OrCAD PSpice® and Circuit Analysis

FOURTH EDITION

John Keown

SOFTWARE ENCLOSED
Book not returnable if software has been removed.
PRENTICE-HALL, INC.

FOURTH EDITION

OrCAD™ PSpice® and Circuit Analysis

JOHN KEOWN
Southern Polytechnic State University

Prentice Hall

Upper Saddle River, New Jersey
Columbus, Ohio

Library of Congress Cataloging-in-Publication Data

Keown, John.

OrCAD PSpice and circuit analysis / John Keown.

p. cm.

Includes bibliographical references and index.

ISBN 0-13-015795-3

1. Electric circuit analysis—Data processing. 2. PSpice.

I. Title.

TK454.K465 2001

621.319'2'0285—dc21 00-029851

CIP

Vice President and Publisher: Dave Garza
Editor in Chief: Stephen Helba
Acquisitions Editor: Scott J. Sambucci
Editorial Assistant: Lara Dugan
Production Editor: Stephen C. Robb
Production Supervision: Tonia Grubb, York Production Services
Design Coordinator: Karrie M. Converse-Jones
Cover Designer: Rod Harris
Production Manager: Pat Tonneman
Marketing Manager: Ben Leonard

This book was set in Times Roman, Helvetica Condensed, and Courier New by York Graphic Services. It was printed and bound by R. R. Donnelley & Sons. The cover was printed by Phoenix Color Corp.

Earlier editions entitled *PSpice and Circuit Analysis*, © 1993 and 1991, by MacMillan Publishing Company.

Prentice Hall

10 9 8 7 6 5 4 3

ISBN: 0-13-015795-3

Preface

Methods of circuit analysis vary widely, depending on the complexity of the problem. Whereas some circuits require nothing more complicated than the writing of a single equation for their solution, others require that several equations be solved simultaneously. When the response of a circuit is to be performed over a wide range of frequencies, the work is often both tedious and time consuming. Various tools ranging from trig tables and slide rules to calculators and computers have been used by those eager to ease the burden of lengthy computations.

In many cases the problem to be solved requires that the student have an understanding of which basic laws and principles are involved in the solution. In some cases, if the topology of a network is known, along with complete descriptions of the elements that are connected among the various nodes, computer programs can be used to perform the analyses.

Such programs have been under development for several decades. If you have access to a computer language such as BASIC, Pascal, or FORTRAN, you can devise your own programs to readily solve certain types of problems. More powerful programs, capable of solving many types of electrical networks under a variety of conditions, require years to develop and update.

What Is SPICE?

Such a program is SPICE, which stands for Simulation Program with Integrated Circuit Emphasis. The version of SPICE used in this book is PSpice, a commercial product developed by the MicroSim Corporation. In 1998, the company merged with OrCAD, producing release 9 of the software. The evaluation version of the program, which is packaged with this text, is sufficient to perform all the exercises and simulations in this book.

The evaluation version is fully functioning, but it has file size limitations. A more specific description of the software contents can be found in the README file on the accompanying CD-ROM.

The SPICE program is both powerful and flexible. At the same time, it can be intimidating and bewildering to the beginner, who might well ask, How do I use this mighty tool in the most elementary way?

Although it might appear foolish to use a powerful hammer to drive a tack, if novices can solve problems with SPICE *for which they already know the answers,* they will gain confidence to move ahead. Thus, this text begins with dc circuit analysis, proceeds with ac circuit analysis, then goes into the various topics involving semiconductors.

PSpice is widely used in industry for the main purpose of allowing the designer to investigate the behavior of a circuit without having to actually breadboard the circuit in the laboratory. This allows for a considerable savings in materials and labor. If the design needs to be modified or tweaked, changes can easily be submitted to the computer for another look at the results. The designer is familiar with the components that will eventually be used in the actual circuit. He or she understands their electrical properties and behavior. How large numbers of these components will interact, however, is sometimes difficult to predict. This is where the computer program takes over, going through the tedious solutions much more quickly and with far less chance for mistakes than the human approach.

Should every electrical student, practitioner, and designer learn SPICE and use it? I believe the answer is an unqualified *yes*. It has become a standard in both the academic and professional worlds. Your education will not be complete without an exposure to this valuable tool.

Will SPICE teach you what you need to know to perform both circuit analysis and design? I believe the answer is an unqualified *no*. A study of the basic laws that govern circuit behavior is just as important today as it ever was. SPICE and other computer aids of the same nature will merely free you of the drudgery of lengthy and repetitive computations. You will surely gain some additional knowledge in the process, which you might otherwise overlook. You will also enjoy using Probe, a feature of PSpice that allows you to plot circuit response involving functions of frequency and time, among other things.

The motivation for this book comes from a desire to present a simple, easy-to-follow guide to PSpice to students who want to learn more about computer aids to circuit analysis. The material is presented in such a way that anyone who is studying or has studied the various electrical topics will be able immediately to put PSpice to practical use.

An important feature of the book is the development of models for such devices as the bipolar junction transistor (BJT), the field-effect transistor (FET), and the operational amplifier (op amp). The models need be no more complicated than necessary for the problem at hand. For example, if you are interested in bias voltages and currents for the BJT, there is no need for a model of the transistor that takes ac quantities into account. It is hoped that the readers will be able to develop their own models for other devices, especially those where linear approximations are all that is needed.

When reading this book, be aware that you will learn much more by going through each example on the computer. It is important that you produce the required input (circuit) files, submit them to the PSpice program, then look at the output files and/or Probe to see the results. Only by actual experience with the computer will you begin to appreciate the power at your disposal and the satisfaction that comes from seeing the solutions appear on your monitor and printer.

Schematics and Capture CIS

The product that allows the circuit designer to place the various components of a circuit on an electronic drawing board prior to carrying out the analysis of a circuit in PSpice is called Schematics. MicroSim supported Schematics until the merger with OrCAD. Then, OrCAD's Capture CIS superseded Schematics. The two programs bear little resemblance. Therefore, if you have learned to use Schematics, you have much to learn before even attempting to simulate circuits in Capture. This further supports the author's decision to introduce SPICE in the form in which it historically developed.

You begin with a hand-drawn sketch of an electrical or electronic circuit in which nodes are labeled, usually in numerical sequence. The ground point is the zero node, and you must label all other nodes. Then you identify the elements of the circuit one by one on a single line of a file that is called a circuit (or input) file. Such files always have the extension *cir*. When the entire circuit has been characterized, the analysis (or simulation) takes place. The results will tell you a great deal (sometime more, sometimes less) about the behavior of the circuit under a variety of conditions.

If you choose to use Schematics or Capture, the entire electrical or electronic circuit is placed on a drawing board (on the computer monitor) and you choose, from among the available options, the kind of analysis you would like to perform. The end result is the same as you would get if you went directly to the PSpice program. The choice to use PSpice directly or to let Schematics or Capture create the circuit file is yours to make. It should be pointed out that PSpice can become an effective tool in short order, while learning Schematics or Capture is much more tedious and involved.

WHAT'S NEW IN THE FOURTH EDITION

As you might expect with the merger of MicroSim and OrCAD, the look and mechanics of PSpice are different from what was available in past versions of the software. The material for this edition is based on the evaluation software, version 9. This software (or a later version if available) is included with this text in the form of a CD-ROM.

Some familiarity with Microsoft Windows 95 or 98 by the reader is assumed. The installation of the software is described in Appendix C, or you may simply insert the CD-ROM and follow the directions on the screen. Either the OrCAD main menu will appear or you can use the Start, Run sequence followed by typing *d:orcadstart.exe* for the file name (where *d* is the letter assigned to your CD-ROM drive).

Chapters 1–13 cover most of the topics that are included in dc and ac circuit analysis, semiconductor devices and circuits, operational amplifiers, two-port networks, and filters. Chapters 14–17 are devoted to the same topics using the tools available in Schematics. The appendices have been refined to reflect a wider availability of digital parts in the device libraries.

All the example problems have been reworked using Windows 98 and the latest available version of PSpice, and all the Probe traces have been revised to show the newer look of output files from Probe. There is little difference in the results most recently obtained and those obtained using earlier versions of the software. One difference is worth mentioning: Depending on the printer that you use, the plots obtained when using Probe will not look exactly alike. Generally speaking, the print drivers for laser printers produce the better results when compared with inkjet printers.

ACKNOWLEDGMENTS

The author gratefully acknowledges the following reviewers of the fourth edition for their insightful suggestions: Thomas E. Brewer, Georgia Institute of Technology; James N. Downing, Holyoke Community College; John D. Polus, Purdue University; and Russell E. Puckett, Texas A&M University.

Contents

1
DC Circuit Analysis 25

2
AC Circuit Analysis (for Sinusoidal Steady-State Conditions) 75

3
Transistor Circuits 119

9
An Introduction to PSpice Devices 289

10
The Bipolar-Junction Transistor and Its Model 319