

Figure 1 A user can directly model circuit traces, or the layout package can feed back the physical configuration (a). The field solver uses this input (b) to calculate the fields, voltages, and currents (courtesy Applied Wave Research).

Figure 2 A design flow includes a field solver to better simulate real-world issues, such as board and IC layout or component interaction.

Figure 3 Field solvers can provide results for intractable problems involving nonlinear magnetics as well as complex 3-D spiral inductors (courtesy Comsol).

Figure 4 A full-blown design environment allows schematic entry, time- and frequency-domain simulation, layout, and field solvers for signal-integrity and RF design (courtesy Applied Wave Research).

Figure 5 The legacy Orcad layout tool can export to the Hyperlynx the file format from the session shell.

Figure 6 This scope shot shows one eye in an eye diagram (courtesy Tektronix).

Figure 7 Russian naval officer Alexei Krylov invented the mathematical technique that modern harmonic-balance simulators use.



FIELD-SOLVER SOFTWARE CAN PREDICT THE BEHAVIOR OF CIRCUITS THAT OPERATE AT SPEEDS AND DENSITIES BEYOND THE CAPABILITIES OF SPICE TO PROPERLY MODEL. WHETHER YOU ARE TRYING TO VERIFY SIGNAL INTEGRITY OR DESIGNING AN RFIC, YOU SHOULD KNOW HOW FIELD SOLVERS COULD HELP YOUR DESIGN.

BY PAUL RAKO • TECHNICAL EDITOR

BEYOND SPICE

Spice is a popular circuit simulator. As electronic designs move to higher frequencies and occupy less space, Spice is often insufficient to predict the behavior of even simple circuits. When this scenario occurs, engineers must turn to the use of field-solver software. Field solvers, sometimes called EM (electromagnetic)-field solvers, calculate Maxwell's equations for a physical representation of your circuit (**Reference 1**). You must provide the field solver with a representation of the physical configuration of the circuit, which you base on the output of either an IC- or board-layout package or a solid model. The solid model also must

include the properties, such as resistance and dielectric constant, of all the materials in the area of interest. **Figure 1a** shows the modeling of a transmission line, and **Figure 1b** shows the resultant field distribution. Note that field solvers do not solve the quantum-mechanics problem of what occurs in transistors or other active devices. Field solvers instead focus on traces, inductance, capacitance, and the way these devices interact.

SPICE AND ITS LIMITATIONS

The Spice software tool tries to pre-

dict the performance of electronic circuits. It works in the time domain—that is, it solves for the voltage and current at small, discrete time steps and then displays a transient simulation, which looks similar to an oscilloscope trace. When you want a frequency response, it solves the time-based problem at a set of frequencies and displays the result as a Bode Plot. The best explanation available for the operation of Spice is *Inside Spice: Overcoming the Obstacles of Circuit Simulation* (**Reference 2**). **Reference 3** provides a good online summary of

Spice. Spice is famous for the headaches it can cause design engineers (see **sidebar** “The trouble with Spice”).

Note that Spice does not solve for physical things, such as board layout and signal interactions. The schematic has no data to convey that information; it is simply a collection of lumped elements and subcircuits. The vagaries of Spice output became clear 15 years ago when not one of five Spice programs agreed with the performance of the actual circuit (**Reference 4**). Spice does have transmission lines and sometimes even lossy transmission lines. These lines represent a lumped-element subcircuit with a parameter to account for the length of the line. Spice does not base the calculation of the lossy characteristic on the layout but simply accounts for it in a formula that bases the prediction of losses on simple parameters. Spice also fails to model physical phenomena, such as skin effect, although lumped-element subcircuits can approximate these phenomena (**references 5 and 6**).

Field-solver programs, which can model physical phenomena, now enter the picture. Thomas Quan, vice president of marketing for Applied Wave Re-

search, notes: “As operating frequency increases in analog circuits, analog designers will benefit greatly from employing high-frequency-design techniques and tools that have been in use by microwave designers for many years.”

A field solver calculates how a strip-line or spiral inductor will act. The user enters a virtual representation of the device as a solid model. This solid model can come from a layout package, or the user can draw it using the field-solver program itself. Many field solvers accept solid models from mechanical-CAD software such as Solidworks and Unigraphics. In addition to the physical shapes of the circuit elements, the field-solver software must know the electrical properties of the material the design uses. This requirement includes not just the conductance of copper and aluminum, but also the dielectric constant of the pc board, glass oxide, or solder mask that is near the conductive elements. At the extreme, the solid model could include the glass fibers inside an FR4 pc board as well as the solder-mask marking that may lie on top of a trace. The IC model could include not just the glass oxide on which you place a spiral inductor, but also the metal layers underneath and the epoxy that encapsulates the die in a finished package. For designs operating at high frequencies, all these factors will matter in how the circuit behaves.

Spice creates a conductance matrix from a circuit netlist and attempts to solve that matrix over various time steps. Field solvers use meshing techniques, much like finite-element-analysis programs

AT A GLANCE

- ❑ Spice cannot accurately solve high-speed or small, interacting circuits.
- ❑ Field solvers calculate Maxwell's equations for any passive circuit.
- ❑ Field solvers provide time- or frequency-based data to a simulator.
- ❑ Modern simulators can solve in the time or frequency domain.
- ❑ You use field solvers for signal-integrity and RF design.

for mechanical stress and strain. Once the software meshes the virtual physical representation into small elements and the user provides the dielectric constant and conduction of those elements, the field-solver program uses various mathematical techniques to deduce the fields and, hence, voltage and currents that are associated with the circuit element. You should remember the admonishment of Henry Ott, a noted authority on signal integrity: “Remember: Fields make currents, not the other way around.”

When the field solver has examined the physical representation of the circuit, it can then output an electrical representation of that configuration as either an S-parameter or LRGC (inductance/resistance-conductivity/capacitance) model. The S-parameter model is a frequency-domain representation, and the LRGC model is a representation for time-domain analysis by Spice engines. Even if there is only an S-parameter output, programs can perform a mathematical convolution to achieve an inverse-Fourier

transform. This step gives a time-domain representation of the network more suitable for solution in Spice, although at a risk of inaccurate results.

FIELD-SOLVER SHORTCOMINGS

On the downside, field solvers can cause several design problems. The field solver must use an adequate mathematical method for the proposed problem. If you use a 2- or 2.5-D solver in designs having 3-D fields, then expect to be suspicious of the answer the field solver produces. As always, the universal software problem of garbage-in/garbage-out also arises. If you incorrectly model the structures or incorrectly enter the dielectric constant or other properties, then the field solver is hardly to blame when you get inaccurate answers. There are also sophisticated mathematical problems that may not be a function of the field solver itself but, rather, how the simulator converts an S-parameter frequency-domain output from a field solver to a time-domain representation for use in signal integrity or any non-steady-state analysis. Doing a convolution on the S-parameter data performs the inverse-Fourier transform. This transform does the conversion from the time domain to the frequency domain. Rational-fitting algorithms, or state-space equations, may also perform the transformation from the frequency domain to the time domain. Two problems can arise if the algorithms are inadequate for the S-parameter data set: causality and passivity.

If the field solver violates causality, a time-domain model that you generate

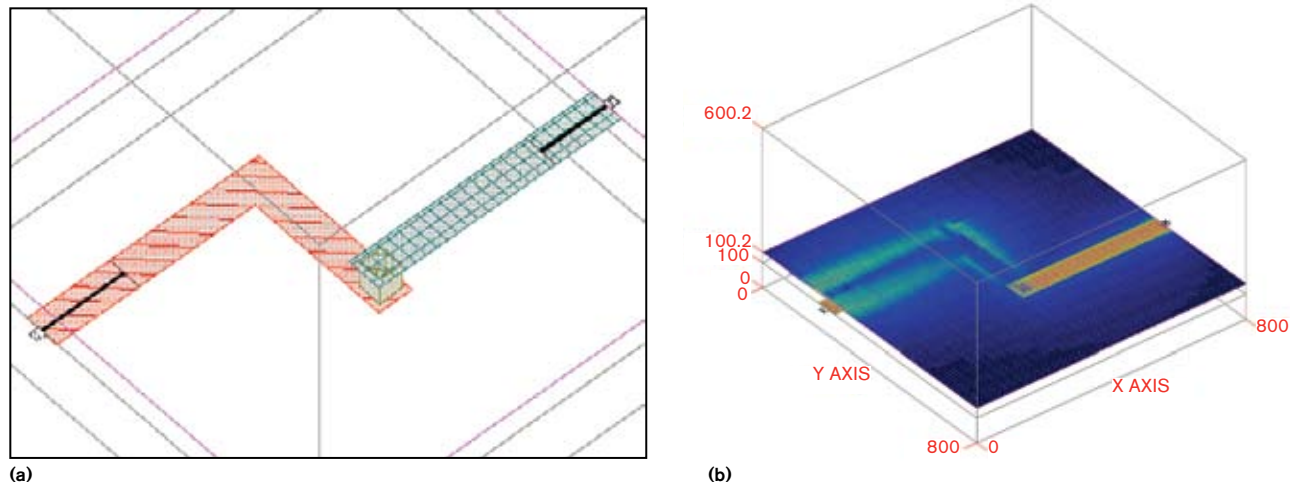


Figure 1 A user can directly model circuit traces, or the layout package can feed back the physical configuration (a). The field solver uses this input (b) to calculate the fields, voltages, and currents (courtesy Applied Wave Research).

from the S-parameter data may behave erratically. For example, a negative-going reflection may appear instantaneously when you launch the signal into a trace or a conductor. This behavior means that the model has neglected to account for the finite transit time the wave takes to reach any reflection-causing discontinuity. Problems with causality result in inaccurate results. Passivity arises when the transformation creates a time-domain model that adds energy to the circuit despite the fact that no active devices are in the subcircuit. Problems with passivity cause the simulation to blow up and fail to converge.

These problems involve complicated mathematical algorithms and should remind diligent engineers that they are designing for the real world. Without real-world verification, problems will occur because of too great a reliance on computer simulations. You need three tools to validate field-solver outputs: experience, a VNA (vector-network analyzer), and TDR (time-domain reflectometry).

You should have filed away in your

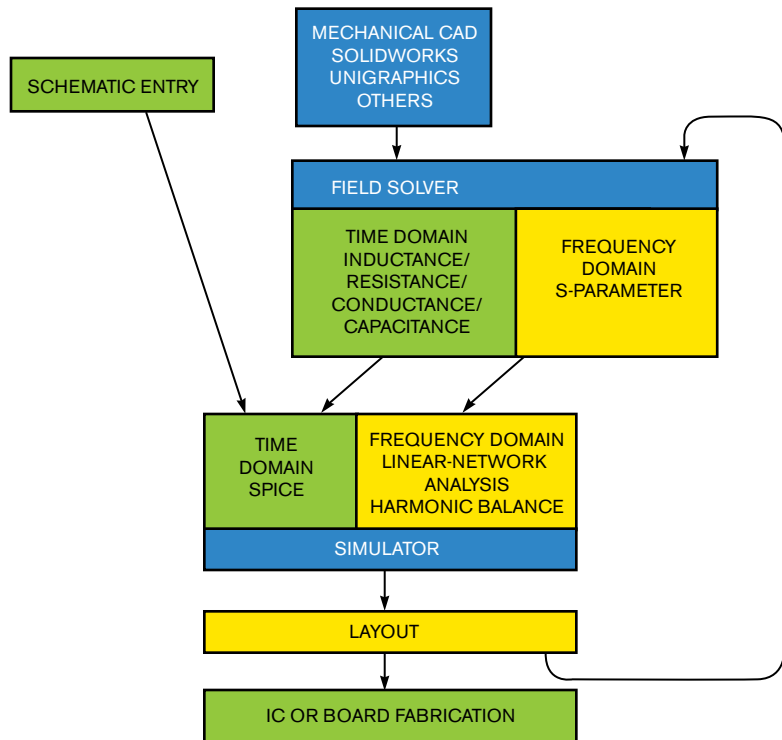


Figure 2 A design flow includes a field solver to better simulate real-world issues, such as board and IC layout or component interaction.

THE TROUBLE WITH SPICE

Spice is a powerful tool that can solve intractable problems. It solves Kirchoff's laws in the time domain. Spice uses matrix math to solve the conductance matrix of the circuit, which the schematic netlist describes. The program iterates from an initial first guess until it gets a result close enough to declare that it is accurate; it then moves to the next time step and repeats the process. Spice models networks of resistors, inductors, and capacitors as simple elements or, better yet, with subcircuits that reflect equivalent series and parallel elements, as well as temperature effects. It models active devices as combinations of passive devices and controlled voltage sources and current sources. It also allows you to insert equations into models to "fudge" for noise and similar effects that are too hard or indeterminate to model.

However, Spice runs into trouble in modeling. The results of Spice are only as good as the models. IC designers swear by Spice, often with disastrous results. Almost all board-level designers have had unpleasant experiences with models. As a result, they are more suspicious of the results and usefulness of Spice. IC designers get to use models that cost tens and even hundreds of millions of dollars to ensure that the Spice model matches the results of the actual transistors and passives in the process. Even with these efforts, temperature or proximity effects often make the first silicon behave unpredictably.

"Spice lies—or fails to tell the truth—all too often," says Bob Pease, staff scientist at National Semiconductor. "Use your head when at all possible. Design using pencil and paper, rather than any simulation. And, no matter what Spice tells you, you do not have to believe it. On the other hand, Spice might tell the truth. But, for you to say that, you have to analyze what the circuit will do all by yourself. Otherwise, how would you know what the truth is?"

Engineers contemplating the vagaries of Spice output must assume that the program will give an output, which is often not the case. Spice must converge on every attempt to solve the conduction matrix and every time step it takes to give a response. The problem is that Spice bases the matrix math on linear solutions, and diodes, just like transistor physics, are not linear. Spice must guess an operating point on the exponential curve that is the diode IV chart. As a result, it often fails to converge when a small movement on the curve in the I or the V axis results in a large change in the other variable. If each subsequent guess the program makes does not result in a smaller error, then Spice returns with the dreaded "failed-to-converge" error. Because this iteration procedure has to occur for each time step, a huge number of chances for nonconvergence exist. When Spice attempts to solve nonlinear-magnetic elements, such as transformers, the convergence problems become even more daunting.

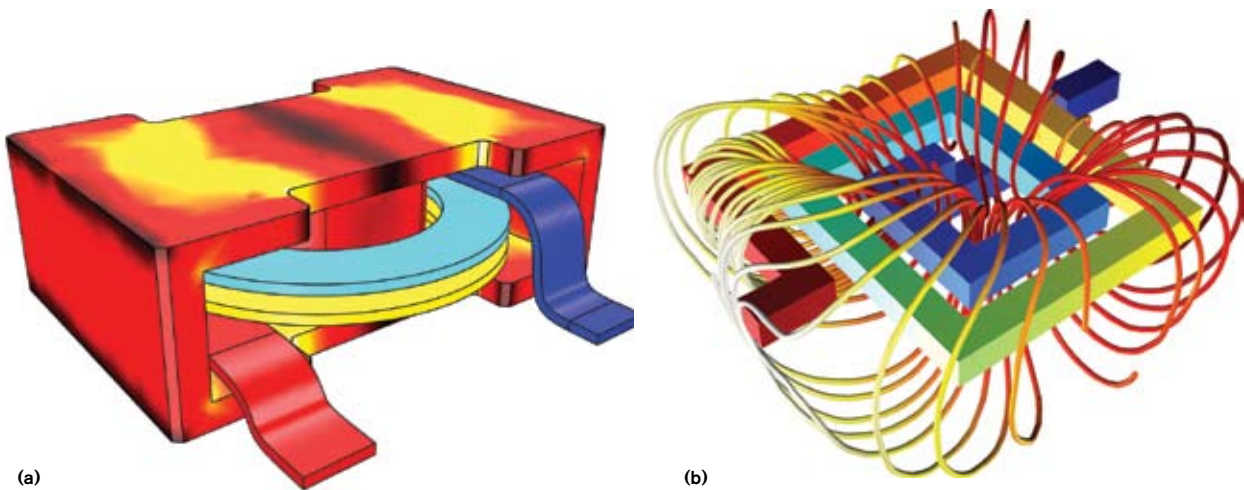


Figure 3 Field solvers can provide results for intractable problems involving nonlinear magnetics (a) as well as complex 3-D spiral inductors (b) (courtesy Comsol).

mind any circuit block, whether on an IC, a board, or a cable, that you have used previously. If the simulator shows a block's acting in an unpredictable way, you should investigate and resolve the conflict. Andy Mastro, former vice presi-

dent of Teledyne CME, performed finite-element analysis starting when the input was a “deck”—of IBM Hollerith cards. “I have never done a single analysis where the answer was right the first time with my initial assumptions,” he reports.

“There was always a need to evaluate a problem with a known or a simple solution just to verify that the software was acting correctly.”

A second technique to check up on your field solver is using a VNA. A VNA

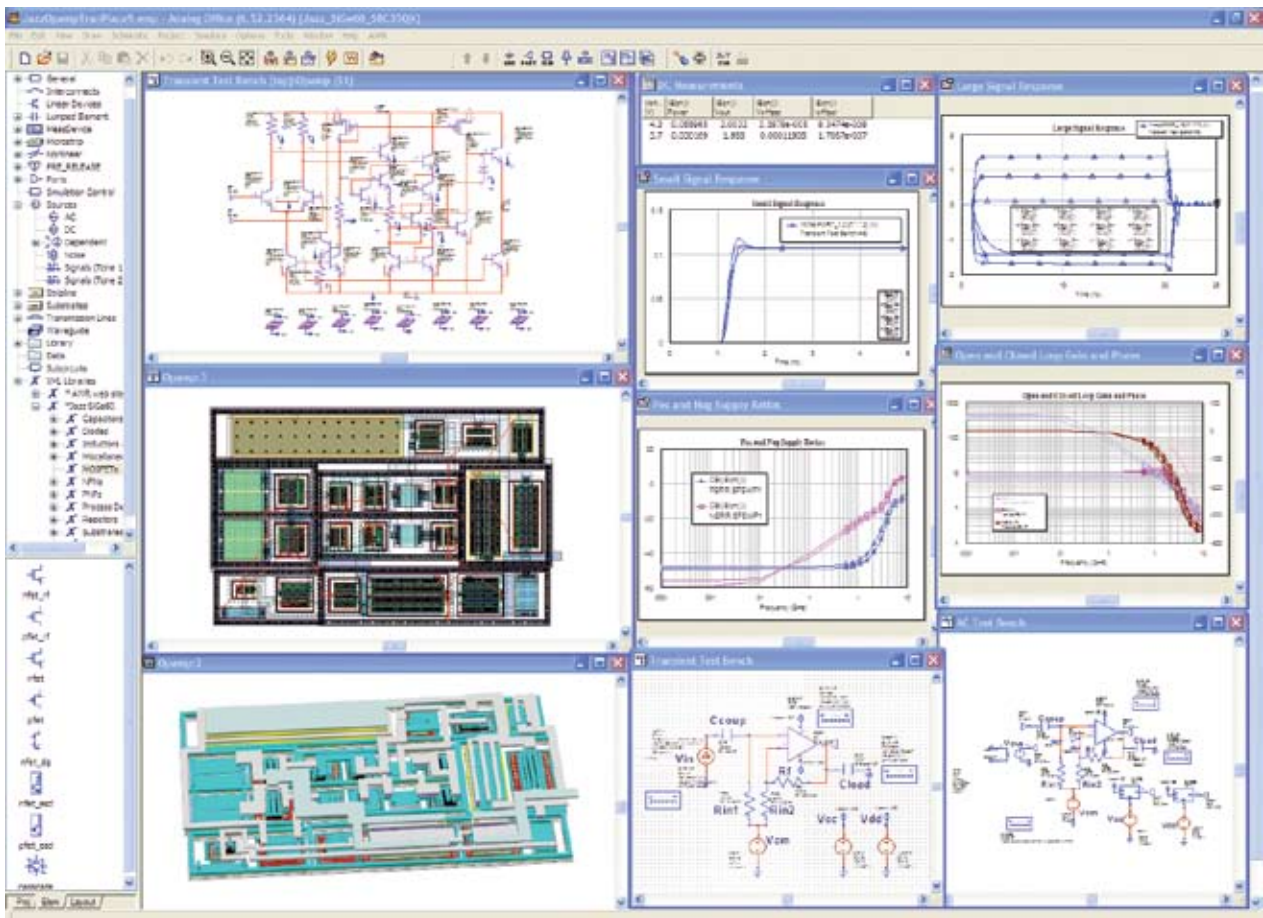


Figure 4 A full-blown design environment allows schematic entry, time- and frequency-domain simulation, layout, and field solvers for signal-integrity and RF design (courtesy Applied Wave Research).

is similar to a spectrum analyzer, but it also records the phase changes of the signal as it passes through a system. By either adding on a separate box or buying an analyzer that has the function built in, the VNA can measure S-parameter data over wide frequency ranges. The 200-MHz, “green-machine” HP 3577A can use the HP 35677A/B to take S-parameter data in 50 or 75Ω systems. For higher frequencies, the 8753ES can work to 3 or 6 GHz and has a built-in S-parameter box. Use caution with instruments such as the Tektronix DSA8200, which does not work like a typical VNA. A traditional VNA sweeps a tuned filter with a narrow bandwidth across the frequencies you are measuring. This technique eliminates out-of-band noise and provides dramatic SNR benefits. The fast DSA8200 scope calculates S-parameters by performing Fourier transforms on a set of time-domain-sampled data to generate frequency-domain results. By its nature, the DSA8200 must be a wideband instrument, meaning that it also has to live with wideband noise, limiting the accuracy of measurements. It would be absurd to use complicated math to convert time-domain data to frequency-domain data, only to have your simulator take the inverse transform to get back to the time domain.

The DSA8200 really shines in TDR measurements. This technique sends a fast rise-time pulse down a transmission line or structure, such as a ribbon cable or another circuit element. It then records the reflections that impedance discontinuities generate. The vertical divisions on the scope when in TDR mode represent impedance, not voltage amplitude. An older tool for TDR measurement is the Tektronix 11801A/B/C series scope with the SD-24 plug-in. Agilent offers the 86100A model, and no discussion of TDR testing should omit the fast-pulse tools from Picosecond Pulse Labs. The company’s single-ended 4020 and differential 4022 models set the standard for the industry. The TDR data can run through a program such as Tektronix’s (formerly, TDA Systems’) iConnect software to generate time-domain Spice models that you can compare with the models that the field solver creates. Note that you can enter data from real-world objects directly into the simulation en-

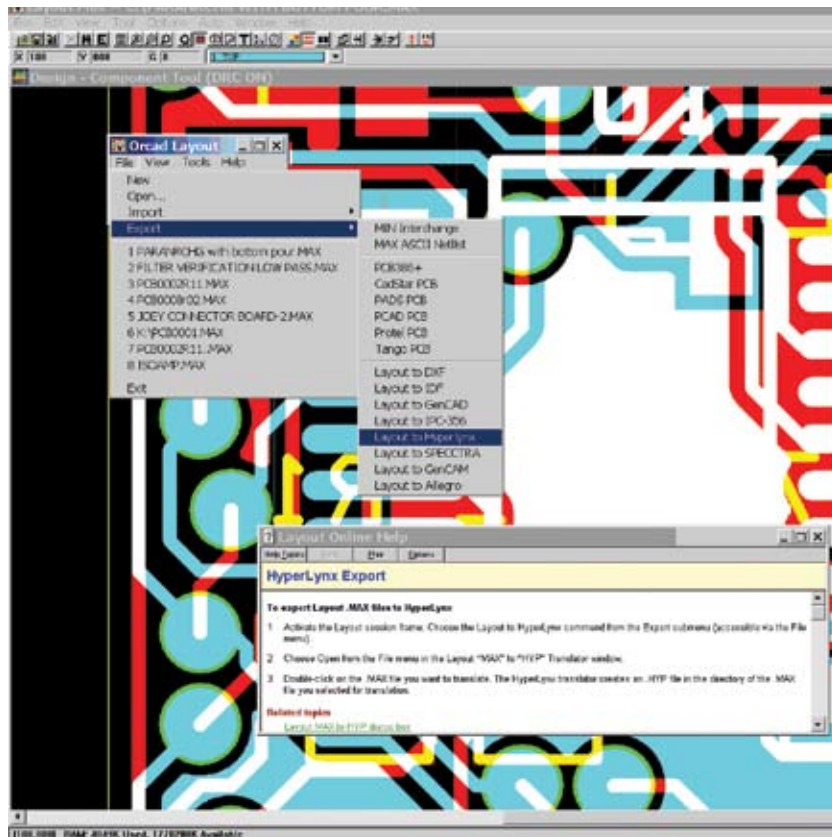


Figure 5 The legacy Orcad layout tool can export to the Hyperlynx file format from the session shell.

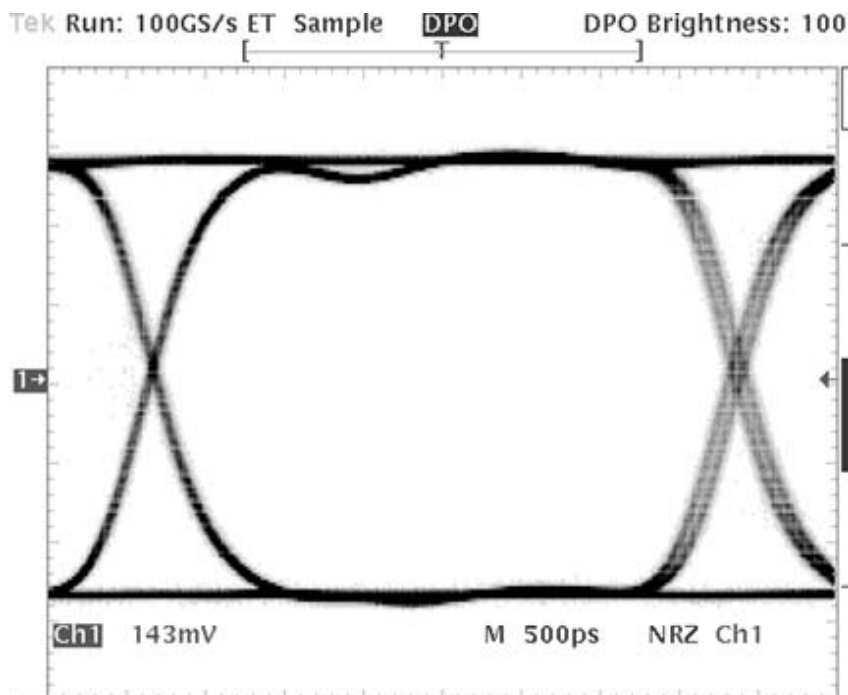


Figure 6 This scope shot shows one eye in an eye diagram (courtesy Tektronix).

vironment, thereby sidestepping the field-solver approach.

TYING IT ALL TOGETHER

Most engineers do not use a field solver as a stand-alone tool but, rather, as part of a simulation environment. This environment provides various mathematical techniques that accept a combination of ideological inputs from the schematic netlist, as well as some layout or physical modeling from the field-solver module. Ben Mika, design engineer at On Semiconductor, states: "Our designs operate at 3 to 5 GHz; therefore, [you must include] the parasitic contributions of interconnects and long-range couplings between inductive elements, both intentional and parasitic ... in the simulations at an early stage ... to make any meaningful prediction about the circuit performance." The simulator can have Spice, LNA (linear-network-analysis) harmonic balance, and other simulators. LNA techniques input S-parameters or Spice models from real-world networks to the simulator based on data from VNAs or TDR. A designer may also contribute "fudge" factors based on experience or design safeguards. **Figure 2** shows how a field solver can fit into a signal-integrity or RF-design flow. The user inputs the schematic, and perhaps a mechanical group supplies 3-D CAD models for the field solver. The simulator can teach the user to change the schematic, but two levels of abstraction are available at the first pass. For instance, you could use a fixed inductance to represent a spiral inductor or a slightly more complex lumped-element Spice model that takes into account that the inductance will change with frequency.

Alternatively, you could enter the physical representation of the inductor into the field solver as a 3-D model, or you could get the 3-D CAD data from a mechanical package, such as Solidworks. You can use that representation, along with user-entered data about dielectric constants and conductance, to generate an S-parameter or an LRGC model that represents the inductor better than a fixed-inductance val-

ue. After simulating and perhaps changing the schematic or some schematic values, the user can then lay out the IC or board. At this time, the user can feed physical-layout information of metal traces back to the field solver so that it can model the interconnections. Better yet, the field solver can solve for mutual effects between conductors and spiral inductors. The user can then send this information back into the simulator for time- or frequency-domain simulation. This approach yields a design that has a far greater chance of meeting all the design criteria and behaving properly.

The field-solver simulation environment can support two broad categories of design: signal integrity and RF design of an IC, a module, or a board. In the context of providing useful design work, you should view a field-solver program as a component of the total simulation environment.

Several companies make high-end simulation environments for signal integrity. Cadence, with its Spectre simulator, dominates the IC-design market. Mentor offers Eldo for RF design, and Synopsys offers the well-known HSpice environment. Synopsys also offers HSpice-RF, which adds harmonic-balance and frequency-domain analysis to the traditional time-domain Spice tool. Ansoft, with its Nexxim simulator, is also well-known in simulation. The simulator plugs into the company's Designer SI GUI that can then call on Ansoft's well-respected HFSS (high-frequency structure simulator) 3-D field solver or the company's SiWave or Q3D solvers.

Perhaps the most sophisticated field solver is the Comsol multiphysics solver. Comsol chose to solve partial differential equations for the problems as a general case, allowing users to employ the company's solver for fluid flow, structural mechanics, heat transfer, and EM. The multiphysics capability enables the solver to solve for the EM fields of materials that are also changing properties or shape due to thermal expansion or magnetic saturation. **Figure 3a** shows the field solution for a power



Figure 7 Russian naval officer Alexei Krylov invented the mathematical technique that modern harmonic-balance simulators use.

inductor, and **Figure 3b** shows the fields for a spiral inductor. The program can read and operate on Spice files, although the Spice engine has fewer features than those from Cadence, Ansoft, Synoptics, Applied Wave Research, and Agilent.

Coming in from the microwave world, but able to handle signal-integrity issues as well, Agilent offers both the enterprise ADS environment and the desktop Eagleware product that targets individuals and small companies. Applied Wave Research offers Microwave Office and Analog Office, a product that supports signal-integrity and frequency-domain analysis (**Figure 4**). The MathWorks has just announced an RF-design and signal-integrity package for its Matlab environment.

ELIMINATING JUST ONE SPIN OF SILICON FOR AN IC OR ONE REDESIGN OF A COMPLEX BACKPLANE OR DAUGHTERCARD CAN OFFSET THE COST OF ALL THE SOFTWARE AND HARDWARE.

PC-board designers have long grappled with signal-integrity issues. In this area, the well-known brand is Mentor Graphics' Hyperlynx 2.5-D field solver that works intimately with Mentor's PADS brand of schematic and layout tools. It also accepts output from Cadence's Orcad layout tool, which exports to Hyperlynx format (**Figure 5**). Do not confuse the Orcad layout program with the Orcad Editor suite, which uses a stripped-down version of the enterprise-class Allegro board-layout package. When using Orcad Editor, it makes more sense to use Cadence's signal-integrity package, which works seamlessly with Allegro. One notable board-level product that includes signal integrity is Altium's Designer package, which costs less than \$5000 for a single license and \$5000 more for the schematic-entry and Spice packages. In contrast, the Hyperlynx tool alone costs \$5000 to \$48,000, depending on fixed- and floating-license requirements as well

as functions, such as lossy-transmission-line support. All the "big-iron" environments from Cadence, Ansoft, Synopsys, Agilent, and others have prices starting at approximately \$5000 for simple modules, but users can expect to spend \$50,000 and up to get a complete simulator with field-solver and layout functions. The desktop-class products such as Microwave Office and Eagleware generally cost \$5000 to \$25,000 for a usable system. One remarkable exception in the vagaries of pricing is Zeland Software, which provides a full pricing sheet for all the packages and suites that it sells.

Scads of third-party tools exist for high-end design environments, such as those from Cadence. These tools can plug a field solver or other specialized simulator into the design flow. Indeed, Ansoft has taken care to ensure that you can separately plug all of its parts, including the simulator, layout package, and field solvers, into Cadence's and other companies' design flows. IBM even offers the free, "method-of-the-moment," 2-D CZ2D field solver (**Reference 7**). IMST (Informatik und Mikrosystemtechnik) offers its Empire field solver that runs under Windows and Linux. Helic's VeloceRF RFIC-design tool plugs into current tool flows. Sonnet Software offers a 2.5-D field solver that plugs into many design environments, including those of Applied Wave Research. Zeland offers field solvers, Spice tools, transmission-line tools, and filter-design tools, as well as signal-integrity support. The test equipment you need to verify the field solvers costs approximately \$2000 on eBay to more than \$60,000 for the newest and fastest VNA and TDR models. Microwave and signal-integrity design is not for the faint of heart. Bear in mind, however, that eliminating just one spin of silicon for an IC or one redesign of a complex backplane or daughtercard can offset the cost of all the software and hardware.

When you use field solvers for signal-integrity problems, the stimulation is in the time domain. In this case, a common requirement is measuring an eye diagram (**Figure 6**). In the real world, you obtain this diagram by running a pseudorandom generator into the trace, connector, cable, or system and looking at the receiving end with a scope that can capture and store the repeated traces overlaid on one another. The thickness of the signal

crossing indicates jitter; the bigger the eye diagram, the better the signal integrity. Simulators working with field solvers can simulate this diagram and give designers an indication of how well the interconnects will perform in real life. You can use this channel modeling to evaluate not only signal integrity in the case of eye diagrams, but also the channel itself on more basic analog-performance criteria. If the time-domain analysis shows no reflections along the entire length of the channel, then you can rest assured that any analog signal will be of higher fidelity than if there were significant reflections.

The field solver can output LRGC models directly to the Spice engine, or the simulator environment can convert S-parameter files from the solver to a time-domain representation with the convolution operation. "Signal integrity is all about the time domain," says Sanjeev Gupta, an application engineer at Agilent. "The models are [in the] frequency domain because, at these frequencies, dispersion and radiation take place,

SIMULATORS WORKING WITH FIELDS SOLVERS CAN ESTABLISH ANOTHER VALUABLE CRITERION: THE AMOUNT OF ENERGY THAT THE BOARD OR THE IC IS RADIATING.

and [you can most easily represent these phenomena] in the frequency domain. People use the frequency domain to describe the channel models, but the system specifications are in [the] time domain, essentially as eye diagrams. So, people need to find out the time-domain performance of these frequency-domain models."

Simulators working with fields solvers can establish another valuable criterion: the amount of energy that the board or the IC is radiating. This information can help predict performance for FCC (Federal Communications Commission) radiation limits and susceptibility under European CE (Conformité Européenne) rules. By having the evaluation occur in software at the beginning of the design, an engineer can predict the performance of the system and prevent a crisis when

MORE AT EDN.COM



⊕ For another article on validating Spice models before use, go to www.edn.com/article/CA512147.

⊕ For a roundup of online simulation tools, go to www.edn.com/070118df1b1.

the product fails FCC certification or, worse yet, does not work at all. "With a lot of our customers, we are embarking on EMC [electromagnetic-compatibility]-radiation analysis, says Mary Tolikas, director of business development for Ansoft. "We model the computer box with ventilation holes and other features and evaluate the radiation."

S-parameters are essential to microwave designers, who are often more comfortable designing and analyzing in the frequency domain. Microwave engineers rarely use a time-domain Spice simulator, preferring LNA or harmonic-balance simulations to solve circuit behavior in the frequency domain. Harmonic-balance and LNA programs focus on the steady-state behavior of electronic systems. By operating in the frequency domain, harmonic-balance techniques can save considerable time over Spice. But, just as Spice is not a panacea for the time domain, LNA and harmonic balance are inadequate and may be unnecessary to design in the frequency domain. "Computer programs are productivity tools, not substitutes for talent or thinking," says James Long, an RF-engineering consultant. "All they do is save design time. Look what was around in 1970 before CAD programs: millimeter microwave, communication theory, ICs, and space flight. If you split the system into blocks, [you'll find] numerous free and low-cost utility programs you can use. Spreadsheet programs have math and graphics abilities. You can write your own utility programs." No responsible engineer sends a circuit to production based only on simulations.

Harmonic-balance techniques solve Kirchoff's voltage laws in the frequency domain. The laws state that, at any point in an electrical circuit where charge density is not changing in time, the sum of currents flowing toward that point is equal to the sum of currents flowing away from that point. Harmonic-balance simulators use a technique that Russian mathematician and naval officer Alexei Krylov invented in the early 20th century

(Figure 7). The Krylov technique solves microwave problems more quickly than time-domain solvers can. The harmonic-balance technique can quickly solve for a microwave signal operating at a single frequency. A Spice time-domain solver breaks the analysis into extremely small time steps and then solves for many of them before the circuit reaches its steady state. Harmonic-balance techniques assume that the system has reached steady state. It can then solve the linear-algebra problem as a sparse-matrix problem. This approach solves for one frequency. Entering a command, such as "order=5," commands the simulator to solve for the fundamental and the first four harmonics of that fundamental. A few harmonics contain almost all of the energy in a microwave system, so the speed and efficiency of the harmonic-balance technique are advantages in solving those kinds of problems. Most microwave and RF simulators also have other modules, or S-parameter simulators, such as LNA, that can operate on the S-parameter data and solve for the performance of high-frequency systems.

"It used to be that you could do pretty good millimeter-wave and RF design just from S-parameters," says Scott Wedge, senior staff engineer at Synopsys. "Empirical-modeling techniques would turn geometries into S-parameter models. For years, we were doing this [step] because EM-solver technology was so slow and expensive. Now, several EM tools are pretty powerful for several types of applications. They've reached the point at which they are overtaking these old modeling approaches. You can throw everything into an EM solver these days. The choice you need to make is whether to do a 2-, a 2.5-, or a 3-D full-wave solution."

Spice can take a high-speed-design problem only so far. After that point, designers need to use advanced simulators with field solvers to properly predict the operation of the circuits. The field solver solves for the performance of one of the most important components in your design—the pc-board or IC layout. It also shows you how to get high-speed signals across connectors and through cables. The old days of prototyping with "dead bug" ICs and leaded components doesn't work for fast circuits. By learning how to use field solvers, you can reduce the time it takes to get to a working design and teach yourself the intuitive aspects of high-speed design. **EDN**

REFERENCES

- 1 "Electromagnetic Field Solver," Wikipedia, http://en.wikipedia.org/wiki/Electromagnetic_field_solver.
- 2 Kielkowski, Ron M, *Inside Spice: Overcoming the Obstacles of Circuit Simulation*, McGraw-Hill, January 1994.
- 3 eCircuit Simulator, www.ecircuitcenter.com/index.htm.
- 4 Swager, Anne Watson, "DOS-based analog-simulation software," *EDN*, May 21, 1992, pg 125.
- 5 Mukherjee, Bhaskar, Lei Wang, and Andrea Pacelli, "A Practical Approach to Modeling Skin Effect in On-Chip Interconnects," Stony Brook University, Stony Brook, NY, 2004, www.sinc.sunysb.edu/Stu/lewwang/pdf/skin.pdf.
- 6 Deisch, Cecil, "Modeling skin effect in Spice," *EDN*, Sept 28, 2006, pg 107, www.edn.com/article/CA6372836.
- 7 IBM Broadband Transmission-line Characterization Using Short-Pulse Propagation, IBM Corp, March 28, 2006, www.alphaworks.ibm.com/tech/gammazandcz2d.

FOR MORE INFORMATION

Agilent www.agilent.com	Mentor Graphics www.mentor.com
Altium Ltd www.altium.com	National Semiconductor www.national.com
Ansoft www.ansoft.com	On Semiconductor www.onsemi.com
Applied Wave Research www.appwave.com	Picosecond Pulse Labs www.picosecond.com
Cadence www.cadence.com	Solidworks www.solidworks.com
Comsol www.comsol.com	Sonnet Software www.sonnetsoftware.com
Helic www.helic.com	Synopsys www.synopsys.com
Henry Ott www.hottconsultants.com	Synoptics www.synoptics.co.uk
Hewlett-Packard www.hp.com	Tektronix www.tektronix.com
IMST www.imst.com	Teledyne CME www.teledyne.com
James Long www.analog-rf.com	Unigraphics www.ugs.com
The MathWorks www.mathworks.com	Zeland Software www.zeland.com

You can reach
Technical Editor
Paul Rako at
1-408-745-1994
and paul.rako@reedbusiness.com.

