Tool Report:

PCB Design

A user-centric look at the state of the industry

By Mike Maisen and Peggy Aycinena

Articles written about design tools, or at least ones under the heading of “Tool Report,” can typically be relied on to offer a summary of whatever technology is in the title, categorizing the companies, what tools they offer, the functions said tools provide, all in a (typically) vendor-oriented manner. This article will attempt to not do that for the sake of conducting an unscientific survey, a kind of “state of the industry” look into PCB Design from a mostly user point of view. Vendor perspective is offered as well, as it’s obviously appropriate to include, but with an admitted bias towards hearing from smaller, more focused PCB tool vendors and less from the larger companies. This bias might make the whole piece useless in some eyes, and this is a good point, but if you read on there’s some interesting ideas.

In the big picture of PCB Design, an always interesting question if perhaps not a practical one is what can or should be done to further integrate the PCB design world with that of chip design. Mike Olivarez, a System Architect from Motorola’s Semiconductor Product Division, spends most of his time developing SOCs and laments that he doesn’t stay current with PCB technology. “We unfortunately don’t keep up with PCB technology, which is probably a bad thing, since an SOC can essentially be considered a PCB Design on silicon.”

Olivarez finds many similarities in the challenges the two design worlds face. “It seems many of the problems [our SOC design group has] are similar to PCB design, but just at a more magnified scale.”

From the PCB side of things, Oliver Engel, President of PCB design company Mettrix Technology Corp (`Hopewell Junction, NY`) sees the issue starting at the university level.

“It’s complicated. The universities train electrical engineers, but it’s not the place to educate board designers. The market demand is for chip designers,” Engel states.

“I know there’s a rift, but the interesting thing is that chip layout designers are not that different from board designers. Many of them, for instance, do not have engineering degrees either. Numerous companies that do just chip layout don’t realize that board layout is quite close to chip layout. Board designers know all about vias and copper and layers. If companies did realize this, they would hire experienced board designers and reconfigure them as chip layout designers.

"The board layout designers don't realize that they could double their incomes by learning to do chip layout. Lots of chip layout guys use Cadence tools or whatever and don't need college degrees."

Good and bad times

In the past several years PCB and Multi-Chip Module (MCM) layout together have been one of the most prosperous parts of the design tool industry. Up until the second quarter of 2001, PCB and MCM Layout growth had posted double-digit revenue increases for 10 consecutive quarters according to EDAC's Market Statistic Services (MSS) Report. In the second quarter however, according to MSS, revenue for PCB and MCM Layout came to $90 million, an 8 % drop from the second quarter in 2000. Though not a drastic downturn when compared to overall EDA revenue growth that was less than 5 % for the 2nd Quarter of 2001, this does signal some cause for worry among tool vendors in an economy that was unsettled even before the September 11 terrorist attacks.

Engel notes that he started seeing a downturn in business in the late summer. "We started seeing a distinct downturn in August. What happened in September didn't help. That's when we started seeing big customers pulling back from projects. People are hunkering down in the industry, waiting to see what's going to happen."

He remains positive though, feeling confident that things will turn around for his business. In fact, when asked about the availability of good PCB Designers circling in the industry today, he seems to have deeper concerns about this than the economy.

"I have trained most of my current employees. They have come to me from a variety of backgrounds, not necessarily technical…and many times are looking to change careers from a previous industry.

"One of the problems is that we are not in Austin or Silicon Valley. If we were in those places, we would have less difficulty finding people to work for us. We are next to IBM here in the Hudson Valley, but they just don't do that much technical work here, so we are hard pressed to find people. We have to work with what's available and hope they will work out."

Engel notes a fundamental difference in recruiting 4 year college graduates for board design instead of chip design,

"It's rare to find someone doing board design who has a college degree in electrical engineering. Colleges don't teach board layout, although some engineers may do layout as part of an overall job. They may do the chip design and want to do the design all the way to the board level. But, in general, board designers, have AA degrees from community colleges. If you get a degree in electrical engineering, companies will hire you to do chip design, not board design."

Design Tools – A user's comments

When asked about the general status of PCB design tools in use today, Engel has a lot to say, starting with the one CAD tool that's essential for PCB design, an autorouter. "Everybody needs an autorouter, they're definitely useful for boards that you couldn't have done manually. For some big boards, we just wouldn't have gotten the job done without the autorouter. For our small jobs, however, we may still be doing the work by hand – if the board is only a few inches by a few inches."

Engel admits that his selection process for choosing his company's design tool inventory wasn't an in-depth one. "We knew the router from Cooper and Chang (CCT) [now part of Cadence], Specctra, was the industry standard and that's what we bought. We didn't research it thoroughly, we didn't have the time.

"Specctra is a fairly complex tool with lots and lots of options. It took some time to learn, but eventually you discover that there are only about a dozen features that you use on a regular basis. I pick up new tools fairly fast, so I learned how to us it."

Engel, like most users, is never satisfied with design tools (all vendor-supplied tools, not just PCB ones), yet his opinions in this context seem more valid than the average user vent-session because he clearly has a detailed understanding of the vendor's perspective as well as the users. "I like to quote the old saw: The problem I have with the software that I use is that software is written by programmers, not users. We sit in front of a tool as users and are always asking: Why does it have this particular feature and not one that we really need? Why would anybody *want* to do a design the way this tools is making us do the design?

"Clearly the programmers are not board designers and don't know enough to write software that actually thinks like a PCB designer thinks."

"However," Engel continues, "in their defense, the CAD tool providers like Cadence write general purpose tools for huge companies like Loral and IBM that allow you to do things your own way. The tools can and need to be customized to meet the needs of the users.

"Sometimes when I think about the tools that are available for chip design, I wish that those types of capabilities were available in PCB design tools. For instance, we could really use ground rule checking for spacing and widths in board design CAD tools. In chip design, you can use design rules to the n-th degree of accuracy, and we could really use that capability at the board level."

Engel adds that hisexperience with a particular interface for a chip design tool developed in-house by IBM, has been better than with the comparable interface of the tool offered by PADs (now Innoveda).

"As another example, IBM makes a nice chip design tool that could be adapted for board-level design. The human interface of the tool is much better than the comparable tool

from PADS, for instance. It would be helping in getting board people to actually use the tool. We get a fair amount of work from IBM, so I know it would be good to have that tool as well."

Vendors and users share the same ideas

Many of the opinions Engel has seem to be shared by tool vendors, many of who seem especially concerned about the makeup of PCB designers and the consequences that their often diverse career backgrounds have on their design experience. David Ross, manager for brand communications at Altium (Sydney, Australia), seems to agree with the characterization of PCB designers as part nomad, part jack-of-all-trades. "The PCB engineer is a multi-disciplined professional, and they often come to PCB Engineering from another discipline. To be trained as PCB engineers they need circuit theory, suitable materials science, and good spatial comprehension."

Ross also points to a broader goal of supporting organizations, not just schools, that help educate designers. "To deal with the current lack of good PCB engineers we need to improve the availability and quality of our educational resources, and we need to assist the formation and growth courses at the educational institutions. An important point to make is that we should also support the industry specialists, such as the standards bodies with their training, as well as the specialist educational institutions, such as the Copper Connection in Santa Clara. These types of organizations provide very focused and up-to-date training, an essential requirement in the electronics industry."

Talking Point: Signal Integrity

Getting back to the specific challenges of PCB design today, a certain buzzword, or phrase, that's becoming increasingly common in PCB Design is signal-integrity. Smaller PCB design companies such as Engel's face a difficult task in keeping their tool inventory, as well as their employees, up to speed on this growing challenge. Taylor Shull, an application engineer specializing in verification and signal integrity at Mentor Graphics (Wilsonville, OR), sees this problem as being difficult to solve for both users and the "commodity" EDA vendors. "High-speed design is done before, during, and after schematic and PCB Entry. PCB Designers and Engineers will need to co-educate themselves in the other's environment. Engineers need to have a vested interested in how the layers are stacked, how the rules are entered, and what the traces look like on a PCB. On the other side, PCB Designers need to have the basics of transmission line theory to understand what needs to be done to make the design work. Due to the complexity of the skills and tools, the one thing that can bridge these is the 'integration' of tools. Commodity EDA Vendors don't have all the tools that need to be combined to facilitate a smooth design process."

Pawel Chadzynski, vp of product development and operations for PCB design collaboration vendor OHIO Design Automation (Nashua, NH) doesn't see the signal-integrity challenge in the same light that Shull does, making the case that the commodity vendors will make inroads into signal-integrity technology because they've already

priced most of their non high-speed technology down so much in competition that they have to tackle and add a new technology to their capabilities. “Definitely, commodity vendors are incorporating elements of high speed design into their tool suites. They have to because so much of their non-high-speed functionality became a commodity with intense price pressures. This expansion is a slow process since so much of it depends on constraint management and SI libraries both of which are quite difficult to implement. In fact, they often require re-architecture of the entire tool suite.” An interesting side-effect of this apparent trend, Chadzynski points out, is the decline of business in the schematic-tool area. “Since all high-speed issues involve physical dimension which schematics do not handle… schematics are becoming less and less attractive as a business for the PCB tool industry.”

Finally, taking the attitude that vendors may be overdoing the intense focus signal integrity problems receive now, Chadzynski wisely points out that there will always be a “new” signal integrity problem, that’s a consequence of technical evolution. “In a sense [a vendor-driven] high-speed solution is a never ending game since what is high-speed today becomes a routine and not-so-high-speed tomorrow and the cycle starts again. This is good news for the tool vendors because it implies a constant need for evolving functionality within the software. Today this essentially means transmission line analysis and the related placement/routing constraint management.

A unifying idea

Olivarez, though not a PCB designer, offers a uniform vision that could be applied to PCB Design big and small. “I can see EDA companies trying to jockey for [position so as to] merge technologies to allow for design at the various levels of abstraction and create a true systems design capability where all element can be simulated, from the board level to the transistor level, depending on how many CPU cycles you are willing to spend.” Whether this vision materializes or not, it’s certain that PCB tool users, much like any other users, won’t wait for such an ideal situation; Engel implies as much when asked about creating his own tools. “We do write our own software. We write little utilities to speed up things that we do over and over again and that can be automated. I’ve got a full-time programmer on the staff and I write some of it as well. We write code in C, or Borland Delphi. We [have] thought about writing software to sell to other board designers.”