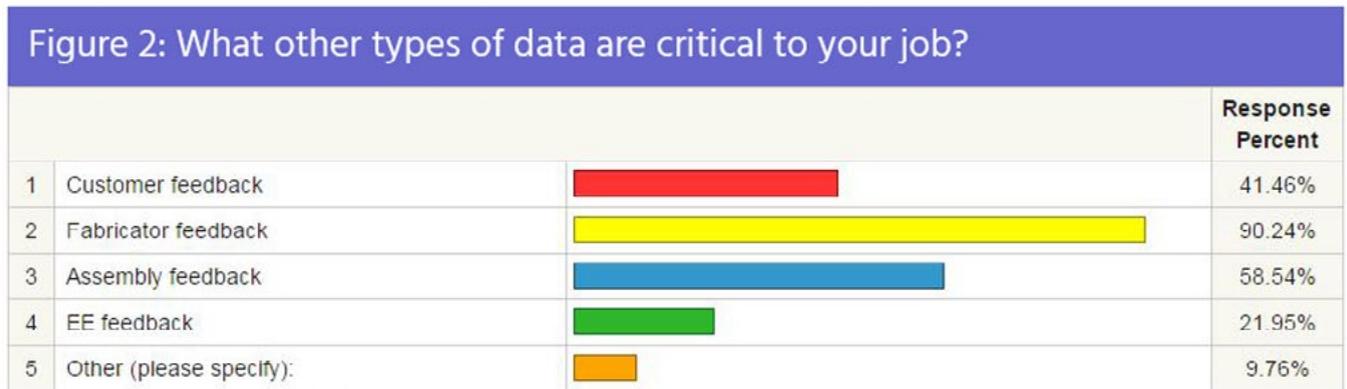
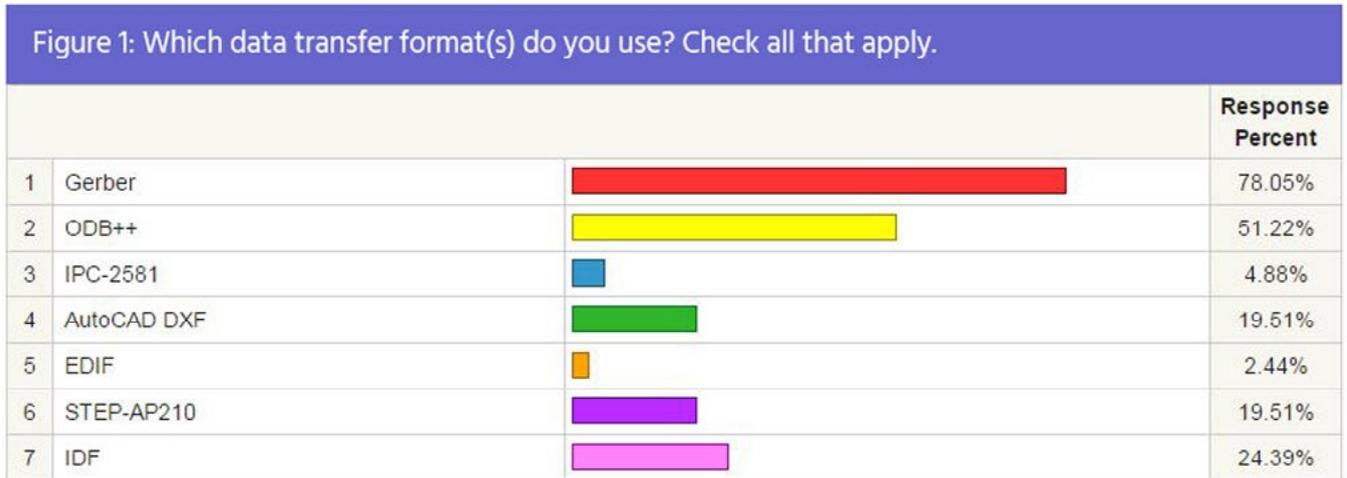
- Import filters for all major tools (designs and libraries)
- Very easy to learn and use
- 5-Star Service and Support

[Download a Free Trial](#)

info@pulsonix.com
www.pulsonix.com

PULSONIX


ARE YOU DROWNING IN DATA?



EMA Design Automation founder Manny Marciano explains how EMA focuses on helping users manage schematic data, libraries, 3D models, simulation models, netlists, Gerbers, and component data. And Mark Gallant of DownStream Technologies discusses how the company focuses primarily on making sense of designers' least favorite step in the design process: post-processing.

Next month, we take a look at the associations and organizations in our industry, what

they can do for you, and what you can do for them.

Happy Thanksgiving! **PCBDESIGN**



Andy Shaughnessy is managing editor of *The PCB Design Magazine*. He has been covering PCB design for 16 years. He can be reached by clicking [here](#).



Nothing Left to Chance: It's Our Hands-on Approach to Managing Your Business.

JIM LIU, Commercial Manager and **NANCY LUO**, Customer Support NCAB Group China, on the way to a factory with **WAYNE ANTAL**, Key Account Manager NCAB Group USA.

Business is conducted between people, not companies. To be successful you need to begin with a strong relationship built on people that will bridge the entire PCB supply chain. From the factory to your dock NCAB Group integrates best-in-class local support with a strong global presence to ensure that your PCB is manufactured to the highest standards of quality and delivery at the lowest total cost. The NCAB Group's factory management organization is an

on-site resource that has become a strategic member of each factory's manufacturing team. Our technicians, quality engineers and inspectors are in the factories to provide rapid problem solving support, process monitoring and technical assistance so that customer requirements are consistently met.

Nothing is left to chance. That is why NCAB Group is **Your Key Component** – on the spot in China and wherever you are.

For more information contact or visit your local office.

EASTERN DIVISION:
10 Starwood Drive,
Hampstead, NH 03841, USA
Phone: +1 603 329 4551

CENTRAL DIVISION:
1300 Norwood Avenue
Itasca, IL 60143, USA
Phone: +1 630-562-5550

WESTERN DIVISION:
1499 SE Tech Center Drive, Suite 120,
Vancouver, WA 98683, USA
Phone: +1 503 351 5424

Email: info@ncabgroupusa.com or visit www.ncabgroup.com for more information.





Brian Paper



Stephen Garcia

GOOD IN, GOOD OUT: Bay Area Circuits Discusses Data Strategies

by Barry Matties

A lot of companies talk about the importance of good data management, but for some firms, this amounts to little more than lip service. Then there are companies like fabricator Bay Area Circuits. I recently sat down with Bay Area Circuits President Stephen Garcia and COO Brian Paper to discuss how automating and upgrading their data systems has significantly cut down overall process time, as well as their drive to educate young PCB designers and actively promote the industry to the emerging electronics industry workforce.

Barry Matties: *Let's talk about how you manage your data. Are you a completely electronic company?*

Stephen Garcia: We have tried to go as paperless as possible, and we're talking actual production data starting from the Gerber. We use Ucamco's Integr8tor software, which is the entire company's backbone. That is our database. Everything is automated. And then we've tied in our own software to connect supplement Integr8tor.

Matties: *When you say your own software, is that something you developed?*

Garcia: Yes, it's basically a customized CRM platform. Some production data, like process cards, are still printed, but eventually everything is digitized and stored on our servers for easy access which is especially important given that a good portion of our engineering staff works remotely and needs real-time access to data.

As we continue to grow and identify bottlenecks, we make sure employees have the access they need. Not long ago, QC may have been inspecting a job but may not have all the details. Maybe they just had a fab print and the process card, so would have to walk over to Planning and say "I'm seeing something out of the ordinary; was this approved by the customer?" Today they have full access. Each job gets a tool number, and then everything is saved on our server for employees to access the data required.

That's the hardest part of our job: incomplete or conflicting customer data. Every customer is different when it comes to Gerbers and



High Reliability Material for High Speed Applications

(Click on a product logo for more info!)

Synamic 4

Thin Core: S7439 / Prepreg: S7439B

Low Loss and High Reliability Material for Complicated PWB Design

Synamic 6

Thin Core: S7338 / Prepreg: S7338B

Very Low Loss and High Reliability Material for Complicated PWB Design

Synamic 6N

Thin Core: S7335 / Prepreg: S7335B

Ultra Low Loss and High Reliability Material for Complicated PWB Design

25+ Gbps Transmission Applications



Photo Chemical Systems

www.photochemicalsystems.com

PARAMOUNT

www.paramount-usa.com

GOOD IN, GOOD OUT: BAY AREA CIRCUITS DISCUSSES DATA STRATEGIES

their fab prints. We have a handful of customers who often provide us with Gerbers that rarely match the fab print. It might be a non-plated hole on their fab print, but then their Gerber is plated, things of that nature. Our system allows employees to have access to the data to investigate any issues and prevent production hiccups.

Brian Paper: We've put more emphasis on the netlist too, as far as ensuring that the customers are providing us with that type of information. We just posted a whole blog post on what a netlist is, and why it's important.

Garcia: With every customer being different, our biggest challenge is trying to educate them about what we need to successfully manufacture their project and why we need it. It's amazing that even when some of our larger custom-

ers put on a generic fab print, half of it doesn't apply to the actual board, and we need to get approval. We say, "Hey, can we deviate away from your fab print?" and they say, "Yeah, that shouldn't be there. Just ignore it." It definitely slows things down. We've really tried to automate as much as possible from the minute we get Gerbers.

Ucamco's Integr8tor is great software; five to six years ago we had a full-time employee who would analyze data and then give a report to the sales team. Now it's done in a matter of minutes. You drop in Gerbers, and we're able to see the layer count, spacing and tracing, test points, drill hits, and all that real time data in a matter of minutes. That's where we came up with InstantDFM, trying to give our growing online customer base that full access of checking their data, being able to make sure their data is clean, and everything looks right.

Paper: At a high level too, a lot of what Stephen talked about we've actually tasked our account managers with starting the process. They're the ones that interface with the customers. We've actually given them quite a bit of responsibility in terms of doing an initial check on the customer's data, ensuring that we've received everything we need to receive and there aren't any issues. They enter that information into our CRM system which triggers Planning to begin to process the job.

But instructions are also automatically generated for other departments all the way down the chain to Shipping. If there are special shipping instructions for that order or for that customer, all that gets transferred into a specific process card that the shipper has to sign off on. That way we ensure that all that information gets transferred. All of that is automated, but it starts with the account manager ensuring they're collecting the right information from the customer and entering it correctly.

Matties: *The old adage applies: Good in, good out. Or bad in, bad out.*

Paper: Exactly. What we have done is eliminated input steps along the way so that there aren't multiple people collecting and inputting data.



Stephen Garcia, Bay Area Circuits president.

GOOD IN, GOOD OUT: BAY AREA CIRCUITS DISCUSSES DATA STRATEGIES

We're eliminating the room for error there. We put a lot of pressure for accuracy on the front end of the process but if our account managers do their job correctly, there shouldn't be any other breaks in the chain along the way.

Matties: *How many customers a month do you manage, roughly?*

Paper: In addition to our larger enterprise customers, we have a growing base of customers who purchase from our online store. They don't represent as much in revenue, but in terms of numbers of customers and transactions it's significantly higher.

Matties: *I'm just curious because you're talking about how everyone does it differently and it's not standardized. If you're doing, say, 400–500 jobs a month, that's a lot of extra work on you guys to just sort it out in the front end. If you don't do that right everything that follows becomes problematic.*

Garcia: Our online sales growth really forced us to take a close look at automating the front end. We saw that for this to work right we were going to have to have it automated. From the moment an online order is received we capture pretty much everything at the front. The order enters itself into our CRM system and it just continues to move. Like Brian was saying, entering it all at the front is where the data starts so there is a document trail.

Then for the sales team it gets flagged if there's something wrong with that data, and they don't need to go into secondary software. It tells them right there if there are any issues. Again, we're just trying to keep a document trail, and we have specific documents that are automatically generated for several departments. That way everybody has familiarity with what the report looks like and what should be there. We're trying to take the human element out of it.

Matties: *How do you work on getting people to come in with the right documentation, the netlist and all the other pieces that you would require? Do you reject orders if they don't send it?*



Garcia: We don't reject the orders per se, but we're trying to educate the customers on the risk. For example, "Here's the best way to do it. If you don't do it, then make sure you understand the consequences that this could happen." We just try to educate them on why it's a good idea and what could potentially happen if they're not providing us with us that data.

Matties: *How much additional work does it take from your customer's side to provide that information?*

Garcia: The netlist is one of the simpler things for a customer to provide. It really should feel like low-hanging fruit for them. They're already working with the design in their application. I think fab prints are probably tougher for some customers to create. We've seen some customers that have started small and they're starting to grow. They should be growing to that stage where they're providing us with the fab print/fab notes and haven't quite gotten there yet. I don't know if it's just because they haven't evolved to that point, but it's something we'd like to see out of more customers. But netlist, I

GOOD IN, GOOD OUT: BAY AREA CIRCUITS DISCUSSES DATA STRATEGIES

think, should be pretty simple for them to provide.

Matties: *I would think so. I didn't see any real challenge to doing it, but boy, there's sure an upside to providing it. It just seems like people would be foolish not to do it in today's world. How do you look at your processes in terms of improvement, and what sort of approach do you take in data there?*

Garcia: The way we track the jobs internally when something goes wrong has really improved. Again, it's all about automation and customizing the software there to make sure we have all the data we need. We have it set up so that any sort of issue that comes up on a job is reported to our quality team. We want to know every little mishap to figure out how we could fix those things. We collect the data, review it, and make the necessary improvements.

Matties: *And for process flow, bottlenecks for example, how do you view that?*

Garcia: That relates back to a lot of the equipment investments we've made by just looking at the high-level view of what's needed to make sure things are moving correctly. Then at the same time on a day-to-day basis we keep track of what's popping up. That may also drive an investment or a purchase. It may even go back to the sales cycle. I'm happy about the sales we win, but I also care a lot about the sales we've lost and what can be done to win them in the future.

Paper: We try to be proactive and not reactive, but I do think, in terms of operating in a lean fashion, a lot of that is reactive thinking. You're constantly trying to figure out how we can react to a situation and fix it quickly. Again, in terms of high level planning, I think it's more of a proactive approach, but down in the trenches it could be very reactive at times.

Matties: *What is your business strategy?*

Garcia: I want us to be the most efficient and lean manufacturer we can be, but it's all about



Brian Paper, Bay Area Circuits COO.

.....

the customer. That's something I talk about in every meeting. It's about the customer. Nothing matters outside of our customers. To ensure a happy customer and a great customer experience we have to be as lean and efficient as possible, even more so as a US-based manufacturer.

Paper: I don't know if there's an employee whose department hasn't seen some type of investment in the last 12 months to support that idea of operating leanly and improving each process, either by improving the quality of the process or shortening the process time. I think the changes have been very visible for everybody to see because the changes have happened across the entire company.

Matties: *They're feeling it.*

Garcia: Yes.



Feel Free to Think **BIG**...We'll Take it From There

Trust our experienced team of Electrical Engineers to take your ideas and specifications and accurately engineer your product from concept to prototype assemblies. In fact, we love a good challenge, so *bring it on!*

- Complete Engineering Project Management
- Circuit / Schematic Design & Analysis
- Power / Voltage Drop Analysis
- Component Analysis / Evaluation
- Signal Integrity Simulation & Analysis
- Mechanical Engineering
- PCB Layout Team Available 24/7
- Largest Privately Owned, Independent Service Provider in the USA

cā dence®

Mentor
Graphics®

Altium

pads

ANSYS

dalTOOLS
design easy

World-Class PCB Design and Engineering Solutions

REQUEST A QUOTE



www.freedomcad.com

Freedom CAD Services, Inc.

HQ, Nashua, NH (603) 864-1300

Sales: • (714) 490-7600 Tustin, CA

• (408) 529-0563 San Jose, CA

• (603) 864-1342 Windermere, FL

GOOD IN, GOOD OUT: BAY AREA CIRCUITS DISCUSSES DATA STRATEGIES



Paper: Their jobs should be getting easier. We have an employee who's a soldermask operator who is now essentially becoming a computer operator because it's less of a manual process and more of an automated, machine-generated process that's trimming hours off of our production time.

Garcia: It's all about team collaboration for us, too. No matter what role in the production process somebody may work at, I want to know if they have a frustration with the process or ideas for process improvement. We want that feedback so we can make ourselves more efficient.

Matties: *What about cycle time? Do you guys measure that? Time is your biggest enemy I would think.*

Garcia: I think that's our biggest driver at the end of the day. We've gone from relying heavily on outside vendors for certain processes to now having almost every process in-house. That was my largest driver of "Hey, if we want to be..."

Matties: *In business.*

Garcia: Right! Quickturn manufacturing is our focus and to deliver for our customers we need to keep cycle time at a minimum. Bringing processes in-house has been key for us.

Paper: The more we can trim down on that process time, the faster we can put something in somebody's hand. It gives these companies more reason to come to us.

Garcia: Every minute we gain is critical.

Matties: *Because that gives you capacity without additional cost. When you look at your cycle time, you collect data by process, I would imagine. You find out where your bottlenecks are just by looking at the investments you made.*

Garcia: Bottlenecks and the amount of employees needed as well.

Matties: *That's the ratio.*

Garcia: We're trying to stay lean. Right now we're looking at automating our plating lines.

Matties: *That's a manual process right now.*

Garcia: Yes and automating it will help keep us lean, but also add consistency from a process standpoint.

Paper: We'd love to grow the business to the point where we're adding jobs to the local economy. That would be optimal, but from an overhead perspective we really have to watch it. It's not cheap to operate here in Silicon Valley, so that's something we have to keep a close eye on.

Matties: *I think you would look to technology solutions before you would people.*

Garcia: That's the human element as well, right? Now for an operator, the computer itself knows the process, and it's going to be the same every time. They hit a button and it runs, where right now you may be relying on an operator to manually follow the process.

Matties: *With specialty knowledge as well. Now with the computerized manufacturing like your new soldermask system, they don't have to be a soldermask expert. They just need to know what buttons to push. I would think all the jobs are being fed in electronically anyway.*

GOOD IN, GOOD OUT: BAY AREA CIRCUITS DISCUSSES DATA STRATEGIES

Garcia: That's the nice part about it, even going back to when we added Laser Direct Imaging. Everything is fabricated off the Gerber data and these machines are smart enough to produce perfect registration based off the data. We've become more reliant on the technology and less reliant on the human operator.

Paper: It can be almost counterintuitive to think that we need to spend a lot of money in order to save money. It's sometimes hard for people to get over that mental hurdle, but that's exactly what we've done. We've invested a lot of money in equipment because we know it's going to pay off in the long run in terms of managing our overhead.

Matties: *And there's so much old equipment still in use in this industry that there's a lot of room for opportunity to improve. How much cycle time have you reduced by bringing in all those processes and bringing in the level of automation that you've already achieved?*

Garcia: I would say, depending on the job and technology, anywhere from 4–8 hours overall from start to finish, maybe even more. In terms of automation, our account managers used to manually enter everything. We had four or five different steps where that same dataset was being entered for different areas of production. Now it's done just once. The automation also extends to accounting functions like customer invoicing and purchase order generation for our vendors.

Matties: *Does your system automatically check stock for base materials and such?*

Garcia: We're working on that right now. We're hoping by the end of the year that process will be fully automated.

Paper: We have automated the stock on the other end of the process, related to jobs we produce. If there is a higher than anticipated yield, we have a stocking system. If a customer calls up and they're interested to know if we've got a particular part in stock, our account managers or service representatives have real time ac-

cess to that information. We know exactly how many pieces we have and exactly where it's stored.

Matties: *With all your drive to improve your automation and your new technology, your competitors must be doing the same thing, I would imagine?*

Garcia: Yes, I imagine some are.

Matties: *They're headed in that direction if they want to stay in business. Because what happens is you accelerate, and they have to keep pace, right? How do you maintain your advantage?*

Garcia: We believe a lot of things we're working on and the way we've set our system up is quite different than a lot of our competitors; we have a uniquely custom implementation built from the ground up. The one big key is that we've really tailored it to what we feel is going to help us grow, and obviously it's always improving and we have to stay on top of that. We also be-



GOOD IN, GOOD OUT: BAY AREA CIRCUITS DISCUSSES DATA STRATEGIES

lieve our location, Silicon Valley, is a strategic advantage for us. This is still the hub of technology and innovation.

Paper: The city of Fremont has really transformed itself into the manufacturing hub to Silicon Valley.

Matties: *And you're sitting right next door to Tesla, which is a driving force.*

Garcia: Fremont has been great for us. Since Tesla moved here the city has really seen a lot of growth in terms of technology companies relocating here. We feel like we're a couple of steps ahead of everybody right now with what we're doing internally. It all goes back to the customer experience. That's the most important thing to me is that every one of our customers love us. I want them to love us. We need to be as seamless and as easy as possible for them. I think that's our driving force. and because we deal with such a variety of customers, we see a lot of the challenges that our customers see. We take that data, and we want to fix it for the customer without them really realizing it and introduce these tools that make them say, "Wow, this makes my life easier."

Paper: You know how you have that consumer buying experience that you almost think is too easy or too seamless? Like if you buy that something on Amazon and it shows up on your door on a Sunday. That's the type of experience we think customers should be having, and I don't think anybody's accomplished that in the in-

dustry yet. There's still tons of room for growth to try to improve that part of the buying experience.

Matties: *Do you think that's an online experience, or are you talking salesperson to customer?*

Garcia: I think it's across the board. Take the quote process for example. You've got to wait for a quote, and when you receive the quote then the buyer has to issue a P.O. and that whole process. That takes time. I think in this day and age it should just be seamless for everybody.

Matties: *With the Web model it almost builds a wall between you and your customer though, in that they can just go and put in an order. They press a button and then the boards are going to arrive at their doorstep. Is that pretty much how it works?*

Paper: Yes.

Matties: *Where do I get the user experience other than that? How do I build that relationship with you?*

Paper: We offer multiple options in terms of buying. We have the account managers if you need that extra level of service or support with your order. Some of our account managers have been with us for more than 30 years. They understand the business, and they've got a depth of knowledge that's helpful for customers. We offer that experience, and some customers prefer that. We also have this growing base of...I was going to say millennials. I don't know if they're all millennials, but a lot of them have just started with PCB design. They're makers. They're hobbyists. Some of them are professional design engineers that just don't want to pick up the phone and talk to anybody.

It really just depends, but there's definitely demand for an online experience. You could call it a walled garden or whatever you want, but people want that experience. They want to be able to be in their house at 3 a.m. and submit an order and not have to go through that process Stephen described where they're waiting for a quote to move forward with an order.



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



Advanced Connectivity Solutions

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 QR code please visit our Support Team website at www.rogerscorp.com/coolspan

GOOD IN, GOOD OUT: BAY AREA CIRCUITS DISCUSSES DATA STRATEGIES

Garcia: Fortunately, too, for that user experience we've tried to provide useful tools including educational articles and a support site which includes a knowledge base. We also offer software including InstantDFM which provides free DFM reports and design software, called PCB Creator. Not to mention a customer support team available to answer questions, via phone, email or online chat. We launched online chat three years ago and I think we really were one of the first in the industry to offer it.

Matties: *How many chats a day would you have now?*

Garcia: I would say more than 10. We've tried to set it up so that, no matter what experience the customer needs, we're able to offer it. With the online experience it's important we offer customers all the tools possible to make it easier to place an order.

Matties: *So for your sales team, what's their role?*

Paper: We try to avoid the word sales. We like to call them account managers because they are the first line of support. We intentionally

chose that term account manager, which a lot of people use. We didn't call them sales manager or salesperson because we don't want them to think like that. We want them to remember that their main purpose is to serve that customer.

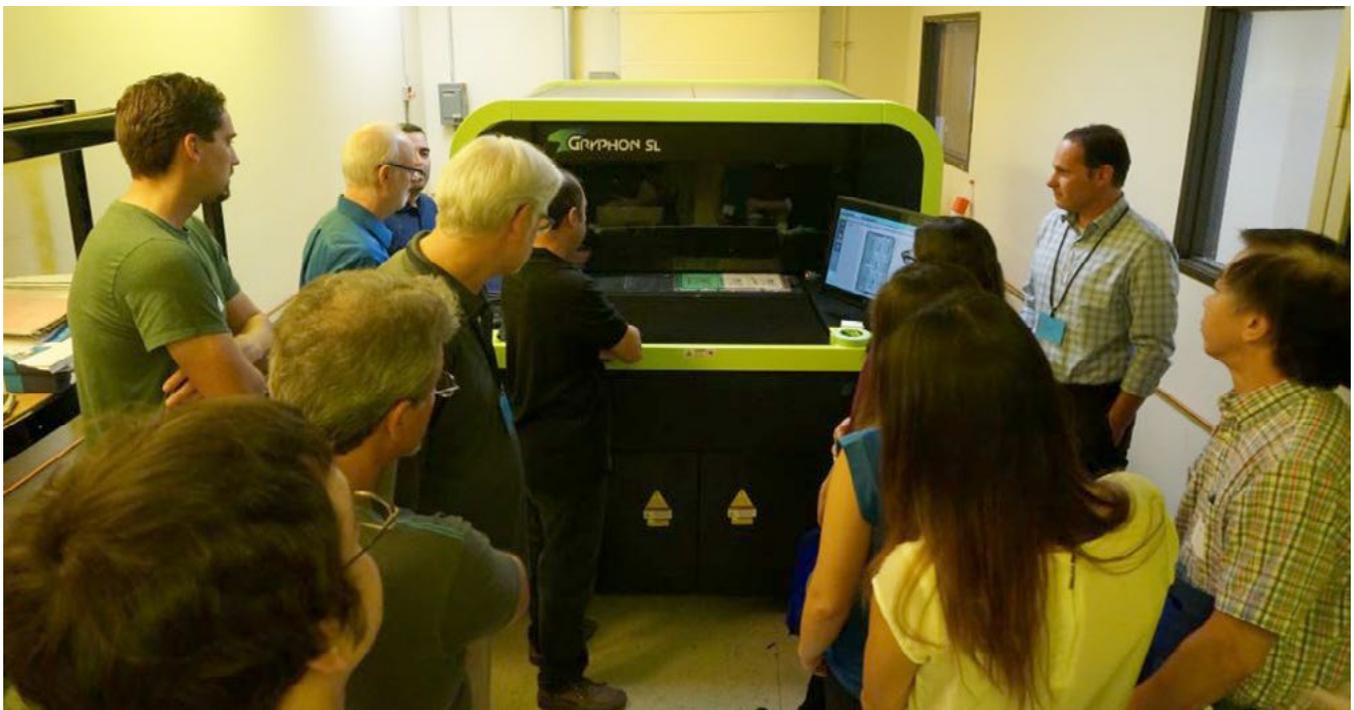
Matties: *When you look at the data for markets and forecasting, what do you guys look at? What's the driving data for you in your mind?*

Garcia: Because we have multiple revenue streams—we have our PCB Creator software, Web, and then B2B—we look at each individually for trends. On the B2B side we'll drill down and look at specific markets. With Web it's just one number and we look at that trend to make sure it's moving in the right direction, which it has been.

Paper: I own that Web experience. That's been one of my focuses.

Matties: *That's been around for a few years now. Has it performed the way you expected it to?*

Paper: This year to date we've already doubled what we did last year.



GOOD IN, GOOD OUT: BAY AREA CIRCUITS DISCUSSES DATA STRATEGIES

Matties: *What do you attribute that to? Is that new markets, or is it increase in the people that were using it?*

Paper: It's marketing. It's experience. Somebody who's tried it perhaps had a good experience, and now they're a repeat customer. We've been able to limit the amount of churn. We have customers that continue to come back, and meanwhile we're continuing to add customers through marketing, content and word of mouth.

Garcia: Even though we've been around for 40 years, since I took over the company's leadership six years ago, we've really tried to act more like an agile startup.

Matties: *So it's really been the last six years that there's been a lot of change in this organization?*

Garcia: Yes, absolutely. This company had and still has a very loyal customer base and was very niche when it was located on the peninsula. The company had to leverage various outside vendors to get by and that is not a long-term recipe for success.

Matties: *There have been pricing pressures. What do you guys feel out there?*

Garcia: The unfortunate part about this industry is the PCB is often considered a commodity. That's really where the pricing has been driven down.

Matties: *It's often called that, but we both know that it's not.*

Garcia: I completely agree.

Paper: The word we use internally is value, and we try to provide customers with good value, meaning high-quality product at a fair price. Our pricing has to stay competitive but we're not always going to be the lowest. Sometimes customers are just so price sensitive they're simply going to take that lowest price. We try our best to educate them that the lowest price isn't always the lowest price, if you know what I mean.

Matties: *Right. There's total cost.*

Paper: You pay for it. There's a total cost of ownership for that lowest price, and you may end up paying more in the long run for it. Some customers understand that, but others learn the hard way.

Garcia: A lot of that relates back to what we've done on the automation side to be lean and keep our cost down. Brian and I have definitely set out for significant growth, and I didn't want to have to add lots of overhead to achieve that growth. Automation helps keep overhead down and as a result, pricing stays low.

Paper: A customer asked us at our recent open house event what the upcharge was to run his board on the Camtek Gryphon soldermask machine, expecting to be charged. "Hey, that's new technology. It's better. How much more are you going to charge me for that?" I said, "No, no, no. That's not the point of this machine. The point is we'll offer you a better quality product but keep the pricing the same." If anything, we're trying to keep pricing competitive while still improving the process and quality. It's not a new feature we're trying to upsell our customers on.

Matties: *You're streamlining your process.*

Garcia: Yes, we work on it continuously.

Matties: *I see you have your loyalty program. How did people respond to that?*

Paper: They've responded really well. You have to opt in to it and they really have nothing to lose by opting in. They start to earn points as they make purchases, and then they take those points and can apply them to get discounts on a future order. It's actually been pretty effective so far, and customers seem to respond well to it. It's one more thing that helps keep us a little sticky, and keeps those customers coming back.

Matties: *Once they're here you have to have a lot of reasons for them to stay. Doing it right, of course, is the baseline. Particularly on the Web, where it can be so impersonal that you don't*

GOOD IN, GOOD OUT: BAY AREA CIRCUITS DISCUSSES DATA STRATEGIES

get that customer service so much either. It's so easy to go to somebody else's website and say "Oh, it's right here."

Garcia: Especially when you're just shopping price.

Matties: Price comparison is easier.

Paper: That's one more reason for somebody to say "Well, you know what, I might save a few cents if I purchase this somewhere else, but I'm part of this loyalty program at Bay Area Circuits, and I'm going to earn points on my purchase. I've already got a bucket of points so I'm just going to stay here." And we like offering something back to the customers for their loyalty, too. It's a win-win situation.

Matties: It costs a lot less to keep them than to get them.

Garcia: Absolutely.

Matties: Is there anything that we haven't talked about that either of you would like to share?

Paper: We mentioned hobbyists earlier. That has been a big area of focus for us. We just hosted a manufacturing day event.

Matties: I saw that. It looked like a good success for you guys.

Paper: We had two groups of about 25 engineering students from a local college come and tour our facility. Their professor brings kids in throughout the year as well. It's a great opportunity for us to open the doors and explain to them what the manufacturing process is like. A lot of them are just getting started with PCB design, and it's great for them to get early exposure to the manufacturing process. We sometimes have professional design engineers who have been in the business for 10 years, who



visit us for the first time and have never seen the inside of a production facility. It is good early exposure and part of our overall plan to expose Bay Area Circuits to students and engineers who are just getting started.

Matties: You're growing your customers. It's a 10-year strategy.

Paper: Yes. We sponsor a lot of projects with universities and build a lot of boards for free for classes. We've sponsored five or six different student racing car groups that universities like U.C. Berkeley put together. We've sponsored a lot of the boards for those types of projects. It's just great exposure. A lot of them are going to eventually become professional design engineers and might remember us then.

Garcia: Also it's great recruitment for future employees too. At our first open house event three years ago, we met a professor from a local college. We stayed in touch and he's been helpful in sending us candidates for engineering positions. As a matter of fact, we recently hired someone based on his referral. That's always a concern of mine, the future of the industry in the sense of the incoming workforce, especially being here in the valley.

Matties: Changing manufacturing towards being a computer operator is going to make it a lot easier to find labor. It's not the specialty knowledge that they need anymore.

Paper: It's a yes and no. Yes, in that people are coming up with that skillset, but it's also a very competitive marketplace. We're obviously trying to control cost as well and have to consider that. If you take an electrical engineer or somebody that's studied that type of work at a four-year university, it could be very difficult or very competitive to get their services with all of the technology opportunities out there.

If you can dream it, we can help you build it.

We provide all the services needed to take your idea into production:

- Board Design
- Product Development
- Embedded Software
- IOT Solutions
- FPGA Design
- Test Engineering

...and deliver them 40-50% less expensively than other service providers! You can use those savings to build the physical product.

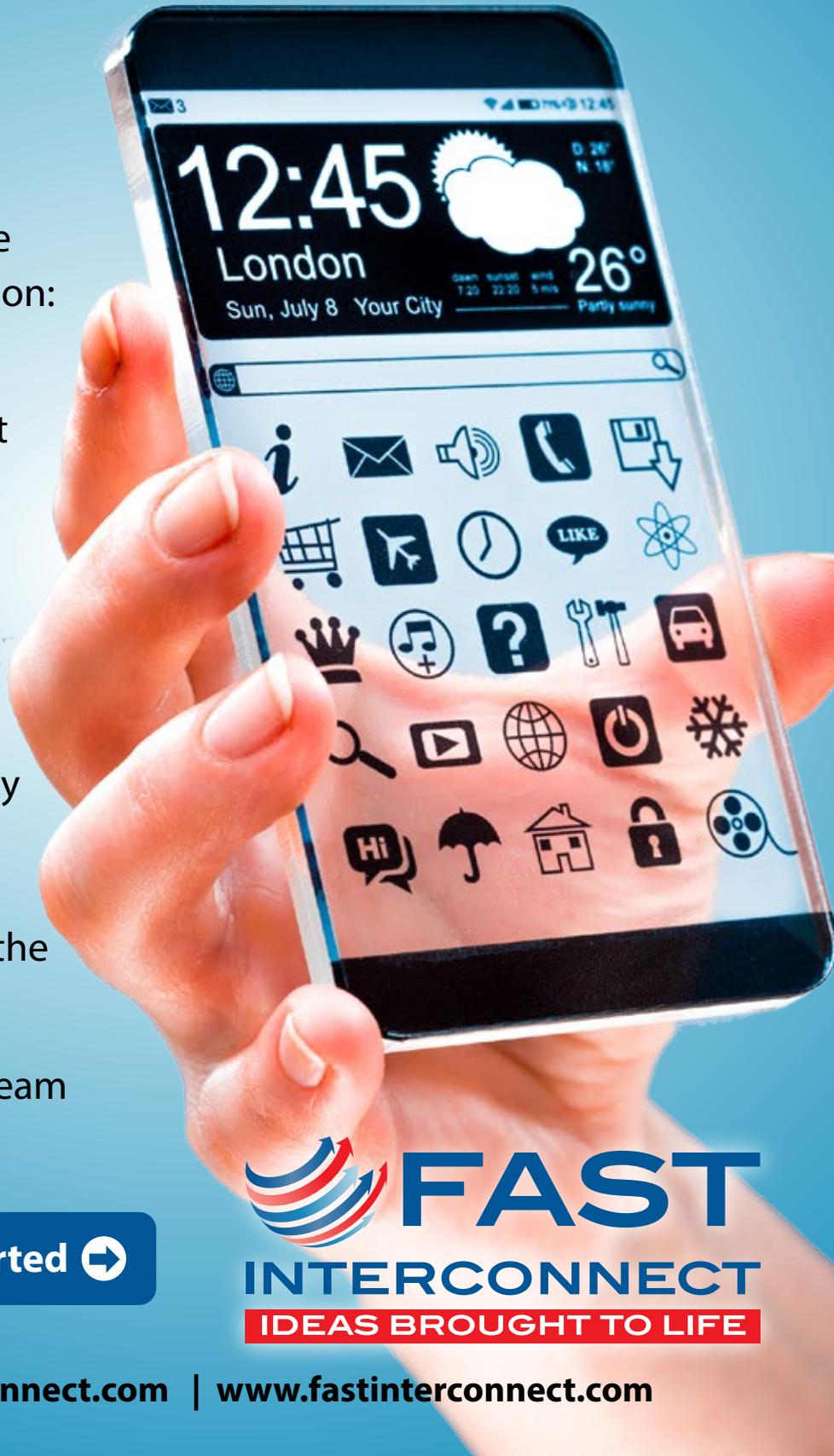
Ready to make your dream a reality?

[Click here to get started](#) 



FAST
INTERCONNECT
IDEAS BROUGHT TO LIFE

sales@fastinterconnect.com | www.fastinterconnect.com



GOOD IN, GOOD OUT: BAY AREA CIRCUITS DISCUSSES DATA STRATEGIES

Matties: *There are certainly jobs of skill like engineering, but in terms of manufacturing, once you get full automation in here, there is limited human intervention and limited handling. It's really just people monitoring the process, loading a parameter or board possibly.*

Garcia: We know some of these kids in these engineering classes may not pan out to be a full-fledged electrical engineer, but they may have a passion for the industry.. That's why we want to get them early exposure so that they are aware of the opportunities that exist for them.

Matties: *How big is your facility?*

Garcia: It is 30,000 square feet.

Matties: *How many people do you have here?*

Garcia: Slightly under 50. We're not 50 yet.

Matties: *Just under that ObamaCare (laughs).*

Garcia: I can't confirm or deny that (laughing), but yes, we've tried to stay under 50.

Matties: *I think a lot of people have. There are a lot of reasons to do it. But with automation, that allows you to grow beyond what you would have had to hire for that sort of growth.*

Paper: It's a number to keep an eye on, but I think also if we're getting to the stage where we need 60 employees, that's a positive thing.

Matties: *Good. So build it as fast as you can for as little money as you can spend on building the highest quality product possible. It's a pretty simple formula.*

Garcia: Yeah. There's a lot of in-between there, but that's the gist of it.

Paper: It's a simple formula, but the execution is a just a little more complicated (laughing).

Matties: *Great, well thanks for sitting down with me today.*

Garcia: Thank you.

Paper: Thank you. **PCBDESIGN**

Interim CEO Jeff McCreary on Changes at Isola

In a recent interview with I-Connect007, Isola's interim President and CEO Jeff McCreary discussed the impetus for the recent personnel reduction and plant closing that took place, mainly in the U.S. With a realignment towards the Asian market and improved plant utilization in the U.S., Isola expects to become more internationally competitive and improve revenue.

McCreary explained that, with long-time president Ray Sharpe stepping down followed by the reductions, many are probably won-



dering about the company's stability. He stated that the company has remained profitable but has suffered revenue-wise in the past few years, due in part to a general market slump in Asia. By improving on their manufacturing utilization rate in the U.S. (savings in the millions) along with more focus on the Asian market, McCreary sees the company as highly competitive going forward.

McCreary also shared his bullish view on the electronics industry in general and on what the industry may expect from Isola going forward. In addition, he explained what the company is looking for in a new CEO, which should be announced in the next few months.

To read the entire interview [click here](#).



DON'T FEAR THE SHADOWS.

Conquer them with secondary curing – and protect the whole board.
Conformal Coatings from Dymax.

Give your PCB the ultimate protection with Dymax. In addition to secondary curing, Dymax offers:

- Faster, tack-free cure
- Ambient moisture cure in just 2-3 days
- Easy re-workability
- Solvent-free coatings

Let Dymax shed some light on your next project. Contact us today.



www.dymax.com/pcbprotect • 877-396-2988



SnapEDA: The Female-Owned Startup Revolutionizing CAD Data

by **Andy Shaughnessy**

The world of EDA tool development is primarily a man's world, and a middle-aged man's world at that. You won't find a lot of young people in this community, not many women, and outside of marketing departments, young women are few and far between.

But SnapEDA founder Natasha Baker may mark the beginning of a new trend in EDA: young female entrepreneurs. (When was the last time we heard about an EDA startup?) As her company prepared for a major launch, Natasha took time to explain the philosophy behind SnapEDA, and how the company is helping designers and engineers manage an ever-increasing volume of CAD data.

Andy Shaughnessy: *Start out by giving us a little background about SnapEDA, and some of your history in EDA.*

Natasha Baker: I started working in the EDA industry in 2006 as an intern at National Instruments in their circuit design software group, and then returned in 2008 full-time. Around the same time, I was also consulting for Analog Devices. During my time with both companies, I saw the tremendous burden of design data creation that affected not only designers, but also EDA vendors and semiconductor companies, and started thinking about better ways to solve these challenges.

I thought it would be great if there was one centralized place for engineers to get all of the CAD data they



Natasha Baker, SnapEDA founder.

WE ARE THE NEW EASY BUTTON FOR PCB APPLICATIONS



At American Standard Circuits, we want every facet of your PCB manufacturing experience to be uncomplicated. From problem solving and technical expertise to unmatched customer service, we make every step easy until your job is complete.

Digital | RF/Microwave | Flex | Metal Clad | RoHS



American Standard Circuits

Creative Innovations In Flex, Digital & Microwave Circuits

www.asc-i.com

SNAPEDA: THE FEMALE-OWNED STARTUP REVOLUTIONIZING CAD DATA

needed, whether it was CAD components, schematic symbols, PCB footprints, reference designs and simulation models, and provided transparency into quality. However, I decided to wait until I had more skills and experience, so I shelved the idea.

As I continued working full-time, the library creation and data management problem kept popping up in conversations with designers. I also saw first-hand how frustrating the design process could be without readily available libraries, while creating demo boards for trade shows.

After a few years of working full-time, the need on all sides of the industry became clear so I decided to start the company. SnapEDA was launched in beta in October 2013, and raised some funding in May 2015, which is when I started working on the company full-time. Since then, our user base of thousands of registered engineers globally has grown over five-fold. Seeing the value that we're providing to designers, semiconductor companies, or EDA companies is extremely rewarding.

Shaughnessy: *What's your company's "sweet spot," so to speak?*

Baker: SnapEDA is focused exclusively on providing electronics design data compatible with popular EDA tools, including Cadence's OrCAD and Allegro, Altium, Eagle, Mentor's PADS, KiCad, and Pulsonix. Engineers use our platform to find CAD components, schematic symbols, and PCB footprints, so this is currently our fo-

cus, although we'd like to expand to simulation models and reference designs going forward.

What differentiates us most is our growing focus on automation technology to create and vet CAD files. One example of this is our recently introduced Verification Checker, which uses a series of algorithms to uncover common manufacturing issues. For example, the Checker can tell whether the centroid of a PCB footprint is at 0,0, which is crucial for pick-and-place machines, and flag when there is silkscreen overlapping copper.

From a business perspective, SnapEDA is building the most targeted user base of design engineers worldwide who are at the pivotal stage of selecting components for their designs. This is particularly interesting for semiconductor companies looking to increase design wins as the electronics industry gets more diverse.

Most of our users are at mid-sized established companies in a broad range of industries, including industrial control, medical devices and consumer electronics. There is also a growing group of smaller organizations using SnapEDA to create connected devices. For example, SnapEDA has been used within Samsung's Think Tank (the group that produced the Galaxy Gear), as well as startups like TeaBOT, which produces a robotic tea machine.

Shaughnessy: *How did you develop such a large library of footprints and schematic symbols?*

Baker: Our data comes from a variety of sources, including data we've created internally, manufacturer and partner-provided data, and user-generated content. Engineers have told us they want more manufacturer-supplied content. So we've been partnering with companies like Texas Instruments, GaN Systems, and Accelerated Designs (a company that works with many semiconductor companies to create CAD libraries) to provide this.

Shaughnessy: *Some engineers say that they spend a lot of wasted effort on footprints. Do you find that to be the case?*

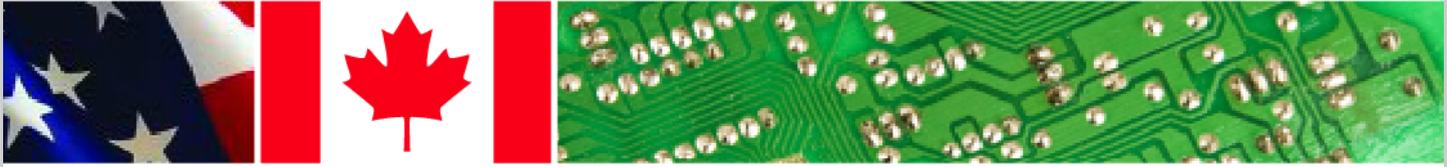
Baker: The Aberdeen Group has estimated that engineers spend nearly half of their days creat-



Natasha Baker with software developer Garret Fick.

In North America

Uyemura is again



#1

for **ENIG
ENEPIG
VIA FILL COPPERS
ELECTROLESS GOLDS**



We are honored, and deeply grateful to our customers,
vendor partners and employees for making us North America's leader in PCB finishes.
Thank you for an extraordinary year, and for the opportunity to be of service.

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



SNAPEDA: THE FEMALE-OWNED STARTUP REVOLUTIONIZING CAD DATA



Intern Adolfo Farias, Natasha Baker, and intern Marcio Moura.

ing or managing CAD data. Engineers are often duplicating the same CAD file created by a colleague in a nearby cubicle or office, so there is a lot of wasted effort.

With SnapEDA, engineers simply search for the part they're looking for, and select their design tool to start the download so it saves considerable time. A core metric we track is the hours we save engineers each week, a function of downloads and pin count, which has grown over 6x in the past six months.

Shaughnessy: *That's impressive. What are the biggest challenges you face in meeting customers' data needs?*

Baker: The biggest challenge is incorporating the wide variety of standards and preferences engineers have when it comes to CAD files. A CAD file that is high quality for one engineer could be poor quality for another engineer who follows another set of standards. So making sure that we provide this transparency into the standards a file follows is what we're focused on.

Furthermore, engineers have their own personal preferences, or need to follow company or application-specific standards. Making the CAD files more customizable is where we're focused next.

Shaughnessy: *How has data management changed over the years?*

Baker: Because of the ubiquity of the Internet and web-based APIs, solving data challenges is becoming much more accessible. For example, electronic component data—such as pricing, availability, datasheets, and specs—is now readily accessible thanks to companies like Octopart (recently acquired by Altium). We use this data at SnapEDA to let engineers know whether a product is available in the marketplace before they download the CAD files.

Shaughnessy: *What sort of opportunities do you see for SnapEDA in the future?*

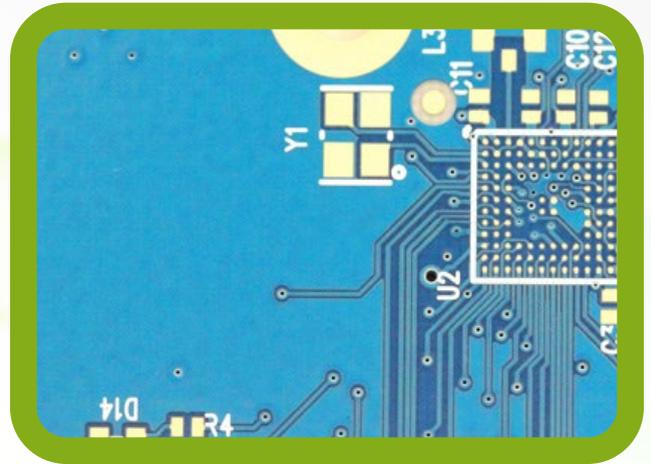
Baker: We're on the cusp of a major revolution in the electronics industry that will see the proliferation of connected devices into every aspect of our lives. Today, the average household owns approximately 10 connected devices. But by 2020, it has been projected that the same household will have over 50 connected devices. Over the next five years, we'll watch as companies launch a "smart" version of nearly every object, from watches to scales, with PCBs being the connective tissue that make these devices a reality.

At the same time, the electronics industry is becoming more diverse as electronic design becomes more accessible. So there's a growing long tail of smaller companies designing new products.

In the past, semiconductor companies could employ an application engineer specifically focused on providing design support to large accounts. But increasingly, innovation is coming from everywhere and it's becoming important to figure out how to provide resources at scale to support the increasingly diverse group of companies designing electronics. SnapEDA is aiming to be the platform that bridges the gap between semiconductor companies and design engineers, helping semiconductor companies increase their design wins by getting their product incorporated into these diverse new products.

We're also planning to launch more premium services for teams, such as improved workflows and collaboration as subscription offerings that will make design data management more accessible to companies creating electronics.

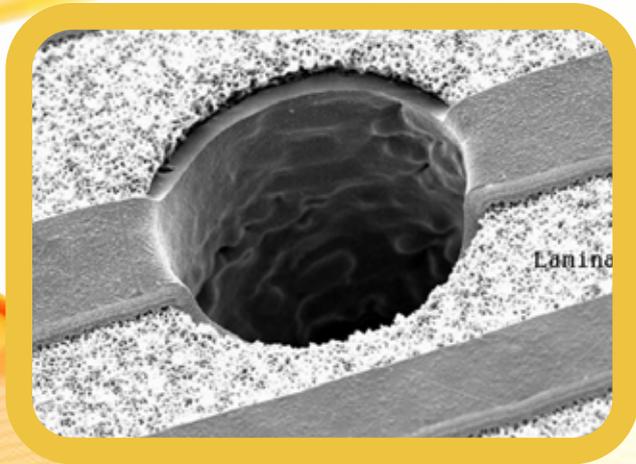
We deliver Advanced PCB Solutions!



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](#)

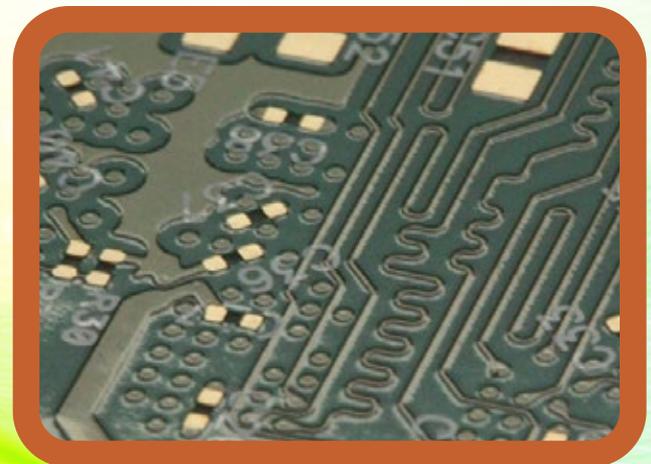


Landless Via Technology

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](#)

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



Impedance Tolerance <5%!

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

[CLICK TO LEARN MORE](#)



CANDOR INDUSTRIES, INC.
ADVANCED PCB SOLUTIONS

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

SNAPEDA: THE FEMALE-OWNED STARTUP REVOLUTIONIZING CAD DATA

Shaughnessy: *Is there anything else you'd like to add?*

Baker: What has been really interesting is seeing the way designers choose new parts. We often hear that they'll choose one part over another just because there's a symbol or PCB footprint available for it on SnapEDA. On some level it makes sense: people will often take the path of least resistance. But at the same time, I wonder if engineers should be making important de-

sign decisions based on the availability of CAD data. So we're looking forward to levelling that playing field going forward by providing semiconductor companies with analytics about the parts engineers are selecting, and making sure their data is well represented on SnapEDA.

Shaughnessy: *I appreciate your time. Good luck.*

Baker: Thank you. **PCBDESIGN**

A New Slant on Semiconductor Characterization

From laptops and televisions to smartphones and tablets, semiconductors have made advanced electronics possible. These types of devices are so pervasive, in fact, that Northwestern Engineering's Matthew Grayson says we are living in the "Semiconductor Age."

Grayson's research team has created a new mathematical method that has made semiconductor characterization more efficient, more precise, and simpler. By flipping the magnetic field and repeating one measurement, the method can quantify whether or not electrical conductivity is uniform across the entire material.

"Up until now, everyone would take separate pieces of the material, measure each piece, and compare differences to quantify non-uniformity," Grayson said. "That means you need more time to make several different measurements and extra material dedicated for diagnostics."

Grayson's research was published on October 28 online in the journal *Physical Review Letters*. Graduate student Wang Zhou is first author of the paper. Undergraduate Heun-Mo Yoo used a Global McCormick Student Ambassador Award to travel to ETH Zurich in Switzerland to help grow the samples and conduct resistivity measurements.

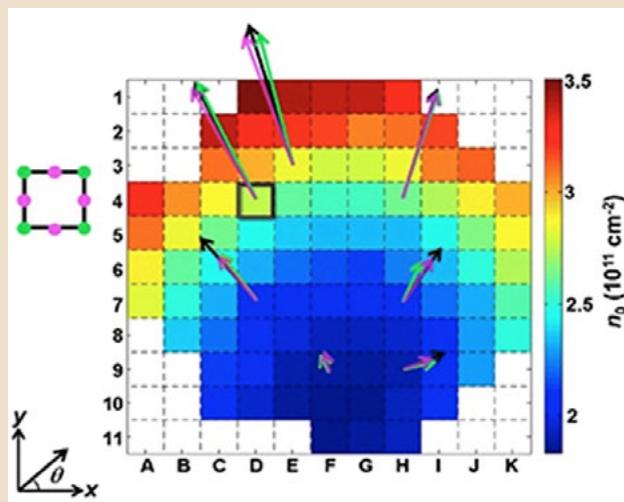
One reason semiconductors have so many applications is because researchers and manufacturers can control their properties. By adding

impurities to the material, researchers can modulate the semiconductor's electrical properties.

Grayson's method can be applied to samples as large as a 12-inch wafer or as small as an exfoliated 10-micron flake, allowing researchers to profile the subtleties in a wide range of semiconductor samples. The method is especially useful for 2-D materials, such as graphene, which are too small for researchers to make several measurements across the surface.

Grayson has filed a patent on the method, and he hopes the new technique will find use in academic laboratories and industry.

"There are companies that mass produce semiconductors and need to know if the material is uniform before they start making individual computer chips," Grayson said. "Our method will give them better feedback during sample preparation. We believe this is a fundamental breakthrough with broad impact."



**APEX
EXPO[®]
IPC 2016**

FORWARD THINKING

FOR TOMORROW'S TECHNOLOGY

Unparalleled Access

What awaits you on the IPC APEX EXPO 2016 show floor? The largest concentration of advanced technologies from the electronics industry supply chain — PCB fabricators, designers, OEMs, EMS companies and more. If you want to experience their forward thinking, you need to be here.

IPC APEX EXPO 2016 is where ingenuity will be unveiled.

Prepare for tomorrow in 2016. Meet customers, network and be inspired by all that you experience on the show floor.

Forward thinking begins at www.IPCAPEXEXPO.org.

LAS VEGAS/NEVADA

LAS VEGAS CONVENTION CENTER

CONFERENCE & EXHIBITION: **March 15-17**

MEETINGS & COURSES: **March 13-17**

www.IPCAPEXEXPO.org



DATA

EMA: Helping Technologists Manage Disparate Data

by Andy Shaughnessy

Today's EDA tools are better than ever, but managing design data, from schematics through Gerbers, can be an unwieldy task. I recently interviewed Manny Marcano, president and CEO of EMA Design Automation. He discusses EMA's approach to managing a variety of types of complex data, the need for seamless data processes, and the future of compliance-aware design.

Andy Shaughnessy: *Why don't you start by giving us a little background about EMA.*

Manny Marcano: EMA was started in 1989 when I was selling P-CAD software in upstate New York. In 1998 when EMA had grown to about 10 employees, we signed on as a Cadence Design Systems value added reseller (VAR) selling Allegro PCB design tools in the New York and mid-Atlantic area. That went well, and in 2003 Cadence chose EMA to be the exclusive provider of OrCAD software in North America. EMA expanded quickly and began working with

customers to understand their processes and their challenges. This led to a tight working relationship between EMA and various functions within Cadence, including both their product marketing and R&D groups. EMA also began creating custom software to solve specific customer challenges, which later became generalized and turned into products that complement the OrCAD offerings. Through it all, EMA has operated on the premise that EMA's success is directly tied to Cadence's success, and that concept has created the positive and successful relationship we have with Cadence.

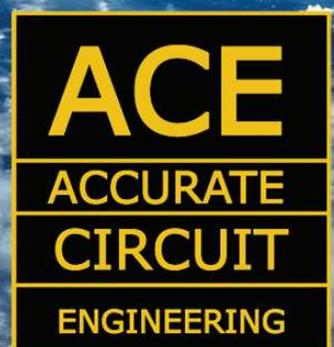
Shaughnessy: *What are your customers' biggest demands when it comes to managing data?*

Marcano: Customers want a complete solution to manage all the work-in-progress (WIP) PCB data before it is released to manufacturing. This can be the design files themselves as well as the BOM, netlist, Gerber, simulation results and even specifications or datasheets. Everything that goes into designing that PCB should be put under data management, or users end up creat-

**When failure is not an option, don't
leave your high tech PCBs to chance.**

**Have an ACE up
your sleeve!**

- Specializing in high technology quick turn prototypes, as fast as 24 hrs
- MIL-PRF-55110, MIL-PRF-31032A, AS9100, and ITAR certified
- Leading experts in RF, Hybrid PTFE/FR4 and Antenna/Microwave



www.ace-pcb.com

sales@ace-pcb.com

EMA: HELPING TECHNOLOGISTS MANAGE DISPARATE DATA



Manny Marciano, president and CEO of EMA Design Automation.

.....

ing workarounds and storing files locally which breaks down the effectiveness of the system.

Customers also stress that the experience for the end user needs to be seamless. In the past our customers have tried to use ad hoc processes or connect with external systems to manage their PCB data. While this is better than nothing, it requires extra manual effort from the engineering team to go outside their CAD environment to push and pull data. Without any checks or processes within the native CAD tool, design teams will often use the data management system haphazardly or forget for a couple revisions then push an update. This is especially prevalent when deadlines are tight or something needs to be done quickly. The extra effort and steps to manage the data are ignored in favor of speed which can have serious consequences later on if data is lost or accidentally overwritten. PCB design teams really need data management to be a natural part of the design process.

Shaughnessy: *What kinds of PCB data are the toughest to manage?*

Marciano: One aspect that makes PCB data tough to manage is the fact that a lot of the design files themselves are in a proprietary or binary format, which makes change management difficult if you don't have a way to read those files. PCB design is also very iterative in nature, which creates large volumes of data during the design phase that needs to be revision controlled. All this data is typically broken up into lots of different files that are all derived from or related to each other in some way. Files like the netlist, models, Gerbers, and simulation results all need to be tracked with the core PCB files from which they were generated. Without having an understanding of the file associations inherent to the PCB design process, things can quickly become out of sync.

For example, you can be doing a great job of posting your PCB files to the shared drive or repository when you have finished editing them, but if you are not also tracking the file outputs along with those revisions there is a serious chance you may inadvertently send an old version of the BOM or the wrong Gerber file. So even though your core design files were managed correctly, the outputs from those files that drive manufacturing and purchasing could be incorrect, leading to re-spins, a bad board, and late nights trying to figure out what went wrong. More importantly, this results in higher cost and lost market opportunities.

Shaughnessy: *Where are the biggest challenges you face in meeting customers' data needs?*

Marciano: One of the tricky items is that PCB design projects involve many sets of related and derived files like the netlist, BOM, and Gerber which are related to the core design files themselves and are often in a binary format. Keeping track of the related PCB data is just as important if not more important than the design files themselves as these derived files are what drive the manufacturing and procurement process. Without an in-depth understanding of how these various file types are inter-related it is easy to lose control of your PCB design data.

Ventec at your fingertips.

Direct access to all the news and technical updates from Ventec is now at your fingertips thanks to the user-friendly new Ventec app, which gives up-to-the-minute information on all developments, ongoing projects, appointments and technical data updates from Ventec International Group.

With this free app, you can:

- Get the latest news from all of Ventec's operations worldwide
- Access the latest datasheets and process guidelines
- Be notified when new datasheets are available for download

Download yours today!

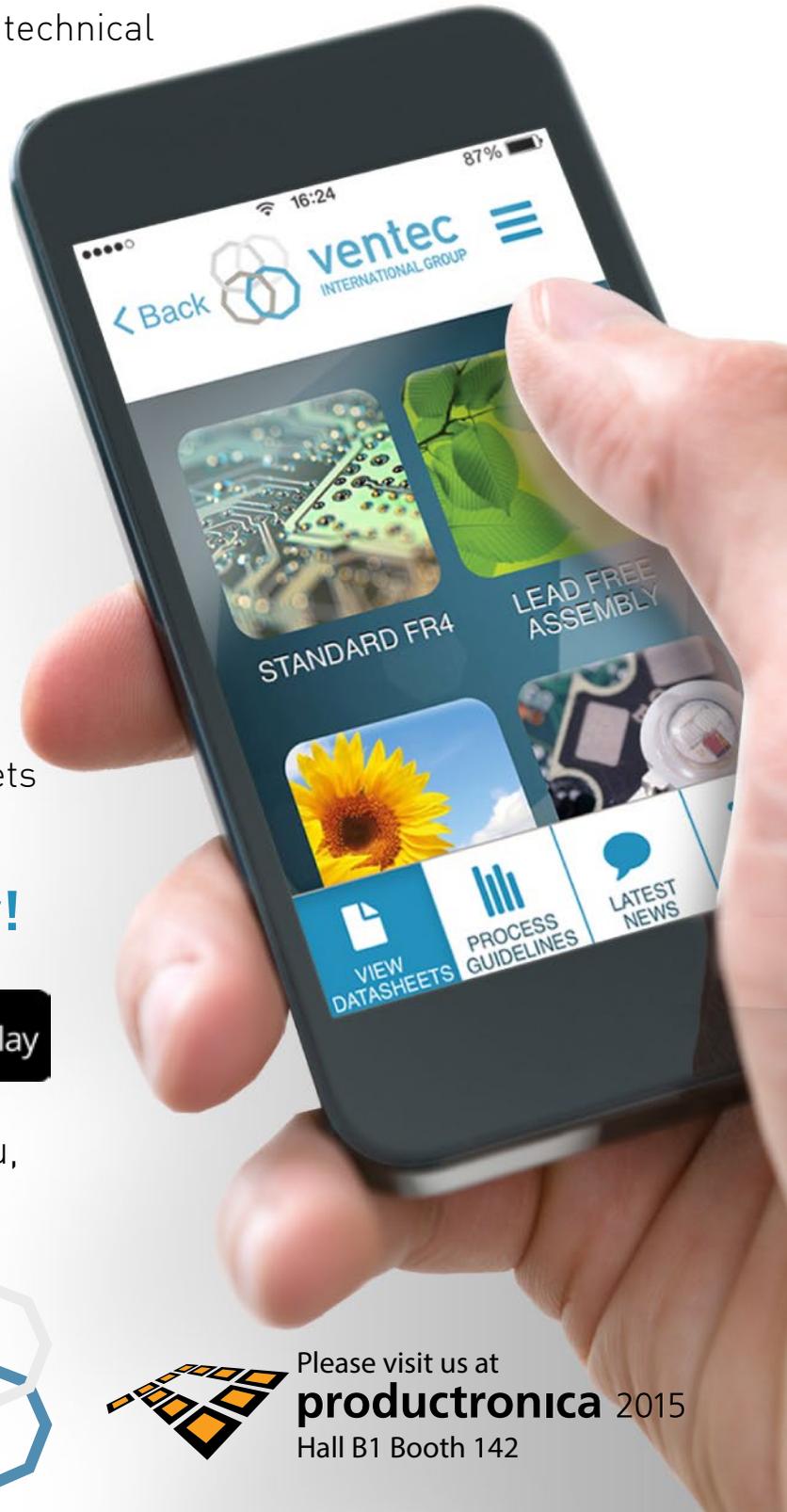


Wherever technology takes you,
we deliver.

ventec



Please visit us at
productronica 2015
Hall B1 Booth 142



EMA: HELPING TECHNOLOGISTS MANAGE DISPARATE DATA

Shaughnessy: *How does EMA help designers manage their data?*

Marcano: We help customers in many ways. First, we provide PCB CAD aware data management tools from Cadence to allow engineers to manage their PCB data in the context of their design environment effectively. We also provide a number of consulting services to help companies adopt best practices around PCB data management. This often starts with process assessments to help them understand where their holes are or where inefficiencies can be eliminated. Then we can help design and implement an effective and efficient environment based on both the needs of the engineering team as well as integration to corporate systems like PLM.

Shaughnessy: *I see that there's a new OrCAD EDM tool that addresses this issue. Tell us how this works.*

Marcano: OrCAD EDM is a PCB data management environment built specifically for PCB design teams using OrCAD. It is integrated directly inside the OrCAD user interface providing a seamless process to check-in and check-out files

.....

“Management and other stakeholders can easily review project status in real-time directly within OrCAD EDM or through a web browser.”

.....

as you need them. It is aware of the different file associations and file types that are derived from your core OrCAD PCB data so it can track them effectively. OrCAD EDM will even generate a set of these derived files as you check-in your design you can be assured they always stay in synch.

Management and other stakeholders can easily review project status in real-time directly within OrCAD EDM or through a web browser.

Full administration and permissions management is also available so organizations can collaborate with their partners while ensuring their critical IP is protected and secure.

Shaughnessy: *EMA helps customers manage WIP PCB data, component data, etc. How does it all work together?*

Marcano: PCB data and component data go hand and hand. WIP PCB data includes all the files associated with the PCB like the schematic, PCB, libraries, 3D models, simulation models, netlist, Gerbers, etc., while the component data covers parametric information about the specific parts being used including things like tolerance, resistance, voltage as well cost and manufacturer part number.

We help design teams manage both these data types together so they can easily map their component parametric data to the PCB libraries and files as they use them. The component data helps drive part decisions and ultimately the BOM while PCB data management ensures the BOM is in synch with the actual design files themselves.

Shaughnessy: *How do PLM tools fit into this equation?*

Marcano: We are completely complimentary to PLM systems that companies may already have in place or are considering. Our solution is focused on WIP PCB data management. This is managing the PCB data when it is still in development stage before it is released for production. During this time change is occurring rapidly as the design is being created and modified to meet the desired design specifications. PLM is designed to manage a product through its life-cycle once it is released. It is really not geared for the rapid change associated with a project before it released.

In the past we have seen customers try to use the PLM for WIP and it never really works out as they expect. We see an opportunity to connect WIP management to the PLM space for PCB similar to what is done in the MCAD world, where each MCAD vendor has their CAD aware WIP environment that can then take the

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



EMA: HELPING TECHNOLOGISTS MANAGE DISPARATE DATA

design data and help push it up to PLM when it is ready for release.

There is also quite a bit of data in PLM that an engineering team could and should have access to as they make design decisions. One of the most important pieces is information on available and approved components. We are able to integrate with most PLM systems to allow design teams controlled access to this engineering information directly inside their CAD tools so they can make better part decisions early in the process. Engineering likes this because they don't have to leave their CAD environment to get this info, and they can see it in the context of their engineering data. Similarly, PLM users like this because it makes sure engineering is using PLM content as the single source of data, so they are confident that the BOMs they receive contain correct and approved parts, and that greatly reduces late stage issues when the BOM is sent to procurement.

Shaughnessy: *How has data management changed over the years?*

Marcano: Shared drives and email exchanges have been the predominant data management methodology for PCB design teams until now. While those may have worked before, companies are finding these methods aren't able to meet the time to market and productivity demands placed on PCB design teams today. This has been the big reason we see customers looking for a much more accurate and efficient way to manage their PCB data. Designs are just getting too complex, project deadlines are too short, and the margin of error is too small for teams to rely on ad hoc and inefficient systems. They need a solution that is geared specifically for managing PCB design data.

I also think technology has really helped bring down the cost of delivering PCB-centric data management solutions while improving ease-of-use. Setting up systems like this in the past could be multi-year efforts costing huge sums of money in hardware and software as well as consulting and customization. With OrCAD EDM we are able to get new customers up and running in just a matter of days with access inside their CAD tools and through a web browser.

Shaughnessy: *What sort of data management opportunities do you see for EMA in the future?*

Marcano: We see a whole range of opportunities in this space. First, we see this as a real driver to help PCB design teams be more productive and more collaborative. I think there is still more we can do there to help PCB designers have access to the information they need, when they need it. For example, compliance-aware design is something we are looking at. The idea is, why wait until you are ready to order your parts to verify they meet regulatory requirements or are orderable or meet your cost targets? Why not provide that information upfront to the designer when they are choosing the parts so they can make those decisions when the cost of change is the lowest? We think there is tremendous opportunity to help empower the PCB design team with more information to make intelligent business and manufacturability decisions as well as engineering ones.

We also see a growing need for PCB design teams to be able to collaborate, share IP, and build out metrics regarding project status and progress. As interfaces become more standardized, PCBs start to become a collection of common parts and connections with specific areas of unique differentiation for that particular product. If engineers are designing in the same interfaces over and over, how do we make it so they can effectively leverage that IP across designs to give them time to focus on the challenging or specialized parts that will differentiate their product from the competition? From a metrics side, once you are able to track and follow designs through their development process you can start to build statistics on how long certain designs take to complete or where problems typically arise. This can greatly help companies with planning and schedule management if they have an accurate historical record based on their own design data from which to build their project estimates.

Shaughnessy: *Thanks for the update.*

Marcano: Thank you. **PCBDESIGN**

ATTENTION BUSINESS OWNERS:

IQ
TEST
TODAY!



BRING YOUR
BRAIN

▶ CLICK HERE TO TAKE THE BizBrain IQ TEST





TTM Technologies Consolidates Manufacturing Operations

TTM Technologies Inc., a major global printed circuit board manufacturer, today announced that it plans to close three facilities in order to improve total plant utilization, operational performance and customer focus.

The Re-industrialisation of Europe

With an inquisitive mind and a head for challenges, besides the ability to think outside the box and the courage to dare to be different and strive to be first, Spirit Circuits MD Steve Driver can be relied upon to grab the attention of an audience of PCB professionals. As keynote speaker at the Institute of Circuit Technology Hayling Island Seminar, he lived up to his reputation with a motivational presentation, the two themes of which exemplified his latest entrepreneurial venture.

Inaugural Boards, Chips and Packaging IMPACT Conference Adds 15 Industry Experts to Presentation List

Isola Group and Semico Research announced today that the inaugural event, “Board, Chips and Packaging: Designing to Maximize Results Conference” is garnering significant interest from the embedded design community with the addition of 15 industry experts added to the agenda.

World PCB Production in 2014 Estimated at \$60.2B

The world market for PCBs reached an estimated \$60.2 billion in value in 2014, growing just 0.7% in real terms, according to IPC’s World PCB Production Report for the Year 2014. Production growth in China, Thailand and Vietnam compensated for declining PCB production in most other regions.

AT&S Highlights the Future of PCB Industry

Miniaturisation and performance enhancement are leading trends not only for mobile devices such as smartphones; applications in the automotive sector and in medical technology are also getting progressively smaller, more efficient and more powerful.

N.A. PCB Book-to-Bill Ratio Reaches Five-year High in August

Total North American PCB shipments increased 1.5% in August 2015 compared to August 2014. Year-to-date shipment growth stood at -0.1, reflecting the negative sales growth rates seen earlier this year. Compared to the previous month, PCB shipments were up 3.6%.

Final Surface Finishes for Automotive: No One-Size-Fits-All Solution

Regardless of whether your application is automotive, medical or military, there are many factors to consider when selecting a final surface finish. Cost, lead or lead-free requirements, end environment, shelf life, fine-pitch components, RF applications, probe-ability, thermal resistance and shock and drop resistance, to name a few. There is not a one-size-fits-all finish. Understanding the advantages and disadvantages of each surface finish allows the designer to select the finish that best fits each particular application.

Becoming an Automotive Supplier—Proceed with Caution

Possibly further fueling a growing desire in our industry towards participating in the automotive electronics sector have been further technology advancements in automobiles, as well as a tear in Tesla’s stock price and headlines.

All Flex Now Offers High Temp Flexible Circuits & Heaters

All Flex, manufacturer of flexible printed circuits and heaters, has developed a polyimide-based material set specifically targeting the market for high temperature flexible circuits and flexible heaters. Increasing demands for high temperature applications have driven the development process at All Flex over the past 18 months.

Testing Flexible Circuits, Part 2: Raw Materials and Components

Even though the raw materials may individually meet certain specifications, it is important to determine if the final product configuration meets those specs, which means that testing done at the finished circuit level is also needed.

53%^{cited}

Limited time to
develop PCBs

32%^{cited}

Need to
support better
collaboration

21%^{cited}

Manufacturing
delays due to data
inconsistencies

19%^{cited}

Better predictability
of design cost

What Are Your PCB Data Management Challenges?

EMA Design Automation commissioned independent research firm The Aberdeen Group® to conduct a study on current trends and challenges with regards to PCB data management.

Download the full research paper to learn how best in class companies are taking control of their PCB design data and turning data management into a competitive advantage.

[Download Report >](#)



DownStream Takes on Data Documentation Management

by **Andy Shaughnessy**

Most PCB designers love their jobs. But designers will also tell you that as much as they enjoy laying out the board, they dislike the final data documentation step, which often involves various formats, including handwritten notes. Enter DownStream Technologies, a company founded 14 years ago to address the challenges related to post-processing the design. Senior Product Marketing Manager Mark Gallant recently discussed the company's efforts to take the pain out of data documentation, even as data becomes more complex.

Andy Shaughnessy: *Why don't you start out by giving us a little background about DownStream.*

Mark Gallant: DownStream Technologies was founded in January 2002 by former PADS Software executives who acquired the assets of Advanced CAM Technologies—products with the brand name CAM350—from Innoveda prior to its acquisition by Mentor Graphics. The newly formed executive team chose the name DownStream Technologies to reflect their goal of delivering technology focused on the “downstream” phase of the PCB design process. Specifically, it's

technology to help bridge the chasm between PCB design and PCB manufacturing domains.

At the time of DownStream's founding, the predominant method of transitioning a PCB design to manufacturing involved post-processing a design to extract multiple files. Each file was independent of the other in format and content, and disjointed from the original design from which they were derived. What we call the “traditional” process for handoff to manufacturing worked well enough to take us from the days of PCB design by light table up to the start of the 21st century. However, as technology continued to evolve in complexity and market windows to profitability continued to shrink, the traditional process led to higher costs, lower quality and ultimately delayed time to market.

Our founders believe the transition from virtual design to the manufacture of the physical product could be, and should be, improved. We observed this trend in several of our early adopter customers—organizations that were the first to implement new technologies and processes which, once proved out, led to adoption by the rest of the market. To reduce manufacturing delays, these early adopters educated themselves on the PCB manufacturing process. Their newfound knowledge led them to intro-

STRIVING FOR **EXCELLENCE** IN EVERYTHING WE SAY AND DO.

- Prototype to Production
- Time Critical Production
- Commitment to Customer Service
- Best Quality & Delivery Performance
- Fair and Competitive Pricing/Costs
- Rapid Response to Quote Requests
- Flexibility Scheduling
- Stock/Consigned Inventory Programs

[Tour our Facilities](#)

[View our Capabilities](#)



MANUFACTURERS OF QUALITY PRINTED CIRCUIT BOARDS

The Absolute Best Value in
High Technology Circuit Boards

847.891.5800 • www.eagle-elec.com

DOWNSTREAM TAKES ON DATA DOCUMENTATION MANAGEMENT



Mark Gallant, Senior Product Marketing Manager, DownStream Technologies.

duce DFM analysis and single file design exchange into their new product introduction process. This revised approach, rather than the “hand it over the wall and hope” method, resulted in significant time-to-market gains. Fundamentally, they were shifting their attention from “upstream” processes to

refocus on “downstream” processes, shifting from how a design is created to how it is to be manufactured. The trend of refocusing on manufacturing and a belief that DFM tools and singular design exchange files were evolving from “nice to do” to “must do” was the catalyst for the creation of DownStream. Since our start we have introduced a full suite of DFM analysis, PCB documentation, and assembly panel design tools.

Shaughnessy: *What are your customers’ biggest demands when it comes to their new product design process?*

Gallant: In the world of electronic product development, and specifically PCB design, the demand is clear: a successful, error-free transition from design to manufacturing. Many of our customers believe the transition process from design to manufacturing is the next frontier of time-to-market gains. Their focus is no longer just faster PCB design, which emphasizes and prioritizes the design phase of new product development. Instead the trend is towards PCB design and manufacturing, and giving equal weight to design and manufacturing. The question, then, is why is this happening, and why now?

Looking back from today’s vantage point, it is an obvious and necessary direction. Electronics manufacturers always list time to market, cost and quality as top priorities in new product introduction. Be the first to market, produce it at the lowest cost, and provide the highest

quality. To be successful, the manufacturing process and its many issues, requirements and challenges must be minimized before production begins.

The demands from this new focus on the transition of design to manufacturing are to support a new downstream-focused engineering process, develop or enhance our tools to analyze designs for “manufacturability” issues, and support industry standard, intelligent design exchange file formats that drive the entire manufacturing process for new products.

Shaughnessy: *What type of PCB data is a challenge to manage?*

Gallant: Under the traditional methodology, the many different files produced in what we refer to as “over the wall” handoff between design and manufacturing are the challenge. Some file examples are soft copy drawing files, Gerber files, bill of materials (BOM) lists, coordinate (centroid) lists, and many more. Maintaining the synchronicity of these files through engineering change orders (ECO) is an even greater challenge for us as an EDA vendor, and our customers.

Shaughnessy: *Where are the biggest challenges you face in meeting customers’ data needs?*

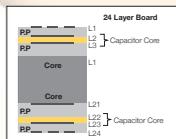
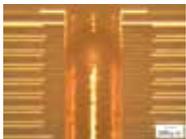
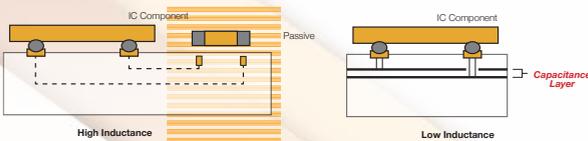
Gallant: This is an interesting question. On the one hand, there is no doubt our customers are in need of improving the transition from design to manufacturing. It is this requirement that accelerated the need for industry standard, intelligent, design exchange files for transferring design data to manufacturing. However, while many of our enlightened customers transition designs to manufacturing using IPC-2581 or Mentor’s ODB++, many continue to utilize the same decades-old handoff process. We have witnessed time-to-market gains by many of our customers who adopted a design exchange file-driven transition to manufacturing. We believe significant gains in time-to-market are not possible without adopting such a design exchange file-driven process.

The time has come for the old handoff to manufacturing process to take its place along-

THE SMART SOLUTION TO IMPROVE SYSTEM PERFORMANCE

FARADFLEX ULTRA THIN MATERIALS FOR HIGHER PERFORMANCE PCBs

Faradflex is a high performance, ultra-thin laminate or resin coated copper material designed as an integral part of the PCB, module or chip package.



FaradFlex® Improves System Performance, Price, Quality, and Reliability by:

- Reducing PCB size and/ or thickness
- Utilizing VSP Copper to provide the lowest loss signal for high speed designs
- Increasing functionality
- Improving power distribution (power integrity)
- Reducing the number of SMT Discrete Caps, Traces, Vias, Pads
- Eliminating hundreds (thousands) of SMT and etched and drilled features sites for possible failure on each PCB

For a free sample contact us at:

sales@oakmitsui.com or call us at 503-313-1905



FARADFLEX®

EMBEDDED CAPACITANCE

DOWNSTREAM TAKES ON DATA DOCUMENTATION MANAGEMENT

side hand taping, light tables, digitizers and joy sticks. We believe the trend towards adopting design exchange files is well underway and will accelerate rapidly in the next few years. Evangelizing and accelerating the adoption of design exchange files is one our biggest challenges.

Shaughnessy: *How does DownStream help designers manage their design data?*

Gallant: At Downstream, we have long recognized the need for eliminating the old multi-file approach of passing design data to manufacturing. In support of enhancing the transition from virtual to physical product, we have assumed a proactive role in the development and adoption of the IPC-2581 intelligent design exchange format, a non-proprietary, robust standard for a design exchange file format.

.....

“At Downstream, we have long recognized the need for eliminating the old multi-file approach of passing design data to manufacturing.”

.....

On the awareness side, we are equally proactive with our membership in the IPC-2581 Consortium, an expanding group of organizations involved in all aspects of PCB design and manufacturing. This group includes OEMs, EDA tool providers, PCB fabricators, PCB contract manufacturers and PCB test companies. We embrace the consortium's goal to create awareness for and accelerate adoption of the IPC-2581 format as an open global standard format.

Shaughnessy: *Have you updated your tools to address this issue?*

Gallant: Yes, we have. Downstream has developed bi-directional interfaces for IPC-2581 and Mentor ODB++ for all of its products. We also

developed our tools to fully utilize these intelligent design formats to perform advanced DFM analysis, design assembly and fabrication panels including flip and mixed-design panels, and define assembly process steps and create visual aids for each assembly step. Our tools can extract centroid files for a single design or a stepped up panel, filtered by process step and/or assembly variant, and automatically create complete PCB documentation sets for engineering and manufacturing. In addition, supporting singular standard exchange file formats facilitates a simplified ECO process. Any analysis or documentation authored in the DownStream tools is updated instantly by simply importing the updated file into our tools.

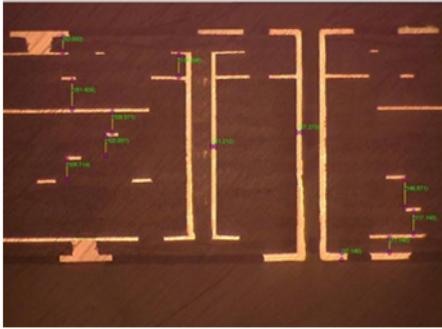
By supporting intelligent files used in the handoff from design to engineering, DownStream can automate the creation of detailed PCB documentation meeting the needs of design and manufacturing. And updates to both engineering and manufacturing's documentation are done automatically simply by re-importing the new design database.

Shaughnessy: *Are designers still providing minimal documentation during data handoff to manufacturing?*

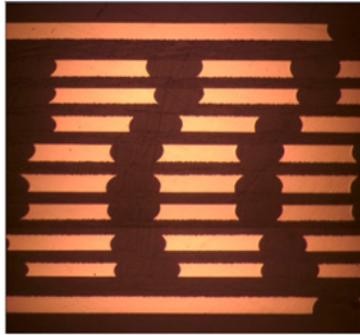
Gallant: PCB designers are accustomed to handing off hard copy fabrication and assembly drawings, Gerber files, and NC drill files. However, designers are hindered by the limited documentation functions in native PCB CAD tools. It is not that the designer wants to provide poor documentation, but the time required to create detailed documentation using current methodologies is challenging with limited resource capacities. As a result, some documentation sets are still very basic. Once the design and documentation are transitioned to manufacturing, whether in-house or to third-party manufacturers, another documentation cycle begins. The manufacturers create manufacturing, assembly, test and other process-based documentation derived from the engineering documentation. Unfortunately, much of the documentation done in manufacturing is primarily a manual effort using a mix of tools like Visio, MS Word & Paint, mechanical tools such as AutoCAD,

EXTRAORDINARY

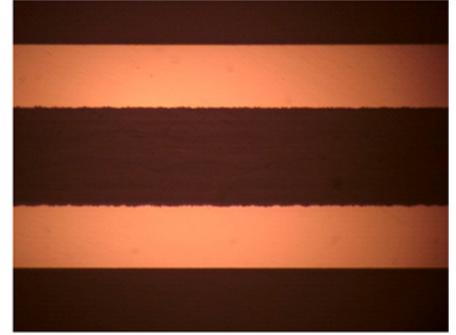
Heavy copper PCB - Microsection



Via in Pad Technology (Micro-via)



10 Layers: 140 μ m copper finished in each layer (4oz cu) with Buried via (L2- L9) and PTH (L1- L10)



2 Layers: 525 μ m copper finished in each layer (15oz cu)

With our strengths in R&D, we are capable of turning out PCBs that meet your specific requirements promptly at competitive prices. ANS is devoted to technological innovation and providing our customers with highly satisfactory and speedy service.



www.anspcb.hk

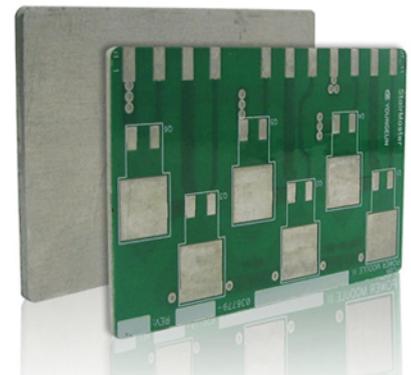
Head office:

ANS Industrial Co. Ltd.
Kowloon, Hong Kong
Tel: (852) 2757 1882

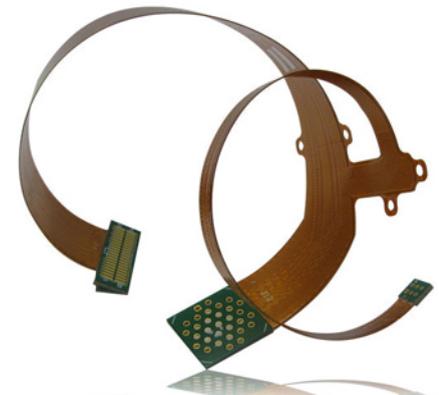
Factory:

Remapp PCB (Foshan) Co.,Ltd.
Guangdong Province, China
Tel: (86) 757-8862 3888

Email: sales@anspcb.hk



Multi-layered aluminum board
lead-free HAL



8 layer rigid-flex
30mm flexible finished thickness



DOWNSTREAM TAKES ON DATA DOCUMENTATION MANAGEMENT

etc. The depth and breadth of documentation requirements for one PCB assembly was a key motivation in our development of Blueprint-PCB, a tool developed explicitly for authoring PCB fabrication and assembly documentation.

Shaughnessy: *What sort of opportunities do you see for DownStream in the future?*

Gallant: As organizations put more focus on how designs are built, and invest in processes and tools to improve the transition to manufacturing, there is great opportunity on both sides. We see opportunities for electronics manufacturers to meet their goals for time to market,

lower their costs for new product introduction, and improve product quality. The same is true for tool providers such as DownStream Technologies, as we recognize the trend towards a design and manufacturing focus. Working together with our customers, developing and enhancing our tools to meet their demands will result in market-leading tools, specifically developed to offer a less error-prone, efficient transition from design to manufacturing.

Shaughnessy: *Thanks for speaking with us.*

Gallant: Thank you. **PCBDESIGN**

Supercomputer with Turbocharger

At the final review, the 16 partners involved in the EU-funded project DEEP presented their innovative HPC platform: a 500 TFlop/s prototype system implements the Cluster-Booster concept which has a lot in common with a turbocharged engine.

The DEEP system achieves the highest density and energy efficiency due to Eurotech's Aurora technology, while it showcases the EXTOLL HPC interconnect, and also leverages Intel multi- and many-core processors. ParaStation MPI, provided as part of ParTec's ParaStation ClusterSuite, has been turned into a Global MPI, the key system software component linking Cluster and Booster.

Scalability, energy efficiency, programmability, and manageability are major challenges on the way to building exascale-class supercomputers. To address them, the collaborative DEEP R&D project implemented the novel Cluster-Booster concept, a heterogeneous architecture that enables applications to always run at the right level of concurrency: highly scalable code parts profit from the throughput of the many-core Booster.

The final DEEP system is up and running at JSC: with a peak performance of 500 TFlop/s, it uses Eurotech's Aurora technology to achieve tight packaging (the whole system uses less than two racks) and high energy efficiency through direct liquid cooling.

DEEP has developed a complete software stack that features an easy-to-use and familiar programming environment for application developers, and can achieve an optimal match between hardware and application characteristics. A global MPI implementation covers both Cluster and Booster, and is based on the fully MPI-3-compliant ParaStation MPI by the Munich-based software company ParTec.

Last but not least, six real-world HPC applications from science and industry were optimized for the DEEP concept. The work resulted in modernized versions of the codes, which are now ready to achieve high performance across a wide range of architectures. Initial results on the final DEEP system show the performance potential and clearly demonstrate the advantages of its architecture, such as its high flexibility and efficiency in using system resources.

The prototype system at JSC will be made accessible to HPC application developers outside the DEEP project.





Grow with Ventec USA!

As we continue to expand our operations in the USA, we are looking for talented and dedicated people to help us turn our vision of being the best technical laminator into a reality.

Our openings include:

- Vice President of Operations (USA)
- Technical Service personnel
- Sales staff in specific territories
- QTA press operator

[LEARN MORE ABOUT OUR USA EXPANSION »](#)

If being part of our industry-leading team interests you, we invite you to join us in our commitment to the US market. Please contact Jack Pattie at 978-265-0734 or by email at jpattie@ventec-usa.

Wherever technology takes you, we deliver.

Why Autorouters Don't Work: The Mindset!

by **Barry Olney**

IN-CIRCUIT DESIGN PTY LTD AUSTRALIA

Ask any group of PCB designers what they think of autorouters and the majority will say that they do not use them because they do not work. I have been battling this mindset for over 20 years now and it still persists today, even with the dramatic advances in routing technology. This way of thinking generally comes from those designers who use the entry-level tools that have limited routing capability. But even the most primitive autorouter may have some useful features. It's all about changing that mindset of the designer and having a crack at it.

I started laying out boards back in the Bishop Graphics days where layout began with a pencil sketch, on graph paper. Then, donuts and fine black tape were stuck to clear film, at twice the actual size, to produce the required connectivity. The 12 mil tape, which we re-

ferred to as "spiderweb," was the thinnest trace width (6 mils finished) manufacturable at that time. It was really a matter of just connecting the dots. Double-sided layouts were sometimes stuck to the same film to improve registration, using red and blue colors to photographically distinguish the layers. But routing has come a long way since then.

The first computer-based PCB design tools that emerged in the late 1970s were grid-based, ran on DOS or UNIX operating systems, and were very basic. Again it was still just connecting the dots, with a graphic trace from point-to-point to build up the layout, and then drawing the circuit on an XY plotter. Basic, but it was effective for the construction on single- and double-sided boards. The next step was to include a netlist for connectivity and then to draw the schematic graphically and extract the netlist to

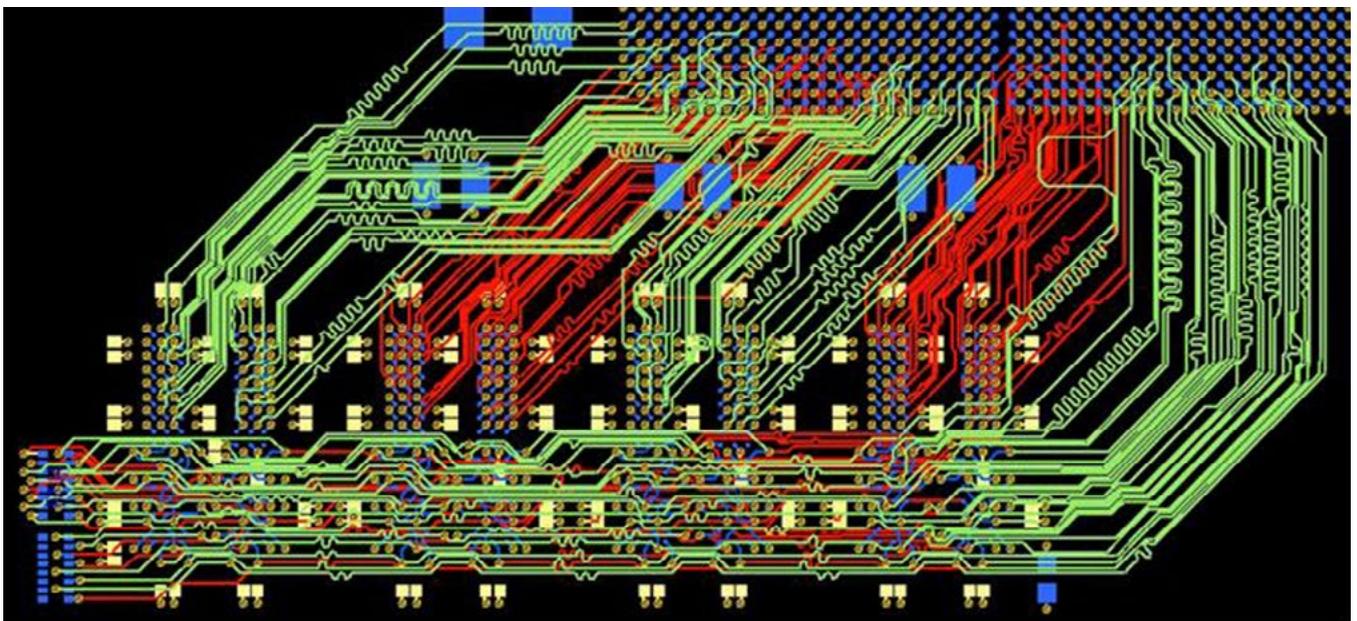


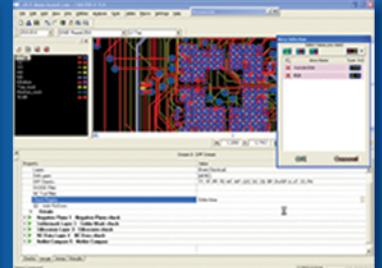
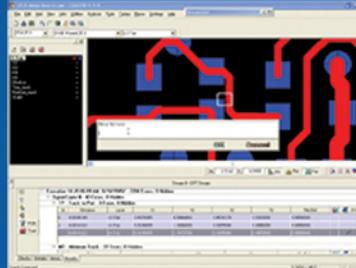
Figure 1: DDR3 memory fly-by address/clock and point-to-point data/strobe tuning.

Solutions to Ensure PCB Manufacturing Success!



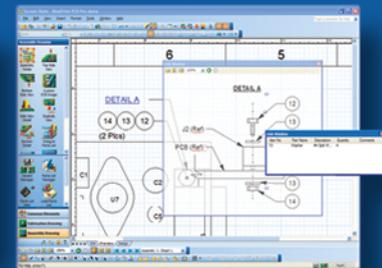
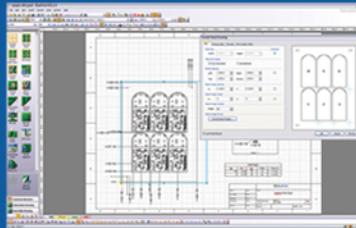
CAM350

Verify and Optimize PCB Designs for Successful Manufacturing.



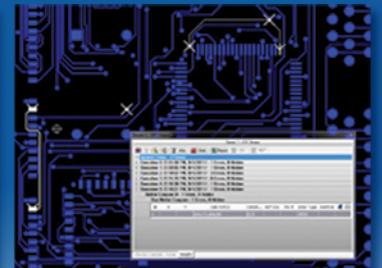
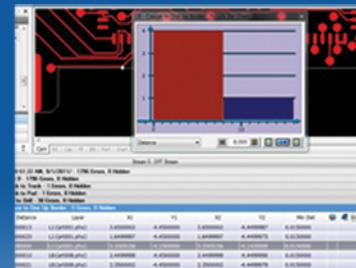
BluePrint-PCB

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.

WHY AUTOROUTERS DON'T WORK: THE MINDSET!

the PCB database. This improved database integrity dramatically.

PCB routers were developed using either the grid-based, gridless, shape-based or geometrical approaches. The first were maze and line searching routers that use an imaginary gridded workspace, while a gridless router uses a workspace with available polygon areas to accommodate the new paths. In a shape-based router, each entity on the board is represented as polygonal geometry with no reference to a specific routing grid. This enables the router to cope easily with boards in which there are SMT devices and fine-pitch BGAs with a variety of pitches and odd shapes. Also, unlike a grid-based router, a shape-based autorouter does not have to work at a particular resolution, so routing of high-density or fine-pitch boards is not significantly slower than for lower density work. Put another way, routing time depends only on the available memory, the number of objects on the boards and on the number of connections to be routed. Later, topology routers allowed designers to plan the strategy for a set of nets with attributes to define routing layers, bias and rules.

The first autorouters were not very capable, limited by computing power and lack of memory. They added too many vias, wasted space due to the strict XY bias, and the quality was

poor compared to manual routing. I recall that I used to set up our Advanced Technology Designer Star router to run on the MicroVax mainframe over the weekend, only to find it 50% complete by Monday morning. However, autorouters evolved, like all technology, to include angle routes, reducing vias, push-and-shove algorithms, rip-up and retry, spreading and gloss passes. But so also has interactive routing.

Probably the most popular shape-based router, 20 years ago, was Cooper & Chyan Technology's Specetra router. The Specetra router was used by many PCB layout tools and interfaces to the router still exist today. Design constraints and routing strategies were setup in a "do file" which contained the sequence of commands. The routing was not graphically visible but the routing status was indicated and updated. Cadence's Specetra for OrCAD is still available today.

In the mid 1990s, Intergraph Electronics (VeriBest Inc.) came up with arguably the best routing technology still available today. Mentor Graphics has made considerable improvements to the router since acquiring this technology, and it is now available in both the Xpedition and PADS flows. They provide a selection of routing tools with each optimized to perform a particular function.

The fanout of a high-pin-count BGA is the

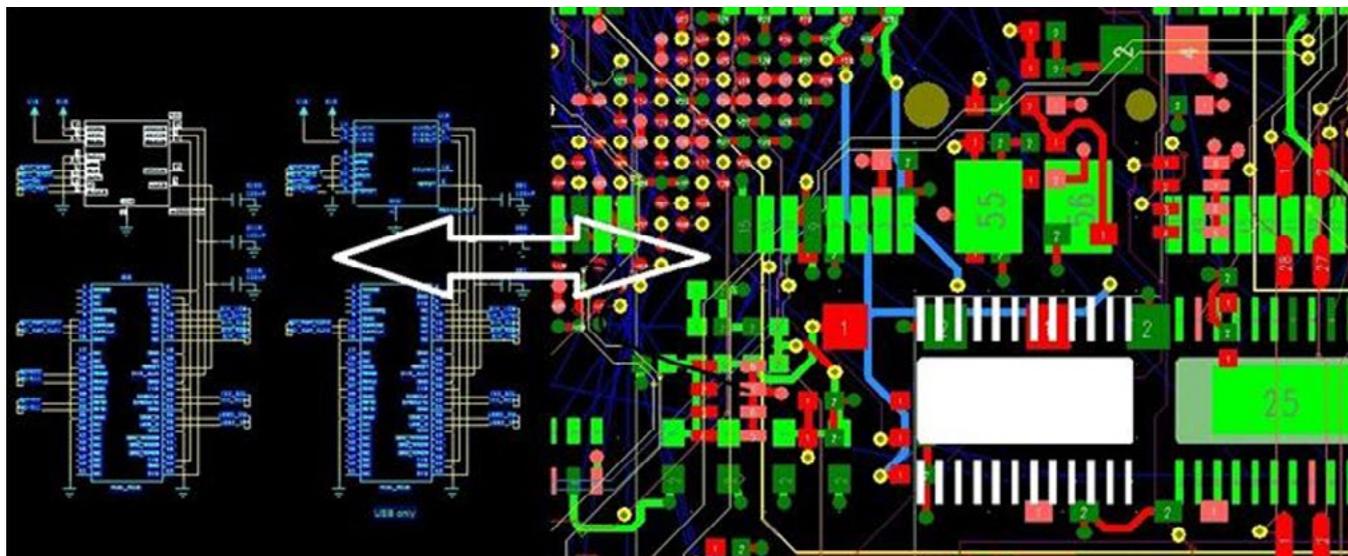


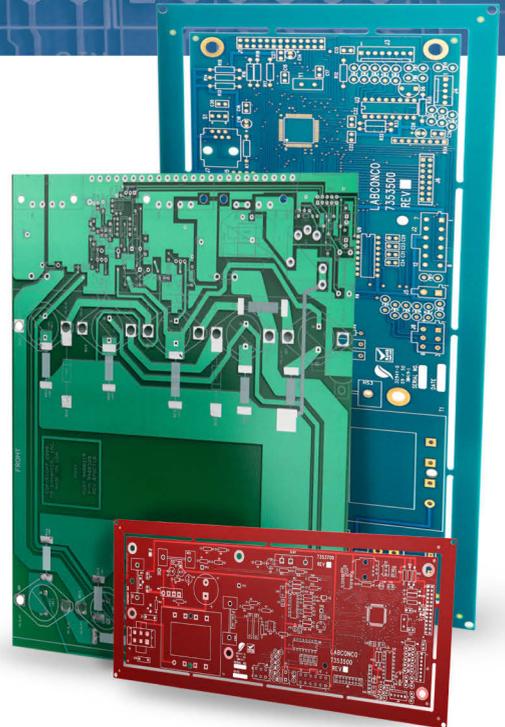
Figure 2: Cross-probing between schematic and PCB.

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



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

WHY AUTOROUTERS DON'T WORK: THE MINDSET!

primary contributor to the number of layers required for routing. An effective fanout solution should provide low inductance supply connectivity, minimal cross-over of signals, reduced crosstalk, breakout on multiple layers and layer reduction. The increasing pin count and decreasing pitch of BGAs proves a challenge to both PCB designers and routing technologies. Placement, orientation of interconnecting devices, swapping of I/O pins to reduce crossovers, together with the fanout to internal layers, are also key factors of routability.

The design constraints need to be established before an attempt to route is initiated. These include:

1. Stackup planning: This should be defined at the time of design entry to improve signal integrity, reduce crosstalk from adjacent layers and provide clear, uninterrupted return paths for all critical signals. There are many stackup options using a myriad of high-speed materials and these should be chosen based on the pre-layout simulation. The stackup should also be designed based on the technologies incorporated on the PCB to include all the single ended and differential impedances used. This determines trace width and clearance of each layer for each technology.

2. Via spans: These should be selected based on the stackups construction and the density of the BGAs to provide a fanout to internal layers. Blind and Buried vias need to be considered in order to fanout from 0.8 mm or less pitch BGAs.

3. Signal integrity: SI should be considered early in the design process to eliminate crosstalk, extended return current paths, and EMI.

4. Power integrity: PI should also be analyzed up-front to determine the number and values of bypass and decoupling capacitors required to reduce the AC impedance to an acceptable level, given the switching regulator properties. Plane resonance should be analyzed to determine the best possible plane definitions.

Design constraints can be established based on the above requirements. These are entered at the schematic level and carried through to the PCB database to control the router. The router needs rules to determine the most effective path but too many rules can also bog it down to such an extent that it will not perform. Care must be used when creating and prioritizing rules.

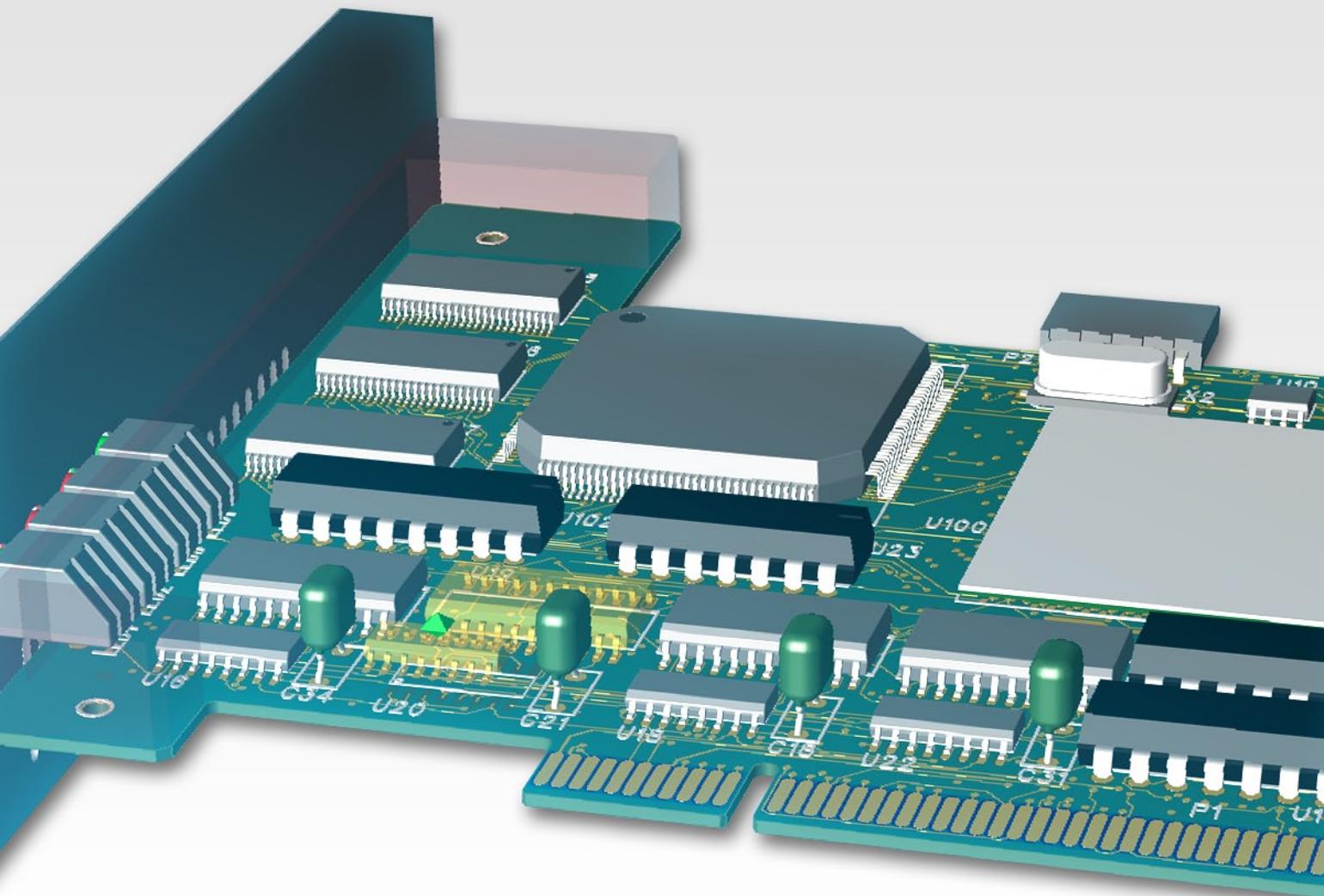
Once the schematic has been completed, the FPGA I/Os need to be evaluated for crossovers and pins swapped where necessary, to assist the router as much as possible. You could do this manually, but it is very time-consuming. Alternatively, Mentor's IO Designer FPGA-PCB co-design tool integrates synthesis and I/O optimization.

To obtain a high route-completion rate, component placement is extremely important. If the board is difficult to route, it may just be the result of poor placement, slots/gates positioned all over the board, or perhaps the sequence of pins on components are flipped. We need to help the router as much as possible by opening route channels and providing space for vias.

In the classic high-speed design flow, timing specifications and simulation results are compared to determine placement and routing constraints. Given a length constraint, a designer can control signal integrity by controlling the PCB trace topology of the various parts of an interface. Included in this topology are any terminations.

Interactive placement is best done by cross-probing, as in Figure 1, and dragging the components one by one from the schematic to place on the PCB, taking functionality and design constraints into account. Once the correct placement and orientation of the major devices is complete, the IO Designer can then be invoked to make some sense of the rats nest.

The trend now, is to put control of the autorouter back into the hands of the designers to enable clean, highly desirable results. The Sketch router can optimize the trace fanouts at both ends of the netlines, avoiding additional vias when completing the routes. It can also gloss the finished route to look much like a manual route. The idea is to give the designer control over the location of the routing, along with some style



3D DESIGN

More than just a pretty picture

Tackle simple or complex PCB design problems with accuracy and ease. PADS® delivers great value with constraint management, best-in-class simulation and analysis, and powerful interactive and automatic routing. New 3D design with STEP import/export provides a virtual prototype of your electronic assembly. 3D editing with DRC makes the layout process quick and efficient.

Find out more at
www.pads.com

pads
personal automated design solutions

WHY AUTOROUTERS DON'T WORK: THE MINDSET!

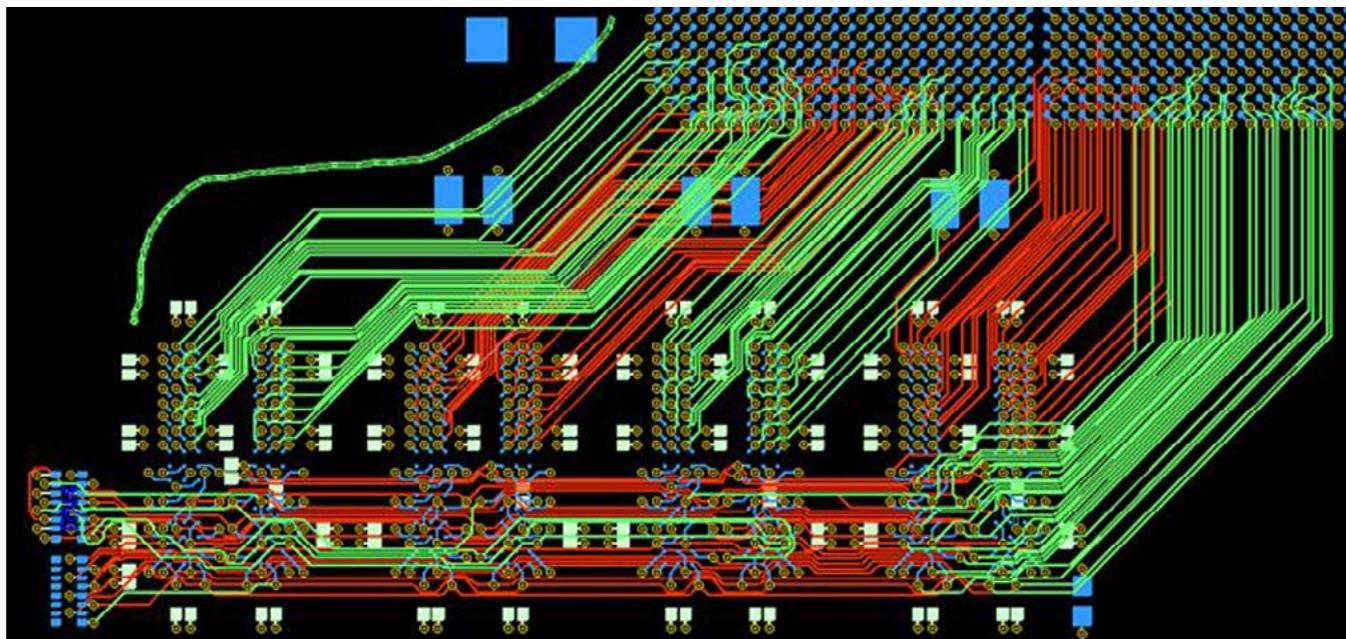


Figure 3: Sketch routing of DDR3 memory.

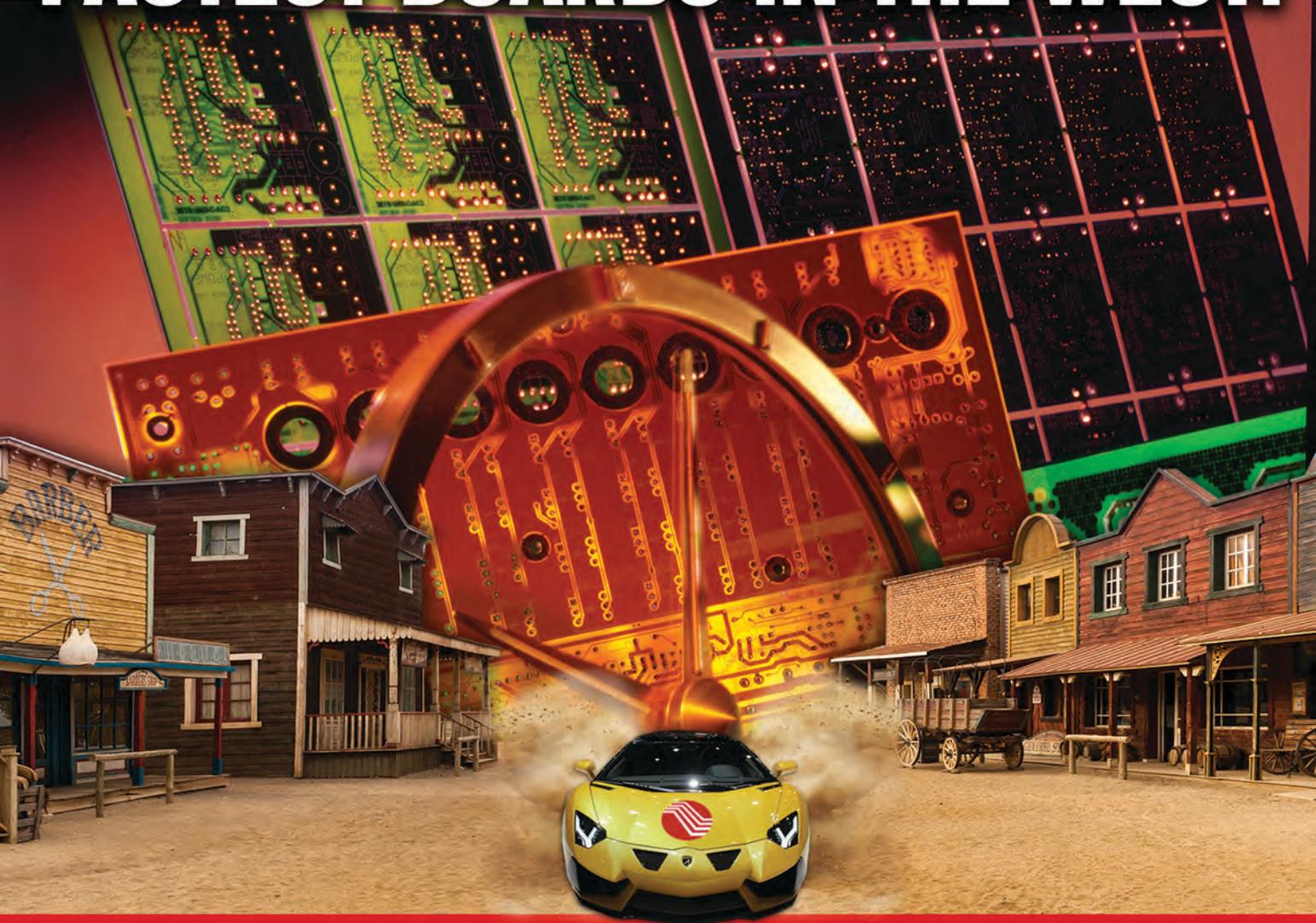
options, using a simple and fast methodology. Figure 2 shows sketch routing using multi-drop topology for address/clock and point-to-point for data/strobe on DDR3 memory. Once the basic routes are connected, the matched length traces are then tuned as in Figure 1.

So why don't autorouters work? I guess you will have to ask yourself that question! They certainly do for me. The trick is to control the router with constraints, cross-probe with the schematics, check as you go, tune and fix critical traces and allow the autorouter to do all the hard work. Believe me, once you know how to control your router, it will definitely save a great deal of time and frustration. PCB layout is a means to combine your artistic side and your creative skills with the power of automation, but you need to uncover the right mix to make it work.

Points to Remember

- Even the most primitive autorouter may have some useful features. It's all about changing that mindset of the designer.
- The first computer based PCB design tools that emerged in the late 1970s were grid-based, ran on DOS or UNIX operating systems and were very basic.
- PCB routers were developed using either the grid-based, gridless, shape-based, or geometrical approaches.
- A shape-based autorouter does not have to work at a particular resolution, so routing of high density or fine pitch boards is not significantly slower than for lower density work.
- The first autorouters were not very capable, limited by computing power and lack of memory. They added too many vias, wasted space due to the strict X/Y bias and the quality was poor compared to manual routing.
- The most popular shape-based router, 20 years ago, was CCT's Spectra router.
- Xpedition is arguably the best routing technology still available today.
- The design constraints need to be established before an attempt to route is initiated.
- The router needs rules to determine the most effective path, but too many rules can also bog it down.
- To obtain a high route-completion rate, component placement is extremely important. If the board is difficult to route, it may just be the result of poor placement.

FASTEST BOARDS IN THE WEST!



When Speed is a Priority
We're Quick on the ~~Draw~~ *TURN*

[MORE INFORMATION](#)

[REQUEST A QUOTE](#)

Prototron
 **Circuits**

High quality PCBs delivered fast, just another reason we are America's Board Shop

sales@prototron.com | prototron.com

WHY AUTOROUTERS DON'T WORK: THE MINDSET!

- The trend now is to put control of the autorouter back into the hands of the designers to enable clean, highly desirable results.
- The Sketch router can optimize the trace fanouts at both ends of the netlines, avoiding additional vias when completing the routes. **PCBDESIGN**

References

1. Barry Olney Beyond Design columns: [Critical Placement](#), [Interactive Placement and Routing Strategies](#), [Routing Techniques for Complex Designs](#)
2. Charles Pfeil: [BGA Breakouts & Routing](#)
3. Michael Pecht: [Placement and Routing of Electronic Modules](#)

4. The ICD Stackup and PDN Planner: www.icd.com.au

5. Sketch router information: www.pads.com/professional



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](#).

Back to the Future: Serviceable Spacecraft Make a Comeback

Ever wonder about the future of space science? Hop inside a time machine that transports you back 40 years and you may get a good idea about where things are headed. History, it would seem, has a funny way of repeating itself.

Serviceable spacecraft—like the NASA-developed Multi-Mission Modular Spacecraft (MMS) and, of course, the iconic Hubble Space Telescope that NASA conceived and developed in the 1970s with servicing in mind—are once again de rigueur.

Case in point: As required by Congress in a law passed in 2010 and then amended five years later, NASA is requiring that proposed flagship astrophysics missions support servicing, even if their orbits are up to a million miles away. The agency also released a Request for Information (RFI) seeking ideas for a spacecraft design that it could use for both its proposed Asteroid Redirect Mission (ARM) and as a vehicle for refueling a government satellite in low-Earth orbit.

“The 40-year cycle is starting all over again,” said Benjamin Reed, deputy project manager of the Satellite Servicing Capabilities Office (SSCO) at NASA’s Goddard Space Flight Center in Greenbelt, Maryland.

WFIRST-AFTA, which NASA plans to equip with an 8-foot (2.4-meter) mirror and a slitless spectrometer and imager, will study dark energy, the mysterious form of energy that permeates all of space and accelerates the expansion of the universe, while providing cosmic surveys. It also will carry a coronagraph that will allow the observatory to image giant exoplanets and debris disks in other solar systems.

Other conceptual missions that various groups currently are studying in preparation for the 2020 Astrophysics Decadal Survey also could operate in more distant orbits. One possible scientific objective would be to find Earth-size exoplanets in the habitable zone in our solar neighborhood and then identify chemicals in their atmospheres that may indicate the presence of life.

To achieve these ambitious goals, WFIRST and the other conceptual observatories ideally would operate from Sun-Earth L2 (SEL2), a thermally stable sun-Earth orbit roughly a million miles away.



the **pcb** list

THE best way to find a PCB fabricator, anywhere.

AMITRON CORP.

Elk Grove Village, Illinois, U.S.A.



As one of the largest and most experienced printed circuit board manufacturers around, we take great pride in delivering world-class products and customer service. We are a one-stop solution provider. Amitron has the flexibility and adaptability to take your design from Prototype and Pre-Production, through Mid-Volume to High Volume. We specialize in Thermal Management and materials for LED applications.

Markets: Automotive, Communication, Computers, Consumer, Industrial, Medical, Military/Aerospace

Board Types: Single-sided, Double-sided, Multilayer, Other: Thermal/Aluminum or Copper, Thermal-Flex, 24 hour Quick Turn Services Available

Mfg Volumes: Prototype, Small, Medium, Large, Other: Specialize in LED Thermal PCB's and Heavy Copper up to 20 oz

Certifications: Blind/buried vias, Carbon contact, Impedance, Filled/plugged vias, Sequential lamination, Silver through-hole thermal management

Standards: ISO 9001, ISO/TS 16949, ITI Registered, MIL-PRF-31032, MIL-PRF-55100 compliant, UL, Other: Minority Certification



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

[Liquid Cooling Moves onto the Chip for Denser Electronics](#)

Using microfluidic passages cut directly into the backsides of production field-programmable gate array (FPGA) devices, Georgia Institute of Technology researchers are putting liquid cooling right where it's needed the most—a few hundred microns away from where the transistors are operating.

[Robots and Us](#)

If you follow technology news—or even if you don't—you have probably heard that numerous companies have been trying to develop driverless cars for a decade or more. These fully automated vehicles could potentially be safer than regular cars, and might add various efficiencies to our roads, like smoother-flowing traffic.

[Lockheed Martin Delivers First Upgraded PAC-3 Missile Interceptors](#)

The U.S. Army significantly upgraded its missile defense capabilities today as it accepted the first PAC-3 Missile Segment Enhancement (MSE) interceptors built by Lockheed Martin. With improved mobility and range, the new interceptors will defend against evolving threats around the globe.

[The Real Martian Spinoffs Part 3: Harnessing the Power](#)

More powerful than the mighty Saturn V that took humans to the moon, the Space Launch System (SLS), NASA's newest rocket currently under development, will have the capability to send astronauts deeper into space than ever before. With SLS and the Orion capsule, humans will no longer have to dream of walking on Mars—they finally will do it.

[Electronic Interconnect Improves Customer Service with Addition of TS16949 and AS9100 Certifications](#)

In an effort to bring a more stringent system into place, PCB fabricator Electronic Interconnect has received both their TS16949 and AS9100 certifications.

[Lockheed Martin to Open Submarine Combat System Laboratory](#)

Lockheed Martin Australia will open a submarine combat system laboratory in Mawson Lakes in November to support the company's pursuit of the Royal Australian Navy's Future Submarine project SEA 1000 Phase 1. Construction began at Mawson Lakes on July 27, 2015 and is expected to open in November. An expanded Phase 2 with a secure area is scheduled to open in the third quarter of 2016.

[FTG's Circuits' Segment Posts 31% Sales Growth in Q3](#)

The circuits' segment sales were up \$3.3M or 31% in Q3 2015 versus Q3 2014. All facilities reported increased revenues. On a year-to-date basis, circuits' sales were up \$7.4M or 23%.

[Epec Hires Todd Barham as Military & Aerospace Business Development Manager](#)

Epec Engineered Technologies recently announced the hiring of Todd Barham as Military and Aerospace Business Development Manager. Todd's main role with Epec will be to help grow the company's market share in the military and aerospace sectors.

[The Application of Advanced Ultrasonics in Metal Plating Processes](#)

Most surface modification chemical treatments by their very nature contain hazardous and oxidising chemicals, and there is a major concern that these chemicals are used maliciously with the intent to manufacture explosives or formulate poisons to harm the public. Cyanide-based plating solutions and high concentrations of hydrogen peroxide used in current techniques are a significant use of these materials in industry.

[UAV Market to Reach \\$10B Mark by 2024](#)

The global defense and security market for unmanned aerial vehicles will expand at 5.5% per year over this decade, from the current figure of \$6.4 billion, to \$10.4 billion by 2024, according to IHS.



Multilayer Technology

“ 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

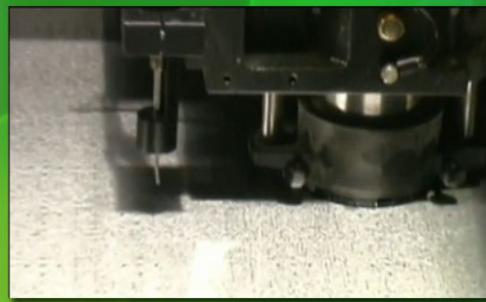
WWW.MULTILAYER.COM



Mil-PRF-55110

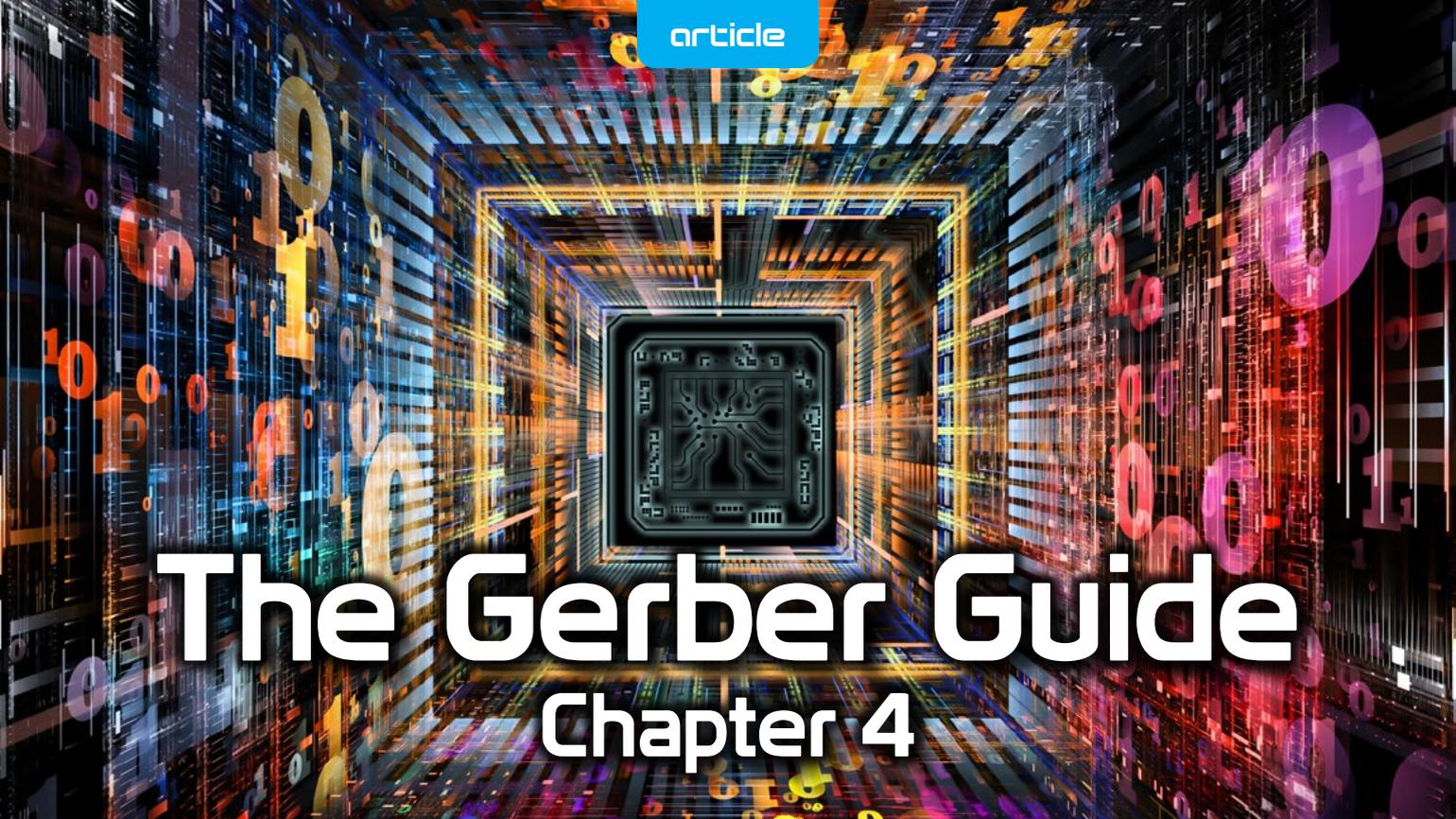


AS 9100



Multilayer Technology
3835 Conflans Rd
Irving, TX 75061-3914

(972) 790-0062



The Gerber Guide

Chapter 4

by Karel Tavernier
UCAMCO

It is possible to fabricate PCBs from the fabrication data sets currently being used; it's being done innumerable times every day all over the globe. But is it being done in an efficient, reliable, automated and standardized manner? At this moment in time, the honest answer is no, because there is plenty of room for improvement in the way in which PCB fabrication data is currently transferred from design to fabrication.

This is not about the Gerber format, which is used for more than 90% of the world's PCB production. There are very rarely problems with Gerber files themselves; they allow images to be transferred without a hitch. In fact, the Gerber format is part of the solution, given that it is the most reliable option in this field. The problems actually lie in which images are transferred, how the format is used and, more often, in how it is not used.

Each month we look at a different aspect of the design to fabrication data transfer process. In this monthly column, Karel Tavernier explains in detail how to use the newly revised Gerber data format to communicate with your fabrication partners clearly and simply, using an unequivocal yet versatile language that enables you and them to get the very best out of your design data.

Chapter 4: Output drill files in Gerber rather than in an NC format

Incoming drill files in particular are never used directly on the fabricator's drilling machines. There are plenty of reasons for this, and we've already seen some of these in earlier chapters. But the specific issue with drill files is that incoming CAD files specify the end-diameter of the hole on the finished bare board, while in reality a bigger hole must be drilled to make room for plating. Furthermore, the PCB designer is unlikely to know the origin and orientation of the fabricator's drilling machine, its working parameters and all the rest, which will also affect the way in which the production panels are drilled and routed.

Therefore, the CAD designer need not fret about whether the drill machine works in metric or imperial, or can handle the resolution, feeds and speeds: the fabricator's CAM system takes care of all that. The designer needs only to concern himself with how to transfer the design data to CAM, optimally, accurately, completely and safely with a minimum of manual work for all parties.

It may come as a surprise, but the best file format for CAD to CAM drill information transfer is a Gerber file, not an NC or Excellon file. The most important reason for this is attributes,



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**

THE GERBER GUIDE, CHAPTER 4

which are essential for automated CAD to CAM data transfer. Attributes specify the span and plating of a file, the function of a drill hole—that it is, say, a via hole—and the drill tolerances. Gerber is the only format that supports attributes. NC formats do not.

In addition, CAM departments often face the following problems with NC files:

- **Drill files and copper layers are not registered as they use a different datum point**

This occurs in nearly 50% of all job data sets. Where this happens, the CAM engineer has to work the data, shifting and rotating each drill file to register it with the copper layers. This is a medieval way of working. The only secure solution is for the CAD professional to generate the drill and copper layers in perfect register, preferably using the coordinates of the CAD system on all layers, to facilitate communication between designer and fabricator. The underlying reason for misregistration is that copper layers are output in Gerber and the drill data in an NC format. The NC output in CAD can possibly be set up to register with the Gerber data, but the simplest way to guarantee registration is to output both in Gerber.

- **Drill holes are not centered on the copper pads**

When this occurs, part of the drill tolerance is gone before drilling even starts, increasing the risk of breakout. The root cause is different

rounding and resolution in the Gerber and NC drill files. To overcome this issue, the CAM engineer may “snap” the copper pads to the drill holes. This manual operation takes time and carries risk, forcing the CAM engineer to manipulate the design, which is something the designer does not want. The simplest solution is to output the drill files in Gerber at the same resolution as the copper layers. Then everything matches perfectly.

- **Incomplete NC file**

All too often—in a whopping 75% of PCB data sets, in fact—the NC files are incomplete and require manual input of scale, unit and tool diameters. Such incomplete files relate to a complete NC file as the (obsolete) Standard Gerber relates to a proper Extended Gerber file. If you use Extended Gerber for image layers, it is not terribly consistent to stick to incomplete drill files. (See Chapter 21.) You must use complete drill files; the question is how to do this. Although it may be possible to set up the NC output of your CAM system to generate complete NC files, but the simplest solution is to output in Extended Gerber, which is guaranteed to be complete.

- **Limited resolution of Excellon file**

The Excellon format suffers from severe limitations on resolutions, so while it is quite adequate for drilling, it is not recommended for design transfer. There is no such limitation in Gerber.

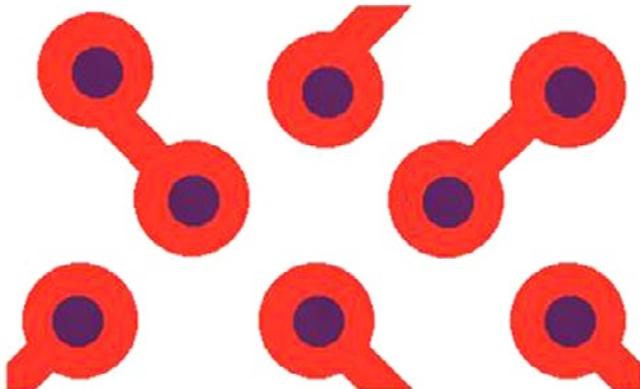


Figure 1: Drill and copper both in the same format, centered.

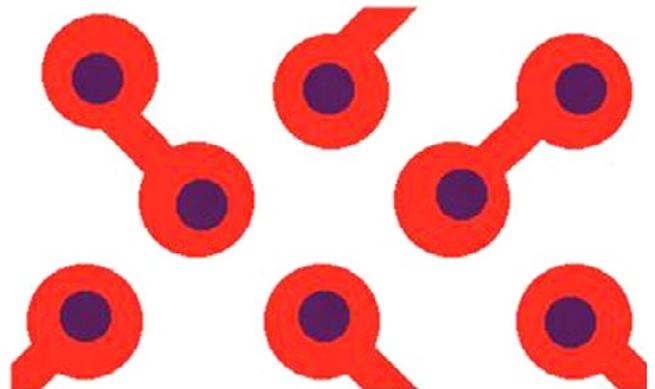


Figure 2: Drill and copper in different formats, not centered.

MERELY SURVIVE? OR DOMINATE.



IPC now delivers

online exams

through our global network of
licensed training centers.

**Get improved accuracy,
increased security and
immediate results.**

Evolve to New Standards Revisions with **IPC Certification Programs**

Success in the electronics industry happens when companies—and individuals—take every opportunity to push themselves to climb to a higher level of performance. Well, this is one of those opportunities. **IPC has updated our most popular standards to the latest industry requirements, including IPC J-STD-001F, *Requirements for Soldered Electrical and Electronic Assemblies* and IPC-A-610F, *Acceptability of Electronic Assemblies*.** The F revisions will help your company dominate through your commitment to outstanding quality assurance and rigorous performance standards. And, among the electronics supply chain, you will raise your profile while you improve communications with your partners.

Now is the time to evolve. Go to ipc.org/evolve to get your IPC specialists or non-certified operators certified on IPC's New Standards Revisions and find your closest IPC Licensed Training Center.



THE GERBER GUIDE, CHAPTER 4

- **Poor implementation of NC output in some CAD systems**

Some NC files are really terrible. They can, for example, have binary junk in the middle of the file, or non-standard commands. As images are considered to be more critical than drill files, the quality of Gerber files is typically much better.

- **Needlessly using two formats where one will do**

Copper layers must be in Gerber as NC files cannot describe a copper image, so Gerber is a given. As Gerber can also describe the simpler drill data, it makes sense to keep things simple by using this format for both copper and drill/rout data. By using an NC format for drill, a second format is needlessly introduced, and the risk of problems is doubled.

“There is not a single disadvantage in using Gerber rather than an NC format to express CAD-to-CAM drill data. On the contrary, it brings many advantages in terms of completeness, accuracy, consistency and simplicity.”

There is not a single disadvantage in using Gerber rather than an NC format to express CAD-to-CAM drill data. On the contrary, it brings many advantages in terms of completeness, accuracy, consistency and simplicity.

One may object that we cannot send Gerber files to a drilling machine. True enough, but as outlined above, this is a non-issue. The CAM system will generate the drill files for fabrication in Excellon, Hitachi, Sieb & Meyer, or whatever format best suits the drilling machine, and will take care of the offset, rotation, feeds, speeds,

tool diameter, resolution, optimisation, size compensation and whatever else is needed for optimal performance on the fabricator's equipment. This means that Gerber is hands-down the best format for input in CAM.

One may object that Gerber is an image description format—true again—and that drill data is not image data. This is actually not true. A Gerber copper file describes where there is copper and where there is no copper. Just as a legend file describes where there is legend ink and where there is no legend ink. These files use images to show the presence and absence of material. So do drill and rout files. In this case they describe where material must be removed: a hole or slot is the absence of material. So a drill file, just like a copper layer, is an image file. Yes, drill holes and copper patterns are produced by different processes, but at design stage, they are images.

Copper layers and legends are also produced by different processes for that matter, but no one claims they must be described by different image formats. The only difference between drill and copper layers is that the drill file applies to a range of layers, the span, and not to a single layer, and of course, the span must be clearly specified, in both Gerber and NC formats. Here too, Gerber is better: NC formats have no inherent mechanism for describing spans, while Gerber's attributes allow span to be specified in a standard, machine-readable manner.

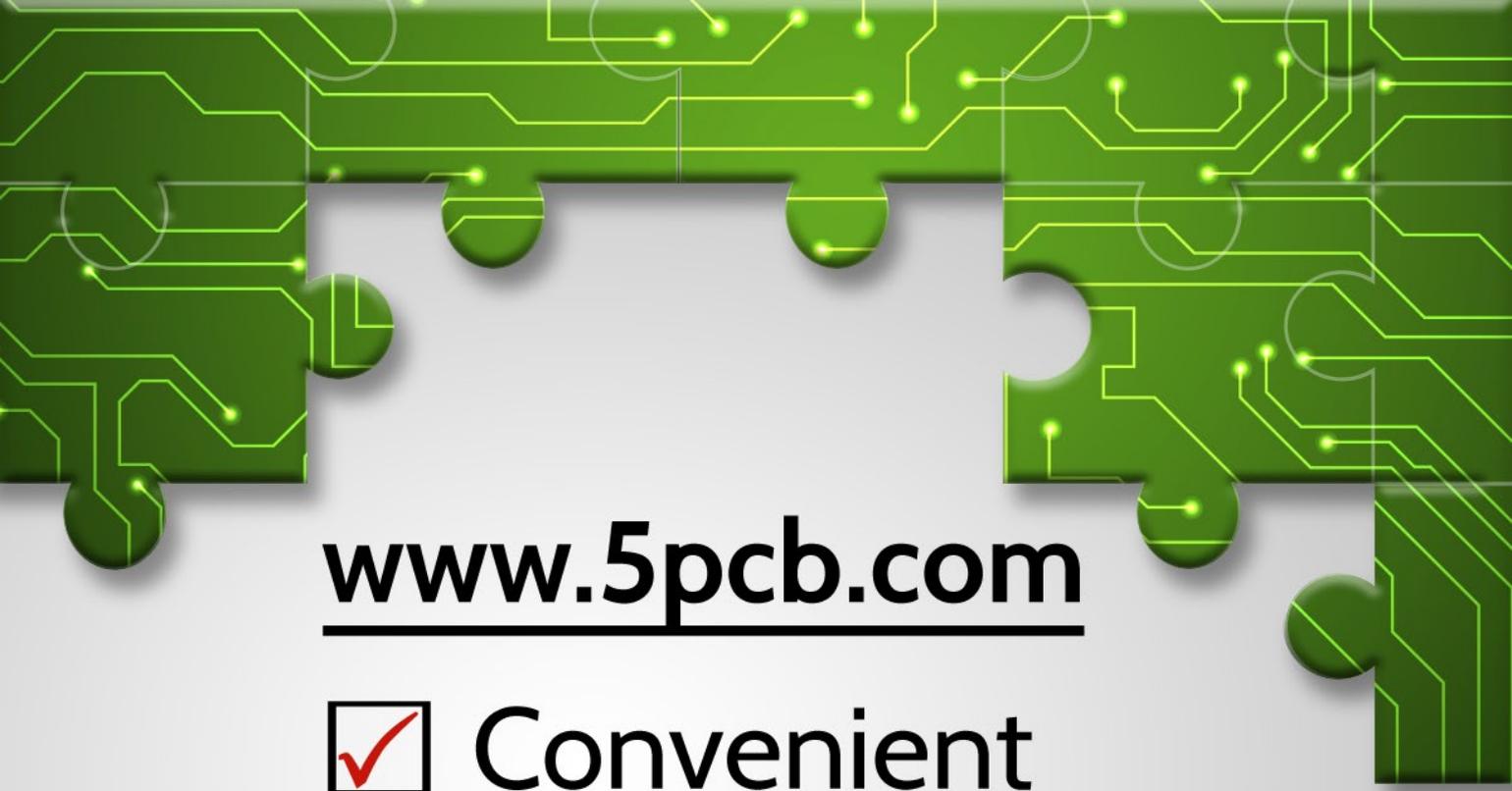
Conclusion: Gerber is far superior to the NC formats for transferring drill and rout data from design to fabrication. Output your drill and rout files in Gerber.

Next month we'll move on to Chapter 5. See you then. **PCBDESIGN**

This column has been excerpted from the [Guide to PCB Fabrication Data: Design to Fabrication Data Transfer](#).



Karel Tavernier is managing director of Ucamco.



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](#)



D.M. Electronic International Co., Ltd.

TOP TEN



Recent Highlights from PCBDesign007

1 Insulectro Hosts Silicon Valley Designers Council Meeting

On October 15, about 25 people gathered for the IPC Designers Council Silicon Valley Chapter meeting at Insulectro's facility in Mountain View, California. The 1.5-hour meeting opened with a warm welcome from Chapter President Bob McCreight, CID, a PCBA designer at Tesla Motors, and Jason Marsh, VP of product management at Insulectro.

2 Mentor Graphics Enhances PADS 3D Tool Capabilities

Mentor Graphics Corporation today announced new PADS 3D tool capabilities plus major usability enhancements to the PADS flow for printed circuit board (PCB) design. The new technology provides visualization, placement, and design rule checking in 3D, minimizing iterations with mechanical and manufacturing teams.

3 The Shaughnessy Report: Squeezing Seconds Out of the Design Cycle

It's almost that bad, isn't it? When you're designing a board, time is always your enemy. Your deadline is around the corner, and you can't be late. (You're going to catch the blame anyway, even if it's not your fault.) So you constantly look for ways to shorten your design cycle, even if it means squeezing out a few seconds here and there.

4 New Version of Valor Process Preparation Connects Lean NPI Flow with Industry 4.0

This release of the Valor Process Preparation product includes enhanced support for rapid test programming, including the ability to intelligently link schematic, PCB layout and bill of materials (BOM) data. Simply selecting a track on the PCB layout immediately shows the node in the corresponding schematic diagram from design, and vice-versa.

5 Accelerating the PCB Design Cycle

In addition to all of the typical design challenges, designers have to be efficient three-dimensional puzzles solvers because time-to-market is still a vitally important objective. After all, time is money. So how do today's designers balance the technical and timing demands? There isn't one answer.

6 Cadence Reports Q3 2015 Financial Results

Cadence reported third quarter 2015 revenue of \$434 million, compared to revenue of \$400 million reported for the same period in 2014. On a GAAP basis, Cadence recognized net income of \$78 million, or \$0.25 per share on a diluted basis, in the third quarter of 2015, compared to net income of \$38 million, or \$0.12 per share on a diluted basis, for the same period in 2014.

7 Lightning Speed Laminates: Impact of Final Plated Finish on PCB Loss

The reason that most plated finishes cause increased insertion loss compared to bare copper is that most plated finishes are less conductive than copper. Electroless nickel/immersion gold (ENIG) is a very good finish. However, the simple fact is that nickel is about one-third the conductivity of copper, and a circuit with ENIG will have more insertion loss than the same circuit with bare copper.

8 Rick Hartley Named Top-Rated Speaker at PCB West

Hartley drew wide acclaim from the attendees in his class, scoring a 9.75 rating out of a possible 10 from the 52 evaluations returned. One hundred percent of attendees also rated the class either extremely relevant or relevant.

9 The Readers Speak: Tips on Accelerating your Design Cycle

This month, we decided to collect feedback from the readers—PCB designers and engineers working in the trenches each day. We asked our readers to provide their favorite tips, tricks, and techniques for speeding up the PCB design cycle. Here are 10 tips for cutting your design time, courtesy of designers just like you.

10 The Material Witness: Nonwoven Aramid Reinforcement is Back

What was Thermount exactly and why was there such a furor when DuPont announced its premature demise? And where do we stand now with the redevelopment of a nonwoven aramid product? The original 85NT product was based on high tensile-strength para-aramid fiber with a meta-aramid fibrid binder.

PCBDesign007.com for the latest circuit design news and information—anywhere, anytime.



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](#).

productronica 2015

November 10–13, 2015
Munich, Germany

2015 EFRA-CEFC Workshop

November 11, 2015
Munich, Germany

International Wafer—Level Packaging Conference

November 11–13, 2014
San Jose, California, USA

Space Coast Expo & Tech Forum

November 12, 2015
Melbourne, Florida, USA

TSensors Summit 2014—San Diego

November 12–13, 2014
La Jolla, California, USA

Indiana Fall Forum & Expo

November 13, 2015
Columbus, Indiana, USA

LED Assembly, Reliability & Testing Symposium

November 17–19, 2015
Atlanta, Georgia, USA

Rapid Oven Setup & PCB Profiling—Seminar

November 24, 2015
Warwickshire, UK

2015 International Printed Circuit & APEX South China Fair

December 2-4, 2015
Shenzhen, China

DesignCon 2016

January 19-21, 2016
Santa Clara, California, USA

IPC APEX EXPO Conference & Exhibition 2016

March 15-17, 2016
Las Vegas, Nevada, USA



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

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

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

EDITORIAL:

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,
TOBEY MARSICOVETERE**

INNOVATIVE TECHNOLOGY: **BRYSON MATTIES**

COVER IMAGE: **BARRY MATTIES**



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.

November 2015, Volume 4, Number 11 • The PCB Design Magazine© is published monthly, by BR Publishing, Inc.

ADVERTISER INDEX

Accurate Circuit Engineering....	37	Mentor Graphics.....	59
American Standard Circuits.....	29	Miraco.....	67
ANS.....	51	Multilayer Technology.....	65
Candor Industries.....	33	NCAB.....	11
Dibble Leaders.....	43	Oak Mitsui.....	49
DME.....	71	Prototron.....	61
Downstream Technologies.....	55	Pulsonix.....	9
Dymax.....	27	Rogers Corporation.....	21
Eagle Electronics.....	47	Schweizer Electronic AG.....	7
EMA/EDA Design Automation...	45	Shengyi Technology.....	13
Fast Interconnect.....	25	Sunstone Circuits.....	57
Freedom CAD.....	17	The PCB List.....	2, 63
I-Connect007.....	76	US Circuit.....	41
In-Circuit Design Pty Ltd.....	3	Uyemura.....	31
IPC.....	35, 69	Ventec.....	39, 53
Isola.....	5		

**Coming Soon to
The PCB Design
Magazine:**

**December:
Associations
and 2016 Events**

**January:
Medical
Electronics
Design**

I-Connect007

GOOD FOR THE INDUSTRY



FREE SUBSCRIPTION



subscribe.icconnect007.com

EDITORIAL CONTACT

Andy Shaughnessy
andy@icconnect007.com
+1 404.806.0508 GMT-5



mediakit.icconnect007.com

SALES CONTACT

Barb Hockaday
barb@icconnect007.com
+1 916 365-1727 GMT-7



www.icconnect007.com