Pspice User S Guide

Schaum's Outline of Basic Circuit Analysis
Distributed Generation Applications
Handbook for Design and Application
Introduction to PSpice Manual, Using ORCad Release 9.2 to Accompany Electric Circuits, Seventh Edition
An Analog Electronics Companion
MicroSim PSpice AD & Basics+
Semiconductor Device Modeling with Spice
ICT Systems-Theory, Radio-Electronics, Information Technologies and Cybersecurity (Volume 5)
Circuits & Networks 4E
Analog Design and Simulation Using OrCAD Capture and PSpice
An Integrated Approach
SPICE
Theory and Practice
Power Integrity for Electrical and Computer Engineers
Personal Wireless Communications
Power-Switching Converters, Second Edition
Pspice
PSPICE and MATLAB for Electronics
Circuits and Networks: Analysis and Synthesis, 5
The 12th IFIP International Conference on Personal Wireless Communications (PWC 2007), Prague, Czech Republic, September 2007
SPICE
SPICE for Power Electronics and Electric Power
LABORATORY EXPERIMENTS AND PSPICE SIMULATIONS IN ANALOG ELECTRONICS
MicroSim PSpice A/D & Basics+
User's Guide
Transmission Lines in Digital and Analog Electronic Systems
A Guide to Circuit Simulation and Analysis Using PSpice
Nanoelectronic Coupled Problems Solutions
Electronic Circuits
PSpice and Probe Circuit Analysis Program User's Guide
Circuit Analysis Software; User's Guide, [version 6.3]
Circuits & Networks,3E
Practical RF Amplifier Design and Performance Optimization with SPICE and Load- and Source-pull Techniques
Complete PCB Design Using OrCAD Capture and PCB Editor
ANALOG ELECTRONICS
Modeling and Control of Fuel Cells
Circuit Simulation with SPICE OPUS
Circuit Analysis Software : User's Guide
The Art of Simulation Using PSPICEAnalog and Digital
Introduction to PSpice Manual Using Orcad Release 9.2 for Introductory Circuits for Electrical and Computing Engineering

Pspice User S Guide ***Downloaded from*** ftp.wtvq.com ***by guest***

JAELYN MORA

Schaum's Outline of Basic Circuit Analysis McGraw Hill Professional
Designs in nanoelectronics often lead to challenging simulation problems and include strong feedback couplings. Industry demands provisions for variability in order to guarantee quality and yield. It also requires the incorporation of higher abstraction levels to allow for system simulation in order to shorten the design cycles, while at the same time preserving accuracy. The methods developed here promote a methodology for circuit-and-system-level modelling and simulation based on best practice rules, which are used to deal with coupled electromagnetic field-circuit-heat problems, as well as coupled electro-thermal-stress problems that emerge in nanoelectronic designs. This book covers: (1) advanced monolithic/multirate/co-simulation techniques, which are combined with envelope/wavelet approaches to create efficient and robust simulation techniques for strongly coupled systems that exploit the different dynamics of sub-systems within multiphysics problems, and which allow designers to predict reliability and ageing; (2) new generalized techniques in Uncertainty Quantification (UQ) for coupled problems to include a variability capability such that robust design and optimization, worst case analysis, and yield estimation with tiny failure probabilities are possible (including large deviations like 6-sigma); (3) enhanced sparse, parametric Model Order Reduction techniques with a posteriori error estimation for coupled problems and for UQ to reduce the complexity of the sub-systems while ensuring that the operational and coupling parameters can still be varied and that the reduced models offer higher abstraction levels that can be efficiently simulated. All the new algorithms produced were implemented, transferred and tested by the EDA vendor MAGWEL. Validation was conducted on industrial designs provided by end-users from the semiconductor industry, who shared their feedback, contributed to the measurements, and supplied both material data and process data. In closing, a thorough comparison to measurements on real devices was made in order to demonstrate the algorithms' industrial applicability.
Distributed Generation Applications John Wiley & Sons

This laboratory manual for students of Electronics, Electrical, Instrumentation, Communication, and Computer engineering disciplines has been prepared in the form of a standalone text, offering the necessary theory and circuit diagrams with each experiment. Procedures for setting up the circuits and measuring and evaluating their performance are designed to support the material of the authors' book Analog Electronics (also published by PHI Learning). There are twenty-five experiments. The experiments cover the basic transistor circuits, the linear op-amp circuits, the active filters, the non-linear op-amp circuits, the signal generators, the voltage regulators, the power amplifiers, the high frequency amplifiers, and the data converters. In addition to the hands-on experiments using traditional test equipment and components, this manual describes the simulation of circuits using PSPICE as well. For PSPICE simulation, any available standard SPICE software may be used including the latest version OrCAD V10 Demo software. This feature allows the instructor to adopt a single laboratory manual for both types of experiments.

Handbook for Design and Application John Wiley & Sons

The International conference on Personal Wireless Communications (PWC 2007) was the twelfth conference of its series aimed at stimulating technical exchange between researchers, practitioners and students interested in mobile computing and wireless networks. The program covered a variety of research topics that are of current interest, including Ad-Hoc Networks, WiMAX, Heterogeneous Networks, Wireless Networking, QoS and Security, Sensor Networks, Multicast and Signal processing.

Introduction to PSpice Manual, Using ORCad Release 9.2 to Accompany Electric Circuits, Seventh Edition Springer Nature

A professional guide to the fundamentals of power integrity analysis with an emphasis on silicon level power integrity Power Integrity for Electrical and Computer Engineers embraces the most recent changes in the field, offers a comprehensive introduction to the discipline of power integrity, and provides an overview of the fundamental principles. Written by noted experts on the topic, the book goes beyond most other resources to focus on the detailed aspects of silicon and optimization techniques in order to broaden the field of study. This important book offers coverage of a wide range of topics including signal analysis, EM concepts for PI, frequency domain analysis for PI, numerical methods (overview) for PI, and silicon device PI modeling. Power Integrity for Electrical and Computer Engineers examine platform technologies, system considerations, power conversion, system level modeling, and optimization methodologies. To reinforce the material presented, the authors include example problems. This important book: • Includes coverage on convergence, accuracy, and error analysis and explains how these can be used to analyze power integrity problems • Contains information for modeling the power converter from the PDN to the load in a full system level model • Explores areas of device level modeling of silicon as related to power integrity • Contains example word problems that are related to an individual chapter’s subject Written for electrical and computer engineers and academics, Power Integrity for Electrical and Computer Engineers is an authoritative guide to the fundamentals of power integrity and explores the topics of power integrity analysis, power integrity analytics, silicon level power integrity, and optimization techniques.

An Analog Electronics Companion CRC Press

This book explains and demonstrates with an exhaustive set of design examples, how common types of radio frequency(RF) amplifiers (classes A, B, AB, C, D, E, F, G and H) can be designed, and then have their performance characteristics evaluated and optimized with SPICE. The author demonstrates the transient analysis features of SPICE, along with industry-standard load- and source-pull techniques to simulate the steady-state, long-term time-domain behavior of any test RF amplifier.· Describes methods for designing and evaluating/optimizing the performance characteristics of an RF amplifier that circumvent the issues involved with existing, traditional methods and don’t require expensive, high-end software tools;· Includes C language executables for each RF amplifier type, eliminating errors that might creep in while computing passive component (capacitor, inductor, resistor) values for a given RF amplifier type;· Demonstrates industry-standard load- and source-pull schemes that can be included easily in text SPICE netlists, allowing accurate calculation of impedance matching and impedance values at the input and output ports of the test RF amplifier, eliminating messy, error-prone S parameter based calculations.

MicroSim PSpice AD & Basics+ Springer Nature

Readers benefit because the book is based on these three themes: (1) it builds an understanding of concepts based on information the reader has previously learned; (2) it helps stress the relationship between conceptual understanding and problem-solving approaches; (3) the authors provide numerous examples and problems that use realistic values and situations to give users a strong foundation of engineering practice. The book also includes a PSpice Supplement which contains problems to teach readers how to construct PSpice source files; and this PSpice Version 9.2 can be used to solve many of the exercises and problems found in the book. Topical emphasis is on the basic techniques of circuit analysis -- Illustrated via a Digital-to-Analog Resistive Ladder (Chapter 2); the Flash Converter (Chapter 4); Dual Slope Analog-to-Digital Converter (Chapter 5); Effect of parasite inductance on the step response of a series RLC circuit (Chapter 6); a Two-Stage RC Ladder Network (Chapter 8); and a Switching Surge Voltage (Chapter 9).

Semiconductor Device Modeling with Spice John Wiley & Sons

"The emerging fuel cell (FC) technology is growing rapidly in its applications from small-scale portable electronics to large-scale power generation. This book gives students, engineers, and scientists a solid understanding of the FC dynamic modeling and controller design to adapt FCs to particular applications in distributed power generation." "The book begins with a fascinating introduction to the subject, including a brief history of the U.S. electric utility formation and restructuring. Next, it provides coverage of power deregulation and distributed generation (DG), DG types, fuel cell DGs, and the hydrogen economy. Modeling and Control of Fuel Cells is an excellent reference book for students and professionals in electrical, chemical, and mechanical engineering and scientists working in the FC area."--BOOK JACKET.

ICT Systems-Theory, Radio-Electronics, Information Technologies and Cybersecurity (Volume 5) Tata McGraw-Hill Education

Significantly expanded and updated with extensive revisions, new material, and a new chapter on emerging applications of switching converters, Power-Switching Converters, Third Edition offers the same trusted, accessible, and comprehensive information as its bestselling predecessors. Similar to the two previous editions, this book can be used for an introductory as well as a more advanced course. Chapters begin with an introduction to switching converters and basic switching converter topologies. Entry level chapters continue with a discussion of resonant converters, isolated switching converters, and the control schemes of switching converters. Skipping to chapters 10 and 11, the subject matter involves an examination of interleaved converters and switched capacitor converters to round out and complete the overview of switching converter topologies. More detailed chapters include the continuous time-modeling and discrete-time modeling of switching converters as well as analog control and digital control. Advanced material covers tools for the simulation of switching converters (including both PSpice and Matlab simulations) and the basic concepts necessary to understand various actual and emerging applications for switching converters, such as power factor correction, LED drivers, low-noise converters, and switching converters topologies for solar and fuel cells. The final chapter contains several complete design examples, including experimental designs that may be used as technical references or for class laboratory projects. Supplementary information is available at crcpress.com including slides, PSpice examples (designed to run on the OrCAD 9.2 student version and PSIM software) and MATLAB scripts. Continuing the august tradition of its predecessors, Power-Switching Converters, Third Edition provides introductory and advanced information on all aspects of power switching converters to give students the solid foundation and applicable knowledge required to advance in this growing field.

Circuits & Networks 4E McGraw Hill Professional

Engineers and scientists frequently find themselves having to get involved in electronic circuit design even though this may not be their specialty. This book is specifically designed for these situations, and has two major advantages for the inexperienced designer: it assumes little prior knowledge of electronics and it takes a modular approach, so you can find just what you need without working through a whole chapter. The first three parts of the book start by refreshing the basic mathematics and physics needed to understand circuit design. Part four discusses individual components (resistors, capacitors etc.), while the final and largest section describes commonly encountered circuit elements such as differentiators, oscillators, filters and couplers. A major bonus and learning aid is the inclusion of a CD-ROM with the student edition of the PSpice simulation software, together with models of most of the circuits described in the book.

Analog Design and Simulation Using OrCAD Capture and PSpice Springer

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.

An Integrated Approach Springer Science & Business Media

This book is a unique combination of a basic guide to general analog circuit simulation and a SPICE OPUS software manual, which may be used as a textbook or self-study reference. The book is divided into three parts: mathematical theory of circuit analysis, a crash course on SPICE OPUS, and a complete SPICE OPUS reference guide. All simulations as well as the free simulator software may be directly downloaded from the SPICE OPUS homepage: www.spiceopus.si. Circuit Simulation with SPICE OPUS is intended for a wide audience of undergraduate and graduate students, researchers, and practitioners in electrical and systems engineering, circuit design, and simulation development.

SPICE PHI Learning Pvt. Ltd.

This text offers a comprehensive introduction to a wide, relevant array of topics in analog electronics. It is intended for students pursuing courses in electrical, electronics, computer, and related engineering disciplines. Beginning with a review of linear circuit theory and basic electronic devices, the text moves on to present a detailed, practical understanding of many analog integrated circuits. The most commonly used analog IC to build practical circuits is the operational amplifier or op-amp. Its characteristics, basic configurations and applications in the linear and nonlinear circuits are explained. Modern electronic systems employ signal generators, analog filters, voltage regulators, power amplifiers, high frequency amplifiers and data converters. Commencing with the theory, the design of these building blocks is thoroughly covered using integrated circuits. The development of microelectronics technology has led to a parallel growth in the field of Micro-electromechanical Systems (MEMS) and Nano-electromechanical Systems (NEMS). The IC sensors for different energy forms with their applications in MEMS components are introduced in the concluding chapter. Several computer-based simulations of electronic circuits using PSPICE are presented in each chapter. These examples together with an introduction to PSPICE in an Appendix provide a thorough coverage of this simulation tool that fully integrates with the material of each chapter. The end-of-chapter problems allow students to test their comprehension of key concepts. The answers to these problems are also given.

Theory and Practice John Wiley & Sons

After nearly a decade of success owing to its thorough coverage, abundance of problems and examples, and practical use of simulation and design, Power-Switching Converters enters its second edition with new and updated material, entirely new design case studies, and expanded figures, equations, and homework problems. This textbook is ideal for senior undergraduate or graduate courses in power electronic converters, requiring only systems analysis and basic electronics courses. The only text of such detail to also include the use of PSpice and step-by-step designs and simulations, Power-Switching Converters, Second Edition covers basic topologies, basic control techniques, and closed-loop control and stability. It also includes two new chapters on interleaved converters and switched capacitor converters, and the authors have added discrete-time modeling to the dynamic analysis of switching converters. The final two chapters are dedicated to simulation and complete design examples, respectively. PSpice examples and MATLAB scripts are available for download from the CRC Web site. These are useful for the simulation of students' designs. Class slides are also available on the Internet. Instructors will appreciate the breadth and depth of the material, more than enough to adapt into a customized syllabus. Students will similarly benefit from the more than 440 figures and over 1000 equations, ample homework problems, and case studies presented in this book.

Power Integrity for Electrical and Computer Engineers PHI Learning Pvt. Ltd.

Electrical drives lie at the heart of most industrial processes and make a major contribution to the comfort and high quality products we all take for granted. They provide the controller power needed at all levels, from megawatts in cement production to milliwatts in wrist watches. Other examples are legion, from the domestic kitchen to public utilities. The modern electrical drive is a complex item, comprising a controller, a static converter and an electrical motor. Some can be programmed by the user. Some can communicate with other drives. Semiconductor switches have improved, intelligent power modules have been introduced, all of which means that control techniques can be used now that were unimaginable a decade ago. Nor has the motor side stood still: high-energy permanent magnets, semiconductor

switched reluctance motors, silicon micromotor technology, and soft magnetic materials produced by powder technology are all revolutionising the industry. But the electric drive is an enabling technology, so the revolution is rippling throughout the whole of industry.
Personal Wireless Communications Springer
Modeling and Simulation have become endeavors central to all disciplines of science and engineering. They are used in the analysis of physical systems where they help us gain a better understanding of the functioning of our physical world. They are also important to the design of new engineering systems where they enable us to predict the behavior of a system before it is ever actually built. Modeling and simulation are the only techniques available that allow us to analyze arbitrarily non-linear systems accurately and under varying experimental conditions. Continuous System Modeling introduces the student to an important subclass of these techniques. They deal with the analysis of systems described through a set of ordinary or partial differential equations or through a set of difference equations. This volume introduces concepts of modeling physical systems through a set of differential and/or difference equations. The purpose is twofold: it enhances the scientific understanding of our physical world by codifying (organizing) knowledge about this world, and it supports engineering design by allowing us to assess the consequences of a particular design alternative before it is actually built. This text has a flavor of the mathematical discipline of dynamical systems, and is strongly oriented towards Newtonian physical science.
Power-Switching Converters, Second Edition PSpice and Probe Circuit Analysis Program User's GuideMicroSim PSpice A/D & Basics+Circuit Analysis Software : User's GuideAnalog Design and Simulation Using OrCAD Capture and PSpice
In the last 30 years there have been dramatic changes in electrical technology--yet the length of the undergraduate curriculum has remained four years. Until some ten years ago, the analysis of transmission lines was a standard topic in the EE and CpE undergraduate curricula. Today most of the undergraduate curricula contain a rather brief study of the analysis of transmission lines in a one-semester junior-level course on electromagnetics. In some schools, this study of transmission lines is relegated to a senior technical elective or has disappeared from the curriculum altogether. This raises a serious problem in the preparation of EE and CpE undergraduates to be competent in the modern industrial world. For the reasons mentioned above, today's undergraduates lack the basic skills to design high-speed digital and high-frequency analog systems. It does little good to write sophisticated software if the hardware is unable to process the instructions. This problem will increase as the speeds and frequencies of these systems continue to increase seemingly without bound. This book is meant to repair that basic deficiency.
Pspice McGraw-Hill Education
PSpice and Probe Circuit Analysis Program User's GuideMicroSim PSpice A/D & Basics+Circuit Analysis Software : User's GuideAnalog Design and Simulation Using OrCAD Capture and PSpiceElsevier
PSPICE and MATLAB for Electronics CRC Press
For improved comprehension of circuit analysis, less time spent studying, and better test scores, you can't do better than this powerful Schaum's Outline! It's the best study tool there is. It gives you hundreds of completely worked problems with full solutions on the information that you really need to know. Hundreds of additional problems let you test your skills, then check the answers. This comprehensive study guide can be used with any textbook, but it's so complete it's ideal for independent study!
Circuits and Networks: Analysis and Synthesis, 5 CRC Press
Complete PCB Design Using OrCAD Capture and PCB Editor, Second Edition, provides practical instruction on how to use the OrCAD design suite to design and manufacture printed circuit boards. Chapters cover how to Design a PCB using OrCAD Capture and OrCAD Layout, adding PSpice simulation capabilities to a design, how to develop custom schematic parts, how to create footprints and PSpice models, and how to perform documentation, simulation and board fabrication from the same schematic design. This book is suitable for both beginners and experienced designers, providing basic principles and the program's full capabilities for optimizing designs. Presents a fully updated edition on OrCAD Capture, Version 17.2 Combines the theoretical and practical parts of PCB design Includes real-life design examples that show how and why designs work, providing a comprehensive toolset for understanding OrCAD software Provides the exact order in which a circuit and PCB are designed Introduces the IPC, JEDEC and IEEE standards relating to PCB design
The 12th IFIP International Conference on Personal Wireless Communications (PWC 2007), Prague, Czech Republic, September 2007 CRC Press
Anyone involved in circuit design that needs the practical know-how it takes to design a successful circuit or product, will find this practical guide to using Capture-PSpice (written by a former Cadence PSpice expert for Europe) an essential book. The text delivers step-by-step guidance on using Capture-PSpice to help professionals produce reliable, effective designs. Readers will learn how to get up and running quickly and efficiently with industry standard software and in sufficient detail to enable building upon personal experience to avoid common errors and pit-falls. This book is of great benefit to professional electronics design engineers, advanced amateur electronics designers, electronic engineering students and academic staff looking for a book with a real-world design outlook. Provides both a comprehensive user guide, and a detailed overview of simulation Each chapter has worked and ready to try sample designs and provides a wide range of to-do exercises Core skills are developed using a running case study circuit Covers Capture and PSpice together for the first time