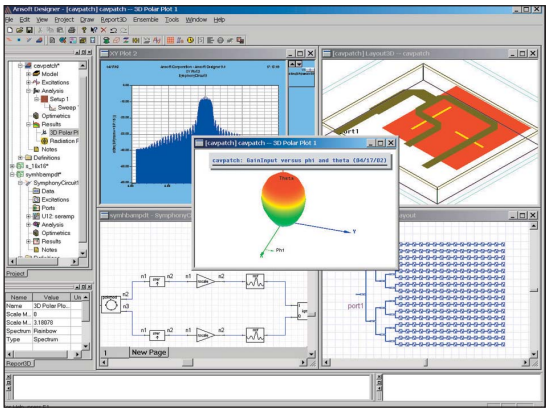
From July 2002 *High Frequency Electronics.*

Design Environment Combines Circuit, System and EM Simulators

Ansoft Designer™ brings together the company's circuit, system and electromagnetic modeling tools in a single system, with design automation features that help speed projects from concept to reality.

Software simulators are intended to make design engineers more efficient. But many software design suites require a long learning curve before a user can become proficient with a set of modeling tools. Ansoft Designer from Ansoft Corp. goes a long way toward solving this problem by combining a powerful suite of design and analysis tools that are accurate and powerful, while also fully integrated and easy to use. The suite's "solver on demand" technology can even automatically choose the best solver for a particular problem based on a user's application.

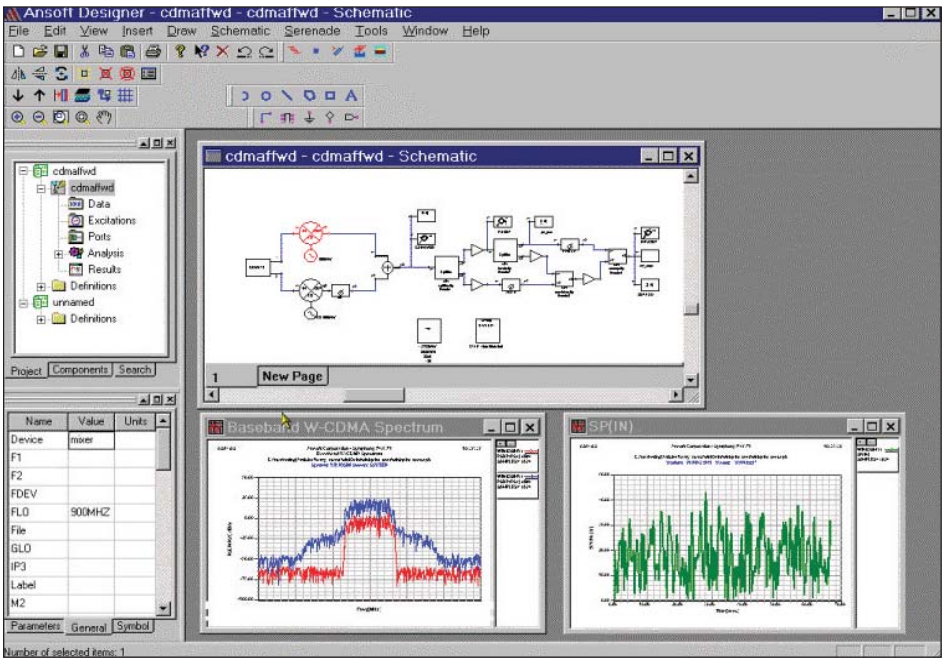


Ansoft Designer can handle a combination of component- and system-level simulations, such as this single and multicell antenna array, co-simulated within a communications system.

Ansoft Designer is a software environment in which circuit, system, and electromagnetic simulators reside and seamlessly interact. Operators can move back and forth among the simulators, or can design a circuit or system and let the "solver on demand" select the tools. When a problem is analyzed at one level, the data representing the solution at that level is available at all other levels through the use of a knowledge-based solution manager. There is no need to re-simulate a design when switching from one level to another, unless the user has made modifications to a design that render the previous solution invalid.

This flexibility stems from the fact that the circuit, system, and electromagnetic solvers in Ansoft Designer all use the same linear circuit components. In addition, schematic capture and layout views are linked, allowing operators to move back and forth between views. And since the linear circuit components can be embedded in any kind of simulation (circuit, system, or electromagnetic), it is possible to develop and optimize a design simultaneously for electrical performance and layout requirements.

Since the schematic and layout editors are linked, a component added to a schematic diagram as an electrical symbol automatically appears in the layout editor as a layout symbol. System-level behavioral models can be accounted for in a layout as precisely sized footprints. In addition, complex systems with multiple layouts (such as multichip modules) can be saved as a single file, to simplify archiving. A three-dimensional viewer assists those working on multilayer designs by allowing circuits to be rotated, sliced, and zoomed in and out.



This screen capture shows the W-CDMA standard applied to system- and circuit-level heirarchical simulation of an RFIC.

Ansoft Designer features a large library of active and passive component models. In addition, users can create their own models by defining the current-voltage, charge-voltage, and noise equations for a generic component. These user-defined models can be created in a schematic diagram without tedious code generation and compilation. Operators may also place electrical elements with no physical counterparts, such as RF ports and parasitic capacitances, directly into a layout as a component symbol.

Once a user has created a schematic and/or layout, Ansoft Designer provides a wealth of analysis tools for predicting performance, including time-domain, frequency-domain, and system-level analysis to perform multiple simulations on devices, circuits, or systems. Its high-frequency circuit simulator provides fast, robust convergence to accurate solutions for large communications networks and high-speed computer designs. A harmonic-balance simulation engine with Krylov iterative subspace solver efficiently solves problems containing large numbers of nonlinear components and unlimited RF tones. Digital modulation analysis capability (FastACPR™) offers a fast and accurate approach to distortion characterization and spectral regrowth for circuits operating with analog and digitally modulated RF signals. The software includes preconfigured modulation sources, such as W-CDMA, GMSK, pi/4DQPSK, PSK, QASK/QAM, EDGE, and CDMA.

The system-level capabilities of Ansoft Designer provide time-domain, frequency-domain, and mixed-mode analysis for arbitrary system topologies. The system tools can simulate most modern communications parameters, including adjacent-channel power, bit-error rate (BER), crest factor (peak-to-average power ratio), error vector magnitude (EVM), and signal-to-noise ratio (SNR). Also available are comprehensive noise analysis capabilities, which enable simulation of the spectral distribution of a circuit's noise power under small- or large-signal conditions. Using source- and load-pull capabilities, circuits can also be evaluated with different terminating impedances.

Method-of-moments (MOM) three-dimensional planar EM simulation is provided, along with integration with Ansoft's well-known High-Frequency Structure Simulator (HFSS), a three-dimensional, finite-element full-wave electromagnetic simulator. The planar electromagnetic simulation capability allows circuit and system designers to include the electromagnetic field effects of high-frequency structures, such as via holes, coupled lines, spiral inductors, interconnections, filters, and patch antennas. HFSS considers three-dimensional geometries and material variations, and is supported by a wide range of analysis and optimization algorithms. Advanced post-processing techniques allow operators to look inside components for additional performance insights.

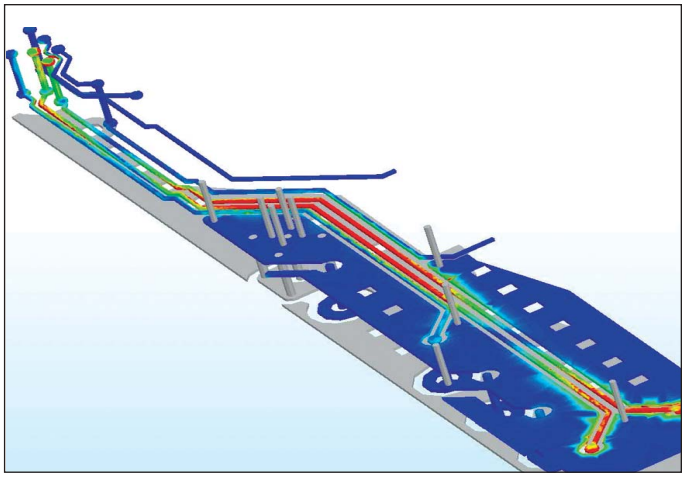
Ansoft Designer also includes transient-analysis capability, with its inclusion of Full-Wave SPICE, a proprietary technology that provides efficient analytical transformations between EM and circuit levels. It uses reduced-order models of the dominant poles and zeros of the EM system to provide broadband fast-frequency sweeps and rapid detailed transient waveforms at the circuit level.

Full-Wave SPICE is ideal for simulating transient events, such as oscillator start up and amplifiers under power-up conditions. It includes an embedded convolution engine to handle frequency-domain models or *S*-parameter data files and a modulation-based harmonic-balance simulator (a circuit-envelope simulator) for analyzing designs with complex waveforms.

Ansoft Designer supports a PSPICE netlist syntax that will accommodate the direct use of many existing SPICE libraries. Ansoft has also developed links into the most commonly employed CAD tools via the AnsoftLinks interface. The Ansoft Neutral file format supports Ansoft-developed translators to and from IC and PCB EDA such as Virtuoso and Allegro from Cadence Design Systems and Board Station from Mentor Graphics.

Ansoft Designer is available for both personal computers and UNIX workstations. It is designed for use with most leading operating systems, such as Microsoft Windows NT 4.0 (SP6 or higher) and Windows 2000 Professional (SP2 or higher).

Ansoft Corp.
Tel: 412-261-3200
E-mail: information@ansoft.com
www.ansoft.com.
HFeLink 302



This interconnect model of a high-speed package can be analyzed with the proven electromagnetic simulation tools in Ansoft Designer.