From August 2008 *High Frequency Electronics*

High Speed Digital Design Benefits from Recent EDA Tool Developments

By Gary Breed
Editorial Director

EDA tools are finally handling both the RF and digital aspects of design, in response to users' needs for better simulation of high speed circuitry, as well as for integrating digital functions into RF systems

Simulation and analysis of RF and microwave circuits using advanced EDA tools has become a well established engineering practice. However, high speed digital circuits have not followed the same path, despite the fact that they involve the same frequency ranges and share common design requirements such as packaging, transmission lines and crosstalk.

With more "radio" circuits being implemented digitally, and with continued increase in clock rates of digital circuits, the realms of RF and digital are now more closely tied than ever before. Those essential EDA tools are also improving how they work with both technologies. And that's what this tutorial note is about, so here are some notes on various software packages.

Key Issues

Simulation of digital circuits began to adopt RF/microwave techniques when clock speeds reached several hundred MHz. The classic "low frequency" techniques using only R-L-C models were no longer capable of accurately representing actual behavior.

In this frequency range, what are sometimes called "microwave" effects become significant. Frequency-dependent dielectric losses are greater. Current flow also becomes nonuniform. With increasing frequency, currents migrate more strongly to conductor surfaces, often becoming concentrated at edges in typical PCB layouts. Radiation and coupling increase with frequency as well.

Perhaps the single biggest issue is packaging—the above effects are sufficient to significantly change chip behavior when it is placed in a package without prior analysis.

As a practical matter, the long-time separation of RF/microwave and digital signal simulation also resulted in great disparity in the way circuit element models were defined, how the models' digital formats were structured, and how the EDA tools interfaced to one another. There has been a gradual evolution from converting common elements with special translation routines between digital and RF/microwave tools, to more recent efforts between EDA tool vendors to harmonize the way they define circuit models. For example, a PCB layout tool simply thinks of traces as bits of copper, but in the high frequency realm, they are transmission lines with specific RF characteristics, including interaction with other metal and dielectric materials.

While there is much progress yet to be achieved, EDA tools have reached an unprecedented level of compatibility with both digital and RF/microwave domains. They can provide valuable design assistance in this cross-domain environment.

Agilent—Digital Signals within ADS

Agilent Technologies' (www.agilent.com/find/eesof) ADS allows you to perform signal integrity analysis of designs such as PCI Express, DDR3, DDR4, RapidIO, Serial ATA, 10 Gigabit Ethernet, and Infiniband.

The Convolution Simulator extends High Frequency SPICE, letting you bring components defined in the frequency-domain, such as *S*-parameter models. Jitter and BER analysis uses the patented EZJIT Plus technology, crossverified with Agilent instruments. Integrated full-wave method of moments and finite-element method EM solvers allow analysis of the effects of physical structures. ADS also supports design flow integration to connect to Cadence Allegro and Mentor Board Station and Expedition layout databases.

The most advanced option in ADS is co-simulation with Agilent's Ptolemy numeric simulator, supporting digital signal processing with SERDES components such as 8B10B and 64B66B coder/decoders, FIR filters, decision feedback (DFE), and feedforward (FFE) equalizers.

ADS' proven RF/microwave simulation allows users to combine digital and analog circuitry within the same simulated system. This makes it possible to do a mixed-signal analysis, such as BER at the digital output versus device bias in the RF chain.

AWR—Signal Integrity Design Suite

Applied Wave Research (www.awrcorp.com) offers a package for simulation and analysis of high speed/high frequency circuits. Signals can be derived from hardware measurement results, IBIS models, or from MatLab™ co-simulation.

Analysis tools include AWR's harmonic balance simulator, EMSight electromagnetic simulator and HSPICE. Because many users have a favorite EM simulation package for their unique application, AWR has the EM Socket open interface that allows inputs from leading vendors. Design flow connects to industry-standard PCB tools like Mentor Graphics BoardStation.

Ansoft—Nexxim Circuit Simulator

Ansoft's Nexxim is simulation software that was created with mixed-signal circuits, high frequency effects, and the large-size and complexity that are characteristics of many high speed/high frequency designs. Nexxim supports HSPICE® and Spectre® models, as well as IBIS and Verilog-A models. User models in C++ are also supported.

One strength of Nexxim is its dynamic link to the popular HFSS electromagnetic simulation tool. When analysis in HFSS is completed, the results incorporated into a single model for subsequent time-domain (e.g. transient) and frequency-domain (e.g. harmonic balance) analysis. As with other companies' offerings, there is a wide range of output plots, including digital-specific ones like BER, jitter, eye diagrams and constellation plots.

CST—Crosstalk Analysis Application

CST's website (www.cst.com) features an article describing work by Cisco Systems engineers, "3D Full Wave Cross-Talk Simulation of Multilayer PCB." The work was carried out with CST MICROWAVE STUDIO® (CST MWS) and summarizes the work by Zianmin Zhang, Kelvin Qiu and Qinghua Bill Chen of Cisco Systems, Inc, presented at DesignCon 2008.

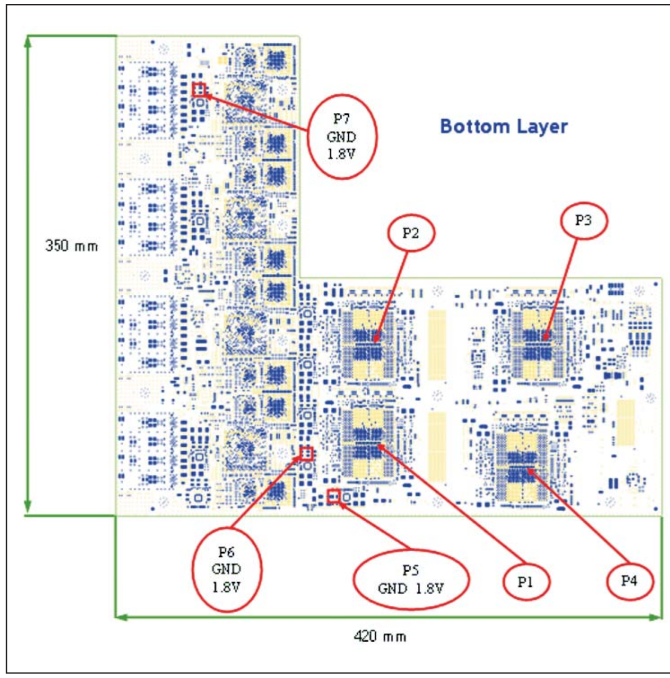

Figure 1—PCB layout and port assignments for crosstalk analysis.

Until recently, designers performing high speed PCB simulations were mostly concerned with finding IBIS models for drivers and receivers, but currently the complexity in high speed signals demands additional types of models, not only for IC buffers, but also for packages, vias and connectors. This is a really challenging task, therefore 3D EM field solvers have to be used in order to generate touchstone files or equivalent circuit models to be used in, for example, SPICE-based circuit simulators.

Figure 1 illustrates a complex multilayer (26 layers) PCB imported into CST MWS by using the ODB++ file, which was directly generated from the Cadence® Allegro® *.brd file, including the port assignment and main dimensions of the PCB.

Summary

It took a long time for the EDA tool vendors supporting digital and RF/microwave technologies to reach the necessary interoperability for truly useful simulation across both realms. Reasons include differences in circuit definitions, model formats and data interfaces, as well as the increased computational load.

Several EDA tools are now available with significant capabilities that cross the analog/digital boundary. While the products noted above are powerful tools for designers, there is still work to be done in models and compatibility across multi-vendor platforms.