McGraw-Hill
ELECTRONIC ENGINEERING

ELECTRONIC ENGINEERING

50 Completely Verified Circuit Simulations

SPICE Circuit Handbook

- ✔ 50 circuits simulated, constructed, and bench-tested
- ✔ 4 simulators—PSpice, IsSpice, Micro-Cap, and SIMetrix
- ✔ Filters, power, logic, instrumentation, and more

Steven M. Sandler
Charles Hymowitz

SPICE Circuit Handbook

Steven M. Sandler

Charles Hymowitz

McGraw-Hill

New York Chicago San Francisco Lisbon London Madrid
Mexico City Milan New Delhi San Juan Seoul
Singapore Sydney Toronto

The McGraw-Hill Companies

CIP Data is on file with the Library of Congress

1 2 3 4 5 6 7 8 9 0 DOC/DOC 0 1 2 1 0 9 8 7 6

P/N 146858-7
PART OF
ISBN 0-07-146857-9

The sponsoring editor for this book was Stephen S. Chapman and the production supervisor was Richard C. Ruzycka. It was set in Century Schoolbook by TechBooks. The art director for the cover was Margaret Webster-Shapiro.

Printed and bound by RR Donnelley.

This book is printed on recycled, acid-free paper containing a minimum of 50% recycled, de-inked fiber.

SPICE Circuit Handbook

This book is dedicated to my wife Susan and my daughters Shanna and Rachel. It is you who encourage me to be the best I can be.

Steven M. Sandler

This book is dedicated to my wife Teresa and my three wonderful blessings, Mitchell, Olivia, and Makenna. You make it all worthwhile.

Charles Hymowitz

Acknowledgments

We would like to thank AEi Systems personnel, including Mark Kwamusi, Greg Boger, and Danny Chow, for performing all of the simulations for this book in an effort to obtain the best relative run times possible, capturing and running most if not all simulations on the same computer.

Thanks to Steve Chapman, the publisher at McGraw-Hill, for continuing to provide us these opportunities to write.

Thanks to John Wagner and his guys at Catena Software Ltd. for creating SIMetrix, Andy Thompson and the guys at Spectrum Software for creating Micro-Cap, Larry Meares and Intusoft for creating IsSpice, and OrCAD for creating PSpice.

Thanks to Priyanka Negi and the staff at TechBooks for the outstanding effort they put into creating this book.

Thanks to Ron Rohrer, Larry Nagel, and all the students at the University of California, Berkeley, who worked hard in 1969 and 1970 to develop the first computer simulation software, Cancer (Computer Analysis of Non-Linear Circuits Excluding Radiation). This effort would result in the release of SPICE into the public domain in 1971.

Steven M. Sandler
Charles Hymowitz

Contents

Chapter 1

Introduction

Since its introduction in 1971, SPICE (Simulation Program with Integrated Circuit Emphasis) has become the most popular analog simulation tool in use today. In the last 15 years, we have seen explosive growth in the use of SPICE, with the addition of Berkeley SPICE 3 enhancements, and support for C code model and mixed-mode simulation using XSPICE (Cox et al. 1992, Kielkowski 1994).We have also seen many new companies emerge as developers of SPICE-based simulation tools, most of which are currently available for the PC platform.

Each vendor of SPICE simulation software has added features such as Monte Carlo analysis, schematic entry, and post simulation waveform processing, as well as extensive model libraries. In most cases, the manufacturers have modified the algorithms for controlling convergence and have added new parameters or syntax for component models. As a result, each electronic design automaton (EDA) tool vendor has the basic Berkeley SPICE 2 features and a unique set of capabilities and performance enhancements.

We have also seen component manufacturers providing SPICE model support. Many of these manufacturers provide models of components such as MOSFETs, transistors, and operational amplifiers. Most of these models are available for free via the manufacturer's web sites, though not all are accurate or well documented. One company filling the void in the modeling area, especially with respect to power electronics, is AEi Systems, LLC (AEi Systems 2005; www.AENG.com). The ability of computers to simulate electronic circuits is increasing every day. The often-quoted "Moore's law" states that the speed of microprocessors doubles nearly every 18 months. As computers become more powerful and more capable, computer simulation is becoming a significant tool in the design process.

Unfortunately, there is still unwillingness in the electronic design community to embrace the abilities of computers to emulate circuit behavior. Many engineers still don't take SPICE simulation seriously. Typically, a design engineer, on being shown a SPICE model of the impeding failure of his or her circuit, will reply, "That's nice, but let's see what the hardware does." Even when the hardware fails, the engineer is more likely to investigate the charred and smoking breadboard than the SPICE model that predicted the result.

The purpose of this book is to showcase the ability of SPICE, via the simulation tools of several EDA vendors, to accurately predict the behavior of electronic circuitry.

The time it takes to run a simulation is orders of magnitude less than the time it takes to build the equivalent circuit on a breadboard. A simulation can be run through any number of environmental conditions with ease–conditions often unavailable or impractical to duplicate in a laboratory environment. Circuit stimulus and tolerances and their effect on the operation of the circuit can be easily evaluated.

Still, there are limitations to the capabilities of SPICE and similar circuit simulators. While the sophistication of simulation increases, the hardware breadboard will still remain a necessary step in the design process. This book will aid the engineer in using SPICE simulation as a very powerful tool in the design process.

This book is a compilation of all various types of electronic circuits. Such compilations are not unusual; in fact, there are several excellent circuit encyclopedias on bookshelves. However, this book goes several steps further. Instead of simply presenting the circuit to the reader, it also provides a SPICE schematic and details about the equivalent hardware performance. The intricacies involved in developing an accurate SPICE model of the circuit are also included. This format benefits readers in numerous ways. First, it allows them to emulate the correlation techniques introduced in this book in order to make their own SPICE models accurately mimic the behavior of the hardware. Secondly, it allows them to clearly see where SPICE excels in its ability to represent real hardware performance.

SPICE simulation gives design engineers a vast array of information that can help ensure a successful and optimal design of their hardware. If designers have circuit designs that they know operate correctly under nominal conditions and also have a SPICE model that can accurately reflect the design's behavior they are much more likely to be able to produce a design that will operate under all operating conditions. Clearly, SPICE simulation can be a much more integral step in the design process and prove its worthiness to engineers of any circuit discipline.

The beginning of the book concentrates on the basics of computer simulation of electronic circuits. A brief overview of four popular SPICE programs is provided along with their basic differences.

We have selected a broad cross section of analog and mixed-mode designs, which we have simulated, as well as constructed. The circuits are grouped into logical chapters. Generic topics, such as oscillators, amplifiers/receivers, power converters, and filters, all head their own chapter. Each chapter starts with a brief overview of the function of the circuits in the chapter. This is followed by several circuit examples. For instance, in the chapter on reference circuits, the beginning details what reference circuits are and their uses at the system level. This is followed by a detailed discussion on a single type of reference circuit, the band gap reference.

The theory of operation of each circuit is discussed, followed by the circuit schematic, the simulation results, and a comparison to laboratory data. Advantages and disadvantages of each circuit are added, along with any tips or hints useful in modeling the circuit accurately. We have attempted to perform each simulation using several versions of SPICE for comparison. Also included are the run times for each circuit simulation.

Four simulation programs were used to simulate the circuits in this book: ICAP/4Windows/IsSpice4™ v8.11, OrCAD®/PSpice® v10.5, SIMetrix™ v5.1, and Micro-Cap™ v8.0.

The simulations in this book were performed using a PC desktop computer running a 2.8 GHz Intel® microprocessor, 512 MB RAM, and Windows XP Professional®.

The run times of the circuits are highly dependant on the CPUs and memory capabilities of the computers running them, as well as the .TRAN and .OPTIONS settings in the simulation. It should be noted that any simulation program can be made to run faster or slower than any other program just by changing various variables, even though comparable output results are obtained. With slight changes in parameters like RELTOL, ABSTOL, VNTOL, TRTOL, or TMAX, simulations have been shown to run 14 times faster (Sandler 1996). Each circuit can be optimized for speed differently, and each EDA vendor's SPICE program has its own set of enhanced simulation optimization and modeling features. Similarly, the same function or individual component can be modeled in different ways, causing dramatic differences in simulation performance. Tricks that speed up simulations in one circuit may not work in another, or even have the opposite effect on speed. Invariably, SPICE simulations are a tradeoff between simulation speed, accuracy, and convergence (Kielkowski 1994).

We have made a reasonable effort to make apples to apples comparisons between the simulation speeds of the software in this book by using commonly available Berkeley SPICE 2 OPTIONS. The reader will notice that it is not predictable which software package will run the fastest on any given circuit. The real purpose of including the run times is to provide the user with an estimate as to how long the circuit will take to simulate on his or her own computer, nothing more. That being said, the simulation times noted after the simulations are reasonably accurate.

The reader will also note that in some circumstances, one or more of the simulation software results did not match the hardware results. We have attempted to explain the reasons why this might have occurred. Bear in mind that SPICE is one of those labors in life where you get out of it what you put into it. If you put very little effort into understanding what the models and circuit are doing, chances are your simulation accuracy will be poor.

The CD-ROM that comes with this book contains four simulation file folders, one for each of the four simulators. Each folder contains the relevant simulation files for that particular simulator. Schematics in their native format are provided in all cases. The circuit names are provided in the appropriate section for that circuit. For example, Circuit 1, a fourth-order Butterworth low pass filter, lists the file names for that circuit as follows: lp fltr (IsSpice), lpflt (Micro-Cap), lp flt (PSpice). Demonstration versions of each simulation tool set are also included. For SIMetrix both a PC version and a Linux version are included.

To make the circuits in this book and your own simulations more useful, we suggest you investigate the Power IC Model Library from AEi Systems, LLC (www.AENG.com/PSpice.asp). This product provides a wide variety of popular switching regulator and PWM IC models, most of which are verified against hardware and not readily available anywhere else. A multitude of application circuit examples are also included in the library. Modeling components using the data sheet information, as is done by most EDA vendors, is not sufficient to model complex parts like power electronics ICs. AEi Systems has taken the time to develop proprietary relationships with IC manufacturers in order to obtain the necessary information.

We have put a great deal of effort into the construction of this book. It is our sincere hope that the reader benefits from our hard work.

Bibliography

AEi Systems. 2005. "EMA Design Automation and AEi Systems Announce New Power IC Model Library for PSpice," Rochester, NY, June 28. Press release.

Cox, F. L., III, W. B. Kuhn, J. P. Murray, and S. D. Tynor. 1992. "Code-level Modeling in XSPICE," in *Proceedings of the IEEE International Symposium on Circuits and Systems, 1992 (ISCAS '92)*, vol. 2, pp. 871–874, http://users.ece.gatech.edu/~mrichard/Xspice

Kielkowski, Ron M. 1994. *Inside Spice*. New York: McGraw-Hill.

Sandler, Steven M. 2006. *Switch-Mode Power Supply Simulation with PSpice and SPICE 3*. New York: McGraw-Hill.

Sandler, Steven M. 1996. *SMPS Simulation with SPICE 3*. New York: McGraw-Hill.

Chapter

2

Description of the PSpice, IsSpice, SIMetrix, and Micro-Cap Simulators

The development of SPICE was initiated by Ron Rohrer, a junior faculty member at the University of California, Berkeley. Rohrer was teaching a class on circuit simulation, in which he and Larry Nagel developed a simulator using the FORTRAN programming language that was to be named CANCER (Computer Analysis of Nonlinear Circuits Excluding Radiation). It was difficult to test integrated circuits (ICs), but SPICE was thought to be an answer to the quick and reliable design of ICs. Larry Nagel increased the capabilities of CANCER by increasing the 400 component or 100 node limit, adding new and improved components and a macromodeling capability. In 1971, Nagel released this improved version of CANCER as SPICE 1 (Simulation Program with Integrated Circuit Emphasis). In 1975, SPICE 2 was released, which offered equation formulation for voltage-defined elements as well as increased simulation speed. This was achieved through the developments of time step control algorithms. The capabilities of SPICE grew with those of computers.

In 1983, SPICE 2G.6 was released and remained the industry standard for many years. Motivated by the increased use of UNIX workstations and superior programming tools, SPICE 2 was converted into the C programming language and released as SPICE 3. Although SPICE 3 is not entirely backward compatible with SPICE 2, the new features far outweigh this drawback. SPICE 3 has a technical advantage of being readily modified because it is written in C. SPICE 3 also offers more and improved device models and analysis functions.

A major improvement in terms of usability has been the addition of a graphical waveform post processing and schematic capture tools. Waveform post processors greatly facilitate computation and documentation of simulation results. Schematic capture automates the SPICE netlist generation dramatically reducing the number of syntax errors.

Understanding the development of SPICE is useful in making a worthwhile comparison of vendor-offered simulation software. The foundation of many vender-offered simulators is Berkeley SPICE 3F.5 combined with XSPICE from the Georgia Institute of Technology. XSPICE is an add-on to SPICE 3, enhancing it with several key features, including a mixed-mode simulation capability (true digital simulator) and over 40 new primitive functional blocks such as Laplace and state machine elements.

Four of these software manufacturers—OrCAD (PSpice), Intusoft (IsSpice), Micro-Cap V (Micro-Cap), and Catena (SIMetrix)—have their products featured in this book. With the exception of PSpice, which uses a greatly enhanced version of SPICE2G.6, these software manufacturers took the Berkley SPICE 3F.5 core and wrapped schematic and waveform display programs around it. The schematic entry tools translate the user-defined design into an ASCII netlist using SPICE syntax. The circuit is processed by SPICE and an answer is generated. The software then takes the result from SPICE and passes it into a graphics postprocessor in order to display the answers in a meaningful form.

The tool flow used by EDA vendors to enhance the basic SPICE engine is roughly the same. Four separate modules are utilized (see Table 2.1).

The first module is the *schematic capture* program. Originally, using SPICE meant translating a schematic by hand into the SPICE description language for calculation. The schematic capture program allows the user to pull down parts from a menu, wire together the components

TABLE 2.1 The Four Simulation Tool Modules and Their Functions

Schematic capture	Text editor	SPICE Simulator	Graphical post processor
Allows users to quickly generate SPICE-compatible netlists graphically. Cross-probing allows users to easily view various simulation results in the post processor by clicking on the object in the schematic.	Examines output files from SPICE. Examines SPICE netlists generated by the schematic capture program.	Performs numerical iteration of the circuit to determine solutions in various domains (time, frequency, DC, etc.).	Converts the text output of SPICE into more meaningful graphs and waveforms. Has the ability to perform complex numerical calculations on waveforms.