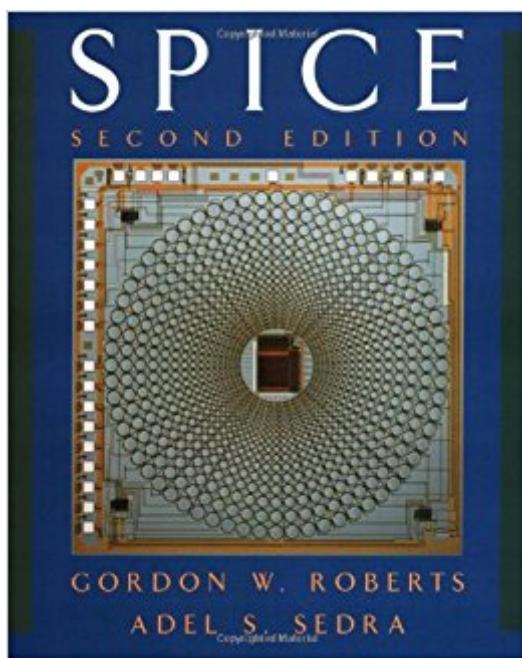


The book was found

SPICE (The Oxford Series In Electrical And Computer Engineering)



Synopsis

SPICE (Simulation Program with Integrated Circuit Emphasis) has become the industry standard for computer-aided circuit analysis for microelectronic circuits, and is used by the majority of IC designers in North America today. Unlike most SPICE books, which simply present SPICE in a how-to-use fashion, this volume outlines how SPICE is used in the process of design itself. It features methodologies for analyzing transistor and op amp circuits, over 100 SPICE examples, and numerous chapter problems. Intended to accompany Sedra & Smith's Microelectronic Circuits, 4/e, this book can also stand alone as a manual for computer-aided circuit analysis for microelectronic circuits. SPICE decks and the examples in this book, as well as examples from the first edition, are all available on-line via the World Wide Web at

<http://www.macs.ece.mcgill.ca/~roberts/ROBERTS/SPICE/>. Most circuit examples can be simulated using a student version of PSpice running on a low cost PC. This new second edition improves upon the first by tightening up the language and shortening the volume's length by almost fifty percent in order to make the materials more useful as a supplement to Microelectronic Circuits 3/e, by Sedra and Smith. Also available from Oxford University Press to accompany Sedra/Smith Microelectronic Circuits 3/E: Laboratory Manual by K.C. Smith (University of Toronto) ISBN 0-19-511103-6 Additional Problems With Solutions by K. C. Smith ISBN 0-19-510586-9 1995 Problems Supplement by K.C. Smith ISBN 0-19-510367-X

Book Information

Series: The Oxford Series in Electrical and Computer Engineering

Paperback: 464 pages

Publisher: Oxford University Press; 2 edition (September 19, 1996)

Language: English

ISBN-10: 0195108426

ISBN-13: 978-0195108422

Product Dimensions: 9.2 x 0.8 x 7.5 inches

Shipping Weight: 1.7 pounds (View shipping rates and policies)

Average Customer Review: 4.3 out of 5 stars 4 customer reviews

Best Sellers Rank: #398,501 in Books (See Top 100 in Books) #54 in Books > Engineering & Transportation > Engineering > Electrical & Electronics > Circuits > Integrated #122 in Books > Engineering & Transportation > Engineering > Electrical & Electronics > Circuits > Design #1026 in Books > Textbooks > Humanities > Linguistics

Customer Reviews

"Best we have used!"--Clifford B. Fallon, Washington State University

SPICE (Simulation Program with Integrated Circuit Emphasis) has become the industry standard for computer-aided circuit analysis for microelectronic circuits, and it is used by the majority of IC designers in North America today. Unlike most SPICE books, which simply present SPICE in a how-to-use fashion, this volume outlines how SPICE is used in the process of design itself. It features methodologies for analyzing transistor and op amp circuits, over 100 SPICE examples, and numerous chapter problems. In many cases, new designers of electronic circuits blindly search for ways to improve the design itself using a brute-force, hit-and-miss approach. The intention of this book is to avoid this pitfall by teaching readers what not to do with SPICE. This is accomplished by keying each example in this text to those presented in Sedra and Smith's Microelectronic Circuits 3/E, where a complete hand analysis is provided. Another benefit of this book is that it can also stand alone as a manual for computer-aided circuit analysis for microelectronic circuits.

Outstanding resource that has been around for years, still excellent...holds up over time.

Excellent Condition! Very Good Book!

Needs to be proofread.

The book was received in a very timely manner. The condition of the book is very good and as advertised. My main concern is that since its publication date is 1997 that it may be out of date with the current software. However, the examples and the answers should still apply well and should still be useful.

[Download to continue reading...](#)

Fundamentals of Electrical Engineering (The Oxford Series in Electrical and Computer Engineering)
SPICE (The Oxford Series in Electrical and Computer Engineering) Fabrication Engineering at the Micro- and Nanoscale (The Oxford Series in Electrical and Computer Engineering) The Science and Engineering of Microelectronic Fabrication (The Oxford Series in Electrical and Computer Engineering) Electrical Engineering Reference Manual for the Electrical and Computer PE Exam, Sixth Edition Modern Digital and Analog Communication Systems (The Oxford Series in Electrical

and Computer Engineering) Electric Machinery and Transformers (The Oxford Series in Electrical and Computer Engineering) Operation and Modeling of the MOS Transistor (The Oxford Series in Electrical and Computer Engineering) Operation and Modeling of the MOS Transistor: Special MOOC Edition (The Oxford Series in Electrical and Computer Engineering) Circuits and Systems: A Modern Approach (The Oxford Series in Electrical and Computer Engineering) Linear System Theory and Design (The Oxford Series in Electrical and Computer Engineering) An Introduction to Mixed-Signal IC Test and Measurement (The Oxford Series in Electrical and Computer Engineering) Probabilistic Methods of Signal and System Analysis (The Oxford Series in Electrical and Computer Engineering) Analog Methods for Computer-Aided Circuit Analysis and Diagnosis (Electrical and Computer Engineering) Microelectronic Circuits (The Oxford Series in Electrical and Computer Engineering) 7th edition CMOS Analog Circuit Design (The Oxford Series in Electrical and Computer Engineering) Digital Integrated Circuit Design (The Oxford Series in Electrical and Computer Engineering) Understanding Semiconductor Devices (The Oxford Series in Electrical and Computer Engineering) Laboratory Explorations to Accompany Microelectronic Circuits (The Oxford Series in Electrical and Computer Engineering) Photonics: Optical Electronics in Modern Communications (The Oxford Series in Electrical and Computer Engineering)

[Contact Us](#)

[DMCA](#)

[Privacy](#)

[FAQ & Help](#)