

THE **pcb**
DESIGN
MAGAZINE

January 2013

AN I-CONNECT 007 PUBLICATION

Kick-Starting a Revolution:
IPC-2581 Meets Gerber p.10

Optimizing PCB New
Product Introduction
Using ODB++ p.28

Gathering Steam (Finally):
IPC-2581 p.34

Data, Data Everywhere,
but... p.56



**DATA
TRANSFER
FORMATS**

Go Round and Round

isola
Booth 819
DesignCon 2013

the **pcb**list

The best way to find a pcb fabricator, anywhere.

Are you on the list?

thePCBlist™?

Yes! The BEST showcase for
a fabricator, anywhere!

Features include:

- Free access to the most comprehensive directory of PCB fabricators worldwide
- Intuitive navigation
- In-depth profiles of PCB fabricators
- Multiple search capabilities including board type, material, region and more!
- The only source OEMs and fabricators need to connect!
- Accessible anywhere, anytime!

www.thepcblist.com

Take our **DESIGNCON 2013**

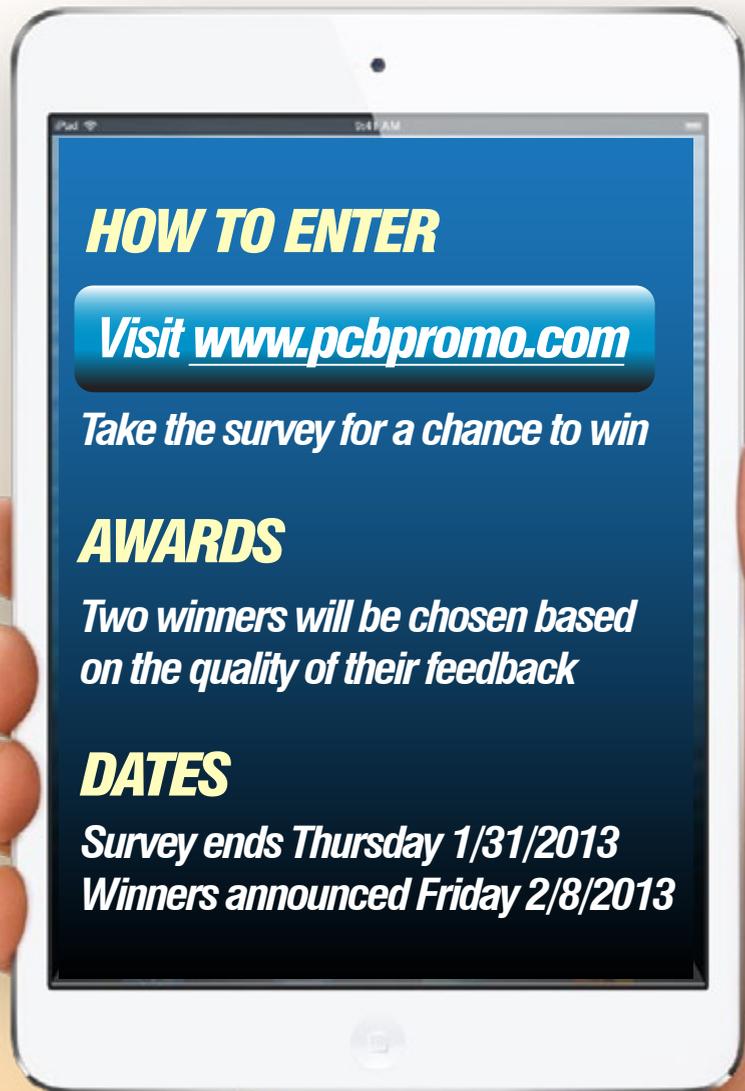
survey for a chance to win an iPad Mini

Visit us at
DesignCon 2013
January 29 and 30
Booth 641

- Get a discount for PCB fabrication and assembly
- Enter a raffle to win a Google Nexus 7
- **FREE** gift to all visitors

Sierra Circuits is an ISO 9001:2008, ISO 13485:2003 and MIL-P-55110 certified manufacturer of printed circuit boards.

We specialize in the quick-turn manufacture and assembly of PCBs, especially HDI designs, as well as medium-scale production.



Sierra Circuits, Inc. 1108, West Evelyn Avenue, Sunnyvale, CA 94086
Phone: 408-735-7137 | Toll free: 800-763-7503 | www.protoexpress.com

THIS ISSUE: DATA TRANSFER FORMATS

FEATURED CONTENT

Yes, most PCB designers still use Gerber, but rival data formats are making big strides. This month, we untangle the maze of design data transfer, with supporters of Gerber, ODB++, and IPC-2581 making the case for their preferred format. Is one truly better than the rest, or should we – as one author suggests – combine the best features of several formats into one super standard?

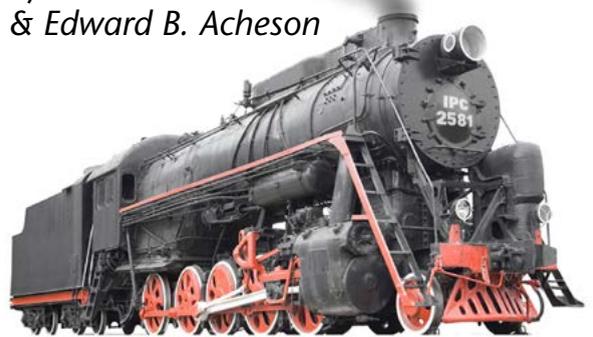
10 Kick-Starting a Revolution: IPC-2581 Meets Gerber

by Karel Tavernier



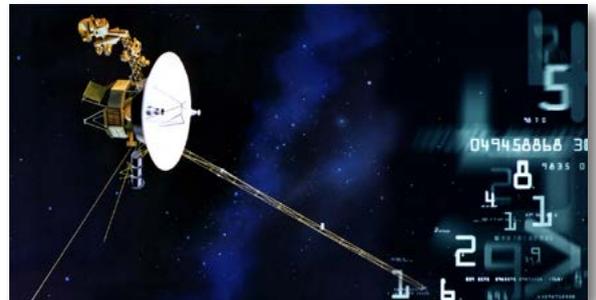
34 Gathering Steam (Finally): IPC-2581

*by Hemant Shah
& Edward B. Acheson*



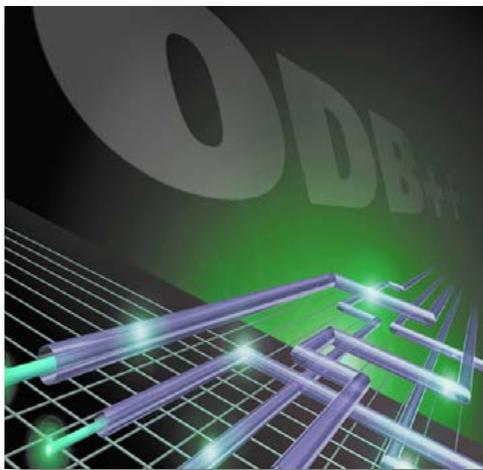
56 Data, Data Everywhere, but...

by Iain Wilson



28 Optimizing PCB New Product Introduction Using ODB++

by Julian Coates



FEATURE COLUMN

46 Dispense With the Gerbers Already

by Amit Bahl

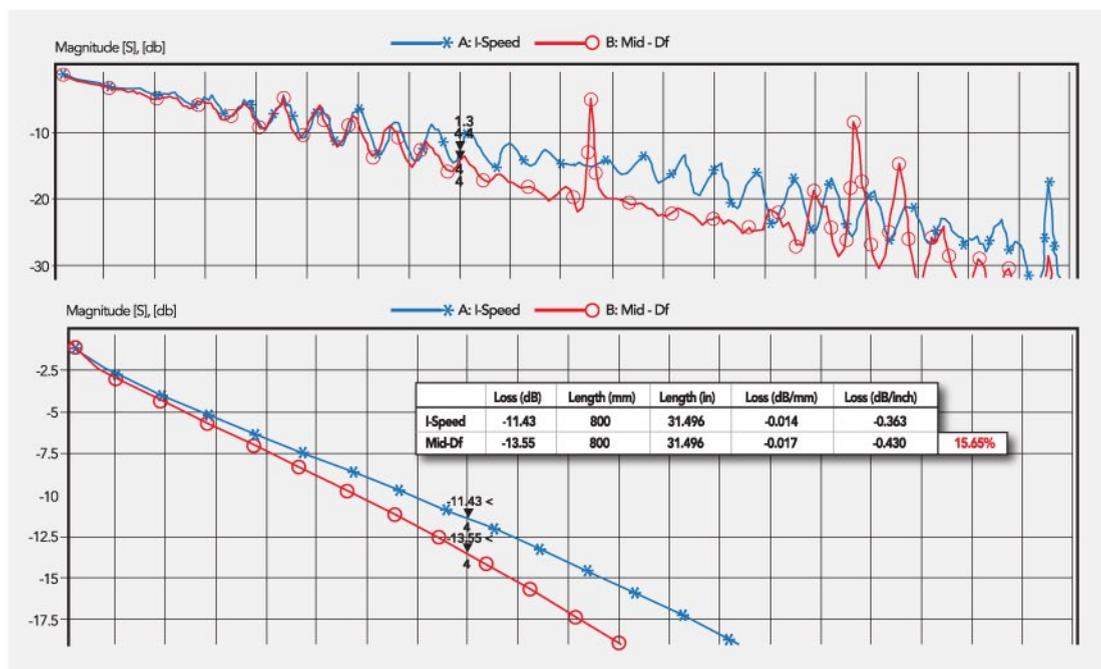


I-Speed[®]

The Next Generation High Speed Product from Isola

- Global constructions available in all regions
- Optimized constructions to improve lead free performance
- I-Speed delivers 15-20% lower insertion loss over competitive products through reduced copper roughness and dielectric loss
- Improved Z-axis CTE 2.70%
- I-Speed – IPC 4101 Rev. C /21 /24 /121 /124 /129
- Offer spread and square weave glass styles (1035, 1067, 1078, 1086, 3313) for laminates and prepregs
 - Minimizes micro-Dk effects of glass fabrics
 - Enables the glass to absorb resin better and enhances CAF capabilities
 - Improves yields at laser and mechanical drilling
- A low Df product with a low cost of ownership

Effective Loss @ 4 GHz on a 32 inch line



I-Speed delivers 15-20% lower insertion loss over competitive low Df products.



<http://www.isola-group.com/products/i-speed>

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

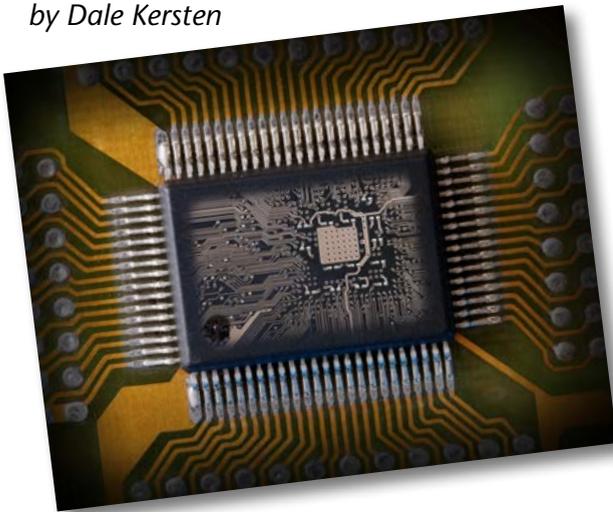
Isola Corp., 3100 West Ray Rd., Suite 301, Chandler, AZ 85226
 +1-480-893-6527
www.isola-group.com

isola
 The base for innovation

CONTENTS

ARTICLES

- 60 PCB Trends in 2013: Smaller and Denser**
by Dale Kersten



SHORTS

- 21 IPC Updates PCB Design Standard**
- 54 Sunstone Reveals "Share Your Story" Winners**

TOP TEN MOST-READ NEWS

- 45 PCB007**
- 55 Mil/Aero007**
- 62 PCBDesign007**



EXTRAS

- 64 Events Calendar**
- 65 Advertiser Index & Masthead**

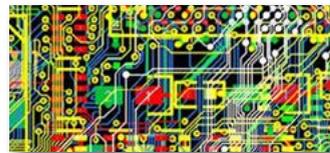


COLUMNS

- 8 The Great Data Format Horse Race**
by Andy Shaughnessy



- 22 Routing Techniques for Complex Designs**
by Barry Olney



- 40 What is a Circuit Board?**
by Jack Olson



- 50 Trace Currents and Temperature, Part 3: Fusing Currents**
by Douglas Brooks, Ph.D.



VIDEO INTERVIEWS

- 9 ODB++: An Open Alliance**
- 21 Rogers Talks New Thermal Product Developments**
- 33 Gary Ferrari Updates IPC-2221 Design Standard**
- 59 A Designer with Flex on His Mind**





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

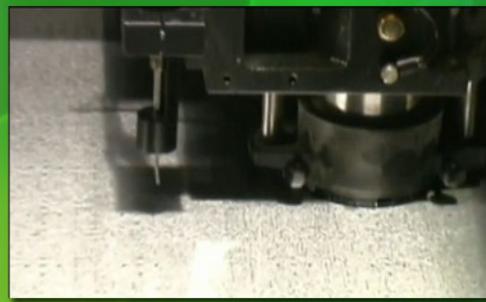
WWW.MULTILAYER.COM



Mil-PRF-55110



AS 9100



Multilayer Technology
3835 Conflans Rd
Irving, TX 75061-3914

(972) 790-0062

The Great Data Format Horse Race

by Andy Shaughnessy

I-CONNECT007

SUMMARY: *There's been an ongoing discussion about design data formats. Now that a PCB has been fabricated with IPC-2581 data, it looks as if Gerber, ODB++ and IPC-2581 are all very capable formats. Which one will win the great data transfer format horse race?*

The ongoing discussion about design transfer formats just gets more interesting every month. With three formats – Gerber, ODB++ and IPC-2581 – all competing for the PCB designer's attention, it reminds me of a horse race, as shown on this month's cover.

Which format is going to be the first to reach the finish line?

Actually, that's probably not the most accurate metaphor – one could argue that Gerber has been sipping mint juleps at the finish line for decades and wondering what all the fuss is about.

The overwhelming majority of PCB designers still use Gerber, despite the fact that it was never designed to describe PCB design data. Gerber may be old and clunky, but designers are accustomed to it and it gets the job done, like a 1991 Nissan pickup truck that just won't quit.

Just out of curiosity, does

anyone have any idea how many Gerbers have been output over time? I'd love to know. How much of the world's armed forces, transportation systems, and communications and power infrastructures were built on PCBs designed with Gerber?

At PCB trade shows, I like to ask designers what data format they use. Nine times out of 10, it's Gerber. Most of the time, they wave off all the talk about formats. For many designers, the prevailing attitude about formats is, "If it ain't broke, don't fix it."

Still, for years manufacturers had been musing about how nice it would be to have a design data format that included all of the necessary information in one file, instead of the multiple files that Gerber entails – all of which have to be run separately through a shop's CAM and DFM tools.

ODB++ fit the bill: Developed by Valor Computerized Systems, it's a single file format with data stored in a hierarchy of file folders, with data that can realistically be termed "intelligent." ODB++ is more streamlined than Gerber, and it seems to do much of the job of data transfer better than Gerber. ODB++ has one huge advantage; with many PCB manufacturers using Valor's CAM tools, this database is installed in shops around the globe.



And Still, Gerber persevered.

Because ODB++ is a proprietary format owned by Mentor Graphics since its 2010 acquisition of Valor, some companies are reluctant to switch from Gerber to ODB++. But the ODB++ Solutions Alliance is open to anyone, and its members include Mentor rivals Cadence and Zuken. There may always be some heartburn over ODB++ being a proprietary format, regardless of what Mentor does to allay any concerns.

Then along came IPC-2581. Released in 2004, IPC-2581 is a result of the “data exchange convergence” (remember that?) effort led by iNEMI a decade ago. This format was designed to combine the best of ODB++X and IPC’s Gen-CAM Version 1.0, and like ODB++, it features “intelligent” data.

We didn’t hear much about IPC-2581 until last year. In September, the first PCB was fabricated with IPC-2581 data. Mentor announced that it supports IPC-2581.

Now, Karel Tavernier, managing director of Ucamco, the company that now owns Gerber,

has an intriguing idea. In this issue, Tavernier suggests combining the best features of IPC-2581 with the best functionality of Gerber, to the point that Gerber could eventually be retired. Whether that idea comes to fruition or not, we need more out-of-the-box thinking like that.

I think it’s safe to say that all three of these formats can do the job, and all three have legions of supporters who are waiting for the other two to just go away.

A new revision of IPC-2581 will be released in early 2013, so stay tuned. Will the ODB++ Solutions Alliance follow suit? Either way, we’ll keep you up to date with news about the great data transfer format horse race. **PCBDESIGN**



Andy Shaughnessy is managing editor of *The PCB Design Magazine*. He has been covering PCB design for 13 years. He can be reached by clicking [here](#).

video interview**ODB++: An Open Alliance**

by *Real Time with...*
Designers Forum



ODB++ has long been considered a de facto standard as a CAD database exchange format. But is it an open format? Julian Coates of Mentor Graphics explains why the term “alliance” is used to describe how ODB++ serves as a data hub for any software tool that must work with other tools to bring a design from layout to manufacture.



realtimewith.com





Kick-Starting a Revolution: IPC-2581 Meets Gerber

by Karel Tavernier
UCAMCO

SUMMARY: *The all-or-nothing approach to improving the CAD-to-CAM workflow benefits nobody. The author proposes keeping what works in Gerber – the image data format – and changing what doesn't work, such as the stackup data format. And stackup is an area in which IPC-2581 excels. Other IPC-2581 sections could be integrated with Gerber in the same way, and "good old Gerber" could eventually be retired.*

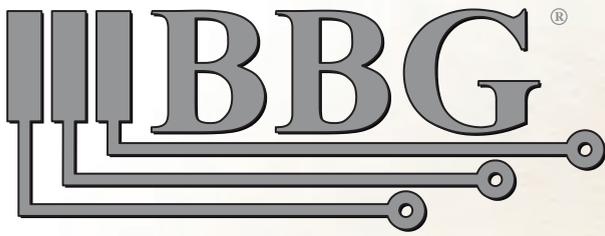
One of the common misconceptions in the world of PCB design centres around what happens to CAD data when it gets to the PCB manufacturer. It is often believed that the Gerber

and Excellon files generated by PCB designers go straight onto the fabricator's NC equipment. This gives rise to all sorts of concerns about how the data must be delivered – whether, for example, a PCB fabricator's drill machine will accept instructions in metric or imperial units, or whether the manufacturing process can handle the resolution, feeds and speeds.

The good news for designers is that their Gerber and Excellon files never, ever go straight into the PCB manufacturing process. One of the several reasons for this is that PCBs are never manufactured as single PCBs as such, but on panels, where they are surrounded by borders and other features necessary for the production process. Incoming files are always read into the PCB fabricator's CAM system, which generates appropriate production data in whatever language and setting necessary for the facility's equipment.

It should be clear from this that designers do not need to concern themselves with how the data will work on the PCB fabricator's equipment. What they really must do, however, and here we come to the purpose of this article, is to make sure that the design data is valid, accurate and complete and can be read into the fabricator's CAM system as easily and reliably as possible.

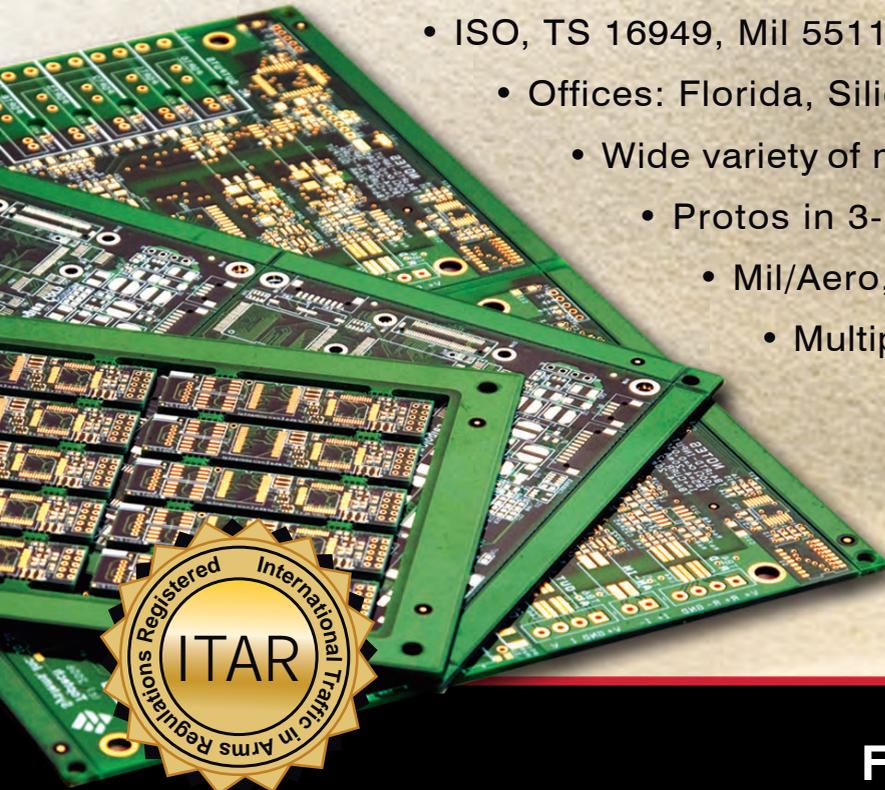
Domestic and Offshore PCB Manufacturing



We Make Printed Circuit Boards Easy.

Now Available in Dine-In and Take-Out!

- ITAR registered for both **domestic** & **offshore** manufacturing
- ISO, TS 16949, Mil 55110, NADCAP, AS 9100 approved
- Offices: Florida, Silicon Valley, Canada and Taiwan
- Wide variety of materials & finishes, 1- 42 layers
- Protos in 3-10 days, production 3-5 weeks
- Mil/Aero, RF, medical, commercial, auto
- Multiple stocking and logistic options



Domestic Manufacturing by



BBG Corporate
8565B Somerset Dr
Largo, FL 33773

BBG SV
3305 Kifer Rd
Santa Clara, CA 95051

For more info reach us at:

727-549-2200

sales@bareboard.com

KICK-STARTING A REVOLUTION: IPC-2581 MEETS GERBER *continues*

CAD data reaches the fabricator as an archive, which are normally contained in the following formats:

- Layer images: Gerber
- Drill files: Excellon, generic NC or Gerber
- Netlist file (roughly 50% of cases): IPC-356
- Function of the image and drill files: text file or drawing
- Stack-up, materials, colors (not always included): text file or drawing
- Other manufacturing instructions: text file or drawing

This information must be read into the fabricator's CAM system. It's clear from the types of data formats listed above that it is only partially standardized and machine readable, even if the process can be partially automated thanks to advanced solutions (such as Ucamco's Integr8tor).

In this article, I would like to explore whether it is possible to improve archive structure and its automatic handling by adopting better data formats. In order to analyze this possibility let's look at each archive's elements separately:

Layer Image Files in Gerber Format

I realise this may seem like marketing hyperbole, but this is truly the most reliable part of CAD-to-CAM data transfer – confirmed by the fact that today's most complex PCBs are all manufactured from extended Gerber files, the vast majority of which will read into a CAM system without a hitch. Extended Gerber is tried and tested; it is a simple, compact, yet precise format whose unequivocal, well-documented presentation is easy to interpret. It's complete in that each layer is described by one single file, and it's portable and easy to debug, as it uses printable 7-bit ASCII characters. Furthermore, it can be read by people as well as all CAM systems with viewers such as GraphiCode's free GC-Prevue viewer.

In fact, Gerber input and output processors are probably the most reliable software in the PCB industry. The freely available Gerber Format Specification^[1] itself is also quite clear and explicit.

That said, I would like to underline the absolute necessity of using proper RS-274X extended Gerber files. Some archives, thankfully fewer each year, are still being transferred in the old RS-274-D Gerber format. This is totally obsolete, severely limited, must be inputted manually, problematic in CAM, and it should be laid to rest as the relic it is.

There is no need for a new format for image transfer.

Drill and Route Information

Problems with drill files are almost exclusively caused by the poor or incomplete use of the Excellon format. In too many instances, so-called Excellon files contain just coordinate data and tool numbers, and the CAM engineer has to search the archive for supporting text files in order to discover which tool sizes, scale and measurement units are to be applied^[2]. Some designers are even using the EIA codes that were already obsolete back in 1980. This too is a choice, but at this point, why not go the whole hog and do the documents in cuneiform script? I can recommend a good font site^[3].

Note that the solution for these shortcomings is not to be found in adopting a more complex new format as this will only aggravate these issues – if files are already being written poorly in the simple Excellon format, imagine the problems in a new and more complex format!

CAM engineers far prefer to receive Gerber drill files as with proper Gerber data there are no problems in transferring drill sizes and locations. Then, when the job is completely cammed, their CAM systems will generate Excellon files dedicated to their drilling equipment.

“
**Extended Gerber
 is tried and tested;
 it is a simple,
 compact, yet precise
 format whose
 unequivocal,
 well-documented
 presentation is
 easy to interpret.**
 ”

There is no need for a new format for drill information transfer, but for better usage of what exists.

Netlist Information

Here too, a standard exists. The good old IPC-356-A standard falls short when it comes to driving today's electrical testers, but it is perfectly adequate for transferring netlists from CAD and CAM. A netlist is, after all, a simple structure. That said, the IPC file must be properly prepared – poor implementation will inevitably result in poor netlist files, a typical problem being incorrect handling of NPTH locations. This is not the fault of the format; problems are generally down to poor understanding of the application by the implementors. Using a different format will not resolve the issue and will introduce further problems. The best solution is to promote better use of the standard through education, tutorials and application notes.

There is no need for a new format for drill information transfer, but for better usage of IPC-356.

Layer Structure, Stackup, Materials, Colors and Tolerances

Stackup design requires a deep knowledge. For many PCB fabricators this is an integral part of their unique selling proposition, perhaps more so than their ability to design and manufacture complex images. However, the description of a stackup is a pretty straightforward list of materials and their properties.

The problem is that there are no standards for transferring stackup information within the framework of a Gerber archive, so informal text files or drawings are the norm. As they do not have a standard structure, such files can often contain incomplete and/or unclear data, forcing CAM engineers to search through accompanying documents, contact the designer, and manually

input data – practices that are frankly unworthy of a high-tech industry like ours.

To recap, we have clear standards for images, drill and route, and netlist information. The misery starts when we get to data describing parameters such as stackup, for which there are no standalone standards at all.

There is an urgent need for a standard format to transfer this information.

“
**The misery starts
 when we get to data
 describing parameters
 such as stackup,
 for which there
 are no standalone
 standards at all.
 There is an urgent
 need for a standard
 format to transfer
 this information.**”

Umbrella CAD-CAM Formats

Numerous attempts have been made to rectify this by creating total CAD-to-CAM data formats such as EDIF, ODB++, Barco DPF and GenCAM. These have all failed, or at best, have achieved limited acceptance. The reason is that they had to be adopted wholesale and nothing was foreseen to combine them with established workflows. Worse, their use imposes the use of new imaging models.

This is a real minefield because all new geometric applications used to create imaging models are initially plagued by tricky bugs – not because geometric programmers are particularly incompetent or sloppy, quite to the contrary, but because this type of programming is very difficult. *The Algorithm Design Manual*^[4], for example, says that: “Implementing basic geometric primitives is a task fraught with peril...There are two different issues at work here: geometric degeneracy and numerical instability...”

And *Computational Geometry in C*^[5] states in a rather resigned tone that “There is no easy solution to the fundamental problems faced here [...] There are several coping strategies...”

The TopCoders blog affirms that “Many TopCoders seem to be mortally afraid of geometry problems.” The fact is that it can take years to sort out the bugs in new image formats, as *The Algorithm Design Manual* intones: “Expect to expend a lot of effort if you are determined to do it right.”

KICK-STARTING A REVOLUTION: IPC-2581 MEETS GERBER *continues*

Yet for the CAD-CAM transition, we absolutely have to “do it right.” Errors in images, fiendishly difficult to detect, are highly likely to lead to scrap. Knowing this, CAD and CAM professionals are reluctant to rely on new image formats – take for example the readme.txt files that frequently accompany CAD datasets containing both Gerber and ODB++ data format, that give the following instructions:

BARE BOARDS MUST BE FABRICATED WITH GERBER, DRILL AND IPC-356 NETLIST PROVIDED. BOARDS ARE NOT TO BE FABRICATED FROM ODB++ FILE.

This does not indicate that there is anything intrinsically wrong with the ODB++ format; on the contrary, it is included because it may contain useful information. There is, however, a concern about the reliability of the images in the newer ODB++ format which is totally understandable given the abovementioned issues.

So the question is: Do we as an industry really want to change our reliable, known image format for one that may take us years to debug? Let’s look at the facts: Images constitute by far the largest and most complex part of any CAD-CAM archive. We have already seen that this part of the data transfer process is pretty solid. The real issue is with the remaining data which, although it is no less important, is far less complex to characterize.

For example, we have a well established format for the flawless transfer of soldermask images, but we do not have a proper way to transfer information about soldermask color. Color is therefore communicated using supporting documentation, and must be entered manually into the CAM system. This is not a good practice and needs to be changed, but it makes no sense whatsoever to ditch a reliable imaging language to add a standard to describe a simple thing such as the soldermask color.

A Simpler Proposal

I would propose a simpler and safer alternative. We keep what works – the image data format – and we change what doesn’t work, such as the stackup data format.

In essence, the stackup of a single-sequence PCB is nothing more than a list of material layers and their properties. Some of these layers, such as copper layers, have images associated with them. Others, such as FR-4, do not. The drill file can be viewed as an image file that goes from top to bottom. This is simple to describe accurately and completely.

A sequential-build PCB is a little more complex, but not much. Here, the PCB is a list of subassemblies and single material layers. Each subassembly is in turn a list of subassemblies and layers. At the lowest level, the subassemblies are a simple list of single layers, just like a simple PCB. Essentially, a sequential-build PCB is described as a list of layers and assemblies,

and the assemblies themselves are again a list. It is an embedded structure. Not terribly complex.

For optimum CAD-to-CAM communication, the stackup must be described clearly in a formal language that leaves no room for doubt. The stackup may be simple to describe, but it takes a lot of application knowledge to define it clearly and completely to ensure that all the necessary fields are included and easily understood.

This is where IPC-2581 excels. It contains outstanding stackup definitions as it is^[6], has been reviewed minutely by a very active team of stackup specialists from a wide range of PCB design and supply chain companies, and is currently being fine-tuned for the next revision, a process that illustrates the advantages of an open organization such as the the IPC-2581 consortium.

As a result, IPC-2581 is an open standard with industry consensus. It also offers the most capable stackup specification published to date, its structure reflecting the essence of a stackup, while layers with an associated image are linked

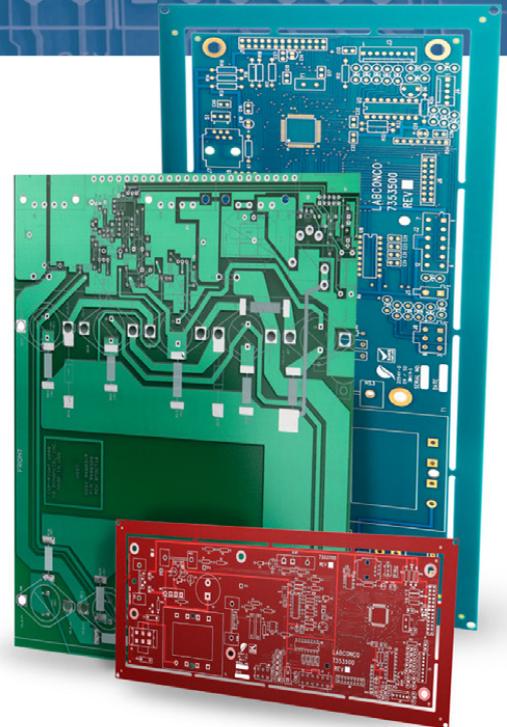
“
**So the question is:
Do we as an
industry really
want to change
our reliable, known
image format for
one that may take
us years to debug?**”

Quality PCBs from the **established** industry leader

With 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 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

KICK-STARTING A REVOLUTION: IPC-2581 MEETS GERBER *continues*

to the description of that image in IPC-2581 format. Furthermore, unlike a typical CAM format which is essentially an image processing format and is therefore image-centric, IPC-2581 is PCB-centric, with its developers' specialist industry know-how built in. IPC-2581 can therefore handle the complexities of specialities like rigid-flex boards as well as a wide range of specialist materials, making it more sophisticated than alternatives such as our DPF format and Valor's ODB++ offering.

I know this from years of experience with IPC-2581. Ucamco was a very early adopter of the standard, and our software may very well be unique in that it uses IPC-2581 routinely. Integr8tor, for example, has been using IPC-2581 since 2006 to describe stackups when it outputs engineering data. Given that this data is input daily by our clients' engineering and ERP systems, integrated IPC-2581 solutions have in fact been in use all over the world for some years now. We and our clients therefore have first-hand experience of the immense advantages offered by IPC-2581 stackup as an integrated part of the CAD-to-CAM communication cycle, and are ever more convinced that this is a real enabler for our industry.

Therefore, I propose the adoption of the IPC-2581 stackup description not only by the broader PCB industry, but as an integral part of conventional Gerber archives. This would mean that for layers with an associated image, the 2581 image description is simply replaced by the Gerber file name describing that image. In other words, I propose that we continue to describe

image and drill files in Gerber format, but add an xml file describing the stackup according to 2581, as illustrated in Figures 1 and 2.

This can also represent the layer structure rather than the full stackup, reflecting real life CAD to CAM workflows, which often start by transferring the layer structure first and adding materials later – and it is a route that is made entirely possible by the flexibility of the xml structure.

If such an xml structure were included in the Gerber archive, the CAM system could then read the xml file, create the proper job structure, and load the associated images with its existing Gerber input processor, without any operator intervention.

The highlighted .gbr files shown in Figure 2 point to the Gerber files in the same archive. The archive would then fully describe the PCB and contain the following files:

- Stackup.ipc2581.xml
- mm620601.gbr
- mm620632.gbr
- mm620660.gbr
- mm620641.gbr
- ImageOutline.gbr
- netlist.356

Benefits of Combining IPC-2581 Stackup with Gerber Images

Compatibility

Such archives would be compatible with existing systems, enabling the Gerber and

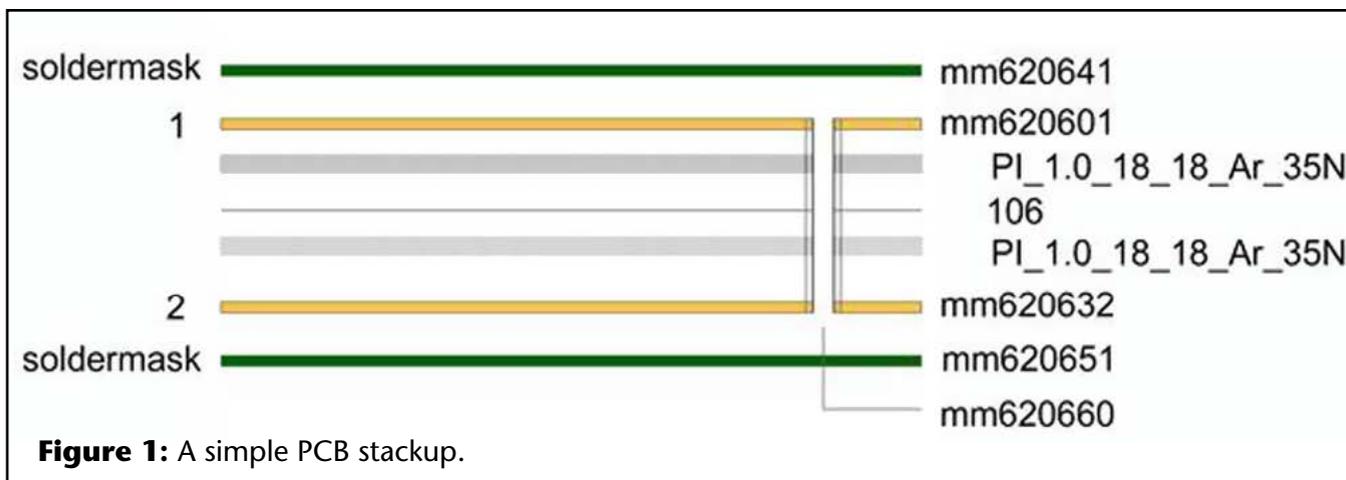


Figure 1: A simple PCB stackup.

KICK-STARTING A REVOLUTION: IPC-2581 MEETS GERBER *continues*

netlist files to be read as they are now, while stackup information would be read from the documentation and entered manually as per current practice. CAM operators would reap benefits from the use of this format because the 2581 structure provides unequivocal stackup description data that can be read either in ASCII or using a generic xml viewer of which there are many available as freeware.

And nobody would be forced to buy new software, so PCB designers would be happy in the knowledge that all their manufacturers can handle the Gerber/2581 archives.

Lower cost

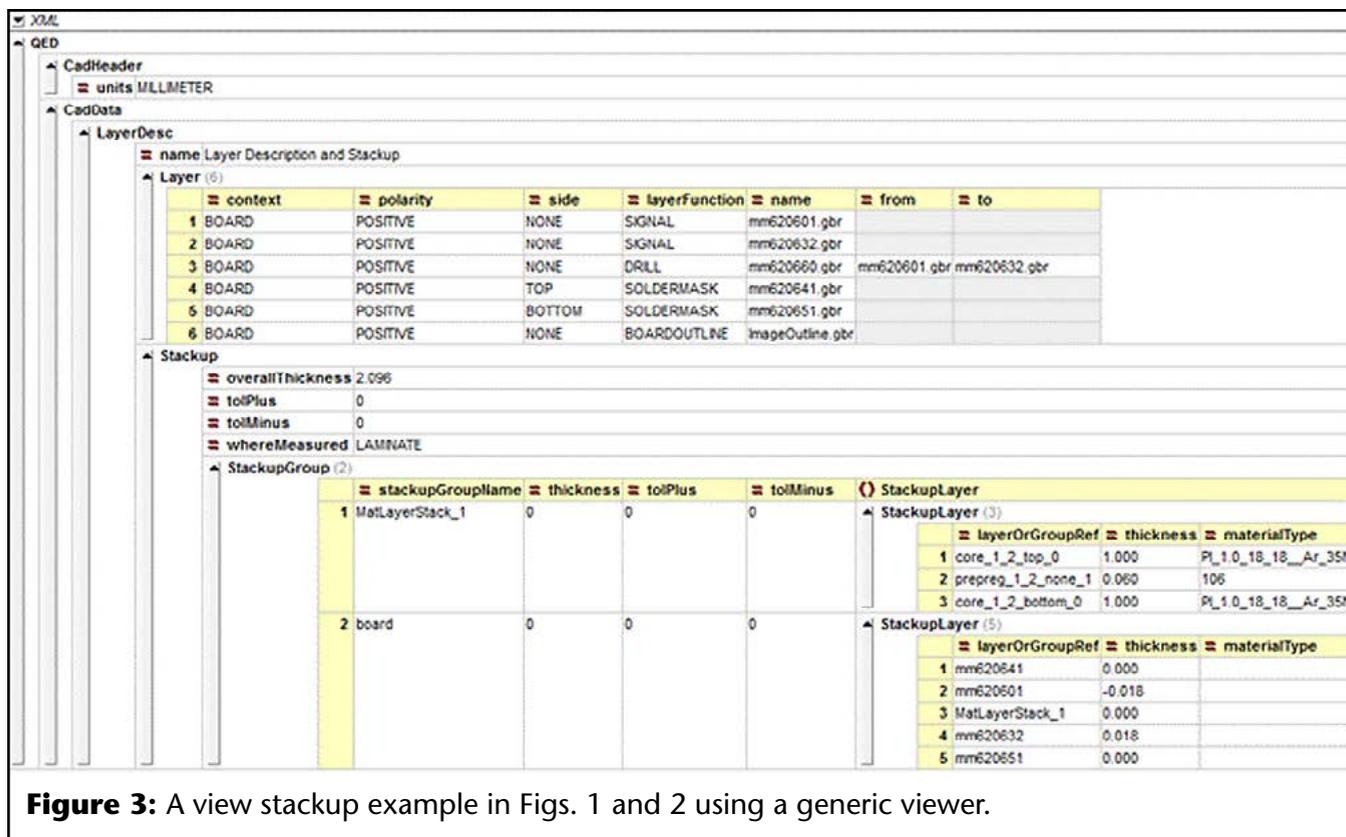
Implementing a new image format is a major undertaking and requires long and painstaking validation, something that would

```

<CadHeader units="MILLIMETER"/>
<CadData>
  <LayerDesc name="Layer Description and Stackup">
    <Layer context="BOARD" polarity="POSITIVE" side="NONE" layerFunction="SIGNAL" name="mm620601.gbr"/>
    <Layer context="BOARD" polarity="POSITIVE" side="NONE" layerFunction="SIGNAL" name="mm620632.gbr"/>
    <Layer context="BOARD" polarity="POSITIVE" side="NONE" layerFunction="DRILL" name="mm620660.gbr"
from="mm620601.gbr" to="mm620632.gbr"/>
    <Layer context="BOARD" polarity="POSITIVE" side="TOP" layerFunction="SOLDERMASK" name="mm620641.gbr"/>
    <Layer context="BOARD" polarity="POSITIVE" side="BOTTOM" layerFunction="SOLDERMASK" name="mm620651.gbr"/>
    <Layer context="BOARD" polarity="POSITIVE" side="NONE" layerFunction="BOARDOUTLINE" name="ImageOutline.gbr"/>
  <Stackup overallThickness="2.096" tolPlus="0" tolMinus="0" whereMeasured="LAMINATE">
    <StackupGroup stackupGroupName="MatLayerStack_1" thickness="0" tolPlus="0" tolMinus="0">
      <StackupLayer layerOrGroupRef="core_1_2_top_0" thickness="1.000"
materialType="PI_1.0_18_18_Ar_35N"/>
      <StackupLayer layerOrGroupRef="prepreg_1_2_none_1" thickness="0.060" materialType="106"/>
      <StackupLayer layerOrGroupRef="core_1_2_bottom_0" thickness="1.000"
materialType="PI_1.0_18_18_Ar_35N"/>
    </StackupGroup>
    <StackupGroup stackupGroupName="board" thickness="0" tolPlus="0" tolMinus="0">
      <StackupLayer layerOrGroupRef="mm620641" thickness="0.000" materialType=""/>
      <StackupLayer layerOrGroupRef="mm620601" thickness="-0.018" materialType=""/>
      <StackupLayer layerOrGroupRef="MatLayerStack_1" thickness="0.000" materialType=""/>
      <StackupLayer layerOrGroupRef="mm620632" thickness="0.018" materialType=""/>
      <StackupLayer layerOrGroupRef="mm620651" thickness="0.000" materialType=""/>
    </StackupGroup>
  </Stackup>

```

Figure 2: How the stackup in Figure 1 would be described in a 2581-style xml structure.

KICK-STARTING A REVOLUTION: IPC-2581 MEETS GERBER *continues*

likely be beyond the reach of smaller software vendors. Make no mistake, it is a costly affair for all parties involved. To quote *The Algorithm Design Manual* again, “Expect to expend a lot of effort if you are determined to do it right.” By contrast, implementing the 2581 stackup model and combining it with an existing Gerber processor costs far less. This benefits the industry in general, and it translates into lower user costs.

Low Risk

The risks involved in adopting the 2581 stackup format are negligible. Not least because this would be a massive improvement on the chaos that reigns now, but more importantly because a stackup transferred via 2581 can be verified visually for plausibility, or compared to conventional drawings. This is impossible with the highly complex layer images, where errors are likely to escape notice, enter production, and create scrap – a risk that, as we have seen, is greatly amplified when implementing a new image format.

The Route to Full IPC-2581 Implementation

In this article we have thus far addressed stackup and materials, the area which most urgently needs a standard, by proposing a solution that combines the appropriate section of the 2581 standard with the incumbent Gerber image format. The same principles could be applied to other new elements in the 2581 standard, such as its component description.

The same cannot be said for its image section. At this moment in time, there is no significant benefit in adopting it, but should the 2581 standard evolve to a point where the benefits of integrating the image description are commensurate with the costs of doing so, there would be good reason to adopt it instead of the Gerber format.

Parallels with the Printing Industry

In looking for the route forward, our industry would do well to take a leaf from the graphic arts industry, which faces challenges similar to our own. The way in which data flows within the PCB industry can be compared to how data

moves within the graphic arts industry, where the printer receives a digital description, mostly image data, of a magazine or consumer package, and then produces the required number of copies.

In the 1980s, data transfer from customer to printer was even more dismal than it was in our industry. Then the “PostScript Revolution”^[7] kicked in as Adobe’s PostScript page description language was used to transfer data digitally. PostScript was developed through three major iterations as the industry placed ever greater demands on it, and then in the 1990s, the PDF format was created^[8]. Using exactly the same imaging model as PostScript, PDF has been developed over the years to the point that today it offers powerful wide-ranging functionality, interactive options such as annotation and dialogue, and the security of certification.

Its development took time, and there were discussions over the years over whether or not to “kick out” the format’s “stupid” forerunners, but as PDF expanded, even the graphic arts industry’s last hold-outs were finally won over.

Today, virtually all graphics production uses PDF, a great format that enables PR agencies’ magazine ads to go straight to offset print without operator intervention or even visual checks – a feat that our own industry can only dream about. And it’s not because the graphics industry is any less demanding than ours: Listen to an ad manager insisting on the precise color contrast of his full page advertisement, or a product manager worrying about the shape and color of a new consumer package, and you’ll understand what I mean.

The graphic arts industry got there by gradually improving its existing, functioning workflow. This in turn was made possible by progressively developing its existing imaging model rather than attempting to overthrow it.

Our industry too went through something of a revolution in the 1980s as manufacturers started to take digital data rather than film – by analogy we could call it the Gerber Revolution. But we have made little progress since then. I believe that this is because the only alternatives that have been proposed have focused on completely replacing the image format instead of addressing the shortcomings in the workflow as the graphics industry did.

I believe that we can learn some valuable lessons from the tremendous success achieved in graphic arts, and that we too should follow the route to progressive improvement by making our workflows increasingly compatible. I am not suggesting that we should aim for total hands-off operations, but I think that, with intelligent and step-by-step improvements we could foreseeably arrive at the point at which simple, repetitive boards could be manufactured without operator intervention.

“
***Our industry too
 went through
 something of a
 revolution in the 1980s
 as manufacturers
 started to take
 digital data rather
 than film – by analogy
 we could call it
 the Gerber Revolution.***
 ”

Conclusion

PCB designs are typically transferred from CAD to CAM in Gerber-based archives. These leave much to be desired, but the issues have little to do with the RS-274X extended Gerber format: Proper extended Gerber files can be read in without a problem. What is lacking is a standard, machine-readable way to transfer non-image information, such as the stackup and components. In other words, the so-called problems of Gerber are not about what Gerber does, which it does superbly, but about what Gerber does not do, and was never designed to do.

This issue could be resolved simply and cost-effectively by using the IPC-2581 standard, which has a well-designed stackup description format. The problem is that as the standard is defined now, in order to use this gem, PCB professionals are also obliged also to use the image section of the IPC-2581 format. This new image format offers no material benefits,

KICK-STARTING A REVOLUTION: IPC-2581 MEETS GERBER *continues*

if any, over Gerber. Developing, debugging and validating a new image format is a daunting task, and carries the risk of creating a lot of expensive scrap. The industry dislikes this prospect, and has accordingly shunned new image-based formats altogether, or has only adopted them to a limited degree, as in the case of ODB++.

In my opinion, the all-or-nothing approach repeatedly attempted in improving the CAD-to-CAM workflow benefits nobody. We are currently in a deadlock because adopting IPC-2581 demands the new software simply to do what can already be done now, so it can only take off once enough users have adopted it. Yet the new software will only be acquired once the new format is used widely. IPC-2581 would in fact be adopted faster and more broadly, and its benefits enjoyed by the industry sooner and more generally, if what is new in it could be accessed without having to adopt a new image format, buy new software and upset existing workflows.

This is eminently possible, and surprisingly simple. If slightly tweaked, the IPC-2581 stackup description would allow linking to Gerber images rather than to the new image formats. Both could be combined within the same archive – an approach whose development, test and validation would cost just a fraction of the investment needed to introduce a new image format. Everyone would benefit from this: the combined format would kick-start the adoption of IPC-2581, and users, no longer forced to buy new software, would work with the new archives semi-manually, and buy the software later on. Other IPC-2581 sections such as components could be integrated in the same way. And eventually, when there are enough benefits in adopting the IPC-2581 image format, good old Gerber could finally be retired after its many long years of faithful service to the PCB industry.

For now, though, in discussions about CAM to CAM data transfer, large numbers of PCB professionals express their preference to stay with Gerber. It's not broken, after all, so why fix it? That's not to say that they like the way in which other information is currently transferred; on the contrary, they

sorely need a standard for information like stackup and component data. Let us give them what they want, and need: Gerber images and a proper standard for stackup and other information.

A CAM manager to whom I explained these ideas exclaimed, "Good old Gerber files with an IPC-2581 stackup – this is the best of both worlds!" So let's follow the example of the graphic arts industry by keeping what works well and integrate it intelligently with new structures that complement and enhance it, working with care and determination towards a better way of communicating, collaborating, and building quality into our industry.

If you would like to join me in enabling all to move forward with this, I look forward to hearing from you at the PCB Forum at LinkedIn. **PCBDESIGN**

References

1. The Gerber Format Specification. Revision 11, [Ucamco](#) 2012
2. Excellon [Wikipedia](#) entry
3. [Cuneiform fonts](#)
4. The Algorithm Design Manual by Steven S. Skiena, Springer-Verlag, New York, Inc. 1998.
5. Computational Geometry in C by Joseph O'Rourke, Cambridge University Press 2001.
6. [IPC 2581](#)
7. [The History of PostScript](#)
8. [The History of PDF](#)



Karel Tavernier is managing director of Ucamco. He has 30 years of experience with software and imaging equipment for the PCB and electronic packaging industry, including sales, service and R&D. He has been in his present role since July 1995. Tavernier received a master's degree in electronic engineering and a postgraduate in computer science from the University of Gent. He holds a management degree from Vlerick Management School.

Rogers Talks New Thermal Product Developments

by Real Time with...
IPC APEX EXPO



Rogers Corporation's John Coonrod talks with Guest Editor Dan Beaulieu about new product development, including the company's new thermal material, and describes how his company selects new product technologies in which to invest.



realtimewith.com



IPC Updates PCB Design Standard

There's plenty of new information in IPC-2221B, Generic Standard on Printed Board Design. It addresses areas as diverse as testing, surface finishes, and separating boards from panels.

In the fast-paced world of electronics, you don't often hear the old saying "good things come to those who wait." But those in the printed board industry will find a lot worth waiting for when they pick up freshly finished copies of IPC-2221B, Generic Standard on Printed Board Design.

The standard serves as the basis for the design of all types of printed boards. It's been in development for nearly a decade. The last revision came out in May 2003, when technology was vastly different than today.

"We finally drew a line in the sand and said we couldn't do anything more

without releasing a revision with the content we've developed thus far. We will never run out of new things to do with this standard," said John Perry, IPC technical project manager.

Some of these things are new. For example, IPC-2221B includes a section on the many surface finish alternatives now used to protect lands and pads so their solderability won't degrade over time.

The completion of IPC-2221B marks the finish of the three main IPC documents for printed board design.

"We've had a nice trifecta in the last three years," Perry said. "IPC-2222A, Sectional Design Standard for Rigid Organic Printed Boards came out two years ago and IPC-2223C, Sectional Design Standard for Flexible Printed Boards, came out last year. This latest release completes an update of the most widely used standards in the IPC-2220 design series."



Routing Techniques for Complex Designs

by Barry Olney

IN-CIRCUIT DESIGN PTY LTD AUSTRALIA

SUMMARY: *When we analyze customer designs, we often find that crosstalk is a recurrent major issue with manually routed boards. Autorouters are ideal for digital designs as they tend to use all available space, thus reducing the possibility of crosstalk due to proximity. An autorouter is not a push-button solution. However, with a little interactive control and the systematic process outlined in this column, it can be a powerful productivity tool.*

To err is human; to completely mess it up, use software.

Autorouter software is essentially artificial intelligence (AI) software – although fairly basic – that makes certain decisions that mimic what designers do in the process of routing a board. Its capacity to do this, of course, varies by software developer, and is dependent upon algorithm complexity and how easy or difficult it is to control the router. Only so many rules can be practically defined, and every situation is different, requiring unique tradeoffs. The limiting factor with any autorouter is describing just what it is that human decision-makers actually do.

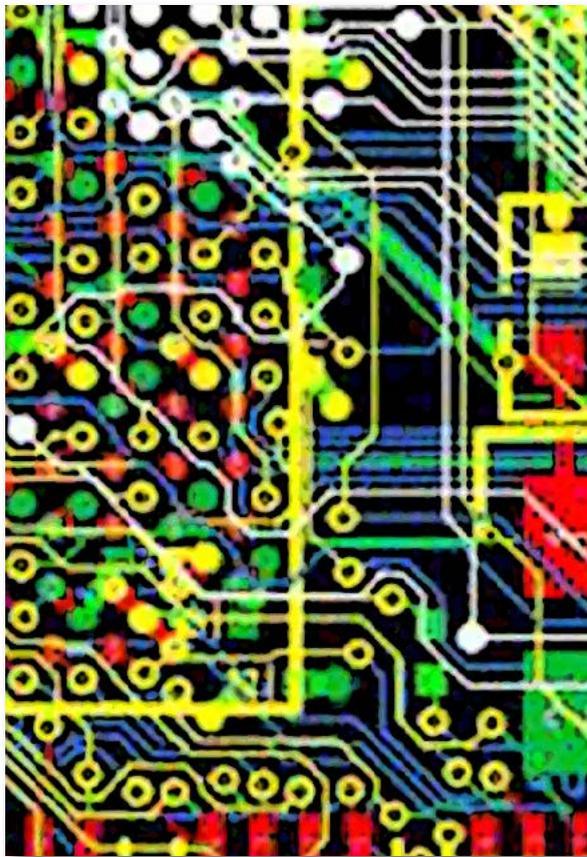
Very often, and most especially in tasks that are highly routine and subconsciously automated, designers may struggle to describe all the steps and conditional rules they employ. Not because they do not want to, but because there is simply a lot that they do not think about in an explicable way.

Layout of an analog circuit or a switched-mode supply, and especially one that incorporates specific placement, routing, thermal and isolation requirements – combined with aesthetic goals – relies on many tradeoffs that a seasoned designer would have trouble describing. Many of those tradeoffs involve a series of complex what-if analyses, like multiple possibilities in a game of chess.

Also, complex digital designs incorporating DDR2 or DDR3 memory, for instance, require matched-length routing of all signals, keeping flight times tight between the clock and address signals and the strobe and data signals. Even with all conditional rules defined, this type of routing requires focused interactive control.

It can be a challenge to get PCB designers to use an autorouter because it introduces unknowns: What it is capable of? How to control it? How much time can it save? Instead, many designers prefer to go back to their comfort zone and complete all connections manually using the autorouter between their ears. This internal dialog ends up being a waste of valuable time and can lead to other problems down the track. Some use an autorouter as a sanity check – if the autorouter can route the board to completion, then they can probably do a better job. But there is more to it.

Employed in the proper context, autorouters can make PCB designers a good bit more productive. For example, they can be used to:





Reach a new dimension with
our complete design flow:
Pantheon, NPI's 2012 winner
for most user-friendly
PCB design tool.

Our superior PCB, RF and Hybrid design software guides you quickly and seamlessly from analysis to schematic to layout to manufacturing, with total design flexibility. Intercept's RF solution includes bidirectional analysis interfaces, parametric RF models, a parametric model route mode and model generator. Design with or without a schematic. Productivity, efficiency and quality are guaranteed. Visit us in Santa Clara, CA at DesignCon on January 29-30 for a full demo of our latest advancements.

XTENT HIGH SPEED DESIGN OPTION
PANTHEON ADVANCED PCB, HYBRID, RF LAYOUT
MOZAIX NEXT GENERATION SCHEMATIC CAPTURE
INDX FLEXIBLE LIBRARY MANAGEMENT
DRAWINGX, QCX, RELEASEX DESIGN AUTOMATION
PALINDROME REVERSE ENGINEERING
INTERFACES TO RF, SIMULATION, AUTOROUTERS
30+ DATABASE TRANSLATORS



www.intercept.com

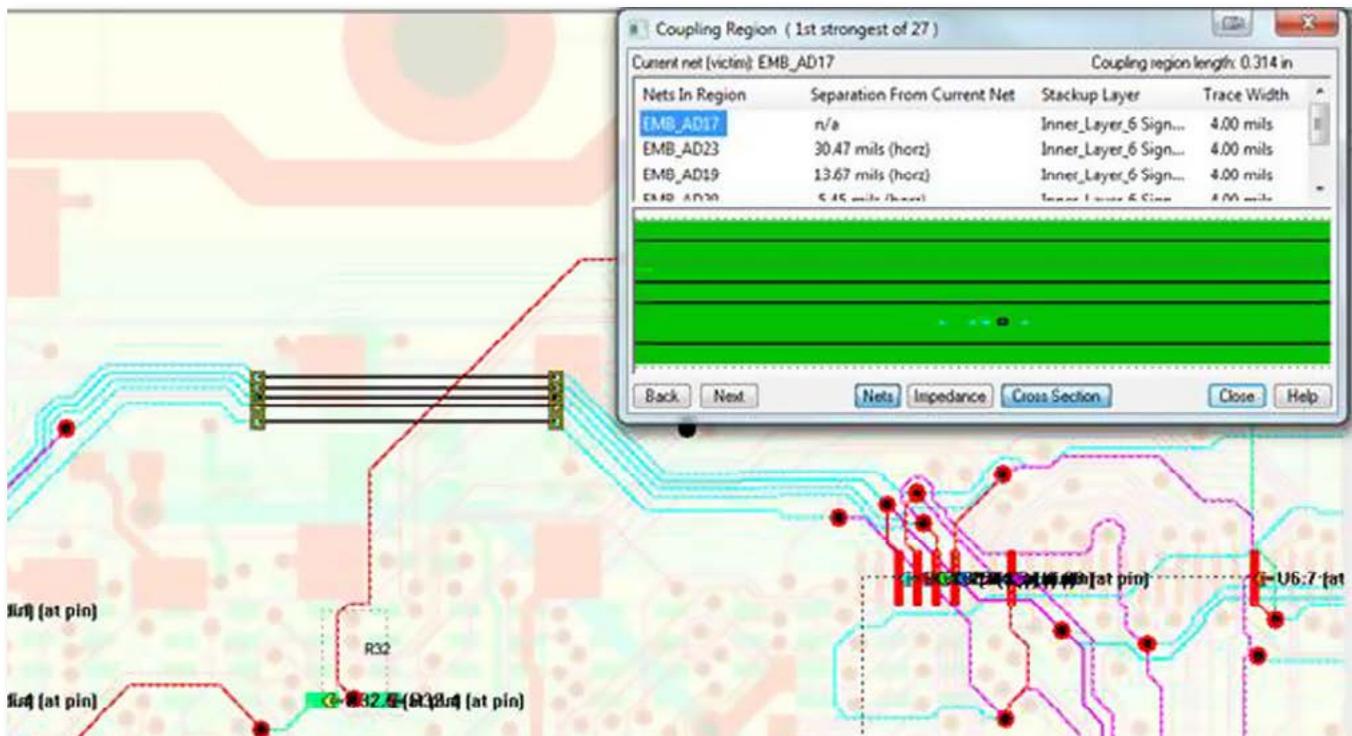
ROUTING TECHNIQUES FOR COMPLEX DESIGNS *continues*

Figure 1: Crosstalk between manually routed parallel segments.

1. Check if design rules have been defined correctly
2. Identify placement congestion
3. Quickly try different routing strategies (after making a backup)
4. Efficiently fan-out from devices
5. Maximize real estate utilization

My company, In-Circuit Design (ICD), provides simulation services such as analyzing customer designs for signal integrity, timing, crosstalk, and EMC issues. More often than not, we find that crosstalk is a recurrent major issue with boards that are manually routed. When we manually route, we tend to use our artistic talents too much, keeping everything nice and neat, coupling traces close together (especially buses) mainly for aesthetics. This may be fine for analogue and low-frequency designs but when we get into the high-speed domain, with rise times < 1 ns, we can only run two trace segments in parallel for less than half an inch before we get excessive crosstalk. Autorouters are ideal for digital design as they tend to use all available space, thus reducing the possibility of crosstalk due to proximity.

Of course, a designer can spend hours setting up design rules to control the autorouter, but I prefer to drive the autorouter from the schematic. This only requires the setup of the most basic rules. When we draw a schematic, we draw it by functionality, and I believe that we should also place and route by functionality. In this way, I can add my own creativity and decision-making on the fly, while still taking advantage of the automation.

Most popular EDA tools have the ability to cross-probe between the schematic and router. This is a fantastic feature that enables a PCB designer to build up an extremely dense, complex route, in a couple of hours – by controlling the router from the schematic. I discussed this in detail in my previous Beyond Design column: [Interactive Placement and Routing Strategies](#).

We do not need to do any routing ourselves to get an acceptable route of the non-critical nets. Of course, matched lengths, differential pairs and other critical signals should be routed with the precision they require. I start by placing all the components by functionality, selecting the desired component on the schematic and dropping them where I want them on the

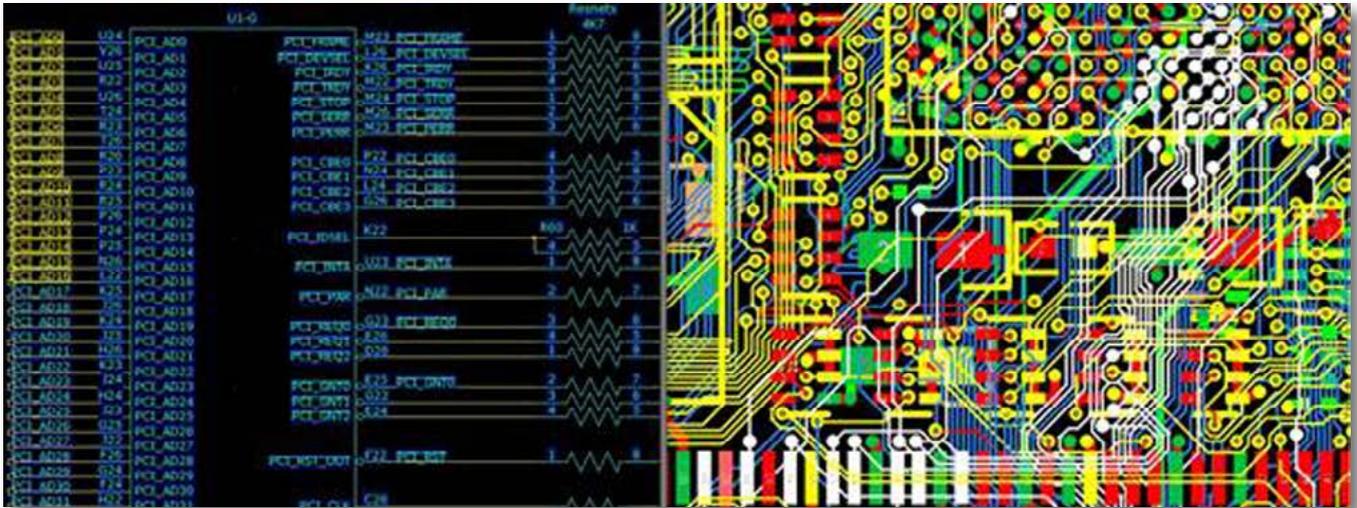


Figure 2: Cross-probing from schematic to router of part of a PCI bus.

PCB. Similarly, when routing, I select a chip on the schematic, the nets are highlighted on the PCB, and I select “Fan-out” on the router. Then, select the critical nets on the schematic “Fan-out,” and “Route” with the autorouter. I use the “Move” command to push and shove the traces where I want them, then move on to the next group of nets and repeat. Each group of routed traces is then verified after completion.

When we drive the router from the schematic, it’s possible to see what needs to be done without entering conditional design rules, and we can later manipulate the traces as if we hand-routed them. Once all the critical nets are routed, I fix them, then turn the autorouter loose on the remainder of the nets to finish off the connections.

The Perimeter Routing Technique

I have used the following technique successfully, over the years, with a number of autorouters. It was first put to the test on the Daisy/Dazix Star Router back in 1987, Cadence Prance-XL Router, Mentor’s Expedition Autoactive, and then the PADS Router.

In other words, it’s safe to say that the technique generalizes.

Let’s assume that you have followed a methodology for placing and routing critical, high-speed signals, fixed them, and that the remainder of the signals are non-critical. We will look at a scenario in which the board is 98% com-

pleted after a series of autorouter and manufacturing routines. A 98% completion rate sounds pretty good, but we all know that the router will leave the most difficult/longest traces for us to complete. Indeed, the last 50 or so traces may take us days of head-banging frustration to complete.

If a board will not route to completion, it may not be the router’s fault. It could just be that we have: (a) poor placement with bus crossovers, (b) poorly defined design rules, or (c) have not planned enough signal layers into the stackup. I guess you get a feel for how many layers are required after doing a few boards. My general rule of thumb is that if I cannot get at least 85% completion before I start tweaking the design, then I will have serious problems. With less than 85% completion out of the blocks, I re-evaluate component placement, redefine design rules if necessary, add a couple more signal layers, or reduce the functionality of the design.

All routers tend to route inward because the algorithms are tuned to make the shortest possible connection of the two open ends. This is why you generally see a tangle of rats’ nests in the centre of the board where all the signals try to cross. Beneath this apparent obstacle is an underlying opportunity.

Here’s the trick: Define a route keep-out perimeter channel 200 mils around the edge of the board. Avoid enclosing component pins that have connections, as they also need to be

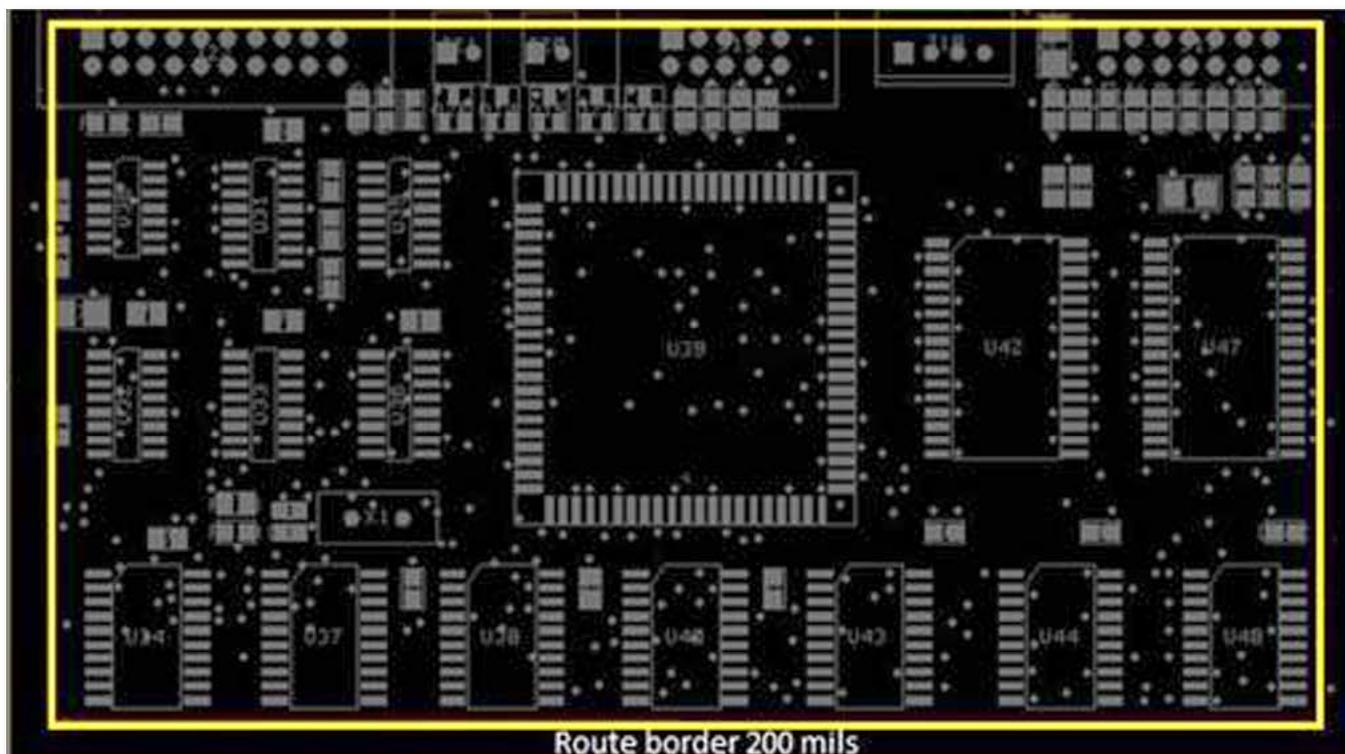


Figure 3: Route border defined 200 mils for the edge of board.

routed. Most components should be 200 mils from the edge anyway, if you are following IPC standards, but there will be connectors and interface devices, etc., that are closer. This channel will prove to be invaluable later to polish off that last 2% of nets.

Autoroute the board to the best completion rate. Some rip-up and retry of connections should be tried and the via minimization and manufacturing passes should be utilized. The autorouter will smooth the lines, remove ghost vias and staircases, eliminate unnecessary vias and reduce the etch length. But, we still have nets to route.

A via fan-out grid should be used initially to avoid blocking route channels. However, at this stage the via grid can be removed as we are only interested in completing remaining connections, since the main routes are in place. Invoke the autorouter again.

Finally, drop the route border to 50 mil perimeter. This gives us 150 mils of extra routing channel on all signal layers around the perimeter of the board. Using 5/5 technology this equates to 15 additional traces per layer. To

complete the remaining nets, manually route each net out to the edge, follow the perimeter around the board in either a clockwise or anticlockwise direction, and then route back in to terminate the connection. This provides an additional 30 traces per signal layer; for example, on a 12-layer board with 8 signal layers, that's 240 additional traces, typically more than enough to complete the route.

Points to Remember

- Autorouters are tools that can make designers more productive, but they don't represent a trivial, push-button solution.
- The limiting factor is describing just what it is that human decision-makers actually do. Many of those judgments involve a nested series of complex what-if tradeoffs.
- Autorouters can be particularly useful for digital designs, as they tend to utilize all available board real estate, reducing the possibility of crosstalk due to proximity.
- A designer can spend hours setting up

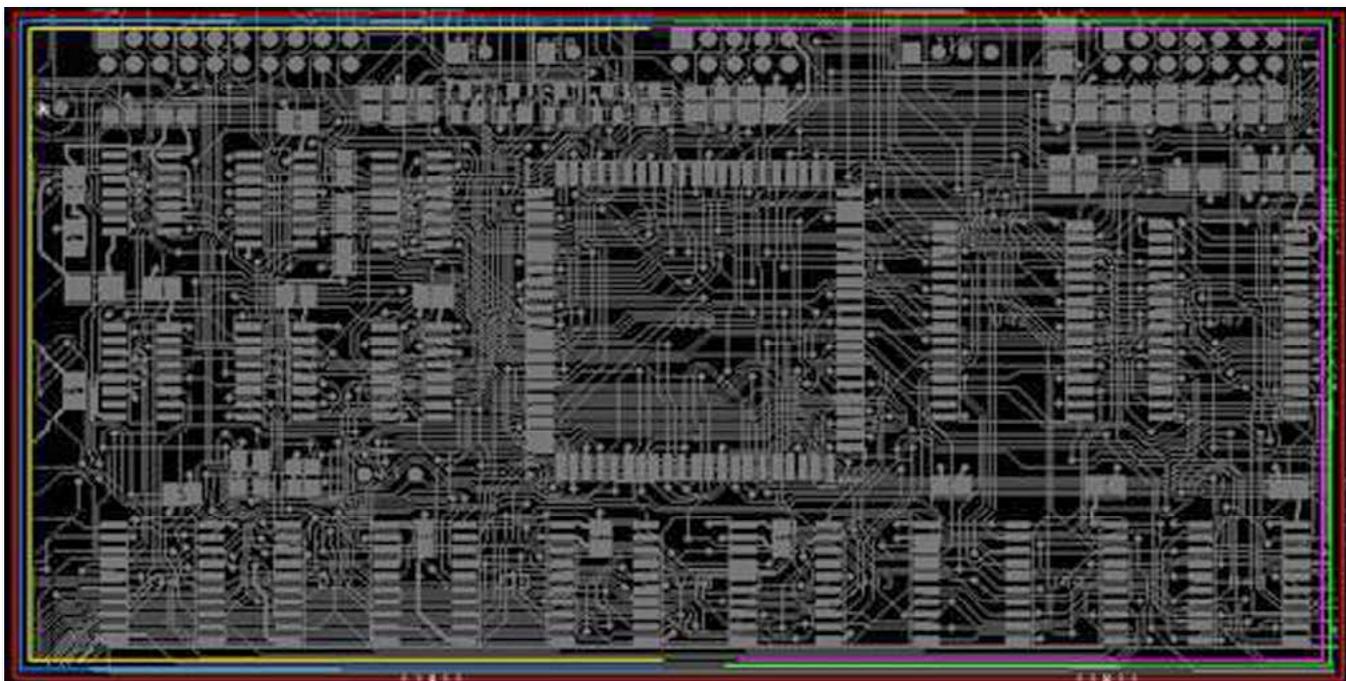


Figure 4: Route border reduced to 50 mils from the edge.

design rules to control the autorouter.

But, when we drive the router from the schematic, it's possible to see what needs to be done without laboring through the process of dialing in complex tradeoffs and conditional design rules.

- Cross-probing between the schematic and router is a fantastic feature that allows the designer to build up an extremely dense, complex route in a couple of hours.
- As general rule of thumb, if you can't reach at least 85% completion with the autorouter without manually tweaking the design, you're setting yourself up for some serious design headaches, as you finish the board.
- If a board won't route to completion, it may not be the router's fault. It may just be that we have: poor placement with bus crossovers, incorrectly defined design rules, or we haven't allowed enough signal layers in the stackup.
- The perimeter routing technique provides an additional 30 traces per signal layer. For a 12-layer board with 8 signal layers, that's a whopping 240 additional traces.

PCBDESIGN

References

1. Advanced Design for SMT – Barry Olney
2. Beyond Design: [Interactive Placement and Routing Strategies](#) – Barry Olney
3. Beyond Design: [Intro to Board-Level Simulation and the PCB Design Process](#) – Barry Olney
4. Beyond Design: [Mixed Digital-Analog Technologies](#) – Barry Olney
5. [PCB Design Techniques for DDR, DDR2 & DDR3, Part 2](#) – Barry Olney
6. [PCB Design Techniques for DDR, DDR2 & DDR3, Part 1](#) – Barry Olney
7. The ICD Stackup and PDN Planner can be downloaded from www.icd.com.au



Barry Olney is managing director of In-Circuit Design Pty Ltd (ICD), Australia. ICD is a PCB design service bureau specializing in board-level simulation. The company developed the ICD Stackup Planner and ICD PDN Planner software.

Optimizing PCB New Product Introduction Using ODB++

by Julian Coates

ODB++ SOLUTIONS ALLIANCE

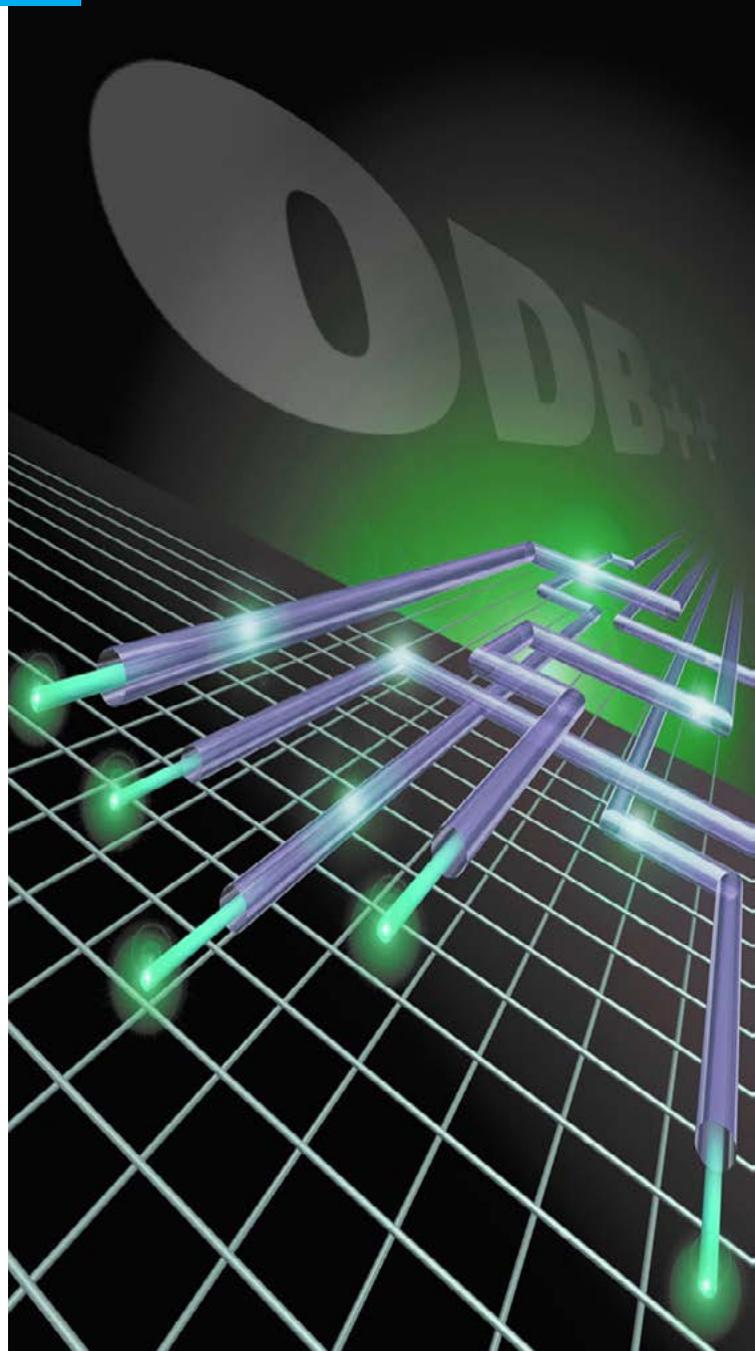
SUMMARY: *The industry does not need any more data transfer formats. On the contrary, the maximum efficiency gains can be achieved at the minimum total cost to everyone by implementing existing industry-proven solutions. This article describes the business advantages of ODB++ intelligent data, along with some comparisons and contrasts with alternative formats.*

A PCB design is not successful until it has been manufactured on time, according to the designer's intentions, and at an acceptable total cost. The traditional approach of outputting multiple legacy file formats and sending them to the manufacturer on the assumption that he will fix any manufacturability problems carries a high indirect cost in terms of supply chain and cycle time risk.

And, in the end, the designer pays that cost. Design and manufacturing must be seen as one business process integrated by intelligent data. This article describes the business advantages of ODB++ intelligent data, along with some comparisons and contrasts with alternative formats.

Intelligent vs. Non-intelligent Data

In general, there are two types of manufacturing data: intelligent and non-intelligent. Non-intelligent data are collections of files in vector formats such as Gerber, HPGL or Excellon



drill files, which use a sequence of single-entity definitions composing a single layer of information. An external source is needed to determine the use of such a single layer. Sometimes this is the filename or a readme.txt document. For a manufacturer to make sense of these files, they must reintegrate them, reconnecting the disjointed information, and then get a view of what the complete PCB looks like – sort of like turning applesauce back into apples.

Non-intelligent data must be imported into a DFM or CAM tool; however, an engineer must use functions to identify the single layers – such

Delivering the highest quality standard for **Aerospace and Defense**



Ventec Accredited to AS9100 Rev. C

We are proud to announce that our parent company, Ventec Electronics Suzhou Co Ltd, is now fully certified to AS9100 Revision C for both the design and manufacture of copper clad laminate and prepreg for aerospace applications.

AS9100 is the quality management standard specifically written for the aerospace and defence industry, to satisfy authorities such as the Federal Aviation Administration, ensuring quality and safety in the "high risk" aerospace industry.

MORE INFO:

**POLYIMIDE & HIGH RELIABILITY
FR4 PRODUCTS**

Wherever technology takes you,
we deliver.



Accredited
AS9100 Rev C



Ventec Europe
www.ventec-europe.com

Ventec USA
www.ventec-usa.com

Ventec International Group
www.ventecclaminates.com

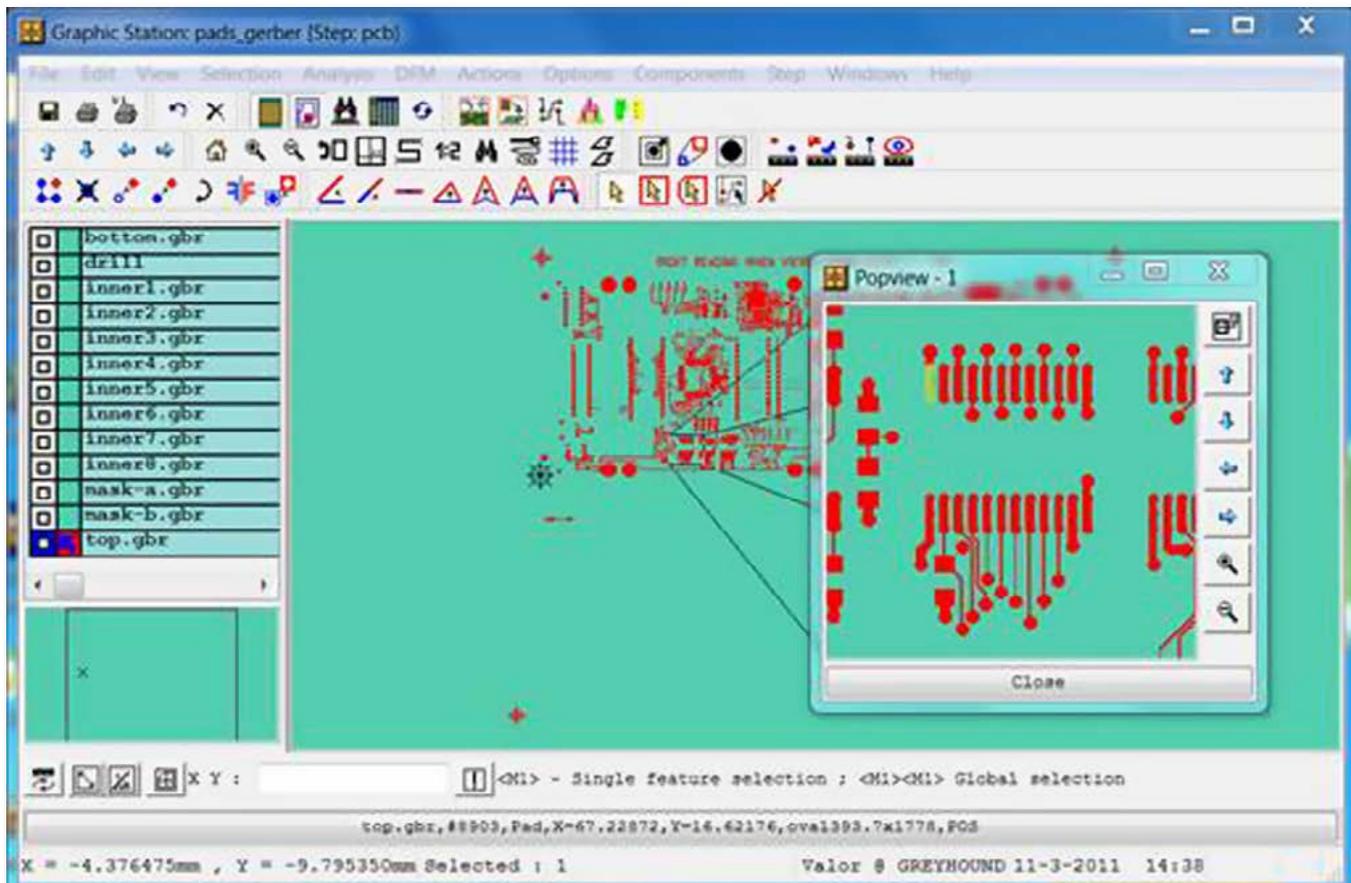
OPTIMIZING PCB NEW PRODUCT INTRODUCTION USING ODB++ *continues*

Figure 1: With non-intelligent data, all that is displayed with the selected element is the information that it is an oval pad at an XY location.

as what represents a signal layer, or what represents the solder mask – using a graphics editor. In Figure 1, a set of Gerber files has been imported and we can see that the layer buildup is in alphabetic filename order. In the figure, the general characteristics of the PCB are visible – pads and traces on this layer – but other information such as physical characteristics, the solder mask, parts outline, etc., are not available. The inset shows information for U37, pin 24 (in yellow) on the top layer (filename top.gbr).

On the other hand, intelligent data comes from the EDA system which already contains objects such as components, pad stacks, traces, and even net names. In the late 1990s, ODB++ was developed by Valor Computerized Systems (acquired in 2010 by Mentor Graphics) as an open, vendor-neutral, data format to address the need for an intelligent data set that could be assembled from PCB design data and man-

ufacturing process rules and data, which then provide the vehicle for DFM analysis and manufacturing process preparation. A few years after the introduction of ODB++, PCB design companies, fabricators and contract manufacturers found ODB++ allowed for an easy interchange of data between them.

ODB++ Delivers Results in the Real World

ODB++ is a directory structure with an index, thus avoiding the need for a single huge file. This allows the CAM or DFM tools to load only the data needed for any specific processing application or task – a very useful characteristic in these times when PCB design data density is outpacing the processing capabilities of typical CAD/CAM hardware platforms.

With ODB++, the manufacturing engineer can, without having to first rebuild the model of the PCB assembly from multiple disjointed

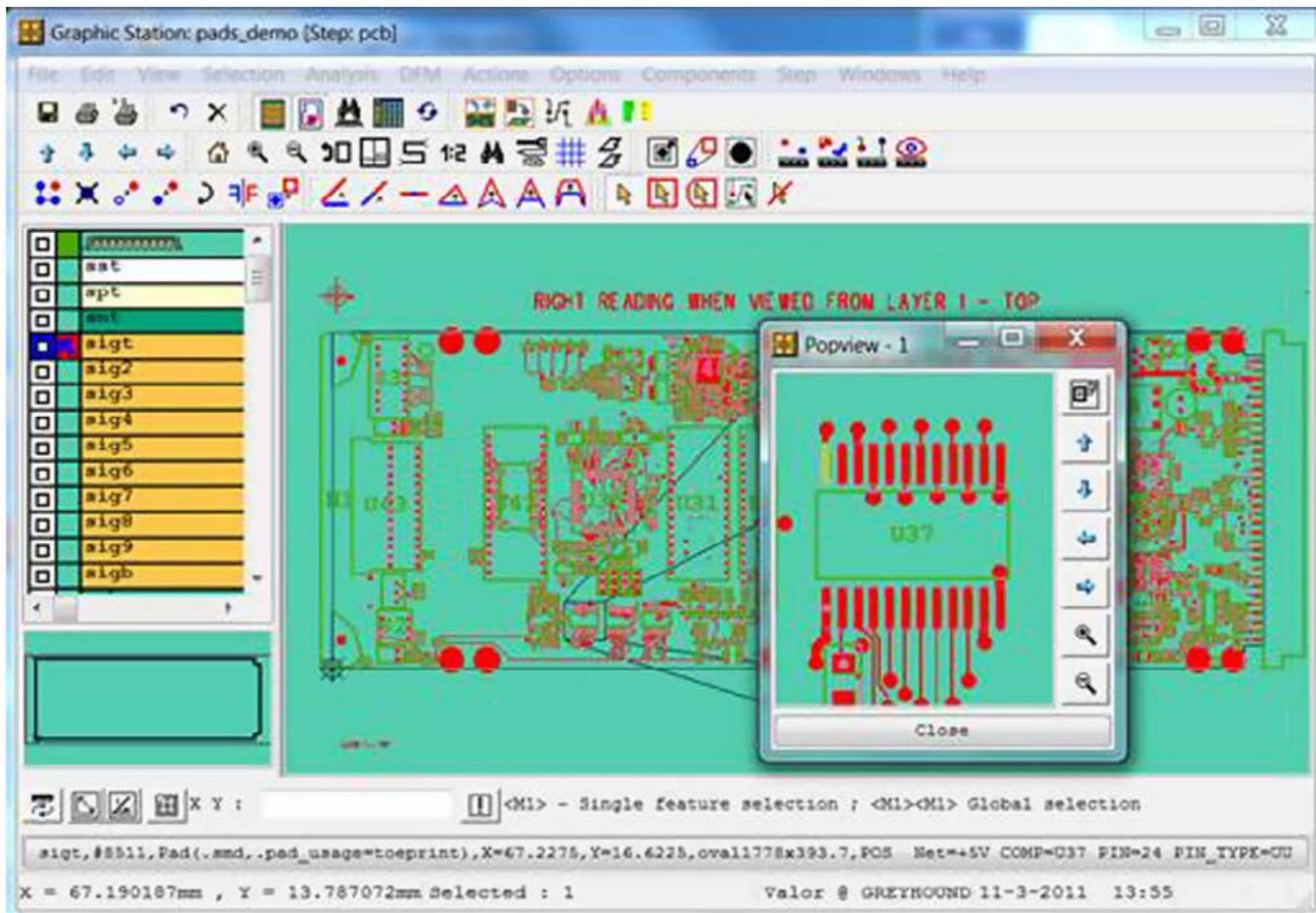


Figure 2: With the intelligent data in ODB++, a full set of information is available. In this figure the RefDes, part number, CAD package, number of pins, XY location, length and width, height, rotation and pin pitch are all shown, and pin number 24 of component U37 is identified as connected to netname +5V.

files, see the same type of objects that the PCB designer used, thus allowing much easier analysis. For example, DFM tools can import data from a wide range of EDA systems, either by importing native CAD formats or by importing the ODB++ that was directly created in the EDA system.

Once an ODB++ model of the product has been created, an engineer can display a layer from the buildup, for example, the “components top,” and then select a component and see not just an outline of the component pad as with non-intelligent data, but specific information (intelligence) such as the RefDes, part number, CAD package, number of pins, XY location, length and width, height, rotation and pin pitch. These are all critical data to the

manufacturing process and simply not available without manual intervention when using non-intelligent data. Selecting a component pad and requesting information displays the full information, as shown in Figure 2.

Essential to the PCB new product introduction (NPI) step is the ability to use data sets assembled during the PCB design phase and to relate that data to the intended manufacturing process. Until fairly recently, data sets used in the PCB design were often incomplete, in terms of not being able to fully support the manufacturing hand-off process. Frequently, a great deal of manual translation was required to do any kind of DFM analysis or manufacturing process preparation. That need gave rise to the invention of the ODB++ format.

OPTIMIZING PCB NEW PRODUCT INTRODUCTION USING ODB++ *continues*

ODB++ is the most intelligent proven CAD/CAM format available today, capturing the complete PCB fabrication, assembly and test knowledge in a single, unified database. ODB++ is an ASCII, fully expandable open format that can capture and store all the information needed for the manufacturing and assembly of a printed circuit board, imported directly from the CAD database and other sources such as PLM and DFM systems. This information includes layer graphics like pads and drills, test points, fiducials, components, netlists, and even any additional documents that may be needed. Data representation is fully WYSIWYG, so displayed objects are exactly the same as they will appear in the manufacturing process.

In addition to the information imported from the CAD system, ODB++ can also store data generated by the DFM application itself, such as the parts list, analysis results or a DFM report. Once stored in the ODB++ file, a standard function from within a graphical editor can be used to view the specific or variant related parts list. It is even possible to store some design specific datasheets or documents in the “user” section of the ODB++ file by using simple drag and drop in a file browser. Thus, the need for parallel files, such as drawings, is eliminated; the risk of mistakes is reduced because users set

themselves up for a “right first time” hand-off to manufacturing.

Summary

Since initial development in the mid-1990s, ODB++ has become the mainstream solution for design-to-manufacturing hand-off for many of the world’s leading electronics OEMs, with thousands of PCB designs being processed in the format every year. Leading manufacturers report up to 80% of their incoming work arriving in ODB++ format, enabling them to focus on the added-value tasks of DFM validation and manufacturing preparation, instead of first having to decode and reintegrate all the legacy files that represent the only proven alternative.

ODB++ is available as a format open to the industry in general, via the ODB++ Solutions Alliance (www.odb-sa.com). Dozens of Solutions Development Partners have supported the format for years, and the alliance has thousands of members who stay connected with its evolution. Since 1995, ODB++ has been under the stewardship of Valor, benefiting from a large multi-vendor user base and continuous investment in improvements to the format and the community of tools that use the data content. This aggregated investment over 15+ years by designers, multiple CAD/CAM software vendors, PCB

“
Once stored in
the ODB++ file,
a standard function
from within a
graphical editor
can be used to
view the specific
or variant related
parts list.”

Format	Supports fabrication?	Supports assembly?	Supports test?	Supported by all leading EDA vendors?	Formal standard?	Years of industry adoption	Industry adoption level
Gerber	partial	no	No	yes	yes	30+	High
IPC2581	yes	yes	yes	no	yes	0	Low
IPCD350	yes	no	No	no	yes	20+	Low
IPCD356	partial	no	No	yes	yes	20+	High
ODB++	yes	yes	yes	yes	De facto	17	High

Table 1.

OPTIMIZING PCB NEW PRODUCT INTRODUCTION USING ODB++ *continues*

fabrication and assembly manufacturers will continue as before to deliver benefits to the industry as the format and tools evolve.

It can accurately be stated that the industry in general has voted for ODB++ as the intelligent PCB manufacturing data format that really works and can be relied upon to deliver time, cost and quality advantages in practical daily reality. From time to time, new formats are suggested, but so far they have not gained critical mass in the market due to not providing significant marginal technical advantage, or requiring an industry-wide investment that cannot be justified, or some combination of the two.

From the viewpoint of the ODB++ Solutions Alliance, the different formats could be compared as follows in Table 1.

Quantified data available at www.odb-sa.com illustrates the main barrier that stands in the way of progress in moving from the legacy formats such as Gerber, component placement lists, and Excellon, to intelligent data such as ODB++: A lack of awareness of the benefits at

the point of generation. While the largest OEM designers have high levels of ODB++ implementation achieved over many years, a large diversity of PCB designers have only a basic notion of the format's benefits, and they need more information in order to move from their current process into intelligent data.

The industry does not need any more data transfer formats. On the contrary, the maximum efficiency gains can be achieved at the minimum total cost to everyone by implementing existing industry-proven solutions. **PCBDESIGN**



Julian Coates is the current director of the ODB++ Solutions Alliance. Julian has held a number of marketing management positions. He holds a degree in engineering science from Exeter University (1979), UK.

video interview

Gary Ferrari Updates IPC-2221 Design Standard

by Real Time with...
IPC APEX EXPO



Gary Ferrari, Director of Technical Support at FTG Circuits and Designers Council member, speaks with Guest Editor Bob Neves about the activities of the Council and the upcoming revision to the IPC-2221 Design Standard.



realtimewith.com



Gathering Steam (Finally): IPC-2581

by Hemant Shah and Edward B. Acheson
CADENCE DESIGN SYSTEMS

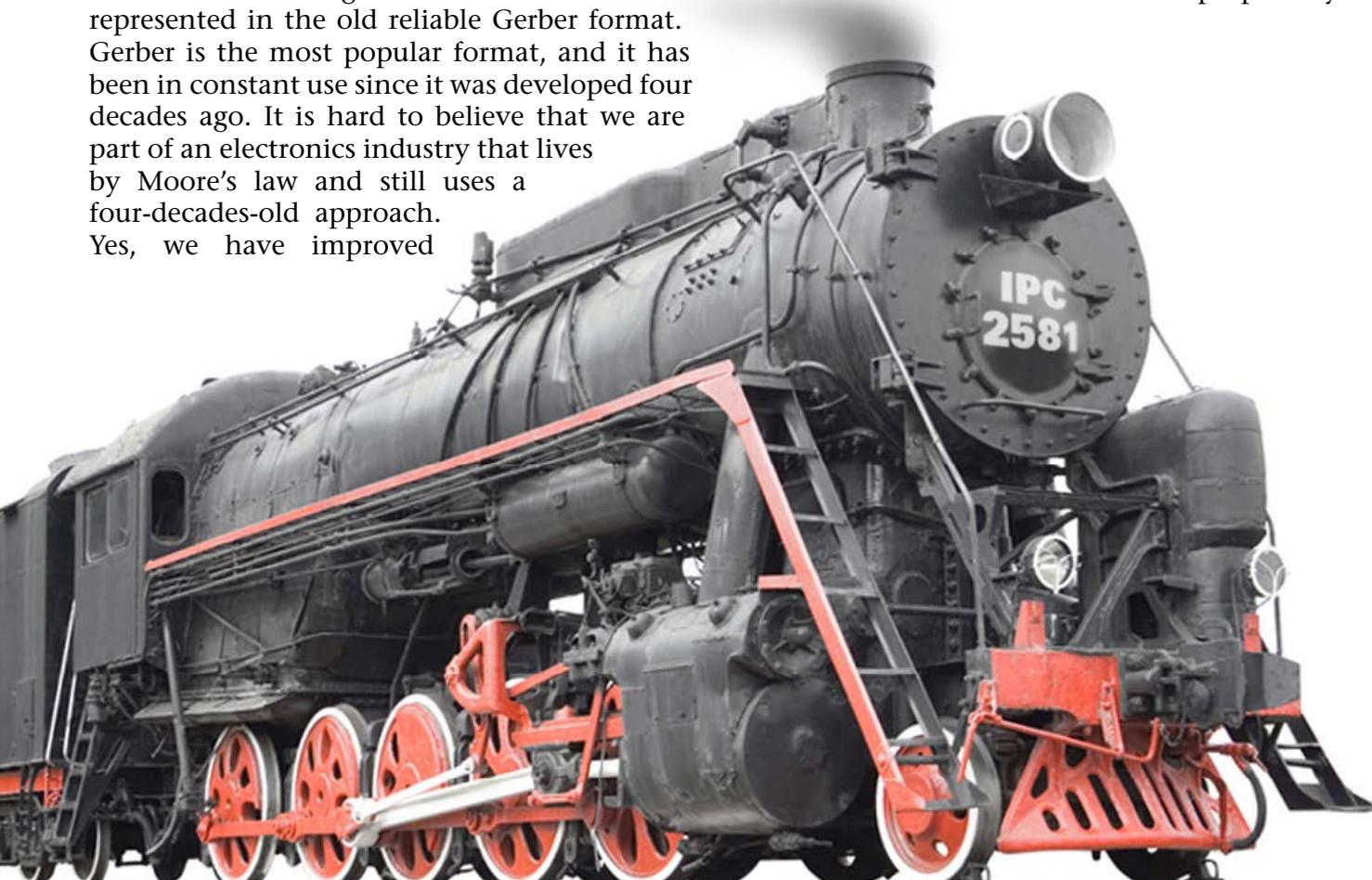
SUMMARY: *The PCB design community faces three potential paths regarding design data transfer. The first is to continue to use Gerber, plus a myriad of files that communicate manufacturing intent. The second is to move to ODB++. The third is to adopt an open, neutral, global standard: IPC-2581. This article provides an update on IPC-2581 and the activities of the IPC-2581 Consortium.*

For decades, efficient PCB design data transfer has been a misnomer. Most designers rely on a set of files that augment data that cannot be represented in the old reliable Gerber format. Gerber is the most popular format, and it has been in constant use since it was developed four decades ago. It is hard to believe that we are part of an electronics industry that lives by Moore's law and still uses a four-decades-old approach. Yes, we have improved

ways of communicating additional information that is not part of the Gerber file to describe the etch on each layer of the PCB. But throughout the years, that approach has resulted in a very inefficient way of transferring PCB design data.

Some people in the industry support ODB++. Yes, this is a better method than Gerber. However, the adoption of ODB++ has been minimal, and many customers who send ODB++ data require their fabricators to use Gerber data files to produce the boards. This was a step in the right direction, but it didn't quite get us there.

With the acquisition of Valor Computerized Systems by Mentor Graphics, ODB++ became a proprietary



POWER TOOLS FROM BAY AREA CIRCUITS

DFM REPORTING TOOL

Eliminate errors before they happen! With just one click, you can upload your file and in a few minutes you will have a full six page report on how a PCB Manufacturer would see your files.

[TRY IT](#)

ARRAY CALCULATOR

It's never been easier to find the maximum fit for your board in an array form. This tool will also make a PDF drawing that can be included with your data to the Manufacturer and Assembler.

[TRY IT](#)

MULTI PART PCB SPECIAL

2 Layers, 5 Days, \$300, Guaranteed, Up to 4 different part numbers

For \$300 you can use up to 352 square inches any way you like (one board or as many that will fit). Up to 4 different designs will run together on this special!

[DETAILS](#)



Quality delivered. On time.

Need a quote? Call toll free 877-422-2472 

Local 650-367-8444 

575 Price Avenue, Redwood City, CA 94063

GATHERING STEAM (FINALLY): IPC-2581 *continues*

format driven by Mentor's business objectives. Why wasn't it proprietary when Valor owned it? It was. But Valor was CAD vendor-independent. Valor provided open access to the ODB++ format with no agreements, no questions asked, no terms to sign; no one could pull the plug on a company's products with a notice to cease and desist within six months.

There is a clear alternative available to the industry – the open, neutral, global standard: IPC-2581. This standard, as many already know, was the result of merging two competing standards in mid 2000s: GENCAM and ODB++. Valor, at the time, donated the ODB++ format to IPC. However, very few software companies actually supported ODB++ after the merged standard was created. Led by Cadence, a group of 12 founding members created a consortium of design and supply chain companies in mid-2011 to get IPC-2581 adopted.

The IPC-2581 Consortium has grown to 42 members. Members represent the following:

- Systems companies: Cisco, EMC, Fujitsu Network Communications, Harris, NVIDIA, Orbital, Qlogic, Qualcomm, and Velus.
- EDA companies: Cadence and Zuken.
- DFM software companies: ADIVA, DownStream Technologies, EasyLogix, Iron Atom, WISE Software, and Numerical Innovation.
- Manufacturing companies: Axiom, CC Electronics, Sanmina-SCI, Screaming Circuits, Sierra Circuits, Newbury Electronics, JD Photodata.
- Software companies that work with manufacturing equipment: Vayo and Ucamco.

Other member companies represent various parts of the PCB design and supply chain.

The Benefits of IPC-2581

For starters, it is an open, neutral, global standard that is driven by the design and supply chain companies' needs. It is not controlled by one CAD tool vendor, and it allows for design and supply chain companies to openly collaborate, innovate and improve the standard. IPC-

2581 doesn't require any agreements to use. It doesn't impose any conditions for usage.

Usage of IPC-2581 improves efficiency in PCB design data transfer to fabrication, assembly and test. On the manufacturing side, data doesn't have to be compiled from various files and data sources. It allows design and supply chain companies to improve on the standard faster. Now, this may sound counterintuitive, but IPC and the IPC-2581 Consortium members have accelerated the process of improving on this particular standard. This standard reduces unnecessary iterations between systems companies and manufacturers, and it's less error-prone because no translation of the data is required to manufacture the PCB.

"As a photoplotting company, we have been involved with receiving hundreds of packets of data on a weekly basis. It sometimes seems like no two companies use the same nomenclature, no two use the same methods to transmit," said John Dingley, business development manager of JD Photo-Tools. "It's for that reason that we welcome the continued development and implementation of IPC-2581. To have one file format that we can import in a single click, then have all of the build setup and manufacturing data ready to go, means that we can now spend less time on sorting data, giving a faster response to the customer."

The Consortium's Progress

When the consortium was created, EDA companies, DFM companies and manufacturing software vendors did not support IPC-2581. As of Jan 1, 2013, six software vendors support the IPC-2581 format. A complete status on support of this standard is shown in Figure 1.

Besides a steady growth in consortium membership, there has been a concerted effort to have software companies validate the IPC-2581 format. Consortium members have been actively reviewing and discussing the need to do more with the IPC-2581 spec. At an IPC meeting in mid-2012, consortium members proposed 16 changes to the specification to improve efficiency, reduce ambiguity and do more with the standard to improve collaboration between systems' companies and their manufacturing partners.

Company Name	Software Name & Release	Support Status	IPC-2581 Release
ADIVA	ADIVAnet Revision 8.7	Q1 2013	IPC-2581A
	ADIVADRC Revision 8.7	Q1 2013	
	ADIVAviiew Revision 8.7	Q1 2013	
Cadence	Allegro PCB Designer 16.5/ 16.6	YES	IPC-2581 Rev1 and IPC-2581A
	OrCAD PCB Designer 16.6	YES	
Downstream Technologies	CAM350	December 1, 2012	IPC-2581A
	Blueprint PCB 3.2	YES	
	DFMStream	December 1, 2012	
EasyLogix	PCB-Investigator 3.4.4	YES	IPC-2581 Rev1 and IPC-2581A
Numerical Innovations	FAB 3000 Version 7	January 1, 2013	To be decided..
	ACE 3000 Version 7.1	January 1, 2013	
	PreflightPCB Version 1.0	February 1, 2013	
Polar Instruments	Speedstack	After IPC-2581B release	IPC-2581B
Siemens	Test Expert 9.3	YES	IPC-2581 Rev1
	UniCam FX 9.2	YES	
	UniDoc FX 9.2	YES	
Ucamco	Ucam V10.1 + 3months ARO	YES	IPC-2581A
	UcamX V1.1 + 3 months ARO	YES	
	Integr8tor V7.3 + 3 months ARO	YES	
Vayo	Vayopro	January 1, 2013	IPC-2581A
WISE	VisualCAM V16.3	YES	IPC-2581A
	GerbTool V16.2	YES	
	WISE2581Viewer V16.2	YES	
Zuken	CR-5000 Board Designer 13.0/14.0/15.0	YES	IPC-2581 Rev1
	CR-8000 Design Force 2013	January 1, 2013	
	CR-8000 DFM Center 13.0/14.0/2012	YES	

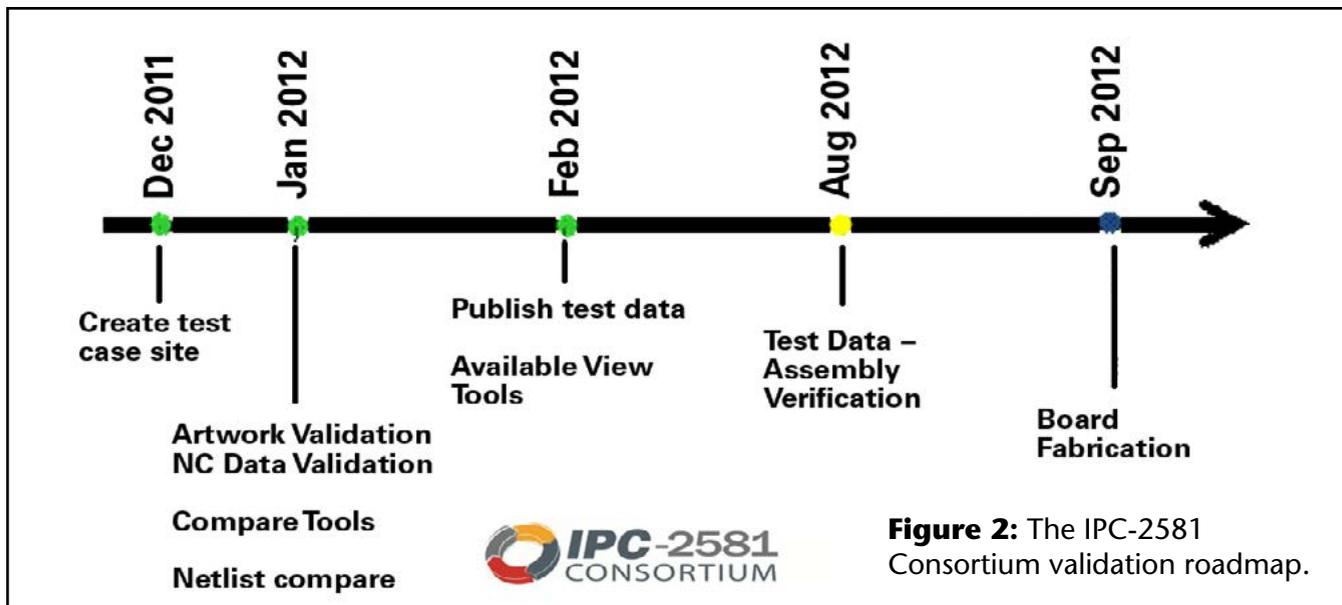
Figure 1: IPC-2581 software support status. (IPC-2581 Rev1, Amendment 1, was released in May 2007. IPC-2581A was released in May 2012, and IPC-2581B will be released in early 2013.)

IPC-2581 Consortium features several working groups, one of which is the technical working group. The technical group's focus is two-fold: 1. Validate that the IPC-2581 data is being produced and interpreted correctly by tools from EDA companies, DFM companies and manufacturing software companies; 2. To develop proposals to enhance the standard based on the needs of the design and supply chain companies.

In June 2012, the technical group reported on the progress of validating bare board manufacturing data defined in the IPC-2581 data standard. It compares data contained in the IPC-2581 files against data in existing formats used today.

Consortium members reached a key milestone last September when Fujitsu Networks Communications fabricated one of its current PCBs using IPC-2581. This board was presented at PCB West in September. The design fabricated is a 10.5" x 8.5" optical plug-in module provided by Fujitsu Network Communications. It features 12 layers (8 signal/4 split planes) and 5,789 vias.

The IPC-2581 data was output by Cadence Allegro PCB Editor and the design was checked and CAM output was provided by WISE Software's VisualCAM tool. The board was fabricated by CC Electronics (Figure 3). CC Electronics was able to reduce setup time by 30% with the

GATHERING STEAM (FINALLY): IPC-2581 *continues*

use of IPC-2581-based data. No iterations were required between CC Electronics and Fujitsu.

“As a manufacturer of printed circuit boards for many years, we have seen a multitude of data formats presented to us,” said Phil Wain, senior IT and front-end engineer with CC Electronics Europe. “IPC-2581 is the next-generation data format and aims to address all the above issues whilst remaining an open and neutral format.”

In November of 2012, Siemens followed by

completing validation of IPC-2581 assembly and test data. Siemens also compared the IPC-2581 data to data formats typically used in today’s PCB manufacturing processes.

Ongoing Technical Activities

IPC-2581 format is designed to be a complete data set that is vendor- and tool-independent. Because of this neutrality, IPC-2581 provides the ability to incorporate multiple types of data that are targeted to the PCB manufacturing processes. This standard is designed in such a way that data is combined into certain categories related to the various processes in the fabrication of a PCB. These categories are divided into five functions: design, fabrication, assembly, test, and full. By selecting a function, the creation tool can provide a data set related to a targeted process. Users may also specify the data to be exported based on their own requirements, as well. Use of these capabilities will reduce the exchange of data not intended for other processes.

Recognizing IPC-2581’s extendable capability, the technical committee, working in conjunction with other IPC groups, is proposing additional data not included in other formats to be included into the IPC-2581 file data. A top priority being explored by the IPC-2581 Consortium, in partnership with the IPC Standards Working Committee, is to introduce full cross-section definition for the next version of

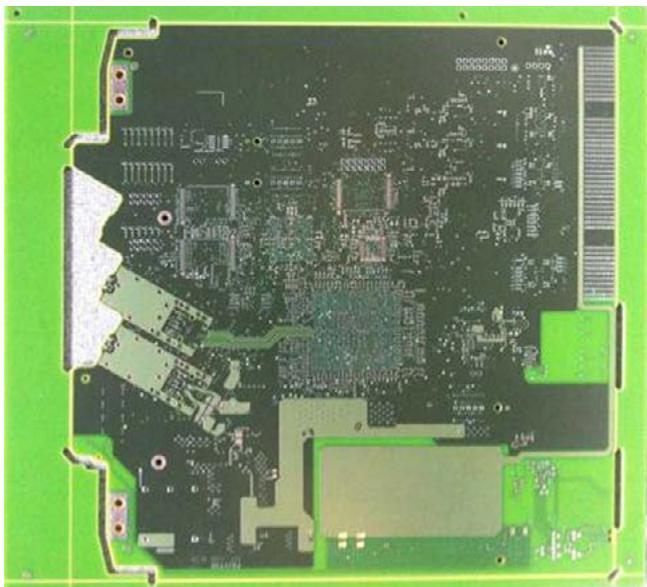


Figure 3: First PCB fabricated using IPC-2581 format. (Image of optical plug-in module courtesy of Fujitsu Network Communications.)

IPC-2581, Rev B. While the current IPC-2581A standard is sufficient for fabricating PCBs, the standard can be enhanced to better exchange design and manufacturing intent.

Using the IPC-2581 format, PCB stackup data may be exchanged between cross-section analysis tools, passing recommended stackup scenarios into CAD tools to get proper impedances in a design as opposed to creating and translating spreadsheets. A CAD tool could also output a stackup to pass on to a board fabricator to define what materials are required to give fabricators proper lead time to acquire resources and ensure design intent. The data contained in the file would be similar to the typical cross-section description found in fabrication drawing today, with perhaps more detail.

IPC-2581 Consortium member companies are working closely with IPC to make enhancements to the format. These discussions that started with the mid-2012 meeting have been accelerated and are expected to be part of an updated standard in early 2013.

As technology progresses, and as long as the data can be described, the IPC-2581 format has the potential to grow as needs dictate. Members of the IPC-2581 Working Standards Committee, who represent many aspects of design centers and tool providers, are continually looking at how the standard can provide all of the data needed to successfully communicate full design intent.

Conclusion

The PCB design and manufacturing industry is in dire need of an improved way to manage design data transfers. Depending on a proprietary format or a decades-old approach is not an option. IPC-2581 is an open, neutral, global standard that is driven by design and supply chain companies. An open standard fosters innovation and collaboration not possible when one company controls the standard. Furthermore, the IPC-2581 Consortium has made more

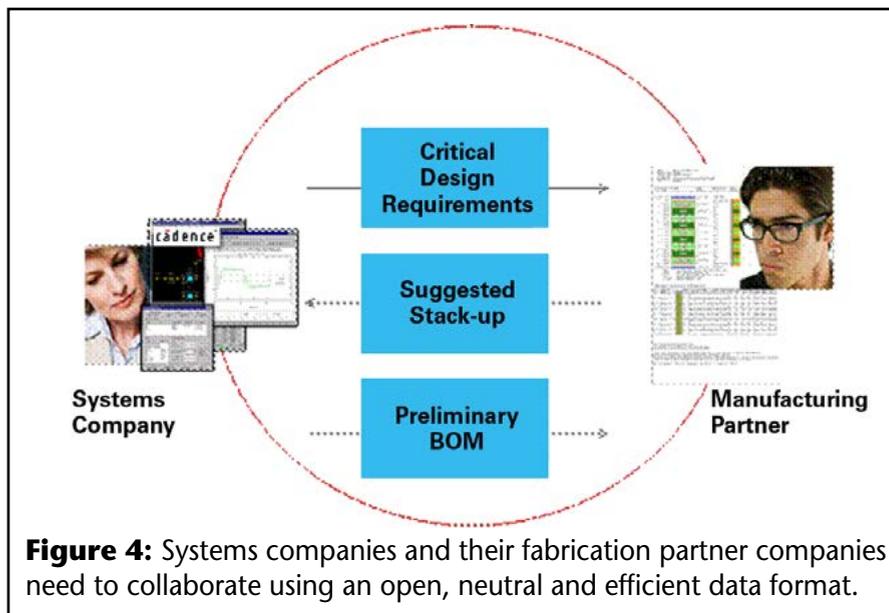


Figure 4: Systems companies and their fabrication partner companies need to collaborate using an open, neutral and efficient data format.

progress in the past year than was made in the past seven years.

It's easy to join the IPC-2581 Consortium. Find out about your supplier's plans to support IPC-2581. With so many software companies supporting the PCB design and manufacturing community, your supplier may already be equipped to supply or accept IPC-2581. Visit www.ipc2581.com to join. **PCBDESIGN**



Hemant Shah is director of product management for Allegro PCB and FPGA products at Cadence. Prior to joining Cadence, he worked at Xynetix Inc. and Intergraph Corporation. Hemant has a BS in EE from BITS Pilani, India and a MS CS from University of Texas at Arlington.



Edward B Acheson is a principal product engineer for Allegro PCB products at Cadence Design Systems focusing on DFM, ECAD-MCAD as well as IPC-2581 Consortium. Ed joined Cadence (Valid Logic at the time) in 1990 and has played several interesting roles including software developer, consulting and process services engineer.

What is a Circuit Board?

by Jack Olson

SUMMARY: *Before starting the design process, we need to have a clear goal in mind. What exactly does a circuit board designer create? This will be a basic introduction to circuit boards from the beginning, exploring the design process step-by-step and connecting the dots along the way.*

What is a circuit board? Doesn't that seem like a silly question to ask in a publication like this? Our readers are involved in all aspects of circuit board development, and many of us have devoted our entire careers to them. But what if you're new to this industry? Where can you go to learn the basics?

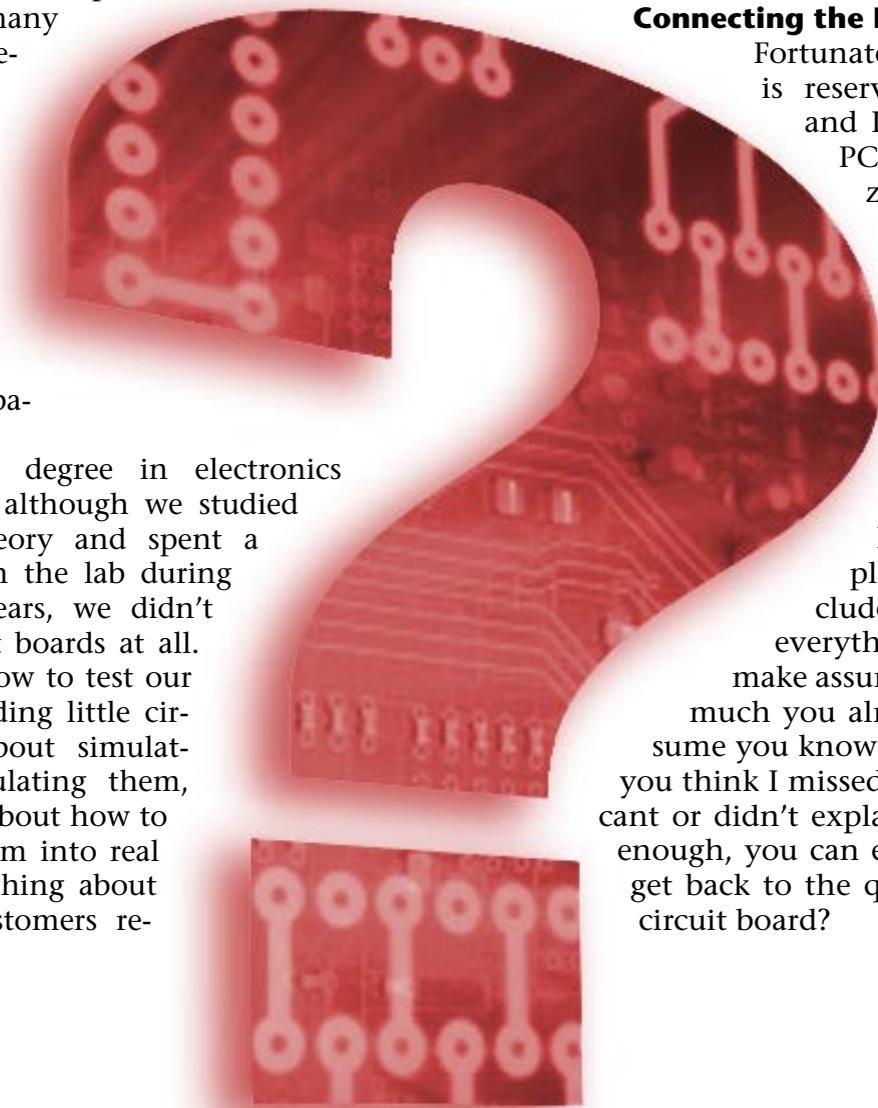
I have a degree in electronics (BSEET), and although we studied electronic theory and spent a lot of time in the lab during those four years, we didn't discuss circuit boards at all. We learned how to test our ideas by building little circuits, and about simulating and emulating them, but nothing about how to transform them into real products. Nothing about what our customers re-

ally need in a production environment. Sure, there's a lot of information available on the Internet, not to mention on-the-job training, and private training, technical conferences, user groups, and publications like this. But many of these resources assume you already know something about the subject. Sometimes it seems as if there is simply too much material to wade through.

Where do you start?

Connecting the Dots

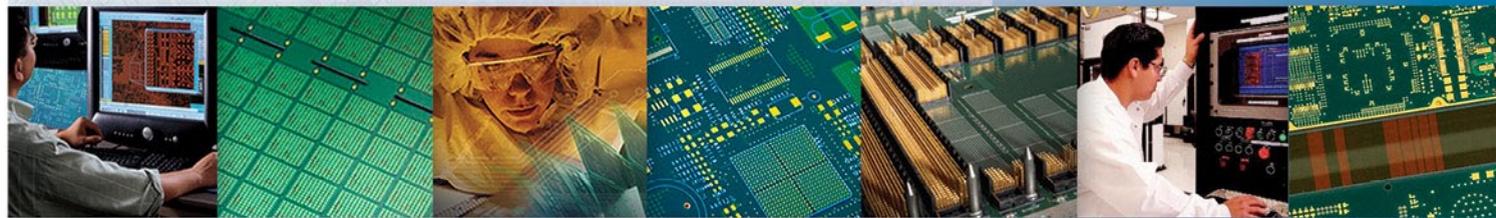
Fortunately, this column is reserved for beginners, and I'm grateful to The PCB Design Magazine for providing it. We now have a blank slate. We can learn step by step. The benefit I have over many other technical writers is that I don't have to decide how much to explain, or what to exclude. I'll try to explain everything. I don't have to make assumptions about how much you already know; I'll assume you know nothing. In fact, if you think I missed something significant or didn't explain something well enough, you can e-mail me. But let's get back to the question: What is a circuit board?



GLOBAL PRESENCE LOCAL KNOWLEDGE

ADVANCED TECHNOLOGY PRINTED CIRCUIT BOARDS & ASSEMBLIES

ISO 9000
ISO 14000
AS 9100



- Advanced HDI
- Flex, Rigid-Flex & Flex Assembly
- High Layer Count / Large Format PCBs
- RF & Microwave Circuits
- Specialty Materials
- Thermal Management
- Custom Backplane Assemblies & Systems

TL 9000
NADCAP
IPC-A-600 CLASS 3
ROHS COMPLIANT
ITAR COMPLIANT
UL RECOGNIZED
MIL-P50884
MIL-P-55110
MIL-PRF-31032

REQUEST TECHNICAL INFORMATION

REQUEST A QUOTE

www.ttmtech.com

TTM Technologies

15 Specialized Facilities – USA, China, Hong Kong

WHAT IS A CIRCUIT BOARD? *continues***Terminology**

Here's the official definition from the IPC-T-50 publication "Terms and Definitions for Interconnecting and Packaging Electronic Circuits" Revision J:

Printed Board (PB)

The general term is for completely processed printed circuit and printed wiring configurations. (This includes single-sided, double-sided and multilayer boards with rigid, flexible, and rigid-flex base materials.)

OK, that didn't teach us very much about what a circuit board is, but it used the term "printed circuit," so let's get that one, too:

Printed Circuit 60.0912

A conductive pattern that is composed of printed components, printed wiring, discrete wiring, or a combination thereof, that is formed in a predetermined arrangement on a common base. (This is also a generic term that is used to describe a printed board that is produced by any of a number of techniques.)

I often turn to IPC when I want to understand something better, and while IPC definitions are technically correct, they just don't create a very good mental picture of what a circuit board is. I'll try to explain more and add a picture or two, but first I need to mention something about terminology. IPC has traditionally used the term "printed circuit boards" or the abbreviation PCB, but the organization has recently been replacing those references with the term "printed boards," or PB.

The reason I prefer "circuit boards" is that I'm not sure boards should be thought of as printed (they are usually etched), and the term "printed board" may soon become confused with the newer "printed electronics." Printed electronic circuits are truly printed, and that industry is now maturing rapidly. And the term PCB can also be confused with the other kind

of PCBs that are environmentally harmful, and PB can be confused with the atomic symbol for lead or even peanut butter.

Seriously, I am a strong proponent of clear communication and the use of unambiguous terms, and I applaud IPC for its efforts to standardize, but in this case I prefer the term "circuit board." Regardless, I predict that the widely used acronym PCB will be used forever, so you should know that "PCB layout" means the same thing as "circuit board layout." We'll learn more about IPC later, but right now we have to get back to discovering what a circuit board is.

The Simplest Circuit Board

From the definitions above we learned that a circuit board is a conductive pattern formed in a predetermined arrangement on a common base. The conductive pattern can be as simple as a single layer of copper, with portions of it etched away to leave the desired connectivity. This is called a "subtractive process" because the material starts out as a full sheet of copper, which is relatively inexpensive and is a good conductor of electricity, and then the unwanted areas are etched away to leave the conductive pattern. The copper used in this process is usually very thin, so it needs to be supported on some type of insulating material, which is the "common base" mentioned in the IPC definition.

In addition to providing support for the flexible copper pattern, the base also provides mechanical support for the electronic components that will be mounted to it. This insulating material is most commonly a thermally cured flame-retardant fiberglass, which is an organic resin system reinforced by one or more layers of glass fibers woven together like cloth.

The most common form of this base material is called FR-4 (flame-retardant), but many other materials are available and each has its own specific properties. For example, some ma-

“
**The reason I prefer
 “circuit boards” is that
 I’m not sure boards should
 be thought of as printed
 (they are usually etched),
 and the term “printed
 board” may soon become
 confused with the newer
 “printed electronics.”
 Printed electronic circuits
 are truly printed, and
 that industry is now
 maturing rapidly.**
 ”

materials are more stable at higher temperatures, while others are better for high-speed circuits and other can flex continuously, for example. We'll learn more about material properties later, but for now we only need to understand that the simplest circuit board is made from a single-sided laminate material, which is a single layer of copper bonded to a base material of the desired thickness. Copper-clad laminates are typically 36 x 48 inches, which are then cut down to 18 x 24 inches for the bare board fabrication process.

Multiple-Layer Boards

Single-sided boards are used for simple circuits in very inexpensive, high-volume products like toys or smoke detectors. Holes can be drilled in the board for leaded components to be inserted from the other side and soldered to the copper conductive pattern, without having to add the expense of plating the holes. Surface-mount components can be soldered directly to the conductive pattern. It is difficult to design a circuit of any complexity on a single layer without using a lot of jumpers, so instead of using a single-sided laminate, it is more common to start with a laminate that has been coated with copper on both sides.

By using a laminate that has copper bonded to both sides, the bare board fabricator can etch a different conductive pattern on each side and connect them together with plated-through holes, which are formed by drilling holes through the laminate and then using a

plating process to deposit copper on the hole walls. This technology allows copper-clad laminate material to be processed in "layer pairs," with the ability to place conductive paths connecting the layers together wherever they are needed. It should be obvious that more complicated circuit designs can be accomplished by using double-sided boards with plated-through-holes, but let's take this idea one step further. Why not process several layer pairs with different conductive patterns, insert more insulating sheets of material between the layer pairs, laminate them all together, and then process the outside layers just like a double-sided board?

The hole drilling and plating process could connect all the layers together, and in this way we could make boards that are 4-layer, 6-layer, 8-layer, 10-layer, etc. The advent of multilayer boards has opened the door to designing extremely complex electronic circuitry, adding layer pairs as needed to make all of the necessary connections. OK, that was a fast-paced and all-too-brief introduction to multi-layer technology, but we need to move on. We can dive into the details of board fabrication later.

The Rest of the Story

A casual glance at almost any modern circuit board will show more than just copper conductive patterns on insulating material. What's the rest of the story here? To continue our general overview of the common circuit board, it might be helpful to have a picture:

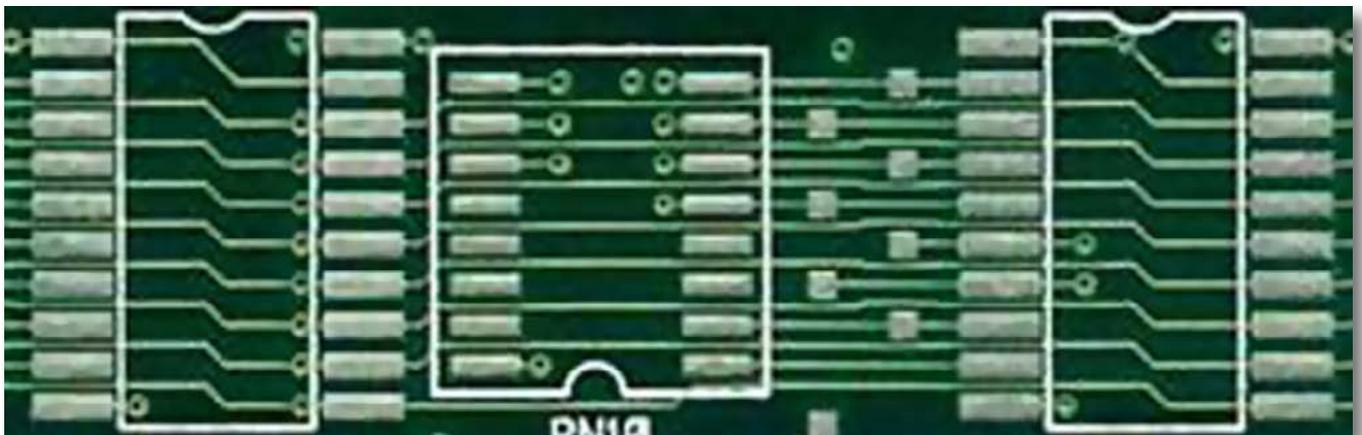


Figure 1: A typical circuit board.

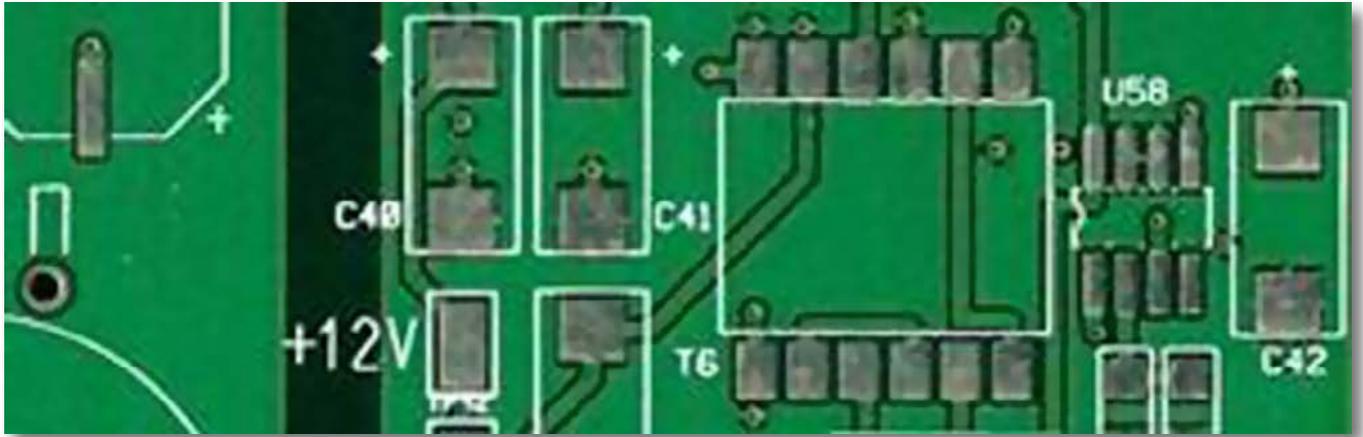
WHAT IS A CIRCUIT BOARD? *continues*

Figure 2: Another view of the same board.

Figure 1 shows one small area of a circuit board I designed many years ago. You can see that the conductive pattern has been formed so that three components can be soldered to it. You can see the connections between the component land patterns, some plated holes called vias, and a few square areas for test points. There are three other important things to notice in this picture:

1 Green soldermask has been applied to the surface of the board, leaving openings in areas that will be soldered and for test point accessibility. This protects the outside layers and also makes the soldering process more reliable.

2 White ink is commonly called silkscreen, and provides identification of parts by reference designator, and sometimes includes component outlines and other info.

3 Bare copper will oxidize quickly (like a new copper penny turning brown), which makes it difficult if not impossible to solder. A final finish has been applied as a protective coating over the exposed copper, keeping it solderable. In this example, it is an alloy of tin and lead (eutectic solder coat applied in a process called hot air solder levelling, or HASL), but it could also have been silver, gold, tin, an organic solder preservative (OSP), or some other coating.

Here's another area of the same board to show some variation. This part of the design has large copper areas on the surface of the board and uses thicker connections between components (Figure 2).

So that's what a circuit board is. Conductive lines that connect land patterns together are called traces. Larger conductive areas are called planes, which are also commonly referred to as copper pours or area fills.

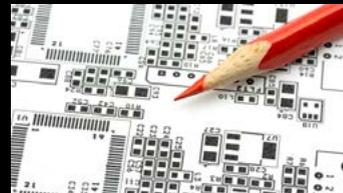
Notice that the silkscreen can be used to provide extra nomenclature (like the "+12V" label in the photo above), which may help people in the lab or during field service. It's a good design practice to consider the needs of others who may be using your product long after you have moved on. Try to include information on the board that will be helpful to them, if possible.

See you next month! **PCBDESIGN**



Jack Olson C.I.D.+ has been designing circuit boards full-time for over 20 years. Jack would like to thank the Orange County PCAD User Group of 1987 for freely sharing so much knowledge and experience, and especially to Jack Miller of RoyoCAD for his sponsorship and support. He can be reached at PCBjack@gmail.com.

Most-Read PCB007 News Highlights



[Sunstone Circuits Opens New Oregon Office](#)

The new building, which houses customer service, finance, marketing, human resources and executive management, represents Sunstone's continued growth and commitment to serving customers 24/7/365.

[IPC APEX EXPO 2013 Highlights 35 Technical Sessions](#)

Over three days, 35 technical sessions with nearly 100 research papers will address additional topics such as military electronics, advanced packaging, fluxes and paste, embedded devices, high-frequency electronics, and ESD. In addition, the areas of rework/repair, cleaning, assembly reliability, solder and alloy reliability, printing and PE will have multiple sessions.

[IPC: N.A. PCB Shipments, Bookings Down in October](#)

Rigid PCB shipments were down 1.1% in October 2012 from October 2011, and bookings decreased 9.2% year over year. Year to date, rigid PCB shipments grew 4.4% and bookings decreased 0.2%. Compared to the previous month, rigid PCB shipments were down 12.4% and rigid bookings fell 14.3%.

[Stickleback Acquires D.E.B. Electronics](#)

PCB manufacturer D.E.B. Electronics, established more than 50 years ago, has enjoyed success with a wide customer base. Recently, the company suffered the effects of one of its largest customers moving its business to China. The family-run manufacturer was not able to recover from the loss and closed in the first week of December 2012.

[Thomas Edman Appointed President of TTM Technologies](#)

TTM Technologies Inc. has announced that Thomas Edman has been appointed as president of TTM Technologies Inc., effective January 7, 2013. Edman will report to Kent Alder, TTM's chief executive officer. Edman brings more than 20 years of executive experience and extensive elec-

tronics industry experience in the U.S. and Asia to his new role.

[IPC/JPCA-4591: Guidelines to Build Printed Circuits](#)

The concept of printing conductors onto a range of substrates has been around a long time, but it's only now beginning to see burgeoning acceptance. IPC is accelerating this growth with standards that will make it easier to specify and build printed electronic circuitry.

[Spirit Circuits Sees 20% Growth in November](#)

Hampshire, UK-based Spirit Circuits is reporting a 20% growth as they finish the year with a record month. November is the year-end for the PCB manufacturer and their records show they have achieved both the highest ever production output and their highest ever sales intake.

[Bare PCB Industry's Revenue Hits \\$5.1 Billion in 2011](#)

The industry's revenue for the year 2011 was reported at \$5.1 billion, with an estimated gross profit of 16.39%. Import was valued at \$1.9 billion from 78 countries. The industry also exported \$1.7 billion USD worth of merchandise to 127 countries.

[K&F Electronics Joins SPF Premier Partner Program](#)

Semblant announces that industry-leading PCB fabricator K&F Electronics has joined its Semblant Plasma Finish (SPF) Premier Partner Program. K&F Electronics joined the program to address growing OEM demand for PCB surface finish advancements.

[Becker & Müller Invests in New Blackhole Production Unit](#)

To better meet increasing market demands, PCB manufacturer Becker and Müller has invested in a new PILL unit for the Blackhole HT direct metallization process. This new production line enables the company to fulfill client requests at an even higher level of quality.

Dispense With the Gerbers Already

by **Amit Bahl**

SIERRA CIRCUITS

SUMMARY: *Most PCB designs are still expressed with Gerber graphic files, despite the clear advantages of the ODB++ intelligent unified file format. Nonetheless, IPC-2581, the open format for PCB description being hammered out by IPC, is likely to become the delivery vehicle of choice.*

Why should you care how data are output from your CAD tool after you've completed your design, provided that the design is sound and the description is thorough?

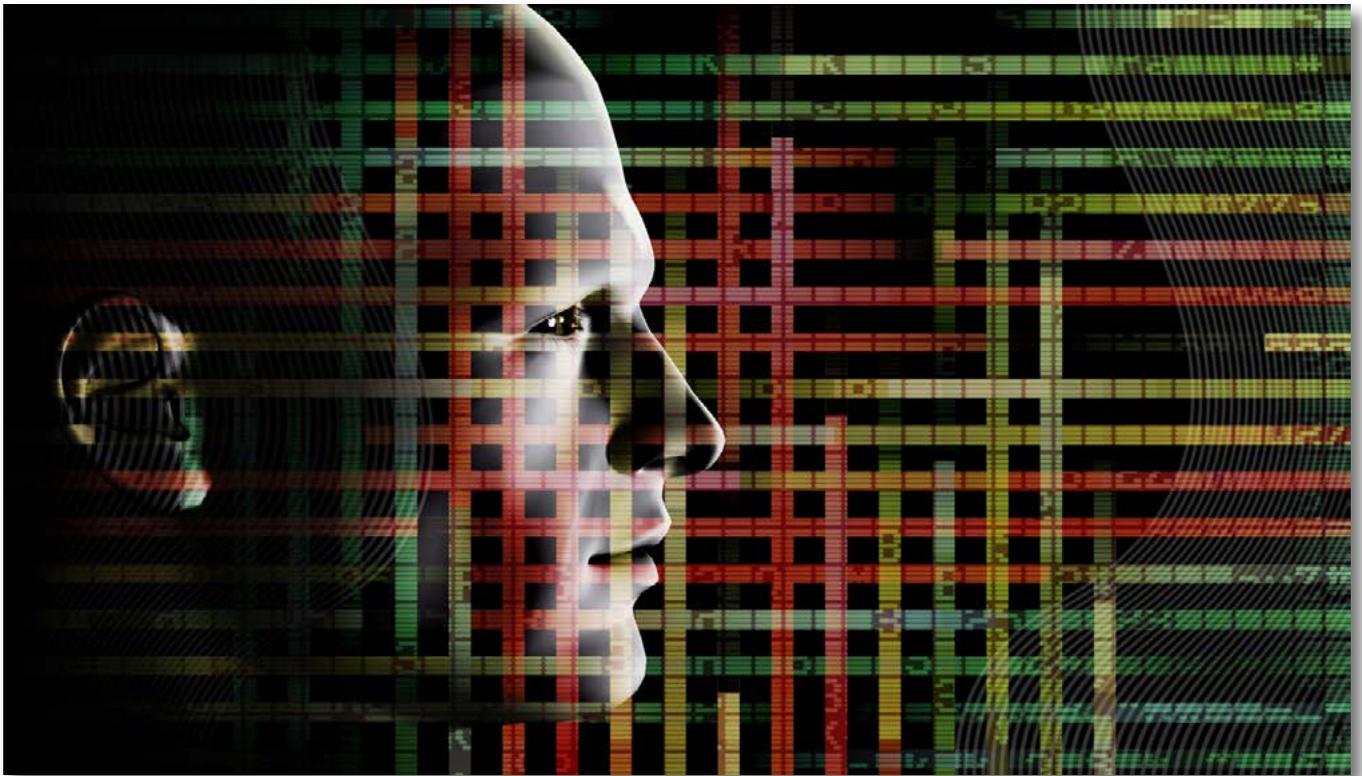
Whether you send Gerbers to your manufacturer (along with a drill file, netlist, BOM, board drawing and readme text), a zipped ODB++ file, or a file in the IPC-2581 format when it's finalized, what's the difference?

How your design data are formatted determines how easily – and perhaps how successfully – your manufacturer can interpret exactly what you intend to have built. Nearly

90% of the orders my company receives for fabrication and assembly are Gerber-based, even though it was more than 15 years ago that Valor introduced the ODB++ format for intelligently describing designs at the manufacturing level.

We would much rather receive ODB++ data, which our CAM tools can analyze in a fraction of the time required to convert and review Gerbers and their accompanying files. But of course, like other manufacturers, we are happy to accommodate our customers' preferences.

I'm not alone in emphasizing the benefits of the richer data in the ODB++ format when compared to Gerber data. Whereas Gerbers convey merely the outlines and locations of features layer by layer, ODB++ data identify the features and enable their sizes, shapes, and positions to be adjusted locally or globally to simplify manufacture and ensure good boards.



Lab Technicians Wear Out... Before **Theta** Circuit Materials Will!

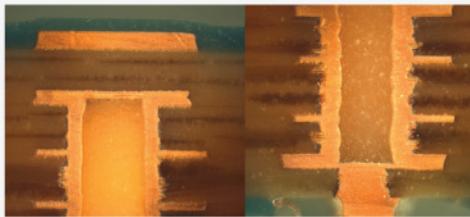
Test engineers will run out of steam trying to drive reliability failures in PCBs using Theta[®] circuit materials. Whether the designs are comprised of high layer counts with fine pitch BGAs and subjected to 260°C lead-free reflow or 3+N+3 stacked microvia designs subjected to thermal cycling, Theta materials continue to demonstrate robust thermal performance and outperform its competition.

Theta materials, environmentally friendly halogen free high-speed digital materials, deliver excellent electrical performance and superior CAF resistance. Whether your high-speed digital needs are lead-free assembly robustness, reliability for your next challenging stacked microvia designs, or CAF resistance, look no further than Theta materials.

Compare the **Advantages** of Theta Materials

- Excellent electrical performance: Dk of 3.8 and dissipation factor of 0.008 at 1 GHz
- Thermally robust: Tg of +180°C and Td of +370°C
- Reliable: 30% less z-axis expansion with temperature than high-Tg FR-4

Example of a Test Scenario



(Magnified @ 100x)

18 layer, .085" thick (2.16 mm), buried and single stack vias.
2 lamination cycles (1+N+1)

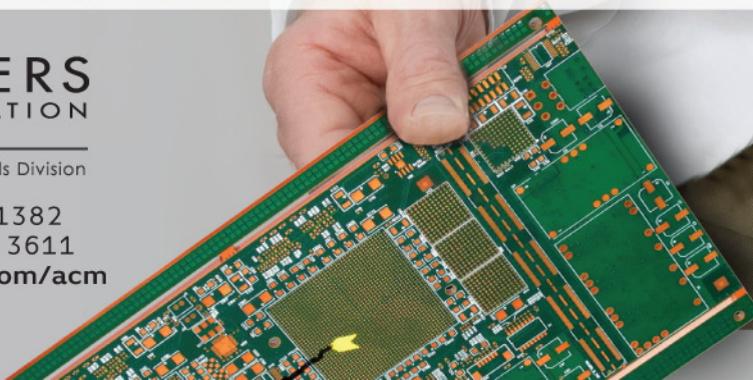
This design has passed:

- 10x solder floats @ 288°C (unconditioned)
- 10x reflow @ 260°C (JEDEC std. lead-free cycle),
- 6x 260°C preconditioning before 1000 cycles IST @ 150°C followed by 100 cycles @ 190°C
- T260 passed more than 30 min
- CTEz (50-260°C) of 3.1%



Advanced Circuit Materials Division

USA: +1 480-961-1382
Europe: +32 9 235 3611
www.rogerscorp.com/acm



DISPENSE WITH THE GERBERS ALREADY *continues*

Gerbers are no more than simple graphic representations, and are therefore hard to physically edit: A round pad is just a filled drawn circle completely independent of all other such circles. However, ODB++ defines pads as pads. For example, if necessary during CAM review, one, some, or all 10-mil pads can be enlarged to 12 mils.

Manufacturers do make slight adjustments in coordination with customers to preserve the integrity of designs. When our Valor system reveals something in a design that would compromise manufacture – such as traces that are too close together or holes that are too big for pads – and it could be resolved with a minor edit, the system automatically checks the edit against the netlist to confirm that change has no effect elsewhere. Each zipped ODB++ file, if complete, consolidates all the information needed for a board to be manufactured and assembled, and it can be directly loaded into the front-end CAM system, which will in turn output the programs to drive all the process equipment – nice, clean and easy.

When we receive a design described by Gerbers and the accompanying files, we convert these files so they can be loaded into our system for review. That takes time. You can appreciate why a quickturn manufacturer would favor production formats that streamline transferring designs to fabrication.

And in This Corner: IPC-2581

Like ODB++, the evolving IPC-2581 standard is an intelligent format for delivering all the data needed to automate PCB manufacturing, assembly and testing, all in one unified file. Unlike ODB++, which has been proprietary to Mentor Graphics since that company purchased Valor several years ago, IPC-2581 will be open to implementation by anyone, with no license required.

ODB++ has been so widely imported by

CAD and CAM vendors that it has become a virtual standard, and as far as I know, no burdensome restrictions are imposed on those vendors licensing ODB++. But I understand why some people may be nervous about relying on a captive de facto standard. Why Mentor Graphics might be reluctant to cheerlead the IPC effort is equally simple to comprehend. From the Mentor (Valor) perspective, why on Earth is there a need for another standard that will simply require conversion to ODB++ and therefore invite errors?

My company actively supports the IPC effort, while recommending the use of ODB++ by our customers because it saves both of us time and uncertainty. IPC introduced the open 2581 standard many years ago, long after my company started business, and there has been little clamor among customers to bring designs to us in that format. However, I am willing to bet that a truly open standard eventually will prevail.

Incidentally, regarding format uniformity, in the 1990s my company compared the Gerber data from several EDA tools for a reference design and found the outputs differed in accuracy and in the size of the data set. There's no guarantee that the ODB++ files output by various platforms would exactly coincide either. My company has volunteered to build a reference design on behalf of the IPC-2581 Consortium when the CAM software is available for testing. We also intend to compare the results of a design output in Gerber, ODB++, and IPC-2581. I'll let you know what happens. **PCBDESIGN**

“
How your design data are formatted determines how easily — and perhaps how successfully — your manufacturer can interpret exactly what you intend to have built.
 ”



Amit Bahl directs sales and marketing at Sierra Circuits, a PCB manufacturer in Sunnyvale, CA. He can be reached via amit@protoexpress.com.

Exclusive Publications for the PCB Design Community



THE pcb
designTM
MAGAZINE

INSIDE DESIGN

WEEKLY NEWSLETTER



SUBSCRIBE NOW!

 I-CONNECT 007
GOOD FOR THE INDUSTRY

Trace Currents and Temperature, Part 3: Fusing Currents

by Douglas Brooks, Ph.D.
ULTRACAD DESIGN INC.

SUMMARY: *This is the third of a four-part series on trace currents and temperature. The [first part](#) discussed the role of resistance and formulated a basic model for analysis. The [second part](#) explored various results empirically obtained. Part 3 explores using the melting temperature of a trace to our advantage, and Part 4 will suggest a way to deal with vias.*

The Question

One day, after I gave a seminar on trace currents and temperatures, a student asked me the following question:

I have a trace that only needs to be able to carry 20 amps for 0.5 seconds. After that, I don't care what happens to it. How do I determine how big the trace needs to be?

There are certain applications where such a requirement is quite reasonable. Consider, for example, a trace that normally carries a reasonable current. But if there is a catastrophic system failure, the trace would be subject to a very large current. If such a failure occurs, you may need to have the time necessary to shut down the system in a controlled manner – perhaps to

prevent even more catastrophic failures, or to prevent the chance of human injury, etc. There are at least three ways to address this type of problem.

1. Design a fuse into the circuit. This is typically a poor solution because (a) it adds component cost, and (b) a fuse does not break the circuit in a controlled manner in a defined length of time.
2. Design the trace large enough to tolerate the maximum amount of current the trace may see. This is also undesirable since the required trace may be unreasonably large.
3. Design a smaller trace that will carry the maximum amount of current (at any temperature) as long as the trace does not get hot enough to melt. Or if it does melt, it lasts at least long enough to allow the controlled shutdown before it melts.

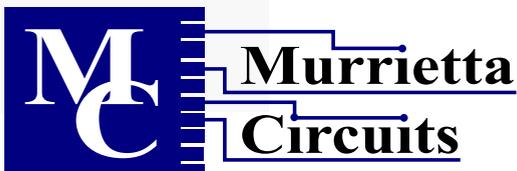
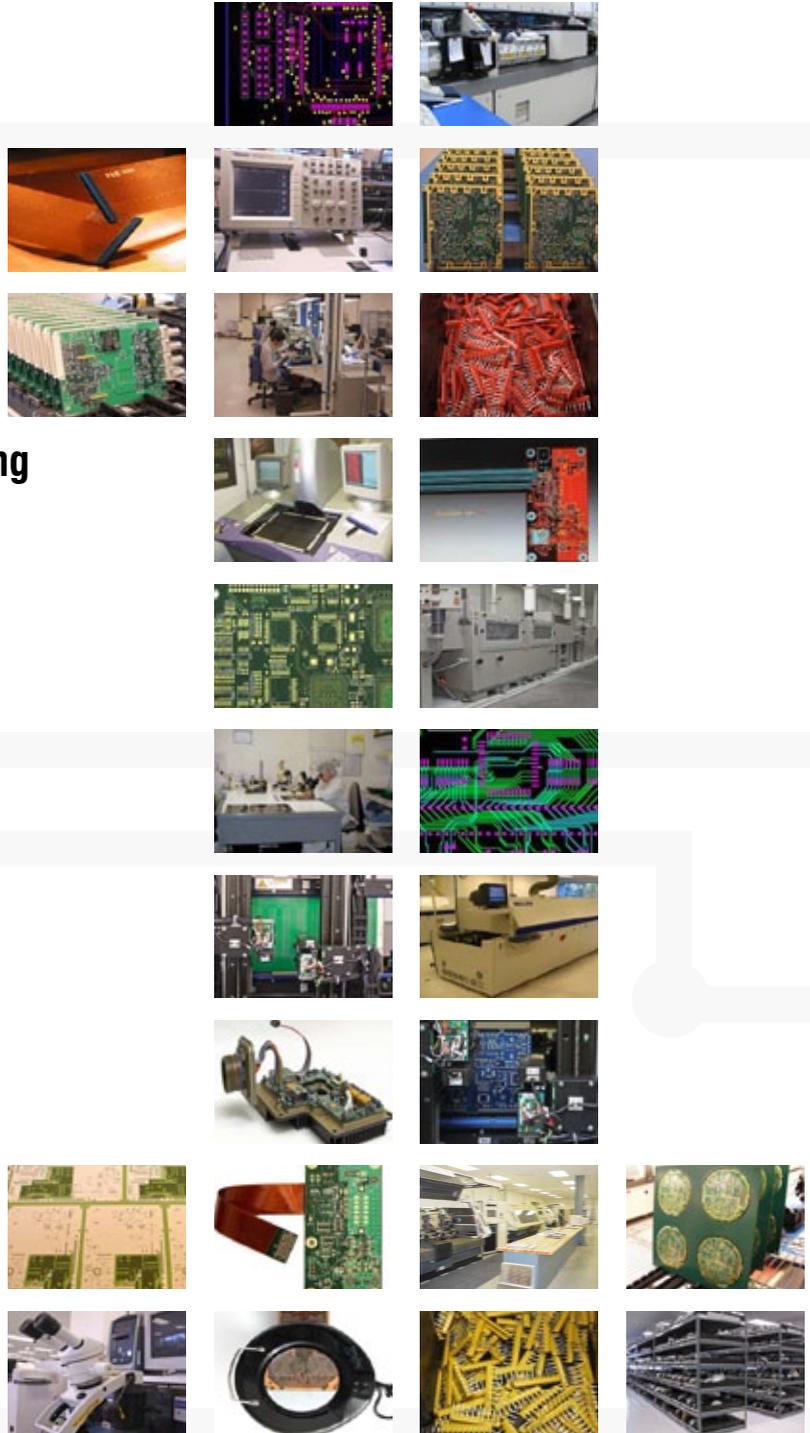
It should be noted that if this last method is used, and if there is a catastrophic failure, the current magnitude and trace temperature will exceed the limits for which the rest of the board was designed. Therefore, the board should be considered irreparably damaged and removed

Murrietta Circuits: Everything you need, from **one** company.

- PCB Design and Engineering
- PCB Fabrication
- PCB Assembly
- Testing
- Re-Tinning
- Adapters and Interposers
- Flex and Rigid-Flex

It's all here under one roof
...why go anywhere else?

The future of PCB
Manufacturing—here today.



5000 East Landon Drive, Anaheim CA 92807
(714) 970-2430 | sales@murrietta.com | www.murrietta.com

TRACE CURRENTS AND TEMPERATURE, PART 2: EMPIRICAL RESULTS *continues*

from service. It should not be repaired and returned to service. If reparability is desired, then other design strategies should be followed.

The Solution

My first reaction was to refer the student to people in the fusing industry. This is a fusing question and they would be the likely people to know the answer. Turns out, I could find no one in that industry with any idea how to approach this question!

Then, one day, while I was looking for some guidance, I stumbled across two obscure equations in a handbook. One of these formulas was attributed to a man named W. H. Preece and the other to I. M. Onderdonk^[1].

I have tried on numerous occasions since I first found this reference to find the original source data for Preece's and Onderdonk's equations. I finally traced Preece's back to a brief, half page summary in an 1884 copy of the Royal Society Proceedings^[2]. But I have never located the source of Onderdonk's equation.

Preece's Equation

Preece's equation (below) relates to a round wire:

$$I = k * d^{3/2} \quad [\text{Eq. 1}]$$

Where:

- I = current in Amps
- d = wire diameter in inches, and
- k = 10,244 for copper

We can make the necessary adjustments to convert this to cross-sectional area, something easier to deal with on circuit boards, as follows:

$$I = 12,277 * A^{0.75} \quad [\text{Eq. 2}]$$

A = area in in²

Or $I = 0.388 * A^{0.75} \quad [\text{Eq. 3}]$

A = area in mil²

According to Preece, this equation would result in the current just sufficient to heat the trace to the melting point in air.

Onderdonk's Equation

Onderdonk's equation is significantly more complex, but it has the added benefit of bringing time explicitly into the relationship. That is, a certain current, for a certain amount of time, across a certain cross-sectional area, will be just sufficient to melt the wire or trace.

$$I = A * \sqrt{\frac{\text{Log} \left(\frac{(T_m - T_a)}{(234 + T_a)} + 1 \right)}{33 * s}} \quad [\text{Eq. 4}]$$

- Where:
- I = Current in amps
 - T_m = Melting temperature of the material, °C
 - T_a = Ambient temperature, °C
 - A = Cross-sectional area in circular mils (take note^{[3]!})
 - S = Time, in seconds

This can be converted to a form more useful for PCB designers as follows:

$$I = \frac{.188 * A}{\sqrt{T}} \quad [\text{Eq. 5}]$$

- Where:
- I = Current in amps
 - A = Cross-sectional area in mil²
 - T = Time in seconds.

Preece vs. Onderdonk

It is instructive to ask, "How do these two equations, ostensibly from totally different sources, relate?" We can explore that by setting them equal to each other and solving for time. That way, we can infer an implied time for Preece's equation. When we do that, we can derive the result in Figure 1.

The result suggests that Onderdonk's equation predicts implied times using Preece's equation of from approximately 1.0 second to about 8.0 seconds for reasonable trace configurations. This is a comforting result in that the two equations are reasonably consistent.

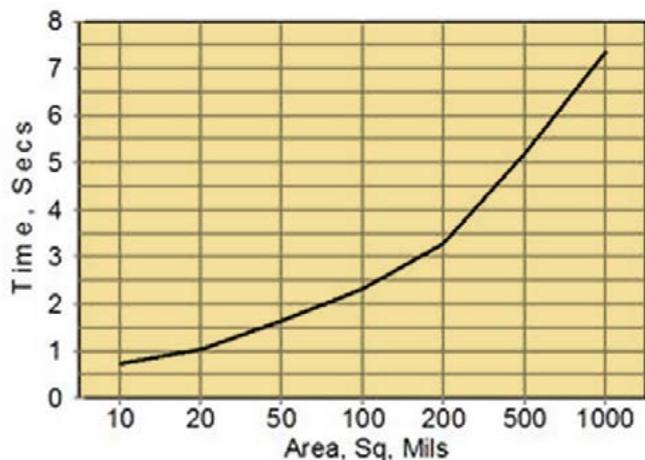


Figure 1: Implied times for Preece’s equation based on Onderdonk’s equation.

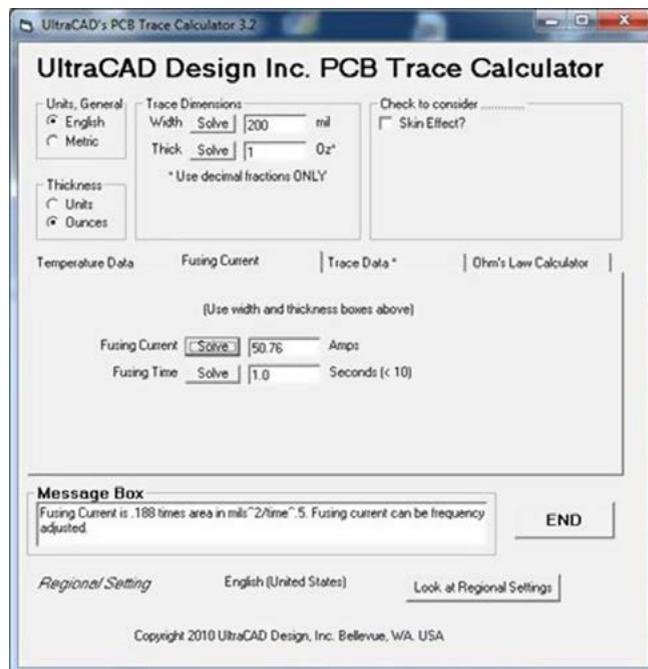


Figure 2: UltraCAD’s UCADPCB3 calculator will do fusing current calculations as well as trace current/temperature ones.

Comparison with Part 2

In Part 2 of this series we developed empirical equations using data from a few data sources. Two of those were provided as Equations 5 and 7 in that series, repeated here as Equations 6 and 7, respectively. The first was derived from the Design News data and the other was derived from the latest IPC 2152 data:

From DN data:

$$I = .040 * DT^{45} * A^{.69} \quad [\text{Eq. 6}]$$

From IPC-2152 (air):

$$I = .063 * DT^{50} * A^{.58} \quad [\text{Eq. 7}]$$

The difference between a typical ambient temperature (20° C) and the melting point of copper (1080°C) is 1063°C. Plugging a DT of 1063° into these equations and converting the area term from in² to mil² results in the melting current of a trace projected to be^[4]:

From DN Data:

$$I = 12,705 * A^{.69} \quad [\text{Eq. 8}]$$

From IPC-2152 (air):

$$I = 6,203 * A^{.58} \quad [\text{Eq. 9}]$$

Compare Equations 8 and 9 with Preece’s equation, Equation 2:

$$I = 12,277 * A^{0.75} \quad [\text{Eq. 2}]$$

The Design News data projection is SO close to Preece’s result as to be suspicious. The results seem almost to be contrived. Yet we reach the results from totally different directions. On the one hand, this would give significant credence to the Design News data. On the other hand, we do recognize that we are projecting well beyond the range of the data to get here. The IPC data suggest a much lower fusing current than does the Design News data, consistent with (as we suggested before) the fact that the IPC data results in more conservative results than does the Design News data.

Calculator

In Part 2 of this series, we showed a calculator that UltraCAD has released for making trace current/temperature calculations^[4]. The same calculator can be used for making fusing current calculations, based on Onderdonk’s equation (see Figure 2).

TRACE CURRENTS AND TEMPERATURE, PART 2: EMPIRICAL RESULTS *continues***Conclusion**

We have looked at two different equations (from Preece and Onderdonk) for approaching the trace fusing question. And we have shown that the two different equations are reasonably consistent with each other. Furthermore, Equations 8 and 9 suggest that the fusing results are within the same order of magnitude as the results we obtained in Part 2. This gives us some degree of confidence that all of these approaches are reasonably credible. **PCBDESIGN**

References

1. See "Standard Handbook for Electrical Engineers, 12 Ed., McGraw Hill, p.4-74.
2. Royal Society Proceedings, London, 36, p. 464, 1884. (I have since learned that Preece was the Chief Electrician of the British Post Office (which included overland telegraph services) at that time).
3. It is important to note that A in Equation 4 is in circular mils. A circular mil is equal to the area of a circle with a diameter of one mil.
4. This calculator, UCADPCB3, can be obtained from www.ultracadm.com. Watch UltraCAD's Web site for an Android fusing calculator (for Android phones and tablets) to be released later this year.

5. In plugging a DT of 1063 into the equations derived in Part 2 of this series is somewhat problematic. We are projecting WAY beyond the range of the data. While this is not a statistically invalid thing to do, the results are nevertheless, statistically speaking, very unreliable.



Douglas Brooks has an MS/EE from Stanford University and a Ph.D. from the University of Washington. He has spent most of his career in the electronics industry in positions of engineering, marketing, general management, and as CEO of several companies. He has owned UltraCAD Design Inc. since 1992. He is the author of numerous articles in several disciplines, and has written articles and given seminars all over the world on signal integrity issues since founding UltraCAD. His book, [Printed Circuit Board Design and Signal Integrity Issues](#) was published by Prentice Hall in 2003. Visit his website at www.ultracadm.com.

Sunstone Reveals "Share Your Story" Winners

Sunstone Circuits has announced the "Share Your Story" contest winners. The contest offered designers a chance to share their PCB-related design successes with their peers. Once a story was submitted, family and friends then voted for the best project. Three lucky winners each took home an Apple iPad, a customized iPad case, plus a \$25 iTunes gift card.

The three lucky winners are:

- **Clive Bolton:** My First Circuit Board – Pong Video Game (488 votes)
- **Steve Ranta:** I Designed my PCBs at Disneyland! (360 votes)
- **Ken Sheets:** Building my First PCB with my Dad (201 votes)

To see a read more of their stories, please [click here](#).



Most-Read Mil/Aero007 News Highlights



[U.S. State Dept Lists PCBs in ITAR Draft Rulemakings](#)

Calling the State Department's decision a "step in the right direction," IPC President John Mitchell said, "IPC appreciates that the State Department shares our view that ITAR's regulation of printed boards should be clearer." But IPC is still troubled by the inclusion of the term "specially designed" in reference to PCBs.

[IPC ITAR Workshop to Raise Awareness](#)

"Domestic printed board manufacturers have sounded an alarm that defense industry confusion over ITAR's treatment of printed boards is undermining national security," said IPC President and CEO John Mitchell. "IPC is grateful for the opportunity to partner with federal officials in workshops like this to clarify current and proposed export control regulations."

[FTG Undergoes Organizational Changes](#)

Firan Technology Group Corporation (FTG) has appointed Claude Bougie as president of FTG Aerospace Tianjin. Claude brings with him a wealth of experience in the Aerospace industry and will use this knowledge and experience to ensure the rapid growth of FTG's business in Tianjin. Kaiyan Gu will continue as general manager at FTG Aerospace, Tianjin and will report directly to Claude.

[Basic Electronics Touts High-Quality Service, Personnel](#)

Basic Electronics Inc. achieved excellent marks in its AS9100 audit report. As per the audit results from NQA, Basic was noted to have zero non-conformities, with mention of its continued customer satisfaction in delivering quality product with an emphasis on cost improvement.

[Call for 21st Century Defense Strategy](#)

The changing global security landscape and worsening fiscal outlook demand significant adjust-

ments to national security strategy and budgeting, according to an extensive, year-long study released today by The Stimson Center: A New U.S. Defense Strategy for a New Era.

[MEMS in Military & Aerospace to Reach \\$283.6M in 2012](#)

Revenue for pressure sensors in both military and civil aerospace applications will reach \$35.7 million by year-end, up 20% from \$29.7 million last year, according to an IHS iSuppli MEMS Market Brief from information and analytics provider IHS.

[Report: Global Military Aviation MRO Market 2012-2022](#)

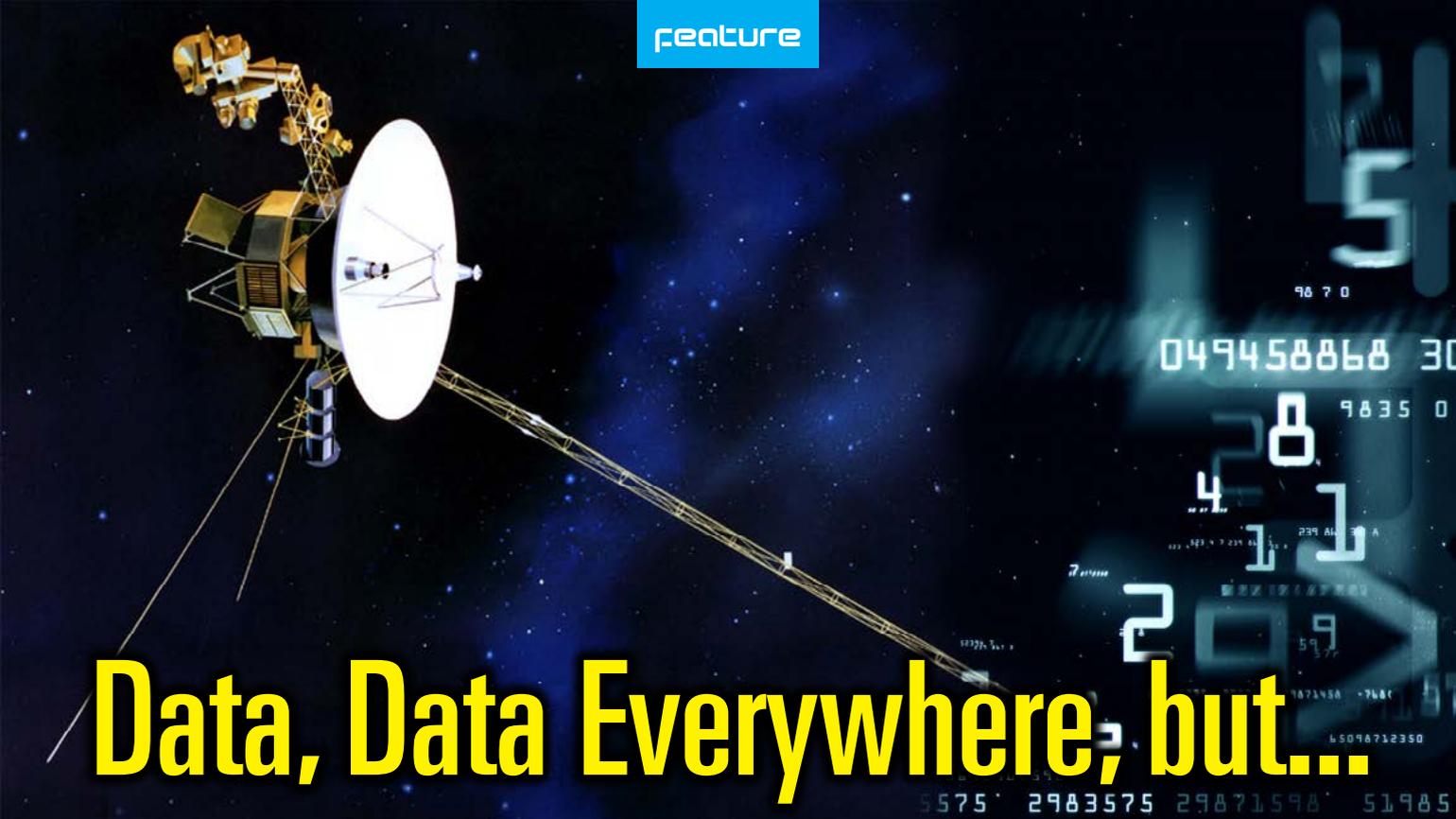
This report provides readers with a comprehensive analysis of the military aviation MRO market through 2012-2022, including highlights of the demand drivers and growth stimulators for military aviation MRO. It also provides an insight on the spending pattern and modernization pattern in regions around the world.

[Air Cargo Security & Screening Systems Market \\$486.5M in 2013](#)

Visiongain's analysis indicates that the global air cargo security and screening systems market will reach a value of \$486.5M in 2013, as airports and air freight companies acquire new screening systems and update existing security systems in order to meet the requirements set aside by relevant government authorities.

[DARPA Unveils PIXNET Technology](#)

PIXNET aims to develop helmet-mounted and clip-on camera systems that combine visible, near-infrared, and infrared sensors into one system and aggregate the outputs. This technology would ingest the most useful data points from each component sensor and fuse them into a common, information-rich image that can be viewed on the warfighter's heads-up display.



Data, Data Everywhere, but...

by **Iain Wilson**
IRON ATOM

SUMMARY: *Transferring PCB data has been a big part of my career, especially for the last 10 years or so. From concept through assembly, a lot of data is used to describe the various aspects of a PCB, the components and final assembly. But I'd like to point out that in reality, there's not enough data. We actually need more.*

Recently, the Voyager 1 spacecraft has made the news by entering the very edges of our solar system, soon to pass into true interstellar space. Remarkably, since its launch in September 1977, the spacecraft continues to transmit useful data to the terrestrial team of scientists monitoring its progress. I wish I could offer similar kudos to the PCB supply chain.

A little more than 10 years ago, I was working with a U.S. PCB manufacturer on an engineering automation project. The engineering manager was a colorful character. One day in the early stages of the project we were discussing various aspects of the information we required. At one point he quipped, "In God we trust; everyone else brings data." I laughed, but only got a wry smile in return; I could see this was a serious point for him.

From concept through assembly, a tremendous amount of data is used to describe the various aspects of a PCB, the components and final assembly. Massive amounts of data, in fact. Someone should print all of the info that goes into today's smartphones. (Note: If you do, please use recycled paper or there's going to be some serious deforestation, I suspect.)

Having said that, I'd like to point out that in reality, there's not enough data. We actually need more.

The big picture of PCB data is being tackled today by the IPC-2581 Consortium. The members are volunteers from a wide range of companies including OEMs, designers, fabricators and software companies. Their mission is to create an open, comprehensive data standard to describe all the aspects of the PCB for the entire supply chain. Sounds good right? I believe it is a just and noble cause and when adopted by the entire supply chain, it will greatly improve the transfer and updating of PCB data.

As such, I've recently involved myself in the consortium's work and I try to help when I can. Much of the work has already been completed, notably a new data format that describes the physical form of the PCB layers. Currently, in the industry, we call this "Gerber." OK, I'm very tempted, but I'll spare you another lecture

Some companies tell you they are great. We let our customers speak for us...

I had mentioned to you upon your last visit that Prototron should feel free to use SelfCharge Inc. as a reference. I believe I speak for everyone in our organization when I say that Prototron has been and continues to be an integral part of SelfCharge's product development. Prototron has had an impeccable ability to meet our high quality standards, on-time delivery performance and price targets.

Thank you for the continued support. We are looking forward to a long and prosperous business relationship.

Brady L. Boyd, C.P.M.
Materials Supervisor
SelfCharge Inc.



I received the 150+ boards yesterday afternoon. Thank you once again for your excellent service and quality. Be sure to thank the people in your factory for me as well. When I order from you, I know I don't have to worry about getting bad boards and going through the purchase cycle again. Let alone the embarrassment of explaining it all to my boss and his boss.

Warmest regards,

Richard Diehl
Maxim Integrated Products



Prototron
Circuits
Quality printed circuit boards

Redmond, WA 425.823.7000

Tucson, AZ 520.745.8515

www.prototron.com

DATA, DATA EVERYWHERE, BUT... *continues*

about our industry's use of that antiquated data format.

I've been working as a PCB engineer or with PCB engineers for most of my career. Transferring PCB data has been an important part of my career, especially for the last 10 years or so. I've spent a decade developing and deploying an engineering automation tool that, among other things, generates stackup and associated impedance requirements, production route/traveler, and so on.

Part of the market requirements we identified for the product was the need to integrate to other systems, and for PCB engineering this primarily means CAM and ERP. CAM data is used as a starting point for engineering jobs whereby we can get the layers and drilling information. From that we can generate the stackup with the manual input of the thickness and impedance requirements. Additional inputs are required to create the traveler and other supporting documentation. This includes items like board finish, mask type, legend color, date code format, specifications, and so on. Overall, it takes about 100-150 attributes to fully describe a PCB for bare board manufacturing, depending on its technology.

Interestingly enough, generating a quote requires only about 20-30 attributes. Let's com-

pare those two figures: About 20 attributes determine the PCB's price, but another 80 or more are needed to build it correctly.

“
Overall, it takes about 100-150 attributes to fully describe a PCB for bare board manufacturing, depending on its technology.
”

Once the engineering work is completed, typically it's exported to an ERP/MES system. The engineering data drives all the critical manufacturing requirements to process the work orders. Prior to the advent of this type of system, data was usually manually extracted from CAM and manually entered into ERP. Bridging this data gap saved a lot of time and eliminated duplicate manual data entry, which is at best time-consuming and at worst-error prone.

Remember that I mentioned that we need more data? Although in the big scheme of things the stackup, impedance and general info are not a huge percentage of the total data, they are still critical to determine the price and actually manufacture the part correctly. So how do we as an industry handle this critical data exchange? PCB engineers can glean this information from:

1. A PDF file of a fabrication drawing.
2. A DXF file of a fabrication drawing.
3. Either of the above and/or
 - a. A text file with notes
 - b. An email with notes
 - c. A purchase order with notes

Interested in supporting the development and adoption of IPC-2581? You can show your support in three ways:

1. Become a member of the IPC-2581 Consortium and participate regularly in meetings and activities.
2. Use the tools that currently support the standard.
3. State your support of the consortium by publishing a note on your company website.

- d. Retained memory of the last time we did a job for you
- e. A customer cheat sheet we drew up last year
- f. General common knowledge of what is probably being asked for

Perhaps there are a few other options I could list, but you get the point. Having all the critical info as part of a simple “button-click” import would be a huge help. Although PCB engineers are very good at manually entering, checking, and then double-checking the data they enter, it’s time-consuming and, by nature, error-prone. Ultimately it’s a waste of time and money.

The IPC-2581 standard should address this glaring data hole. I say “should” because it’s not complete yet. I hope to influence the consortium (along with many others) to ensure that this is taken care of in the standard. We’re in the midst of it right now, and it remains to be seen how it all transpires. I’m hopeful of a good result and a big step forward for our industry.

As an industry, we’ve been here before. The attempt to merge the GenCAM and ODB++ was ultimately a failure. While ODB++ has seen some successes in terms of adoption, it still only accounts for about 10% of data transfers. The rest is made up of “Old Man Gerber.” Industry-wide adoption of an open and comprehensive data standard for PCBs is quite simply a benefit for all. Let’s do what it takes to advance our industry with seamless, critical data transfer from the beginning of the supply chain to the end. After all, it can’t be that hard.

A 34-year old spacecraft is doing it from 10 gazillion miles away. **PCBDESIGN**



Iain Wilson is president and co-founder of Iron Atom. He and Alessandro Federici founded Iron Atom in 2011 after seeing an opportunity to utilize cloud computing to offer on-demand usage of highly automated, expensive software applications to the mass market.

video interview

A Designer with Flex on His Mind

by *Real Time with...*
Designers Forum

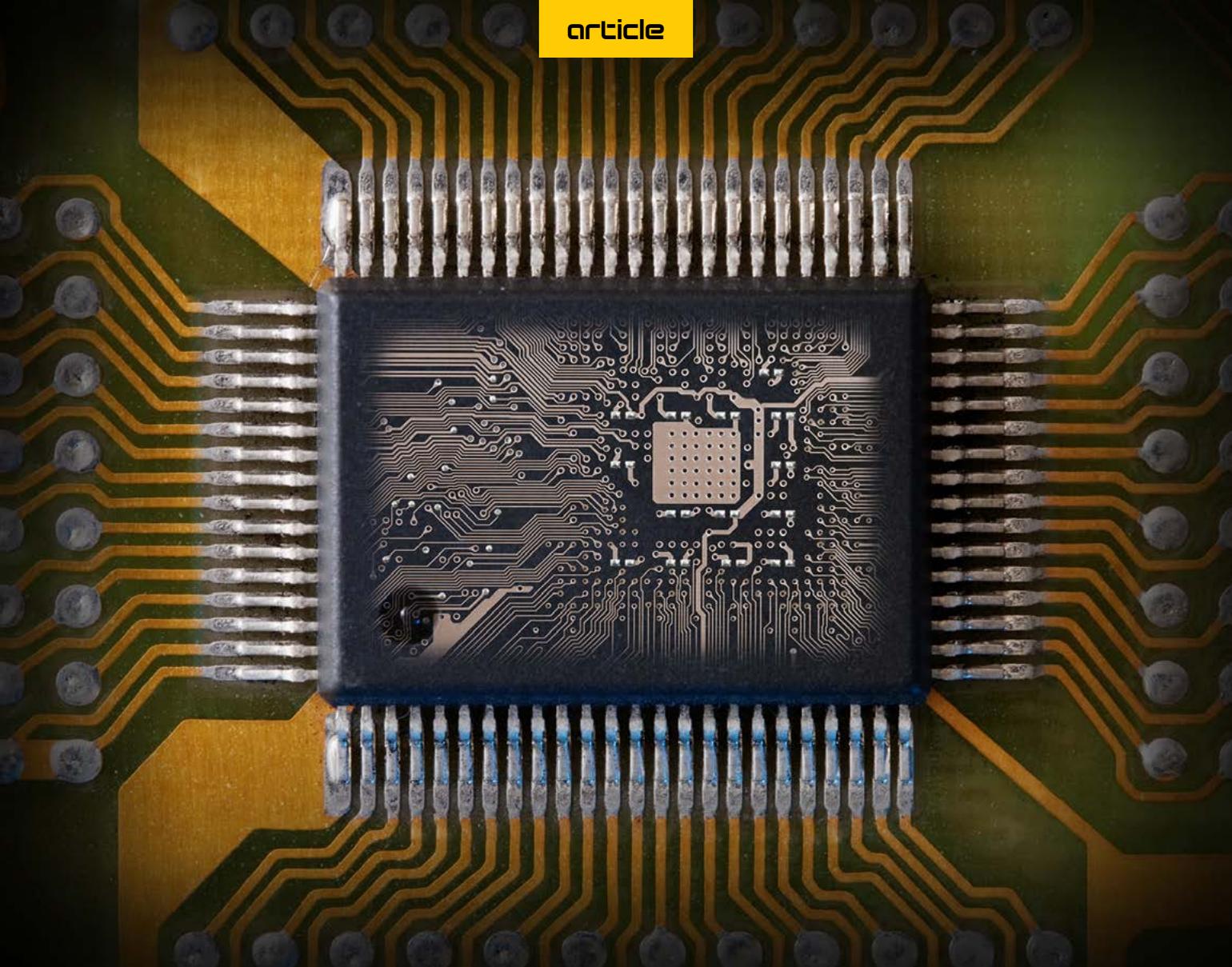


Guest Editor Kelly Dack takes advantage of an intermission during the “Designing with Flex in Mind” class to grill flex designer Scott Bowles on his flex design background, his work with several IPC flex subcommittees, and his satisfaction with the content of the class.



realtimewith.com





PCB Trends in 2013: Smaller and Denser

by **Dale Kersten**
SANMINA

***SUMMARY:** In 2013, we can expect electronics designs to shrink even further. Engineers across the globe will find PCBs to be increasingly challenging to design due to the increased density and thermal issues brought on by miniaturization.*

The ever-increasing volume of electronics built into the cars we drive, the server farms that power our daily Internet usage, medical devices and even elevators is driving demand for smaller, more functional electronics. And, of course, smaller and smarter means more complex, particularly the PCB interconnects.

In 2013, we can expect electronics designs to shrink even further. Engineers across the globe

will find PCBs to be increasingly challenging to design due to the increased density and thermal issues brought on by miniaturization.

As smart phones and tablets tack on functionality and shrink in size, the need for higher-layer count and increasingly denser interconnect is well known. But many other industries, from telecommunications to medical to industrial segments, are also requiring interconnects with increased functionality, smaller footprints and more robustness.

According to this recent study from IBM, 2.5 quintillion bytes of data are created every day. And it's not just the amount of data, but the transfer speed – just a few years ago bandwidth was 8G, but it is now pushing 25G. The telecom, networking and cloud computing companies that process and serve up this data are partnering with manufacturers like Sanmina to help design and produce interconnects that can transfer the magnitudes of data at faster speeds with fewer errors.

The Facebooks and Googles of the world don't want to add more server towers to their computing farms; they want to add more speed and functionality, which means interconnects with higher layer counts. What used to be a four- or six-layer interconnect is now a 10-, 12-, or 14-layer interconnect, and this trend will only increase in the future. However, with that kind of density comes design and manufacturing challenges.

For example, as the layer counts increase, the boards naturally become thicker, and individual cores or material substrates become thinner and less reliable in terms of movement. These boards have many more signal traces, which can create challenges in registration and issues with thermal expansion when materials don't move at the same rate. The diameter of the holes is significantly reduced, while the number of holes increases from 15,000 to 100,000+, which means higher densities, although it also allows for smaller form factors or more components. As interconnects and systems get smaller, thermal considerations such as power, dissipation and hot spots become increasingly more challenging to the product designer.

These issues have required a change in the types of materials used on interconnects. The

industry is moving beyond conventional FR-4. Most of the change is geared toward the raw materials, like resin, glass fabrics and copper, as well as a focus on hybrids' constructions that utilize multiple types of laminates in one PCB. Spread glass is becoming popular again because it has more consistent rates of signal speeds. We are also looking at material types that function differently.

Some layers of interconnect may require high-speed materials, while other parts might only require FR-4, or you may have a combination of materials on different layers. Another reason for the move to hybrids is to lower costs, as high-speed laminates can reach 15x the cost of standard materials. Additionally, there is usually much more material on a higher-layer board. Another important change is the move to lead free-assemblies in order to conform with international standards for eco-friendly materials.

At Sanmina, we've found that our customers are requesting more complex requirements for PCBs. One of our industrial automation customers used to have 10 or 15 control stations on a line. Now, they want only one or two stations, which means the boards have to be that much more intelligent and robust. We expect this trend toward more complex PCBs to continue.

Design engineers must adopt the expertise required for reliable manufacturing of PCBs on a much smaller scale. So long as this is done effectively, innovation will thrive in an increasingly complex and competitive world. **PCBDESIGN**



Dale Kersten is vice president of global engineering, R&D at Sanmina. Kersten joined Sanmina in 2001 and has served in several roles within the PCB division. He has been involved in the design and manufacture of leading edge, high-end complex printed circuit boards throughout his career. Prior to joining Sanmina, Kersten held executive positions at Automated International and HADCO Corporation.

TOP TEN

PCBDesign007
News

Most-Read News Highlights from PCBDesign007 this Month

① Top 10 Most-Read PCBDesign007 Articles of 2012

It's been a wild year in the PCB design community. Naturally, the top PCBDesign007 articles from 2012 cover a maze of topics, from DFM to high-speed design techniques. So, without further ado, here are the Top 10 Most-Read PCBDesign007 articles of the past year.

② Sunstone Circuits Reveals "Share Your Story" Winners

The contest, which launched on October 10, 2012, offered designers and design engineers a chance to share their PCB-related design successes online with their peers. Designers sent story links to their family and friends, who then voted for the best project. Three lucky winners each walked away with an Apple iPad, a customized iPad case, and a \$25 iTunes gift card.

③ "Father of the Gridless Router" Alan Finch Dies

Few people in the electronics industry can honestly claim responsibility for a quantum leap in technology. Alan C. Finch was one of those people. Finch, the "father of the gridless autorouter," passed away this weekend in the UK. I was fortunate enough to talk with him by phone years ago, and found him very unimpressed with his "rock star" status in the EDA world.

④ Mentor Graphics Debuts New Thermal Testing Method

Mentor Graphics Corporation today announced the new T3Ster DynTIM tester, the industry's cutting-edge method of measuring thermal characteristics of thermal interface materials.

5 Agilent Releases ADS 2012

ADS 2012 features new capabilities that improve productivity and efficiency for all applications the system supports and breakthrough technologies applicable to GaAs, GaN and silicon RF power-amplifier multichip module design.

6 IPC Updates PCB Design Standard

In the fast-paced world of electronics, you don't often hear the old saying "good things come to those who wait." But those in the printed board industry will find a lot worth waiting for when they pick up freshly finished copies of IPC-2221B, Generic Standard on Printed Board Design.

7 Bittele Offers Free Design for Manufacturing

Bittele Electronics' design for manufacturing service, which prevents costly electronic design errors and resolves manufacturability problems prior to production, is available at no cost to all electronic PCB manufacturing customers.

8 Zuken CADSTAR Distributor Quadra Solutions Expands

Zuken has announced enhanced support for its Scandinavian CADSTAR users as UK-based distributor Quadra Solutions expands. Scandinavia is an increasingly important region for Zuken. Sales in Scandinavia of CADSTAR have seen consistent growth.

9 EDA Consortium: PCB & MCM Revenue Up 9% in Q3

EDA Consortium's Market Statistics Service reports that EDA industry revenue increased 4.9% for Q3 2012 to \$1.62 billion, compared to \$1.54 billion in Q3 2011. PCB and MCM revenue of \$153.0 million represents an increase of 9% compared to Q3 2011. The four-quarters moving average for PCB & MCM decreased 1.7%.

10 All Flex Acquires Part of TRI-C Design

The acquisition of TRI-C Design's assets and design services adds five people to the expanding company, consisting of 140 employees between the Northfield facilities and a third production facility in Bloomington, Minnesota.



PCBDesign007.com
For the latest circuit design news—anywhere, anytime.

EVENTS

PCB Design Events

IPC Complete Calendar of Events

SMTA Calendar of Events



42nd Internecon Japan

January 16-18, 2013
Tokyo Big Sight, Japan

18th Annual Pan Pacific Microelectronics Symposium

January 22-24, 2013
Maui, Hawaii, USA

DesignCon 2013

January 28-31, 2013
Santa Clara, California, USA

43rd Annual Collaborative Electronic Warfare Symposium

January 29-31, 2013
Pt. Mugu, California, USA

SEMICON Korea 2013

January 30-February 1, 2013
Seoul, Korea

SPIE Photonics West 2013

February 2-7, 2013
San Francisco, California, USA

6th Annual Mobile Deployable Communications

February 7-8, 2013
Amsterdam, Netherlands

Medical Design & Manufacturing

February 11-14, 2013
Anaheim, California, USA

Electronics Manufacturing Korea 2013

February 13-15, 2013
Seoul, Korea

IPC APEX EXPO® Conference & Exhibition 2013

February 19-21, 2013
San Diego, California, USA

CMSE - Components for Mil & Space

February 20-21, 2013
Los Angeles, California, USA

Embedded World

February 26-28, 2013
Nurnberg, Germany

MEDTEC Europe

February 26-28, 2013
Stuttgart, Germany

IEEE CPMT Advanced Pkg Material

February 27-Mar 1, 2013
Irvine, California, USA

Medical Devices Summit

February 28-March 1, 2013
Boston, Massachusetts, USA



PUBLISHER: **BARRY MATTIES**

barry@iconnect007.com

PUBLISHER: **RAY RASMUSSEN**

(916) 337-4402; ray@iconnect007.com

SALES MANAGER: **BARB HOCKADAY**

(916) 608-0660; barb@iconnect007.com

EDITORIAL:

GROUP EDITORIAL DIRECTOR: **RAY RASMUSSEN**

(916) 337-4402; ray@iconnect007.com

MANAGING EDITOR: **ANDY SHAUGHNESSY**

(404) 806-0508; andy@iconnect007.com

TECHNICAL EDITOR: **PETE STARKEY**

+44 (0) 1455 293333; pete@iconnect007.com

MAGAZINE PRODUCTION CREW:

PRODUCTION MANAGER: **MIKE RADOGNA**

mike@iconnect007.com

MAGAZINE LAYOUT: **RON MEOGROSSI**

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

INNOVATIVE TECHNOLOGY: **BRYSON MATTIES**

COVER ART: **BRYSON MATTIES**



The PCB Design Magazine® is published by BR Publishing, Inc., PO Box 50, Seaside, OR 97138 ©2013 BR Publishing, Inc. does not assume and hereby disclaims any liability to any person for loss or damage caused by errors or omissions in the material contained within this publication, regardless of whether such errors or omissions are caused accidentally, from negligence or any other cause.

January 2013, Volume 2, Number 1 • The PCB Design Magazine© is published monthly, by BR Publishing, Inc.

ADVERTISER INDEX

Bay Area Circuits.....	35
BBG.....	11
I-Connect007.....	49
Intercept Technology.....	23
Isola.....	5
Multilayer Technology.....	7
Murrietta Circuits.....	51
thePCBlist.....	2
Prototron Circuits.....	57
Rogers Corp.....	47
Sierra Circuits.....	3
Sunstone Circuits.....	15
TTM Technologies.....	41
Ventec.....	29

Next Month in *The PCB Design Magazine*

For today's high-speed designs, simulation isn't just an option anymore – it's a necessity. In the February issue of *The PCB Design Magazine*, we'll get the lowdown on simulation and analysis techniques from some of the industry's top design engineers.

See you in February!