Add Some Spice To Your Analog Designs

This month's column takes a look at some books with high EE interest, specifically a couple on Spice (Simulation Program with Integrated Circuit Emphasis).[1] As the name implies, Spice was originally developed specifically for IC design, but has long since been used much more generally. Today, the many Spice variants are considered general-purpose electronic circuit design tools, but they are also used in broader ways, such as control system design.

Without question, the capabilities of early Spice simulators have been greatly enhanced. Spice today provides many powerful and useful features, such as Monte Carlo and worst-case analysis, circuit optimization, sophisticated display, and graphical output features, etc. Currently, some simulation packages go so far as to offer a completely integrated design *program suite*. They include schematic capture and pc-board layout tools, in addition to the basic simulator. And, quite obviously, any simulator's performance is tied directly to the power of the host machine, a factor which provides today's 300-MHz-processor machines with some potent computational savvy.

Surely it behooves the wide-awake designer to be fully aware of this simulation power, so as to best harness it for his/her designs. This awareness, in turn, implies some measure of understanding of how Spice and Spice-like simulators work. And, here's where the well-focused and well-documented experiences of others can aid greatly.

Spice books: If you are like me, you've probably noticed that there are a lot of books out there on Spice and Spice-related simulators. You may also have wondered just which books are the good ones. I, myself, have wondered those same thoughts, and have spent money on Spice books, not always finding a full measure of satisfaction. Nevertheless, I hope the Spice books described here are ones that you'll find useful.

Ron Kielkowski of RCG Research Inc., has been presenting a series of three-day workshops on MicroSim's PSpice since 1993, and he seems to have furnished at least some of the answers.[2] He has written two Spice-related books, which are oriented to the SPICE2G.6 version, the most widely used (and also the last Fortran-based) Spice release from the University of California at Berkeley. They are reviewed below.

Inside SPICE: Overcoming the Obstacles of Circuit Simulation, 1994, ISBN: 0-07-911525-X, is a $50.00, 188-page, 6- by 9-in. hardcover book with six chapters, 200 illustrations, an index, and a 3.5-in. floppy diskette with programs. It is available from McGraw-Hill [*www.mhhe.com/engcs/electrical/*, or (800) 262-4729].

SPICE: Practical Device Modeling, 1995, ISBN: 0-07-911524-1, is a $55.00, 272-page, 6- by 9-in. hardcover book with six chapters, five appendices, 302 illustrations, an index, and a 3.5-in. floppy diskette with programs. It is also available from McGraw-Hill.


WALT JUNG

Both of Kielkowski's books simply radiate the fact that he is deeply involved with his topics, and knows them thoroughly. One clear example of this is the fact that he distributes his very own 32-bit, PC-compatible SPICE2G.6 program along with the two books. This program is RSPICE, as well as RGRAPH, a graphical processor. These programs and other specialized utilities are used as live demos, in conjunction with the book, and illustrate the various points made within the text.

The six chapters of *Inside SPICE* include: "What is Spice?," "Understanding Circuit Simulation," "Nonconvergence," "Numeric Integration, Timestep Control," and "Spice Options." Kielkowski does a good job of stating this book's raison d'être, both in the preface and first chapter, and indeed, throughout the book. Simply stated, Spice can be extremely difficult and frustrating when the reasons behind a failure to run or a nonconvergence aren't at all obvious. As the saying goes, "Not user-friendly." In fact, a balky Spice program can evoke a feeling that all such computer operations are best left to masochists.

But, the goal of a design simulation is to learn more about how a circuit performs. Sometimes it may take further effort on the part of the designer to first learn how the simulator works. At which point, he or she is better equipped to give it the correct input, so that Spice can then produce the expected and desired output.

This is the basic premise of *Inside SPICE*, that is, once you better understand how Spice works internally, you'll be much better able to make it do your bidding. You can then avoid the "obstacles," which are those items Kielkowski lists as often causing simulations to fail. These are the topics of Chapters "Nonconvergence," "Numeric Integration," and "Timestep Control."

As a preface to appreciating nonconvergence, Kielkowski discusses the matrix math operating basis of Spice in Chapter 2, "Understanding Circuit Simulation," and how nonlinear elements, in particular, influence convergence sensitivity. With this background, the reader is ultimately ready to absorb the topics of the three obstacle chapters, and more fully appreciate all of the user-adjustable Spice settings which can influence a simulation. The book concludes with a chapter on the various Spice option settings that are available.

One should appreciate that this Spice book goes far beyond a typical user's manual listing of the Spice statements and their modifiers. It does, in fact, give the reader a deeper appreciation of what's going on under the Spice simulator's hood, and offers many useful suggestions as to how to keep things fine-tuned. The bottom line is that *Inside SPICE* has the potential of being useful to anyone using Spice.

WALT JUNG

SPICE: Practical Device Modeling (hereafter referred to as "*Modeling*") deals with the practical problem of getting the appropriate models to use with Spice. These models might be any one or more of the following passive or active devices, as noted by these chapter headings: "Practical Device Modeling;" "Modeling Resistors, Capacitors, and Inductors;" "Modeling Diodes and Zener Diodes;" "Modeling the Bipolar Junction Transistor;" "Modeling the Junction Field Effect Transistor;" and "Modeling the Power MOSFET."

Models are basic to the operation of any Spice simulator, and it isn't always realistic to use simplistic models, even for passive parts. Why? Because almost all passive parts have parasitic elements which modify their *behavior* with varying temperature, applied voltage, or frequency, etc. So, simulations which take these parasitics into account are more complete and accurate. But, a snag here is that with some notable exceptions, few passive component vendors currently offer Spice models for their parts.[3] Hopefully, this situation will change for the better as time progresses.

Active part models are in better shape, because many libraries distributed with popular simulators include IC-vendor-supplied macromodels and simulator-vendor-generated models for many transistors and diodes. This situation, however, is not universal, nor is it complete, and often a model may need to be developed.

Kielkowski's *Modeling* solution for the unavailable model problem is to simply roll your own (at least for those parts as mentioned above). To aid in this process, he has developed a number of utility programs which help to match measured device data and model curves, yielding a useful model for a given part. His goal for the end model is one with a "5% RMS error," with respect to the real part.

The models developed in *Modeling* are subcircuits. As this name implies, they are smaller domain circuits consisting of various Spice elements, both active and passive as required. These elements are interconnected to represent, at the outer terminals, the part being modeled with respect to dynamic changes in current, voltage, frequency, and temperature.

It is a fascinating study to see the development of the various models in the chapters of this book. Even the simpler passive-part models allow good insight into the development, for example, as with a model for a real capacitor. This model would include a nominal capacitance shunted by a leakage resistance, in series with an inductance and series resistance. Temperature-related effects can be added, by modifying both the nominal capacitance and series resistance as a functions of the temperature.

To achieve this, Kielkowski shows us how to use an interconnected combination of controlled sources in the form of an Analog Behavioral Model (ABM). An ABM is a subcircuit which can be configured to perform addition, subtraction, multiplication, and so on. This trick allows the temperature characteristics of a capacitor to be modeled. Of course ABMs are very useful models on their own, and the book's Appendix D is devoted to a series of them. There, ABMs are complete working entities, ready to be applied within larger circuits.

But, of course, the bulk of the book is devoted to the much more detailed processes of model development for diodes and transistors—both bipolar and FET. Familiar parts such as the 1N752 and 2N2222 are used as examples. The developmental processes used give good results.

Throughout these modeling developments, continuous use is made of the supplied modeling software tools, as mentioned above. This technique, alas, brings up the one unfortunate caveat which must be stated of these two books in their present format. These software tools are *DOS-based* utilities, which was understandable for the pre-Windows 95 days of 1994. But, present-day PC users are likely to be using Windows 95, which makes it rather cumbersome to use the tools as designed, with their special DIR structure and CONFIG.SYS-loaded ANSI.SYS driver.

For test purposes, I was able to install the RSPICE software on my Windows 95 machine, but it does not run from a DOS window under Windows 95. It does, however, seem to run just fine, after rebooting from Windows 95 into DOS mode.

The software certainly does offer a low-cost entry into Spice experiences. If desired, one can also purchase three levels of support for this software, at prices ranging from $29.95 to $89.95.

In discussing these operating-system compatibility issues with the author, I learned that an new edition for *Inside SPICE* is planned for introduction sometime around March. This new edition will feature Windows 95-compatible software, so my suggestion to interested readers is watch for this update.

TIP: I enjoyed reviewing these two books, and anticipate gleaning useful material from them for some time to come. And, that's speaking as one who has used Spice on and off for about 12 years. I haven't as yet, taken one of the Kielkowski Spice courses, but hope to do so in the future.

In both of these books, the author's interest and dedication to the topics shows strongly. This fact is communicated to the reader with a style which is direct, clear, and no-nonsense. I can see both books becoming practical tools, and can also easily recommend them as references.

Future topics: On Spice in particular, I'm interested to learn of the general appeal of simulation as a design tool. Are the current simulator tools and available models adequate for your needs? Are integrated packages (suites) of schematic capture, simulators, and pc-board design the way to go? What are the problems you see in these areas? Let's hear from you on these, or other items.

Walt Jung is a corporate staff applications engineer for Analog Devices, Norwood, Mass. A longtime contributor to ELECTRONIC DESIGN, *he can be reached via e-mail at: Wjung @USA.net.*

References:

1. L. Nagel, *SPICE2: A Computer Program to Simulate Semiconductor Circuits*, Report # ERL-M520, University of California at Berkeley, 1975.

2. RCG Research Inc., P.O. Box 509009, Indianapolis, IN 46250-0900, (800) 442-8272 or (317) 877-2244.

3. Tantalum Electrolytic Capacitor SPICE Models, Kemet Electronics, Box 5928, Greenville, SC 29606, (803) 963-6300.