Pspice User S Guide

"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.

Designed for engineers and scientists who are non-specialist in electronic circuit design.

Analog Design and Simulation using OrCAD Capture and PSpice provides step-by-step instructions on how to use the Cadence/OrCAD family of Electronic Design Automation software for analog design and simulation. Organized into 22 chapters, each with exercises at the end, it explains how to start Capture and set up the project type and libraries for PSpice simulation. It also covers the use of AC analysis to calculate the frequency and phase response of a circuit and DC analysis to calculate the circuits bias point over a range of values. The book describes a parametric sweep, which involves sweeping a parameter through a range of values, along with the use of Stimulus Editor to define transient analog and digital sources. It also examines the failure of simulations due to circuit errors and missing or incorrect parameters, and discusses the use of Monte Carlo analysis to estimate the response of a circuit when device model parameters are randomly varied between specified tolerance limits according to a specified statistical distribution. Other chapters focus on the use of worst-case analysis to identify the most critical components that will affect circuit performance, how to add and create PSpice models, and how the frequency-related signal and dispersion losses of transmission lines affect the signal integrity of high-speed signals via the transmission lines. Practitioners, researchers, and those interested in using the Cadence/OrCAD professional simulation software to design and analyze electronic circuits will find the information, methods, compounds, and experiments described in this book extremely useful. 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

This Handbook presents all aspects of memristor networks in an easy to read and tutorial style. Including many colour illustrations, it covers the foundations of memristor theory and applications, the technology of memristive devices, revised models of the Hodgkin-Huxley Equations and ion channels, neuromorphic architectures, and analyses of the dynamic behaviour of memristive networks. It also shows how to realise computing devices, non-von Neumann architectures and provides future building blocks for deep learning hardware. With contributions from leaders in computer science, mathematics, electronics, physics, material science and engineering, the book offers an indispensable source of information and an inspiring reference text for future generations of computer scientists, mathematicians, physicists, material scientists and engineers working in this dynamic field.

This guide to the PSpice circuit simulator provides a tutorial approach to using PSpice through graduated examples. This edition includes enhanced pedagogical features, and coverage of the newest capabilities of this program. It explains: the use of Monte Carlo methods in PSpice for statistically computing estimates of how circuits will behave with variations in component values and derivation and use of two-port parameters, including s-parameters. It also includes an expanded section on group and time delay, and on noise analysis, as well as fuller descriptions and examples for using parameters, functions and values defined by formulas to generalize circuit blocks and specify component values.

A guide to the use of PSpice in common electrical and electronic problems. This revised edition features two-port network analysis, loop gain analysis, and expanded coverage of group and time delay, noise analysis and macros. Software supplements are available for the IBM PC, IBM PS/2 and Mac 2.

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.

PLEASE PROVIDE COURSE INFORMATION PLEASE PROVIDE

Accompanying CD-ROM contains OrCAD Lite version 9.2 to focus on dc analysis, transient analysis, and steady-state sinusoidal (ac) analysis.

This book is aimed at advanced students and practising engineers. It provides step by step instructions in the use of MicroSim PSpice, industry-standard software that simulates power-electronics circuits. Computer-aided simulation is recognised as the most efficient method of power electronics circuit performance analysis, and is widely used in the industrial marketplace. This book presents a clear and concise guide to one of the most popular software packages. The theory is backed up by drills and exercises throughout, building up practical experience in MicroSim PSpice. The book is intended for use alongside a PC, and a free evaluation version of MicroSim PSpice will be supplied on application to Microsim Corporation. Alternatively, the author's site on the Internet can be accessed at the Internet and the software can be downloaded along with free circuit files, library files and zipped solutions to exercises.

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).

This book is devoted to the latest advances in the area of electrothermal modelling of electronic components and networks. It contains eight sections by different teams of authors. These sections contain the results of: (a) electro-thermal simulations of SiC power MOSFETs using a SPICE-like simulation program; (b) modelling thermal properties of inductors taking into account the influence of the core volume on the efficiency of heat removal; (c) investigations into the problem of inserting a temperature sensor in the neighbourhood of a chip to monitor its junction temperature; (d) computations of the internal temperature of power LEDs situated in modules containing multiple-power LEDs, taking into account both self-heating in each power LED and mutual thermal couplings between each diode; (e) analyses of DC-DC converters using the electrothermal averaged model of the diode–transistor switch, including an IGBT and a rapid-switching diode; (f) electrothermal modelling of SiC power BJTs; (g) analysis of the efficiency of selected algorithms used for solving heat transfer problems at nanoscale; (h) analysis related to thermal simulation of the test structure dedicated to heat-diffusion investigation at the nanoscale.
This book provides a comprehensive overview of current research on memristors, memcapacitors and, meminductors. In addition to an historical overview of the research in this area, coverage includes the theory behind memristive circuits, as well as memcapacitance, and meminductance. Details are shown for recent applications of memristors for resistive random access memories, neuromorphic systems and hybrid CMOS/memristor circuits. Methods for the simulation of memristors are demonstrated and an introduction to neuromorphic modeling is provided.
This outstanding textbook provides an introduction to electronic materials and device concepts for the major areas of current and future information technology. On about 1,000 pages, it collects the fundamental concepts and key technologies related to advanced electronic materials and devices. The obvious strength of the book is its encyclopedic character, providing adequate background material instead of just reviewing current trends. It focuses on the underlying principles which are illustrated by contemporary examples. The third edition now holds 47 chapters grouped into eight sections. The first two sections are devoted to principles, materials processing and characterization methods. Following sections hold contributions to relevant materials and various devices, computational concepts, storage systems, data transmission, imaging systems and displays. Each subject area is opened by a tutorial introduction, written by the editor and giving a rich list of references. The following chapters provide a concise yet in-depth description in a given topic. Primarily aimed at graduate students of physics, electrical engineering and information technology as well as material science, this book is equally of interest to professionals looking for a broader overview. Experts might appreciate the book for having quick access to principles as well as a source for getting insight into related fields.
A step-by-step manual providing detailed explanations and examples of PSpice that can be used with any microelectronics text. It introduces students to the fundamental uses of this book in support of basic microelectronics. The organization allows readers to advance quickly to solve a variety of microelectronic problems.
Power electronics can be a difficult course for students to understand and for professors to teach. Simplifying the process for both, SPICE for Power Electronics and Electric Power, Third Edition illustrates methods of integrating industry standard SPICE software for design verification and as a theoretical laboratory bench. Helpful PSpice Software and Program Files Available for Download Based on the author Muhammad H. Rashid's considerable experience merging design content and SPICE into a power electronics course, this vastly improved and updated edition focuses on helping readers integrate the SPICE simulator with a minimum amount of time and effort. Giving users a better understanding of the operation of a power electronics circuit, the author explores the transient behavior of current and voltage waveforms for each and every circuit element at every stage. The book also includes examples of all types of power converters, as well as circuits with linear and nonlinear inductors. New in this edition: Student learning outcomes (SLOs) listed at the start of each chapter Changes to run on OrCAD version 9.2 Added VPRINT1 and IPRINT1 commands and examples Notes that identify important concepts Examples illustrating EVALUE, GVALUE, ETABLE, GTABLE, ELAPLACE, GLAPLACE, EFREQ, and GFREQ Mathematical relations for expected outcomes, where appropriate The Fourier series of the output voltages for rectifiers and inverters PSpice simulations of DC link inverters and AC voltage controllers with PWM control This book demonstrates techniques of executing power conversions and ensuring the quality of the output waveforms rather than the accurate modeling of power semiconductor devices. This approach benefits students, enabling them to compare classroom results obtained with simple switch models of devices. In addition, a new chapter covers multi-level converters. Assuming no prior knowledge of SPICE or PSpice simulation, the text provides detailed step-by-step instructions on how to draw a schematic of a circuit, execute simulations, and view or plot the output results. It also includes suggestions for laboratory experiments and design problems that can be used for student homework assignments.
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.
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.

MicroSim PSpice A/D & Basics+Circuit Analysis Software : User's GuideMicroSim PSpice & Basics : Circuit Analysis SoftwareUser's GuideMicroSim PSpice AD & Basics+Circuit Analysis Software; User's Guide, [version 6.3]LABORATORY EXPERIMENTS AND PSPICE SIMULATIONS IN ANALOG ELECTRONICSPHI Learning Pvt. Ltd.

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.

This book addresses the challenges and opportunities of information/data processing and management. It also covers a range of methods, techniques and strategies for making it more efficient, approaches to increasing its usage, and ways to minimize information/data loss while improving customer satisfaction. Information and Communication Technologies (ICTs) and the Service Systems associated with them have had an enormous impact on businesses and our day-to-day lives over the past three decades, and continue to do so. This development has led to the emergence of new application areas and relevant disciplines, which in turn present new challenges and opportunities for service system usage. The book provides practical insights into various aspects of ICT technologies for service systems: Techniques for information/data processing and modeling in service systems Strategies for the provision of information/data processing and management Methods for collecting and analyzing information/data Applications, benefits, and challenges of service system implementation Solutions to increase the performance of various service systems using the latest ICT technologies

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.

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.

Publisher's Note: Products purchased from Third Party sellers are not guaranteed by the publisher for quality, authenticity, or access to any online entitlements included with the product.

This is a practical approach to control techniques. The author covers background material on analog controllers, digital controllers, and filters. Commonly used controllers are presented. Extended use of PSpice (a popular circuit simulation program) is used in problem solving. The book is also documented with 50 computer programs that circuit designers can use. Explains integration of control systems with a personal computer**Compares numerous control algorithms in digital and analog form**Details the use of SPICE in problem solving**Presents modeling concepts for linear and nonlinear systems**Examines commonly used controllers

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.

Used collectively, PSPICE and MATLAB are unsurpassed for circuit modeling and data analysis. PSPICE can perform DC, AC, transient, Fourier, temperature, and Monte Carlo analysis of electronic circuits with device models and subsystem subcircuits. MATLAB can then carry out calculations of device parameters, curve fitting, numerical integration, nume

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.

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.

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.

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.

This book caters to a course on Circuits and Networks with coverage of both Analysis and Synthesis. Lucid language, fundamental discussions and illustrative examples are some of the excellent features of this text. There are numerous solved examples employing the step wise problem solving approach which helps in easy grasping of the concepts by the students. The numericals employ both AC and DC methods of analysis. Multiple Choice Questions and Practice problems have been provided in plenty and are of graded challenge levels, helping the students to prepare for competitive examinations. PSpice problems have been incorporated to help in simulation.