A hard part of ASIC verification

One of the most important parts of an ASIC design process just happens to be a board design.

Engineers at the Santa Clara campus of STMicroelectronics know this all too well. This global semiconductor manufacturer focuses on multimedia convergence and power management in many applications. Within the data storage segment, the company designs ASICs that function as the brains inside the latest generations of hard-disk drives.

" I can't depend on a large support team, with separate resources for libraries, design, PCB layout. When you are in charge of this type of project, you do everything. Altium Designer lets you do all the work in one piece of software. "

Vincent Himpe, Systems Development Engineer, STMicroelectronics

One critical step in the ASIC production is testing and verification using a custom development board. Each ASIC needs its own development board, designed and manufactured in parallel with the chip. These boards connect the prototypes to the motherboard and the hard-drive assembly so engineers can put the ASIC through its paces. Any slip in the production of the boards threatens the rollout of the chip, as well the bottom line of the company.

Two designers do it all

Last year STMicroelectronics produced 11 of these ASICs, each requiring its own custom development board for evaluation. Just two engineers were responsible for these boards, pumping out a new one nearly every month.

"I can't depend on a large support team, with resources for libraries, design and PCB layout," explains Vincent Himpe, Systems Development Engineer. "When you are in charge of this type of project, you do everything."

Doing everything means parts library maintenance, schematic capture, simulation, system design, PCB layout and generating manufacturing files. But there is an added dimension. Hard drive assemblies are notoriously difficult for their odd mechanical clearance rules. All the parts are sandwiched between the PCB and drive assembly, which has a non-uniform surface. The engineers need to verify the clearance of each component during the layout process to ensure they will not touch the drive assembly.

Unified environment drives productivity

To keep up with the pace of work, Himpe needed to find a design system that could improve the design process and minimize the amount of rework from project to project.

For years, Himpe used ProTel 99 SE and hadn't upgraded because he'd perceived the learning curve as too steep. "New ways to do certain options took many more mouse clicks," says Himpe. "But as technology kept shrinking more, ProTel 99 SE couldn't keep up. For example, we needed less than one mil resolution."

Himpe continues, "When Altium Designer Summer 09 came out, the integrated library made more sense. DXP had that but I found it too hard to learn to use. Now, I can look up suppliers and order codes, linking directly to supplier's information."

In the one new system, Himpe can now manage everything he needs – the libraries, design and simulation, the PCB layout and generating the Bill of Materials and manufacturing drawings. According to Himpe, "Altium Designer lets you do all the work in one piece of software."

The system's integrated device library is the main reason for reducing unnecessary efforts. "All the information is stored behind the one object – what we need for schematic capture, simulation, PCB layout," Himpe says. "For example, when I move into PCB design, I need to know immediately what the footprint looks like. With Altium Designer, it's all there. That boosts productivity dramatically."

The unified functionality of the system encourages rapid product development and prototyping. All of the tools that Himpe uses are in one cohesive environment. "I have control over the entire project," he says, "and I don't need to leave the tool when going from step to step."

Making the first design error-free

In addition to reducing unnecessary design tasks, Altium Designer helps to make sure the first design is error-free through automatic design rule checking. Himpe can modify the PCB layouts and rely on the software to automatically adjust traces to accommodate design rules that he has defined.

"This capability is particularly impressive to users of older software," he says. "When I was visiting a customer in Singapore, I showed them how easy it was to modify a design. Even though they run 'big ticket' EDA software, they still had to shave their PCB traces by hand. Afterwards, the manager of the engineering group pulled me aside and asked me what kind of software I used. I told him, and now they are evaluating Altium."

The system also automates checking for 3D mechanical tolerances as Himpe creates the PCB layout. To make this work, he has updated his parts library with actual part clearances and entered the hard drive assembly mechanical design into the system.

"I retrieved the mechanical drawings directly from SolidWorks, load up the chassis and turn on 3D checking. Altium Designer tells me when each part will touch the chassis, in real time. I can shift the part over, and the system pushes traces out of the way. It will even show me all the details about a connection just by mousing over. These are the tiny little details that make the life of the PCB layout engineer much easier."

The new way is the easy way

Moving up from 10-year-old Protel 99 to the latest Altium Designer did entail a learning curve. "We had to learn new ways of setting the design process up. We had to remind ourselves that the old, familiar way was still the hard way." Bringing over the design IP was easier – Altium Designer read in all the old files and converted them to the new format in a number of minutes.

Himpe has discovered new ways to save time as he learned the new system. For example, he finds it very easy to reuse a block of circuitry from one design to the next using Altium Designer's "sheet as an object" function. He is also starting to use "design variations" to allow one design to satisfy a range of applications simply by varying the values of specific components in the design.

"We were asking ourselves how to avoid doing the same tasks over and over again from project to project," Himpe says. "Altium Designer helps us avoid all the manual steps that take too long and are error prone. The unified system takes all that away."

About STMicroelectronics

STMicroelectronics is a global leader serving customers across the spectrum of electronics applications with innovative semiconductor solutions. ST aims to be the undisputed leader in multimedia convergence and power applications leveraging its vast array of technologies, design expertise and combination of intellectual property portfolio, strategic partnerships and manufacturing strength. In 2009, the Company's net revenues were $8.51 billion.