

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.

OrCAD® PSpice® is a registered trademark of Cadence® Design Systems.

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

Copyright © 2001, 1998 by Prentice-Hall, Inc. Upper Saddle River, New Jersey 07458. All rights reserved. Printed in the United States of America. This publication is protected by Copyright and permission should be obtained from the publisher prior to any prohibited reproduction, storage in a retrieval system, or transmission in any form or by any means, electronic, mechanical, photocopying, recording, or likewise. For information regarding permission(s), write to: Rights and Permission Department.



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

Introduction **1**

- A Bit of Background 2
- Getting Started 3
- A Few Helpful Points 4
- Here's How It's Done 5
 - Creating the Input File 6
 - Examining the Output File 7
 - Changing the Input File 8
 - The Current Directions 9
- Further Reading 10

PSpice Overview **11**

- DC Circuit Analysis 11
- AC Circuit Analysis 14
 - Probe 17
- Transistor Circuit Analysis 20

1

DC Circuit Analysis

25

An Introductory Example	25
Using SPICE to Investigate the Circuit	26
Examining the Output File	27
Another Simple Circuit for Analysis	28
Basic Circuit Laws	30
Getting More from the Output File	32
Current Directions	32
Circuit with Two Voltage Sources	33
Thévenin's Theorem and Applications	35
SPICE and Thévenin's Theorem	36
Practical Application of Thévenin's Theorem	37
Circuit for Thévenin Replacement	38
Practical Current Source vs. Practical Voltage Source	41
SPICE Analysis of Circuit with Current Source	42
Norton's Theorem	43
Using Norton's Theorem	44
Short-Circuit Current in Missing Element	44
Circuit with Current and Voltage Sources	45
Maximum Power Transfer	46
Dependent Sources in Electric Circuits	47
Voltage-Dependent Voltage Source	47
Current-Dependent Voltage Source	50
Current-Dependent Current Source	51
Another Current-Dependent Current Source	52
Voltage-Dependent Current Source	54
Another Current-Dependent Voltage Source	55
Polynomial-Dependent Sources	56
Dependent Source as a Function of Two Other Voltages	57
Mesh Analysis and PSpice	59
DC Sweep	61
Using the .PROBE Statement	61
Nodal Analysis and PSpice	63
A Nonplanar Circuit	65
Summary of PSpice Statements Used in This Chapter	66
Dot Commands Used in This Chapter	68
Problems	69

2

AC Circuit Analysis (for Sinusoidal Steady-State Conditions)

75

Series AC Circuit with R and L	75
Series AC Circuit with R and C	77
Parallel Branches in AC Circuit	77
Parallel Branches with Capacitive Branch	78
Maximum Power Transfer in AC Circuit	79
Resonance in Series RLC Circuit	79
Frequency Sweep for Series-Parallel AC Circuit	82
Effect of Changes in Coil Resistance	83
A Parallel-Resonant Circuit	84
Using the Probe Cursor	85
Finding the Input Impedance of an AC Circuit	86
Input Impedance of a Two-Branch Network	87
A Phase-Shift Network	89
Locus of Admittances	90
Admittance Locus for Series RLC	93
Multiple Sources in AC Networks	94
Transformers	96
Frequency Response of Tuned Circuit	97
Three-Phase AC Circuits	98
Power-Factor Improvement	101
Three-Phase Power-Factor Improvement	102
A Three-Phase Rectifier	105
Voltage Regulation in a Three-Phase System	108
A Two-Phase System	109
Summary of New PSpice Statements Used in This Chapter	111
Dot Commands Used in This Chapter	112
.Probe	113
Problems	113

3

Transistor Circuits

119

The Bipolar-Junction Transistor	119
A Model Suitable for Bias Calculations	119
Saturation Considerations	121

Biasing Example for a Ge Transistor	121
Small-Signal h -Parameter Model of the BJT	123
Common-Emitter Transistor Analysis Using h -Parameter Model	123
Common-Collector Transistor Analysis Using h -Parameter Model	126
Common-Base Transistor Analysis Using h -Parameter Model	127
Other Configurations	128
Using a Circuit Involving Miller's Theorem	128
The Dual of Miller's Theorem	130
CC Circuit with Collector Resistor	133
High-Input-Resistance Amplifier	135
Two-Stage Amplifiers	136
Simplified h -Parameter Model	137
The CE Amplifier Using the Simplified h -Parameter Model	138
Field-Effect Transistor Amplifiers	139
Common-Drain FET with External Drain Resistor	141
Frequency Response of FET Amplifiers	142
High-Frequency Model of the BJT	143
Emitter Follower at High Frequencies	144
DC Sensitivity	147
DC Sensitivity of Biasing Circuit	149
The PSpice Parts Library	150
Sensitivity of Library BJT Circuit	151
Summary of New PSpice Statements Used in This Chapter	152
Dot Commands Used in This Chapter	154
Problems	155

4

Multistage Amplifiers, Frequency Response, and Feedback

161

Low-Pass Filter	161
Low-Frequency Response of High-Pass RC Network	163
Common-Emitter Amplifier with Bypass Capacitor	163
Two-Stage Amplifier at High Frequencies	166
Two-Stage CE Amplifier with Voltage-Series Feedback	167
Two-Pole Amplifier Model with Feedback	172
CE Amplifier with Voltage-Shunt Feedback	175
Current-Shunt Feedback Two-Stage CE Amplifier	176
Three-Stage CE Amplifier Frequency Response	178

Effects of Circuit Modifications	179
Three-Stage Amplifier with Voltage-Shunt Feedback	180
Summary of New PSpice Statements Used in This Chapter	181
Various Forms of Transient Specification	181
The Exponential Source	182
The Pulse Source	183
The <i>PWL</i> Source	184
The Frequency-Modulated Source	185
The Sine-Wave Source	186
Problems	188

5

The Operational Amplifier

193

The Ideal Op Amp	193
Noninverting Ideal Op Amp	195
Op Amp Giving Voltage Difference Output	196
Frequency Response of the Op Amp	197
Using a Subcircuit for the Op Amp	202
Op Amp Differentiator Circuit	203
Op Amp Integrator Circuit	204
Response to Unit Step Function	206
Double Op Amp Circuit	207
Active Filters	210
Second-Order Butterworth Low-Pass Filter	211
Fourth-Order Butterworth Low-Pass Filter	213
Active Resonant Band-Pass Filter	217
Active <i>RC</i> Band-Pass Filter	221
Summary of New PSpice Statements Used in This Chapter	222
Problems	223

6

Transients and the Time Domain

227

Switch Closing in an <i>RL</i> Circuit	227
Nonzero Initial Current in the Transient Analysis	228
Resistor and Capacitor in the Transient Analysis	230

A Double-Energy Circuit	232
Overdamped RLC Series Circuit	232
The Critically Damped RLC Circuit	234
The Underdamped RLC Circuit	235
Step Response of an Amplifier	237
Low-Frequency Response of an Amplifier	238
Circuit with Charged Capacitor	240
Switch-Opening Circuit with L and C	242
Circuit with Current Source	244
Bridge Circuit with Initial Current	247
A Ringing Circuit	249
Problems	249

7

Fourier Series and Harmonic Components

253

Fundamental and Second-Harmonic Frequency	253
Decomposition and Reconstruction of Wave	256
Second-Harmonic Distortion in a Power Amplifier	257
Intermodulation Distortion	259
Adding Sine Waves	263
Adding Fundamental and Second Harmonic	265
Amplitude Modulation	266
Summary of New Dot Command Used in This Chapter	267
Problems	268

8

Stability and Oscillators

271

The Feedback Loop	271
The Wien-Bridge Oscillator with Initial Help	272
The LC Oscillator with Initial Help	274
Measurements with a Test Circuit	276
The Phase-Shift Oscillator	276
The Wien-Bridge Oscillator	279
Another Wien-Bridge Example	282
The Colpitts Oscillator	284
Problems	286

9

An Introduction to PSpice Devices

289

- A Half-Wave Rectifier 289
- The Built-in Model for a Diode 290
- The Filtered Half-Wave Rectifier 292
- The Full-Wave Rectifier 295
- Full-Wave Rectifier with Filter 295
- Simple Diode Clipper 297
- A Double-Ended Clipper 298
- Variable Load Resistor for Maximum Power 299
- Built-in Model for the Bipolar-Junction Transistor 301
- Output Characteristics of the Common-Emitter Transistor 302
- Input Characteristics of the *CE* Transistor 303
- Output Characteristics of the Junction Field-Effect Transistor 305
- Other Active Semiconductor Devices 305
- The Differential Amplifier 305
 - Difference-Mode Gain 305
 - Common-Mode Gain 306
 - Transfer Characteristics of the Differential Amplifier 306
- Logic Gates 309
- The 7402 NOR Gate 309
- Summary of New PSpice Statements Used in This Chapter 313
- New Dot Commands 314
- Problems 314

10

The Bipolar-Junction Transistor and Its Model

319

- The Bipolar-Junction Transistor 319
 - Output Characteristics 319
 - Input Characteristics 321
- A Common-Emitter BJT Amplifier 321
- Biasing Case Study 325
 - The AC Analysis 326
- CE* Amplifier with Unbypassed Emitter Resistor 330
 - Finding the Input Resistance 334
- Using Our Own Model with the *h* Parameters 334
 - The *h*-Parameter Analysis 334

Phase Relations in the <i>CE</i> Amplifier	335
The Amplifier without the Emitter Capacitor	336
The Amplifier with the Emitter Capacitor	338
The BJT Flip-Flop	339
The PSpice Analysis	340
The Astable Multivibrator	343
An Emitter-Coupled BJT Multivibrator	344
Problems	349

11

The Field-Effect Transistor

351

Output Characteristics for the Junction Field-Effect Transistor	351
Input Characteristics for the JFET	352
JFET Biasing Circuit	354
The JFET Amplifier	356
JFET Waveshapes	357
The Power MOSFET	357
The Output Characteristics	357
The Input Characteristics	359
The MOSFET Amplifier	361
The Waveshapes	363
The Gallium Arsenide FET	363
Problems	365

12

Two-Port Networks and Passive Filters

371

Two-Port Parameters	371
Finding the y Parameters	371
Using the y Parameters to Solve a Circuit	374
y Parameters of Network with Dependent Source	375
The Open-Circuit Impedance Parameters	377
The z Parameters for an AC Circuit	379
Using the z Parameters to Solve a Circuit	380
The $ABCD$ Parameters	380
The Hybrid Parameters	383

Another Set of Hybrid Parameters	383
Transmission Lines	384
A Long Telephone Line	385
Constant- k Filter	387
Stop-Band Behavior of the Constant- k Filter	389
Lossless Transmission Line	391
Lossless Line Composed of Several Sections	395
Input Impedance at Points along the Line	397
A Band-Pass Filter	398
Band-Elimination Filter	400
Problems	402

13

Nonlinear Devices

405

The Nonlinear Resistor	405
The Iron-Core Inductor	406
The BH Curve	407
The Iron-Core Transformer	409
Voltage-Controlled Switch for Variable Resistor	411
Current-Controlled Switch	414
Summary of New PSpice Statements Used in This Chapter	416
New Dot Command	416
Problems	416

14

Capture

419

A DC Series Circuit	420
Creating a Schematic in Capture	420
Analyzing the Circuit	423
Examining the Output File	423
Printing the Results	424
Net Aliases	426
Drawing a Tee Circuit	427
Running the Simulation	428
Dependent Sources in Schematics	429
Voltage-Dependent Voltage Source	431

Current-Dependent Current Source	432
An AC Circuit	436
Producing the AC Sweep	436
Finding More Accurate Trace Values	439
Order of the Nodes	439
Sine-Wave Representation for AC Analysis	440
Plotting Current with Voltages	442
Series Circuit with R and C	442
Maximum Power to Load Impedance	444
The Probe Voltage and Current Notation	446
Series Resonance	446
Multiple-Source AC Network	448
Sine-Wave Representation of Multisource AC	450
Sine Waves of Currents	452
Transformers	452

15

Transistor Circuits in Capture

455

Output Characteristics of $Q2N3904$	455
Input Characteristics of $Q2N3904$	458
Common-Emitter Bipolar-Junction Transistor Case Study	459
The AC Analysis	462
The Transient Analysis	462
Modifying the Transistor Parameters	465
Using the h -Parameter Model	468
Field-Effect Transistor Characteristics	471
The JFET Amplifier	472
The Transient Analysis (JFET Waveshapes)	478
Frequency Response of BJT	478
Changing the Transistor Characteristics	480

16

Operational Amplifiers in Capture

481

Noninverting Ideal Op Amp	481
Op Amp for Voltage-Difference Output	483
Frequency Response of the Op Amp	485