Circuit Simulation With Spice Opus Theory And Practice Modeling And Simulation In Science Engineering And Technology

Relaxation Techniques for the Simulation of VLSI Circuits
The SPICE Book
Electronic Circuit Analysis using LTSpice XVII Simulator
Electronics Circuit SPICE Simulations with LTspice
LTspice for Linear Circuits
Fundamentals of Computer-Aided Circuit Simulation
VLSI Circuit Simulation and Optimization
SMPS Simulation with SPICE 3
Analog Integrated Circuits for Communication
Advanced Circuit Simulation Using Multisim Workbench

Circuit Simulation Methods and Algorithms
SPICE for Power Electronics and Electric Power
Electric and Electronic Circuit Simulation Using TINA-TI
Passive Circuit Analysis with LTspice®
Silicon and Beyond
Circuit Analysis with Multisim
Advanced Circuit Simulation Using Multisim Workbench
Computer Simulation of Electronic Circuits
The Art of Simulation Using PSPICEAnalog and Digital
Switch-Mode Power Supply Simulation: Designing with SPICE 3
MOSFET Modeling with SPICE
Electronic Circuit & System Simulation Methods (SRE)
SPICE for Circuits and Electronics Using PSpice
Schematic Capture with Electronics Workbench Multisim
Silicon and Beyond
Relaxation Techniques for the Simulation of VLSI Circuits
Inside SPICE
Circuit Simulation with SPICE OPUS
Circuit Simulation
Analysis and Application of Analog Electronic Circuits to Biomedical Instrumentation

SPICE Circuit Handbook
FET Modeling for Circuit Simulation
Inside SPICE
An Analog Electronics Companion
Mixed-Mode Simulation and Analog Multilevel Simulation
Electric and Electronic Circuit Simulation using TINA-TI®
Analog Circuit Simulators for Integrated Circuit Designers
Analog Circuit Design with Spice
Electric and Electronic Circuit Simulation Using TINA-TI®
SPICE

VANESSA SHAFFER

Relaxation Techniques for the Simulation of VLSI Circuits Springer Nature

This Book On A Very Topical Subject Is Aimed At Engineers Who Either Use Or Develop Cad Tools For Circuit Design, Be It At The Discrete Device Level Or At The Lsi/Vlsi Level. The Book Is Unique In The Sense That It Covers Analog Circuit Simulation, Device Models, Logic Simulation And Fault Simulation. These Topics Traditionally Belong To Different

Areas Of Electrical Engineering And Are Therefore Not Covered In One Book. However, A Person Doing Circuit Design On A Computer Today Needs To Know All Aspects Of The Simulation. This Book Attempts To Satisfy This Need. Many Examples Of Programs As Well As Applications Are Given. Every Chapter Contains Solved As Well As Unsolved Problems. In Addition, Programming Assignments Are Included. Mathematics Has Been Kept To A Minimum And An Intuitive Approach Has Been Taken.The Background Required Is That Of Final Year Undergraduate In Electrical Engineering. It Is Expected That Much Of This Material Would Percolate Down To More Basic Courses In Future Years.

The SPICE Book Springer Science & Business Media

Analog Integrated Circuits for Communication: Principles, Simulation and Design, Second Edition covers the analysis and design of nonlinear analog integrated circuits that form the basis of present-day communication systems. Both bipolar and MOS transistor circuits are analyzed and several numerical examples are used to illustrate the analysis and design techniques developed in this book. Especially unique to this work is the tight coupling between the first-order circuit analysis and circuit simulation results. Extensive use has been made of the public domain circuit simulator Spice, to verify the results of first-order analyses, and for detailed simulations with complex device models. Highlights of the new edition include: A new introductory chapter that provides a brief review of communication systems, transistor models, and distortion generation and simulation. Addition of new material on MOSFET mixers, compression and intercept points, matching networks. Revisions of text and explanations where necessary to reflect the new organization of the book Spice input files for all the circuit examples that are available to the reader from a website. Problem sets at the end of each chapter to reinforce and apply the subject matter. An instructors solutions manual is available on the book's webpage at springer.com. Analog Integrated Circuits for Communication: Principles, Simulation and Design, Second Edition is for readers who have completed an introductory course in analog circuits and are familiar with basic analysis techniques as well as with the operating principles of semiconductor devices. This book also serves as a useful reference for practicing engineers.

Electronic Circuit Analysis using LTSpice XVII Simulator CRC Press

LTspice® for Linear Circuits Introduce yourself to the industry-leading software in electronic circuit simulation The simulation of electronic circuits is a crucial tool in modern electrical engineering. Many currently available software toolkits for circuit simulation are expensive, or nominally free but with significant restrictions on features and applications. LTspice®, a software distributed by semiconductor manufacturer Analog Devices, is not only the most widely used SPICE-based circuit simulator in the industry, but also free and unrestricted. LTspice®for Linear Circuits provides a comprehensive introduction to this software and its circuit simulation capabilities. Focusing on the fostering of practical knowledge, the book develops a six-step strategy for solving circuit analysis problems, beginning with the formulation of the problem, and proceeding through the simulation and the review of results. Readable and built around an easy-to-use, accessible software, LTspice® for Linear Circuits is an essential tool for any would-be electrical engineer. LTspice® for Linear Circuits readers will also find: Practical examples of circuit analysis problems and their solutions Detailed treatment of problems involving DC Circuits, First-Order Circuits, AC Circuits, Frequency Response and more Educational content from an author with decades of experience teaching electrical circuits LTspice®for Linear Circuits is perfect for undergraduates in electrical engineering and adjacent subjects, as well as anyone looking for an introduction to this widely used software.

Electronics Circuit SPICE Simulations with LTspice Springer Science & Business Media

Circuit simulation is widely used for the design of circuits, both discrete and integrated. Device modeling is an impor tant aspect of circuit simulation since it is the link between the physical device and the sim ulate d device. Curren tly available circuit simulation programs provide a variety of built-in models.

Many circuit designers use these built-in models whereas some incorporate new models in the circuit sim ulation programs. Understanding device modeling with particular emphasis on circuit simulation will be helpful in utilizing the built-in models more efficiently as well as in implementing new models. SPICE is used as a vehicle since it is the most widely used circuit sim ulation program. How ever, some issues are addressed which are not directly appli cable to SPICE but are applicable to circuit simulation in general. These discussions are useful for modifying SPICE and for understanding other simulation programs. The gen eric version 2G. 6 is used as a reference for SPICE, although numerous different versions exist with different modifications. This book describes field effect transistor models commonly used in a variety of circuit sim ulation pro grams. Understanding of the basic device physics and some familiarity with device modeling is assumed. Derivation of the model equations is not included. (SPICE is a circuit sim ulation program available from EECS Industrial Support Office, 461 Cory Hall, University of Cali fornia, Berkeley, CA 94720.) Acknowledgements I wish to express my gratitude to Valid Logic Systems, Inc.

LTspice for Linear Circuits Springer Nature

A DEFINITIVE TEXT ON DEVELOPING CIRCUIT SIMULATORS Circuit Simulation gives a clear description of the numerical techniques and algorithms that are part of modern circuit simulators, with a focus on the most commonly used simulation modes: DC analysis and transient analysis. Tested in a graduate course on circuit simulation at the University of Toronto, this unique text provides the reader with sufficient detail and mathematical rigor to write his/her own basic circuit simulator. There is detailed coverage throughout of the mathematical and numerical techniques that are the basis for the various simulation topics, which facilitates a complete understanding of practical simulation techniques. In addition, Circuit Simulation: Explores a number of modern techniques from numerical analysis that are not synthesized anywhere else Covers network equation formulation in detail, with an emphasis on modified nodal analysis Gives a comprehensive treatment of the most relevant aspects of linear and nonlinear system solution techniques States all theorems without proof in order to maintain the focus on the end-goal of providing coverage of practical simulation methods Provides ample references for further study Enables newcomers to circuit simulation to understand the material in a concrete and holistic manner With problem sets and computer projects at the end of every chapter, Circuit Simulation is ideally suited for a graduate course on this topic. It is also a practical reference for design engineers and computer-aided design practitioners, as well as researchers and developers in both industry and academia.

Fundamentals of Computer-Aided Circuit

Simulation McGraw Hill Professional The expert guidance needed to customize your SPICE circuits Over the past decade, simulation has become an increasingly integral part of the electronic circuit design process. This resource is a compilation of 50 fully worked and simulated Spice circuits that electronic designers can customize for use in their own projects. Unlike traditional circuit encyclopedias Spice Circuit Handbook is unique in that it provides designers with not only the circuits to use but the techniques to simulate their customization.

VLSI Circuit Simulation and Optimization McGraw Hill Professional Learn how analog circuit simulators work with these easy to use numerical recipes implemented in the popular Python programming environment. This book covers the fundamental aspects of common simulation analysis techniques and algorithms used in professional simulators today in a pedagogical way through simple examples. The book covers not just linear analyses but also nonlinear ones like steady state simulations. It is rich with examples and exercises and many figures to help illustrate the points. For the interested reader, the fundamental mathematical theorems governing the simulation implementations are covered in the appendices. Demonstrates circuit simulation algorithms through actual working code, enabling readers to build an intuitive understanding of what are the strengths and weaknesses with various methods Provides details of all common, modern circuit simulation methods in one source Provides Python code for simulations via download Includes transistor numerical modeling techniques, based on simplified transistor physics Provides detailed mathematics and ample references in appendices

SMPS Simulation with SPICE 3 McGraw-Hill Professional Publishing

A circuit simulator is a computer program that permits us to see circuit behavior, i.e. circuit voltages and currents, without making the circuit. Use of a circuit simulator is a cheap, efficient, and safe way to study the behavior of circuits. The Toolkit for Interactive Network Analysis (TINA(R)) is a powerful yet affordable SPICE based circuit simulation and PCB design software package for analyzing, designing, and real time testing of analog, digital, VHDL, MCU, and mixed electronic circuits and their PCB layouts. This software was created by DesignSoft. TINA-TI is a spinoff software program that was designed by Texas Instruments (TI(R)) in cooperation with DesignSoft which incorporates a library of pre-made TI components for the user to utilize in their designs. This book shows how a circuit can be analyzed in the TINA-TI(R) environment. Students of engineering (for instance, electrical, biomedical, mechatronics, and robotics to name a few), engineers who work in the industry, and anyone who wants to learn the art of circuit simulation with TINA-TI can benefit from this book.

Analog Integrated Circuits for

Communication CRC Press

From little more than a circuit-theoretical concept in 1965, computer-aided circuit simulation developed into an essential and routinely used design tool in less than ten years. In 1965 it was costly and time consuming to analyze circuits consisting of a half-dozen transistors. By 1975 circuits composed of hundreds of transistors were analyzed routinely. Today, simulation capabilities easily extend to thousands of transistors. Circuit designers use simulation as routinely as they used to use a slide rule and almost as easily as they now use hand-held calculators. However, just as with the slide rule or hand-held calculator, some designers are found to use circuit simulation more effectively than others. They ask better questions, do fewer analyses, and get better answers. In general, they are more effective in using circuit simulation as a design tool. Why? Certainly, design experience, skill, intuition, and even luck contribute to a designer's effectiveness. At the same time those who design and develop circuit simulation programs would like to believe that their programs are so easy and straightforward to use, so well debugged and so efficient that even their own grandmother could design effectively using their program.

Advanced Circuit Simulation Using Multisim Workbench World Scientific

A circuit simulator is a computer program that permits us to see circuit behavior, i.e. circuit voltages and currents, without making the circuit. Use of a circuit simulator is a cheap, efficient, and safe way to study the behavior of circuits. The Toolkit for Interactive Network Analysis (TINA®) is a powerful yet affordable SPICE based circuit simulation and PCB design software package for analyzing, designing, and real time testing of analog, digital, VHDL, MCU, and mixed electronic circuits and their PCB layouts. This software was created by DesignSoft. TINA-TI is a spinoff software program that was designed by Texas Instruments (TI®) in cooperation with DesignSoft which incorporates a library of pre-made TI components for the user to utilize in their designs. This book shows how a circuit can be analyzed in the TINA-TI® environment. Students of engineering (for instance, electrical, biomedical, mechatronics, and robotics to name a few), engineers who work in the industry, and anyone who wants to learn the art of circuit simulation with TINA-TI can benefit from this book.

Circuit Simulation Methods and Algorithms Springer Science & Business Media

-- Learn to use Spice circuit simulation software at the same time as mastering essential analog electronics-- Master Spice through core analog electronics, not from a software guide or advanced circuit design tome-- Includes Free CD-ROM with netlists for all circuits in the book, additional circuits, and a free limited-function version of the circuit simulation application SpiceAge for WindowsDevelop this key skill of modern circuit design through the essentials of analog electronics. Analog Circuit Design with SPICE introduces circuit simulation with SPICE in a way which all electronics professionals, students and amateurs will understand -- through the basics of analog electronics.By introducing Spice through the fundamentals of electronics, professionals and technicians operating at a higher level are given the chance to put Spice through its paces. The comprehensive topic coverage also makes this a useful reference source for anyone using Spice simulation in a variety of circuit design applications.

SPICE for Power Electronics and Electric Power Morgan & Claypool Publishers

This comprehensive volume covers both elementary and advanced analog and digital circuit simulation using PSpice. The text includes many worked examples, circuit diagrams, tables, and code listings. It also compares practical results with those obtained from simulation.

Electric and Electronic Circuit Simulation Using TINA-TI Prentice Hall

Multisim is now the de facto standard for circuit simulation. It is a SPICE-based circuit simulator which combines analog, discrete-time, and mixed-mode circuits. In addition, it is the only simulator which incorporates microcontroller simulation in the same environment. It also includes a tool for printed circuit board design. Advanced Circuit Simulation Using Multisim Workbench is a companion book to Circuit Analysis Using Multisim, published by Morgan & Claypool in 2011. This new book covers advanced analyses and the creation of models and subcircuits. It also includes coverage of transmission lines, the special elements which are used to connect components in PCBs and integrated circuits. Finally, it includes a description of Ultiboard, the tool for PCB creation from a circuit description in Multisim. Both books completely cover most of the important features available for a successful circuit simulation with Multisim. Table of Contents: Models and Subcircuits / Transmission Lines / Other Types of Analyses / Simulating Microcontrollers / PCB Design With Ultiboard

Passive Circuit Analysis with LTspice® Cambridge University Press This text discusses simulation process for circuits including clamper, voltage and current divider, transformer modeling, transistor as an amplifier, transistor as a switch, MOSFET modeling, RC and LC filters, step and impulse response to RL and RC circuits, amplitude modulator in a step-by-step manner for more clarity and understanding to the readers. It covers electronic circuits like rectifiers, RC filters, transistor as an amplifier, operational amplifiers, pulse response to a series RC circuit, time domain simulation with a triangular input signal, and modulation in detail. The text presents issues that occur in practical implementation of various electronic circuits and assist the readers in finding solutions to those issues using the software. Aimed at undergraduate, graduate students, and academic researchers in the areas including electrical and electronics and communications engineering, this book: Discusses simulation of analog circuits and their behavior for different parameters. Covers AC/DC circuit modeling using regular and parametric sweep methods. The theory will be augmented with practical electrical circuit examples that will help readers to better understand the topic. Discusses circuits like rectifiers, RC filters, transistor as an amplifier, and operational amplifiers in detail.

Silicon and Beyond John Wiley & Sons

Mixed-Mode Simulation and Analog Multilevel Simulation addresses the problems of simulating entire mixed analog/digital systems in the time-domain. A complete hierarchy of modeling and simulation methods for analog and digital circuits is described. Mixed-Mode Simulation and Analog Multilevel Simulation also provides a chronology of the research in the field of mixed-mode simulation and analog multilevel simulation over the last ten to fifteen years. In addition, it provides enough information to the reader so that a prototype mixed-mode simulator could be developed using the algorithms in this book. Mixed-Mode Simulation and Analog Multilevel Simulation can also be used as documentation for the SPLICE family of mixed-mode programs as they are based on the algorithms and techniques described in this book.

Circuit Analysis with Multisim CRC Press

Circuit Simulation Methods and Algorithms provides a step-by-step theoretical consideration of methods, techniques, and algorithms in an easy-to-understand format. Many illustrations explain more difficult problems and present instructive circuits. The book works on three levels: The simulator-user level for practitioners and students who want to better understand circuit simulators. The basic theoretical level, with examples, dedicated to students and beginning researchers. The thorough level for deep insight into circuit simulation based on computer experiments using PSPICE and OPTIMA. Only basic mathematical knowledge, such as matrix algebra, derivatives, and integrals, is presumed.

Advanced Circuit Simulation Using Multisim Workbench Prentice Hall

This book is all about Spice Circuit Simulations Using LTspice. LTspice is available free from Linear Technology. LTspice is perhaps one of the most widely used free simulators. It is a powerful simulator with a simple interface to handle. The book covers the requirements of a laboratory course in SPICE simulations at an introductory level. It can be used an aid to practical understanding in any undergraduate engineering course of Analog electronics. The book can also be used as an aid to any standard text on Analog Electronics. Salient Features: - Step by step simulation procedure is presented - Experiments are clearly illustrated. - Brief theory on each topic for understanding is presented.

Computer Simulation of Electronic Circuits Springer

This book introduces the basic

mathematical tools used to describe noise and its propagation through linear systems and provides a basic description of the improvement of signal-to-noise ratio by signal averaging and linear filtering. The text also demonstrates how op amps are the keystone of modern analog signal conditioning systems design, and il

The Art of Simulation Using PSPICEAnalog and Digital Springer Science & Business Media

This is a guide to the SPICE simulation program which provides practical methods for generating simulations that are fast, accurate and convergent. The accompanying CD features a Windows-compatible version of RSPICE, the author's simulator, which can be used to model circuits.

Switch-Mode Power Supply Simulation: Designing with SPICE 3 CRC Press

The first book on the market that teaches how to use the Electronics Workbench MultiSIM software, this most in-depth manual contains step-by-step screen captures that show how to create a circuit, how to run different analyses, and how to obtain the results from those analyses, allowing the user to self-teach. It contains topics that will be useful throughout the users' careers, making it an invaluable reference work.It features simulations of the same circuits using both the MultiSIM Virtual Lab and SPICE analyses to show users the connection between circuit operation, lab measurements, and SPICE simulation results.An invaluable handbook and reference guide for electrical engineers, electronics engineers, circuit simulation specialists, computer engineers, power electronics employees, analog electronics employees, and project managers.