BOARD LEVEL SIMULATION SPECIALISTS

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

- 2D (BEM) field solver precision
- Characteristic impedance, edge-coupled & broadside-coupled differential impedance
- Unique field solver computation of multiple differential technologies per stackup
- Heads-up impedance plots of signal and dielectric layers
- User defined dielectric materials library - over 28,000 materials up to 100GHz

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

- Fast AC impedance analysis with plane resonance
- Definition of plane size/shape, dielectric constant & plane separation for each on-board power supply
- Extraction of plane data from the integrated Stackup Planner
- Definition of voltage regulator, bypass/decoupling capacitors, mounting loop inductance
- PDN EMI Plot with EMC Limits. Frequency range up to 100GHz
- Extensive Capacitor Library – over 5,650 capacitors derived from SPICE models

iCD

In-Circuit Design Pty Ltd

LEARN MORE

www.icd.com.au

The Rise of the Independent Engineer

by Barry Olney
IN-CIRCUIT DESIGN PTY LTD / AUSTRALIA

With the changing demographics, the oldtimers in our industry—the master PCB designers—are about to retire and hand over the exacting job of PCB design to the Gen-X and Ys. These generations, shaped by technology, will tackle the most demanding designs without possessing the experience that we veterans benefit from.

And to top it off, these up-and-coming designers will be degreed engineers who have to cope with both design and layout tasks as the specialized PCB designer's positions are phased out. Apart from a demanding regime of training, what can these guys do to become successful independent engineers?

The majority of veteran PCB designers began their careers on a drafting table. In the late 1970s, basic PCB design software began to emerge in the mainstream market. The computer skills of the PCB designer grew and before you knew it, we were all proficient with the latest EDA software tools. Some argue that since the emergence of EDA, the line between layout and engineering has become blurred. Engineers who are proficient with EDA software can produce a complex PCB, eliminating the need for a PCB designer. Similarly, a PCB designer can perform engineering design with the use of sophisticated analysis software. In theory, this is a good concept but from the engineer's perspective, it runs into practical problems:

1. Engineers must undergo significant training in order to use the software. In many cases, this is simply not feasible and the lost opportunity costs are prohibitive.
2. In order to remain proficient, the engineer must regularly use the software. This is a problem for busy engineers who have a variety of responsibilities and only work on one or two projects per annum.
3. The engineer must have a thorough understanding of the specific requirements of the

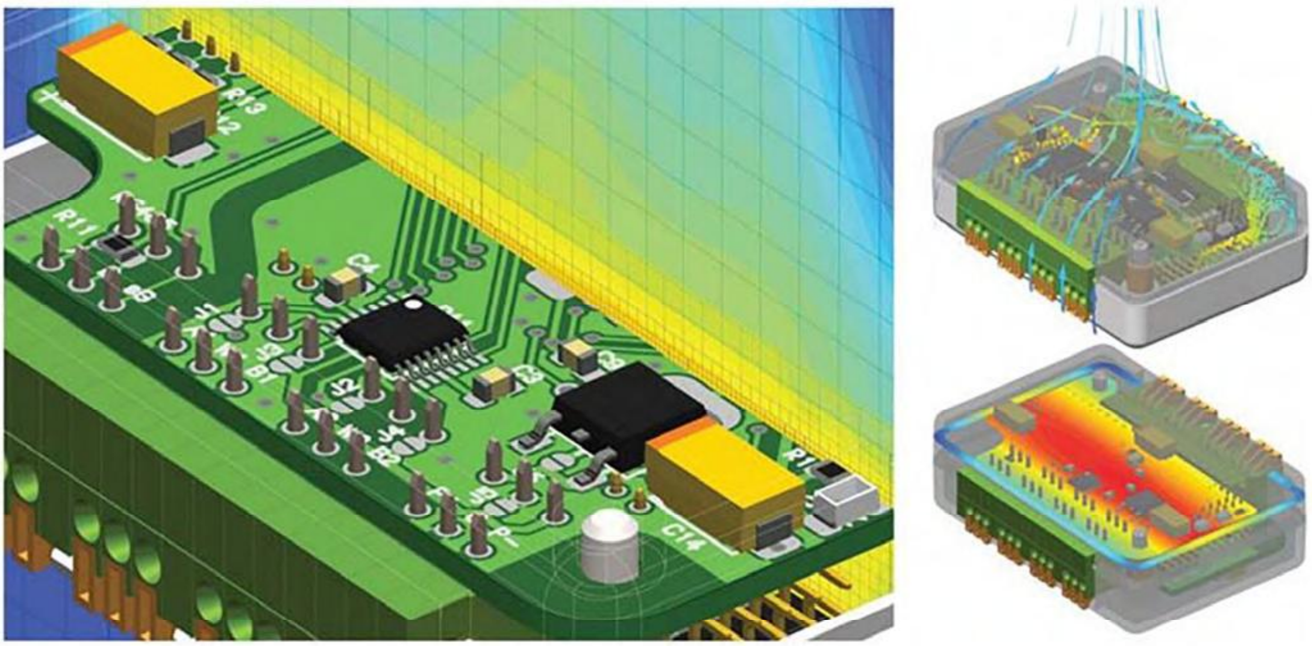

Figure 1: Thermal analysis of the PCB and assemblies. (All images courtesy of Mentor Graphics)

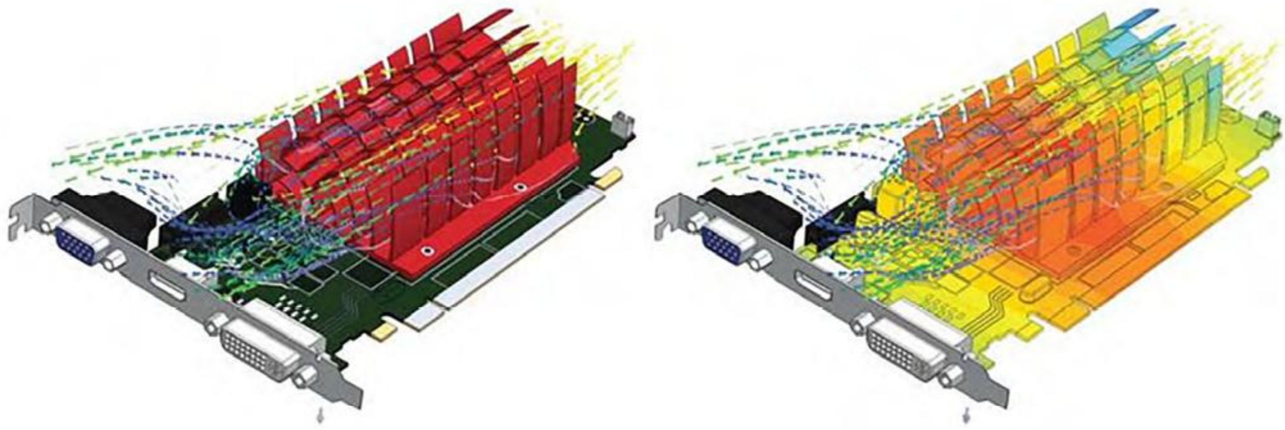

Figure 2: 3D flow field using particle post-processing.

design rules encompassing PCB fabrication and assembly in order to produce a reliable, manufacturable product.

The net result is that the engineer typically does not use the software but rather relies on the PCB designer's application and manufacturing knowledge to guide him through the process. Also, in most cases, the PCB designer struggles to use the analysis software and requires extensive and ongoing training. So, the next innovation in EDA tools must not only be fast to adopt and easy-to-use, but still be packed with all the features today's designers need for the most complex boards.

In the past, engineers were alienated from the layout tools. Many stood over the shoulder of the PCB designer, providing input during the design phase. This methodology is inefficient. It limits the engineer's ability to try alternate scenarios. However, the next generation will take control of the mouse which will enable them to make better design decisions. This may well be a positive aspect, as design complexity, with fast rise-time signals, reaches the point where high-speed digital circuits exhibit RF behavior. Engineers, unlike traditional PCB designers, are more aware of the transmission line issues including signal and power integrity requirements and can investigate the options that are available to them, on the fly, saving valuable development time.

Today, the PCB design process entails much more than just schematic capture and PCB layout. With increasing complexity in electronics systems, engineers need to develop with the whole product in mind. Having access to a design tool that encompasses PCB design, coupled with comprehensive simulation and analysis, really gives design engineers the confidence that their products will be delivered on schedule and at the highest performance and reliability.

To create a stable, reliable product, one must satisfy all the high-speed and manufacturing constraints. As electronic products become smaller, faster and more densely packed, engineers are compelled to consider thermal aspects (Figure 2) and utilize virtual prototyping to meet stringent schedules. Enter the product creation platform. Developed for individuals and small teams, designing electronic products, new tools such as the new Mentor Graphics PADS PCB product creation platform encompasses a variety of productivity enhancements.

The issues that previously impeded the independent engineer have been addressed by providing:

1. Professional layout tools that combine ease-of-use with highly automated functionality to give engineers exceptional control over the creation of the most complex designs. The familiar GUI enables the occasional user to pick it up and become productive quickly.

2. A powerful and easy-to-use constraint management system that provides a common, integrated constraint definition environment for the creation, review, and verification of PCB design rules. The system supports definition

and verification of electrical and physical constraints within one environment, eliminating the need for separate databases, and simplifying a complex constraint entry process while improving design accuracy.

3. Apart from ECAD-MCAD collaboration and EMC verification tools, tight integration with Valor NPI for concurrent DFM validation and optimized hand-off to manufacturing. This ensures that all manufacturing data is included and synchronized, and that the engineer's design intent is maintained. Additional, Valor NPI provides concurrent DfM analysis during the design process, applying almost 700 manufacturing rules, to the design, ensuring compliance and minimum revision spins in manufacturing.

With the release of the product creation platform, PADS has included the FloTHERM XT option that utilizes a powerful solver and mesher for fast and effective electronics cooling simulations, working with both MCAD and PADS design flows. Full geometric and non-geometric SmartParts and library capabilities provide access to a full set of the most popular components for fast and accurate model creation. You can easily define, solve, and analyze results using parametric variations of geometry, attributes (e.g., material, thermal), and solution parameters to significantly enhance the design optimization process.

The tight integration with PCB layout reduces time consuming data translation and prevents costly errors. Board or component layout can be easily modified for position, size, orientation, shape, and modeling level prior to import into FloTHERM.

Insufficient cooling can result in poor reliability, schedule delays, and increased production costs. This is especially relevant in today's high-performance devices where components are densely packed into ever-smaller enclosures. Proper thermal management, of the entire de-

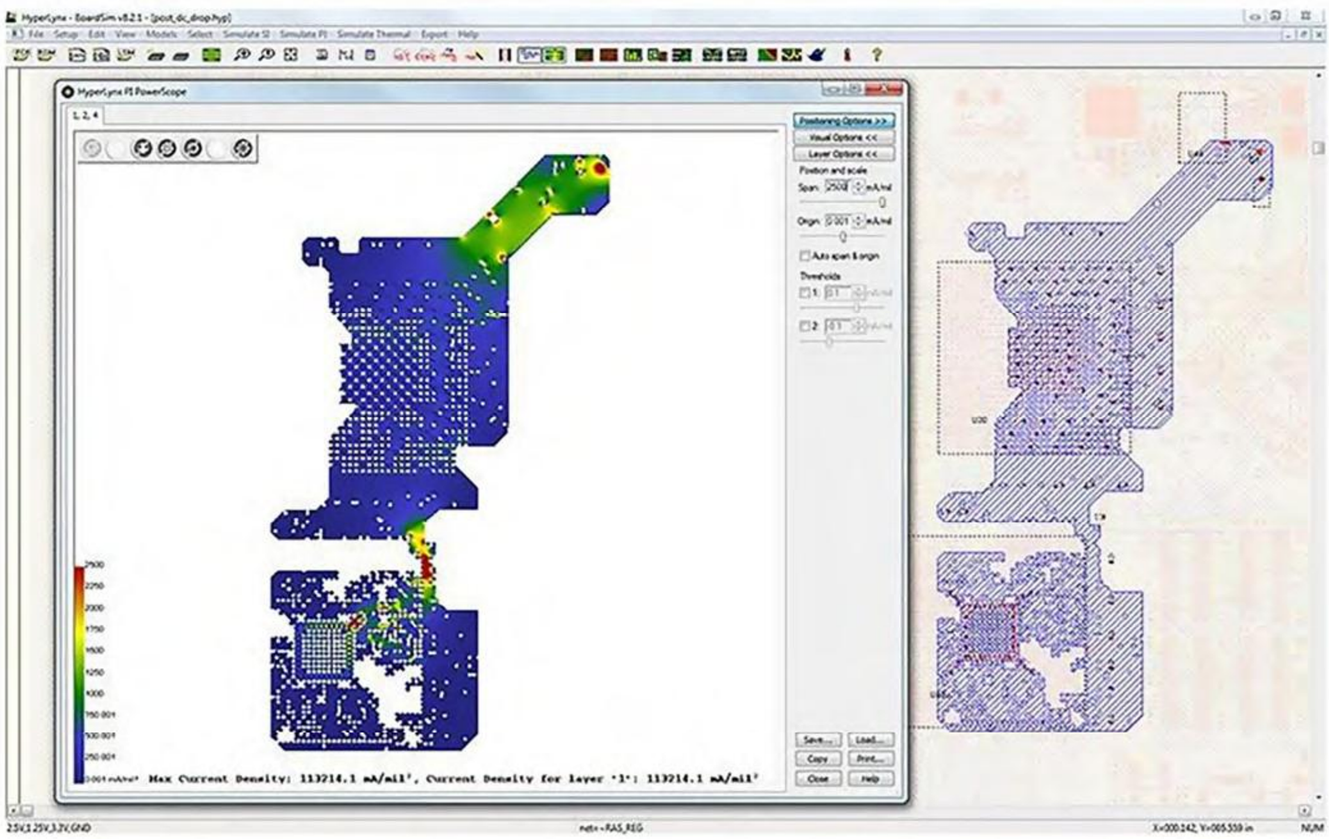

Figure 3: IR drop analysis identifies excessive current density.

sign space, is essential to delivering on-time, on-budget, reliable electronic products.

Also, power delivery and reliability are critical for product creation with today's modern, high-performance digital circuits. Unpredictable, intermittent circuit behavior can be avoided by identifying power delivery issues early in the product creation process. The HyperLynx DC Drop power integrity analysis environment is easy to set up and use, giving simulation results without requiring weeks of software training. Power distribution problems can be identified early in the design, even prior to layout. IR drop issues (Figure 3) can be identified that would otherwise be difficult to spot in the lab, and solutions can be investigated in an easy-to-use "what-if" environment. Once the layout is complete, the results can be validated to ensure that the appropriate guidelines were followed. This will ultimately reduce prototype spins and deliver the product to market faster, while creating more reliable products.

One huge advantage the next generation has in their favor is the availability of information. I recall back in my day, we had to actually buy books and magazines to get technical information. But now, it is just a matter of uttering "OK Google" and a wealth of information is available at our finger tips. Imagine how much we old guys would know now if we had had access to such powerful search engines during our academic years? The even younger Gen Z is the first generation that never experienced the pre-Internet world. To them, a magazine is just an iPad that doesn't work.

Raised on high-technology, the next generation of PCB designer—the independent engineer—is very computer savvy and more adapt to manipulating the application software to achieve the outcomes they need. Given the ever decreasing time-to-market constraints imposed today, these guys are well equipped to tackle a multi-disciplined environment that increases design process efficiency. The younger generation has a lot to offer with new ideas and more efficient processes. Success is built on experience, but with the latest EDA tools that can provide automated features that cover the lapse in experience, the independent engineer will be empowered to push the envelope of design integration and complexity to the next level. **PCBDESIGN**

References

1. Softree: Empowering Engineers with Easy-to-Use Civil Design Software
2. Mentor Graphics PADS documentation
3. PADS software information: www.pads.com

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

Rechargeable Batteries That Last Longer and Re-charge More Rapidly

Materials researchers at the Swiss Paul Scherrer Institute PSI in Villigen and the ETH Zurich have developed a very simple and cost-effective procedure for significantly enhancing the performance of conventional Li-ion rechargeable batteries.

When the battery is in use and thus discharging, the lithium ions pass back to the cathode but are forced to take many detours through the densely packed mass of graphite flakes, compromising battery performance.

These detours are largely avoidable if the flakes are arranged vertically during the anode production process so that they are massed parallel to one another, pointing from the electrode plane in the direction of the cathode. Adapting a method already used in the production of synthetic composite materials, this alignment was achieved by André Studart and a team of research experts in the field of material nanostructuration at the ETH Zurich. The method involves coating the graphite flakes with nanoparticles of iron oxide sensitive to a magnetic field and suspending them in ethanol. The suspended and already magnetized flakes are subsequently subjected to a magnetic field of 100 millitesla, about the strength of a fridge magnet.