cadence®

Designing Complex, High-Speed Boards with Fewer Re-Spins

Imagine Communications and Cadence

About Imagine Communications

Imagine Communications, headquartered in Frisco, Texas, is a leading supplier of live production, playout, multiscreen delivery, and next-generation ad management solutions. Broadcasters, networks, video service providers, and enterprises around the world rely on the company's optimized, future-proof, multiscreen video and revenue enablement solutions every day to support their mission-critical operations. Nearly half the world's video channels traverse Imagine products, and its software solutions drive close to a third of global ad revenue.

Boris Nevelev is a senior hardware design engineer at the company. Nevelev's team works with the PCB design team to develop boards for high-definition video-processing equipment.

Key Challenges

Over the years, the boards that the teams develop have only grown in size and complexity. It's not uncommon for large routers to have boards with more than 50 layers, thousands of differential pairs, long traces, different signal types, and multiple interface connections, for example.

As Nevelev explained, the teams soon outgrew their existing PCB design tools. There were simply too many limitations to what they could do, and having to handle certain tasks manually didn't support their quality or time-to-market requirements. In addition, the teams had simplistic simulation technology that was inadequate for DDR4 and 10G Ethernet. Lacking a way to catch problems with the boards before they reached the fab, the teams had to endure multiple prototype re-spins.

The Solution and Results

The teams evaluated a number of available solutions before choosing Cadence® Allegro® PCB Designer, the Allegro EDM Solution, Allegro Design Entry Capture, Allegro Sigrity Power-Aware SI Option, and the entire suite of Cadence Sigrity™ signal integrity tools. The Sigrity simulation capabilities, tightly integrated with Allegro PCB Designer, sealed the deal.

Challenges

- Meet time-to-market and quality requirements on dense, complex high-speed boards
- Increase team productivity by reducing design iterations
- Eliminate unnecessary board re-spins

Cadence Solutions

- Allegro PCB Designer
- Allegro EDM Solution
  - Allegro Data Manager
  - Allegro Library Manager
  - Allegro Flow Manager
  - Allegro Part Information Manager
  - Allegro Design Authoring
  - Allegro PCB Team Design
- Allegro Design Entry Capture
- Allegro Sigrity Power-Aware SI Option

Results

- Ability to design and deliver larger, more complex, higher quality high-speed boards
- Reduced the number of iterations on large boards from up to four to a maximum of two
- Increased team productivity, with ability to focus more on design enhancements
- Reduced turnaround time by 50% through ability for two engineers to work simultaneously on the same design file
- Configure Allegro EDM Solution upfront for the target application

> *"It was remarkable. To be able to do backdrilling with ease on a big board like that was a dream. Now when we release a board to manufacturing, our confidence is pretty high"*
>
> Boris Nevelev, Senior Hardware Design Engineer, Imagine Communications

With their combination of Allegro and Sigrity technologies, Imagine Communications can complete more complex designs with higher quality, fewer iterations, and in less time. For example, a large design that previously would have required up to four revisions before going into production now needs only one or two revisions. The teams can assess a board comprehensively, rather than having to evaluate critical components manually as they previously did. "I don't have to worry that I missed something," noted Nevelev.

Allegro PCB Designer provides a constraint-driven design flow for concurrent team design, with features including design partitioning, interconnect design planning, and interactive floorplanning and component placement. Allegro EDM Solution provides a collaborative library and design data management environment, while Allegro Design Entry Capture provides the teams with a schematic design tool. The Allegro Sigrity Power-Aware SI Option analyzes source-synchronous parallel buses. With the Sigrity portfolio, the teams have an array of signal integrity analysis as well as board simulation capabilities.

"It's just a pleasure to route and design these boards. We actually broke our previous tool, component-wise and net-wise. Doing these really tight, difficult boards with the Allegro tools is great," said Nevelev, adding, "The Sigrity PowerSI® tool provides an extremely easy and efficient way to extract PCB information into a simulation file. There are quite a few powerful simulation tools on the market, but the PCB extraction portion in the Sigrity tools is probably the best."

Imagine Communications' hardware design engineers can quickly check for issues such as impedance violations, reference plane crossing, and space/coupling violations; optimize via structures; and validate compliance with protocol standards such as Ethernet on the whole board using Sigrity technology. For example, running compliance reports on 10G Ethernet traces before building a board saved a full iteration. Being able to validate integrity and timing by using Sigrity SystemSI™ technology to simulate DDR4 parallel buses—something that's difficult to test on a real board—gave the team confidence in their design. On a design with a large number of DDR4 interfaces, the teams were able to build the board, do one revision, prototype, perform full validation, and move into production on schedule with a high degree of confidence in the design.

"It was remarkable. To be able to consolidate backdrilling depth with ease on a big board like that was a dream. Now when we release a board to manufacturing, our confidence is pretty high," said Nevelev, adding, "One revision was totally unthinkable before."

By working more efficiently, the engineers can focus their efforts on design enhancements. Now, the team can complete a full iteration on a board in a day or two. Previously, the teams had no way to maintain the same kind of schedule with the same level of quality. The teams also appreciate being able to reuse setup components and models in the Sigrity tools, which saves time in subsequent iterations. And using the Team Design Option in Allegro PCB Designer, they can assign two layout designers to work simultaneously on the same file, which essentially cuts the turnaround time in half.

"The quality of our designs is totally different, so it's not just about reducing iterations," said Nevelev. "With our older approach, we wouldn't be able to cover everything with this depth. There's only so much you can do manually or semi-manually. If if takes too long, you start to limit simulation efforts to only critical sections. Now, we can easily enter constraints in Allegro PCB Designer and run our designs against them for results on the fly." The team also expects this improved "whole design quality" will reduce hard-to-diagnose field quality issues related to high-speed signal integrity.

> *"One revision was totally unthinkable before."*
>
> Boris Nevelev

Not only were the designs created with higher confidence and higher performance, but the cost was reduced as well. In previous versions of designs, retimers were used on the high-speed serial interfaces to clean up the signals that move from board to board through connectors. Using the combination of Allegro and Sigrity technologies, high-speed channels are simulated and optimized at the system level to confirm signal quality standards. The tool suite has allowed Imagine Communications to simulate performance with and without retimers before building the board, which saves an iteration of the PCB build to validate the need for retimers.

Lessons Learned

Imagine Communications engineers are based in multiple locations using computers running different operating systems. The Allegro EDM Solution provided a way for the teams to maintain a single library of design data. To make the best use of the tool, the engineers suggest configuring the solution upfront for the target application before rolling it out to the larger user base. The team also worked closely with Cadence support staff and field application engineers to ensure a smooth, fast ramp-up on the tools.

Summary and Future Plans

For Imagine Communications, delivering complex boards with the quality that the engineers are achieving is invaluable. Rather than being frustrated with multiple re-spins or having to address problems with their designs, the company now has a high level of confidence in the boards produced for its global broadcasters and media companies.

cadence®

Cadence Design Systems enables global electronic design innovation and plays an essential role in the creation of today's electronics. Customers use Cadence software, hardware, IP, and expertise to design and verify today's mobile, cloud and connectivity applications. **www.cadence.com**