Pspice Simulation Of Power Electronics Circuits 1st Edition

Market_Desc: · Electrical Engineering Students · Electrical Engineering Instructors· Power Electronics Engineers Special Features: · Easy to follow step-by-step in depth treatment of all the theory.· Computer simulation chapter describes the role of computer simulations in power electronics. Examples and problems based on Pspice and MATLAB are included.· Introductory chapter offers a review of basic electrical and magnetic circuit concepts.· A new CD-ROM contains the following:· Over 100 of new problems of varying degrees of difficulty for homework assignments and self-learning.· PSpice-based simulation examples, which illustrate basic concepts and help in design of converters.· A newly-developed magnetic component design program that demonstrates design trade-offs.· PowerPoint-based slides, which will improve the learning experience and the ease of using the book About The Book: The text includes cohesive presentation of power electronics fundamentals for applications and design in the power range of 500 kW or less. It describes a variety of practical and emerging power electronic converters made feasible by the new generation of power semiconductor devices. Topics included in this book are an expanded discussion of diode rectifiers and thyristor converters as well as chapters on heat sinks, magnetic components which present a step-by-step design approach and

a computer simulation of power electronics which introduces numerical techniques and commonly used simulation packages such as PSpice, MATLAB and EMTP.

"This book uses a top-down approach to introduce readers to the SPICE simulator. It begins by describing techniques for simulating circuits, then presents the various SPICE and OrCAD commands and their applications to electrical and electronic circuits. Lavishly illustrated, this new edition includes even more hands-on exercises, suggestions, sample problems, and circuit models of actual devices. It is an ideal supplement for courses in electric or electronic circuitry and is also a solid professional reference."--BOOK JACKET.Title Summary field provided by Blackwell North America, Inc. All Rights Reserved

This fully updated textbook provides complete coverage of electrical circuits and introduces students to the field of energy conversion technologies, analysis and design. Chapters are designed to equip students with necessary background material in such topics as devices, switching circuit analysis techniques, converter types, and methods of conversion. The book contains a large number of examples, exercises, and problems to help enforce the material presented in each chapter. A detailed discussion of resonant and softswitching dc-to-dc converters is included along with the addition of new chapters covering digital control, non-linear control, and micro-inverters for power electronics applications.

Designed for senior undergraduate and graduate electrical engineering students, this book provides students with the ability to analyze and design power electronic circuits used in various industrial applications. 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

PSPICE A Powerful Simulation Tool for Power Electronics & VLSI Design

Practical Examples Using the PSpice A/d Demo to Simulate Power Electronic and Electrical Power Circuits

Interactive Modelling Using Simulink
Converters, Applications, and Design
Magnetic Components for Power Electronics

Provides a step-by-step method for the development of a virtual interactive power electronics laboratory. The book is suitable for undergraduates and graduates for their laboratory course and projects in power electronics. It is equally suitable for professional engineers in the power electronics industry. The reader will learn to develop interactive virtual power electronics laboratory and perform simulations of their own, as well as any given power electronic converter design using SIMULINK with advanced system model and circuit component level model. Features Examples and Case Studies included throughout. Introductory simulation of power electronic converters is performed using either PSIM or MICROCAP Software. Covers interactive system model developed for three phase Diode Clamped Three Level Inverter, Flying Capacitor Three Level Inverter, Five Level Cascaded H-Bridge Inverter, Multicarrier Sine Phase Shift PWM and Multicarrier Sine Level Shift PWM. System models of power electronic converters are verified for performance using interactive circuit component level models developed using Simscape-Electrical, Power Systems and Specialized Technology block set. Presents software in the loop or Processor in the loop simulation with a power electronic converter examples.

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.

Power Electronics is intended to be an introductory text in power electronics, primarily for the undergraduate electrical engineering student. The text is written for some flexibility in the order of the topics. Much of the text includes computer simulation using PSpice as a supplement to analytical circuit solution techniques.

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.

Advances and Trends

Proceedings of ICAET 2020

SPICE for Power Electronics and Electric Power

SPICE for Circuits and Electronics Using PSpice

Power Electronic Converters

Power electronics systems are nonlinear variable structure systems. They involve passive components such as resistors, capacitors, and inductors, semiconductor switches such as thyristors and MOSFETs, and circuits for control. The analysis and design of such systems presents significant challenges. Fortunately, increased availability of powerful computer and simulation programs makes the analysis/design process much easier. PSIM® is an electronic circuit simulation software package, designed specifically for use in power electronics and motor drive simulations but can be used to simulate any electronic circuit. With fast simulation speed and user friendly interface, PSIM provides a powerful simulation environment to meed the user simulation and development needs. This book shows how to simulate the power electronics circuits

in PSIM environment. The prerequisite for this book is a first course on power electronics. This book is composed of eight chapters: Chapter 1 is an introduction to PSIM. Chapter 2 shows the fundamentals of circuit simulation with PSIM. Chapter 3 introduces the SimviewTM. Simview is PSIM's waveform display and post-processing program. Chapter 4 introduces the most commonly used components of PSIM. Chapter 5 shows how PSIM can be used for analysis of power electronics circuits. 45 examples are studied in this chapter. Chapter 6 shows how you can simulate motors and mechanical loads in PSIM. Chapter 7 introduces the SimCouplerTM. Simcoupler fuses PSIM with Simulink® by providing an interface for co-simulation. Chapter 8 introduces the SmartCtrl®. SmartCtrl is a controller design software specifically geared towards power electronics applications. https://powersimtech.com/2021/10/01/book-release-power-electronics-circuit-analysis-with-psim/

This book is covered with simulation procedure of Power Electronics and VLSI circuit in detail using PSPICE Simulation tool. The purpose of this Book is to provide a guideline how to simulate and analyze power electronics and VLSI circuits which are building block of a complex circuit. It is possible to analyze the circuit in different ways using PSPICE Simulation tool. This book is useful for simulation of Power Electronics circuits, making simulation project useful for UG, PG and research scholar subjected to power electronics and VLSI design.

Most power electronics textbooks use PSpice for the simulation of circuits, even though MATLAB is a much easier and user-friendly tool. Fundamentals of Power Electronics Using MATLAB teaches students and engineers how to use MATLAB as a simulation and computational tool for power electronics. Designed as a hands-on reference, the scope of the material in the text is not as broad as other reference-style texts, thus making the material less intimidating and more

attainable to the reader. Each portion of the text starts with an example based on the section material, followed by a detailed solution. A conclusion is then drawn to emphasize the ?point? of the problem and finally an exercise similar to the example is presented to challenge engineer. This format provides an immediate illustration of how to use the material and an opportunity for students to apply the material on their own. The text also introduces sliding mode control (SMC) of converter circuits where the converter is treated as a variable structure system, in addition to traditional pulse-width-modulation (PWM) control. SMC is a relatively new method of control and is a robust and attractive alternative to PWM. Engineers and students do not need to be proficient in MATLAB to work along with the text because a toolbox is provided on the companion CD-ROM that allows them to use MATLAB and obtain results immediately. The toolbox provides functions to perform power computations, waveform analysis, and power converter circuit design and simulations.

Simulation of Software Tools for Electrical Systems: Theory and Practice offers engineers and students what they need to update their understanding of software tools for electric systems, along with guidance on a variety of tools on which to model electrical systems—from device level to system level. The book uses MATLAB, PSIM, Pspice and PSCAD to discuss how to build simulation models of electrical systems that assist in the practice or implementation of simulation software tools in switches, circuits, controllers, instruments and automation system design. In addition, the book covers power electronic switches and FACTS controller device simulation model building with the use of Labview and PLC for industrial automation, process control, monitoring and measurement in electrical systems and hybrid optimization software HOMER is presented for researchers in renewable energy systems. Includes interactive content for numerical

computation, visualization and programming for learning the software tools related to electrical sciences Identifies complex and difficult topics illustrated by useable examples Analyzes the simulation of electrical systems, hydraulic, and pneumatic systems using different software, including MATLAB, LABVIEW, MULTISIM, AUTOSIM and PSCAD

Computer Simulation, Analysis, and Education Using PSpice Schematics

Computer Simulation, Analysis and Education Using PSpice

Introduction to Power Electronics

converters, applications, and design

Switch-Mode Power Supply Simulation: Designing with SPICE 3

This book presents best selected papers presented at the International Conference on Advances in Energy Technology (ICAET 2020) organized by Gandhi Institute for Education and Technology (GIET), Bhubaneswar, India, during 17–18 January 2020. The proceeding targets the current research works that may lead to sustainable development of new products and techniques. Carefully reviewed works from the submission are selected to include in the book. It is broadly having four divisions based on the tracks – energy systems, energy technology, green technology, and renewal energy. Emphasis is mainly given on inclusion of original research works within the scope.

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

To be accredited, a power electronics course should cover a significant amount of design content and include extensive use of computer-aided analysis with simulation tools such as SPICE. Based upon the authors' experience in designing such courses, SPICE for Power Electronics and Electric Power, Second Edition integrates a SPICE simulator with a po

An introductory textbook in power electronics for electronic engineers. Acknowledging the very wide scope of power electronics, this book aims to approach the subject from the low power end of the spectrum. The first three chapters review the background technology of power electronics, covering active devices, thermal modelling and magnetics, while the rest of the book examines techniques and applications, in particular high frequency switching techniques. There are numerous review questions and worked examples; coverage of DC power supplies from simple to SMPs; case studies of switching regulations; and full listings provided for computer simulation examples using PSpice.

PSPICE and MATLAB for Electronics

Analog Design and Simulation Using OrCAD Capture and PSpice

Power Electronics

Digital Signal Processing in Power Electronics Control Circuits

Introduction to Modern Power Electronics

PSpice for Circuit Theory and Electronic Devices is one of a series of five PSpice books and introduces the latest Cadence Orcad PSpice version 10.5 by simulating a range of DC and AC exercises. It is aimed primarily at those wishing to get up to speed with this version but will be of use to high school students, undergraduate students, and of course, lecturers. Circuit theorems are applied to a range of circuits and the calculations by hand after analysis are then compared to the simulated results. The Laplace transform and the s-plane are used to analyze CR and LR circuits where transient signals are involved. Here, the Probe output graphs demonstrate what a great learning tool PSpice is by providing the reader with a visual verification of any theoretical calculations. Series and parallel-tuned resonant circuits are investigated where the difficult concepts of dynamic impedance and selectivity are best understood by sweeping different circuit parameters through a range of values. Obtaining semiconductor device characteristics as a laboratory exercise has fallen out of favour of late, but nevertheless, is still a useful exercise for understanding or modelling semiconductor devices. Inverting and non-inverting operational amplifiers characteristics such as gain-bandwidth are investigated and we will see the dependency of bandwidth on the gain using the performance analysis facility. Power amplifiers are examined where PSpice/Probe demonstrates very nicely the problems of cross-over distortion and other problems associated with power transistors. We examine power supplies and the problems of regulation, ground bounce, and power

factor correction. Lastly, we look at MOSFET device characteristics and show how these devices are used to form basic CMOS logic gates such as NAND and NOR gates.
This textbook, designed for undergraduate students of electrical engineering, offers a comprehensive and accessible introduction to state-of-the-art power semiconductor devices and power electronic converters with an emphasis on design, analysis and realization of numerous types of systems. Each topic is discussed in sufficient depth to expose the fundamental principles, concepts, techniques, methods and circuits, necessary to thoroughly understand power electronic systems.
Power electronics is interdisciplinary and is at the confluence of three fundamental technical areas - power, electronics and control, .and is used in a wide variety of industries from computers to chemical plants to rolling mills. The importance of power electronics has grown over the years due to several factors. Computer simulation can greatly aid in the analysis, design and education of Power

Electronics. A computer simulation (or "sim") is an attempt to model a real-life or hypothetical situation on a computer so that it can be studied to see how the system works. By changing variables, predictions may be made about the behavior of the system. In our work towards this we have ensured to bring out the different responses of current and voltage in the power electronics circuits. However, simulation of power electronics systems is made challenging by the following factors: 1) Extreme non-linearity presented by switches, 2) Time constants within the system may differ by several orders of magnitude and 3) A lack of models. Therefore, it is important that the objective of the computer analysis be evaluated carefully and an appropriate simulation package be chosen. In view of the above considerations, a SPICE based simulation package PSpice and PSIM have been chosen by us for this very purpose. They have had the detailed device models and have been able to represent the controller portion of the converter system by its functional features in as simplified a manner as possible.

Magnetic Components for Power Electronics concerns the important considerations necessary in the choice of the optimum magnetic component for power electronic applications. These include the topology of the converter circuit, the core material, shape, size and others such as cost and potential component suppliers. These are all important for the design engineer due to the emergence of new materials, changes in supplier management and the examples of several component choices. Suppliers using this volume will also understand the needs of designers. Highlights include: Emphasis on recently introduced new ferrite materials, such as those operating at megahertz frequencies and under higher DC drive conditions; Discussion of amorphous and nanocrystalline metal materials; New technologies such as resonance converters, power factors correction (PFC) and soft switching; Catalog information from over 40 magnetic component suppliers; Examples of methods of component choice for ferrites, amorphous nanocrystalline materials; Information on suppliers management changes such as those occurring at Siemens, Philips, Thomson and Allied-Signal; Attention to the increasingly important concerns about EMI. This book should be especially helpful for power electronic circuit designers, technical executives, and material science engineers involved with power electronic components.

Modelling Photovoltaic Systems Using PSpice

Fundamentals of Power Electronics with MATLAB

Power Electronics Circuit Analysis with PSIM®

PSpice for Circuit Theory and Electronic Devices

POWER ELECTRONICS

Many digital control circuits in current literature are described using analog transmittance. This may not always be acceptable, especially if the sampling frequency and power transistor switching frequencies are close to the band of interest. Therefore, a digital circuit is considered as a digital controller rather than an analog circuit. This helps to avoid errors and instability in high frequency components. Digital Signal Processing in Power Electronics Control Circuits covers

problems concerning the design and realization of digital control algorithms for power electronics circuits using digital signal processing (DSP) methods. This book bridges the gap between power electronics and DSP. The following realizations of digital control circuits are considered: digital signal processors, microprocessors, microcontrollers, programmable digital circuits. Discussed in this book is signal processing, starting from analog signal acquisition, through its conversion to digital form, methods of its filtration and separation, and ending with pulse control of output power transistors. The book is focused on two applications for the considered methods of digital signal processing: an active power filter and a digital class D power amplifier. The major benefit to readers is the acquisition of specific knowledge concerning discussions on the processing of signals from voltage or current sensors using a digital signal processor and to the signals controlling the output inverter transistors. Included are some Matlab examples for illustration of the considered problems.

Photovoltaics, the direct conversion of light from the sun into electricity, is an increasingly important means of distributed power

generation. The SPICE modelling tool is typically used in the development of electrical and electronic circuits. When applied to the modelling of PV systems it provides a means of understanding and evaluating the performance of solar cells and systems. The majority of books currently on the market are based around discussion of the solar cell as semiconductor devices rather than as a system to be modelled and applied to real-world problems. Castaner and Silvestre provide a comprehensive treatment of PV system technology analysis. Using SPICE, the tool of choice for circuits and electronics designers, this book highlights the increasing importance of modelling techniques in the quantitative analysis of PV systems. This unique treatment presents both students and professional engineers, with the means to understand, evaluate and develop their own PV modules and systems. * Provides a unique, self-contained, guide to the modelling and design of PV systems * Presents a practical, application oriented approach to PV technology, something that is missing from the current literature * Uses the widely known SPICE circuit-modelling tool to analyse and simulate the performance of PV modules for the first time * Written by respected and well-known academics in the

field
Provides comprehensive coverage of the basic principles and methods of electric power conversion and the latest developments in the field This book constitutes a comprehensive overview of the modern power electronics. Various semiconductor power switches are described, complementary components and systems are presented, and power electronic converters that process power for a variety of applications are explained in detail. This third edition updates all chapters, including new concepts in modern power electronics. New to this edition is extended coverage of matrix converters, multilevel inverters, and applications of the Z-source in cascaded power converters. The book is accompanied by a website hosting an instructor's manual, a PowerPoint presentation, and a set of PSpice files for simulation of a variety of power electronic converters. Introduction to Modern Power Electronics, Third Edition: Discusses power conversion types: ac-to-dc, ac-to-ac, dc-to-dc, and dc-to-ac Reviews advanced control methods used in today's power electronic converters Includes an extensive body of examples, exercises, computer assignments, and simulations Introduction to Modern Power

Electronics, Third Edition is written for undergraduate and graduate engineering students interested in modern power electronics and renewable energy systems. The book can also serve as a reference tool for practicing electrical and industrial engineers.
CD-ROM contains SPICE3 and ISPICE simulation models and examples from the book, allowing easy customization
Power Electronics and Motor Drives
Multiphysics Simulation by Design for Electrical Machines, Power Electronics and Drives
Power Electronics: Computer Simulation and Analysis
An Integrated Approach, Second Edition
Power Electronics Handbook
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.

Power Electronics and Motor Drives: Advances and Trends, Second Edition is the perfect resource to keep the electrical engineer up-to-speed on the latest advancements in technologies, equipment and applications. Carefully structured to include both traditional topics for entry-level and more advanced applications for the experienced engineer, this reference sheds light on the rapidly growing field of power electronic operations. New content covers converters, machine models and new control methods such as *fuzzy logic and neural network control. This reference will help engineers further understand recent technologies and gain practical understanding with its inclusion of many industrial applications. Further supported by a glossary per chapter, this book gives engineers and researchers a critical reference to learn from real-world examples and make future decisions on power electronic technology and applications. Provides many practical examples of industrial applications Updates on the newest electronic topics with content added on fuzzy logic and neural networks Presents information from an expert with decades of research and industrial experience*

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 Editionillustrates 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.*

This course provides a well-organized, step-by-step demonstration of how SPICE/PSpice can be used in the simulation and verification of power electronics converter performance. Students will learn how to obtain device I-v characteristics, time-to main transient and steady-state waveforms, frequency domain fourier data and important *performance indices such as average values, forms values, ripple factor, power factor and THD. The course is useful for engineers, engineering managers, and technicians who are interested in the applications of SPICE simulation for analysis and design of power electronics circuits and systems. A B.S. in Engineering, Engineering Technology or equivalent experience is recommended.*

Software Tools for the Simulation of Electrical Systems

An Introductory Guide

Circuit Analysis and Design

Theory and Practice

Power electronics

This book shows how to use PSpice to quickly analyze common industrial power electronic and power circuits. It would be most useful to an electrical engineer. The book begins with a brief review of PSpice with DC, AC, and transient analyses of simple circuits. It follows with examples that solve typical industrial circuit problems. One of the examples predicts the waveform of the electrical noise that would be transmitted through an inductor. In that example, PSpice would help the engineer properly size a filtering inductor. This can be important if the inductor is large or a custom item. Other examples *find steady state and transient solutions for unbalanced three phase faults. PSpice's Probe program is used to make realistic output traces of transient analysis voltages, currents, and powers. All of the books examples are done with the free (Demo) Release 16.0 version of PSpice. Sources for obtaining free (Demo) copies of PSpice and other Spice programs are provided.*

*Power electronics, which is a rapidly growing area in terms of research and applications, uses modern electronics technology to convert electric power from one form to another, such as ac-dc, dc-dc, dc-ac, and ac-ac with a variable output magnitude and frequency. Power electronics has many applications in our every day life such as air-conditioners, electric cars, sub-way trains, motor drives, renewable energy sources and power supplies for computers. This book covers all aspects of switching devices, converter circuit topologies, control techniques, analytical methods and some examples of their applications. * 25% new content * Reorganized and revised into 8 sections comprising 43 chapters * Coverage of numerous applications, including uninterruptable power supplies and automotive electrical systems * New content in power generation and distribution, including solar power, fuel cells, wind turbines, and flexible transmission*

Presents applied theory and advanced simulation techniques for electric machines and drives This book combines the knowledge of experts from both academia and the software industry to present theories of multiphysics simulation by design for electrical machines, *power electronics, and drives. The comprehensive design approach described within supports new applications required by technologies sustaining high drive efficiency. The highlighted framework considers the electric machine at the heart of the entire electric drive. The book also emphasizes the simulation by design concept—a concept that frames the entire highlighted design methodology, which is described and illustrated by various advanced simulation technologies. Multiphysics Simulation by Design for Electrical Machines, Power Electronics and Drives begins with the basics of electrical machine design and manufacturing tolerances. It also discusses fundamental aspects of the state of the art design process and includes examples from industrial practice. It explains FEM-based analysis techniques for electrical machine design—providing details on how it can be employed in ANSYS Maxwell software. In addition, the book covers advanced magnetic material modeling capabilities employed in numerical computation; thermal analysis; automated optimization for electric machines; and power electronics and drive systems. This valuable resource: Delivers the multi-physics know-how based on practical electric machine design methodologies Provides an extensive overview of electric machine design optimization and its integration with power electronics and drives Incorporates case studies from industrial practice and research and development projects Multiphysics Simulation by Design for Electrical Machines, Power Electronics and Drives is an incredibly helpful book for design engineers, application and system* *engineers, and technical professionals. It will also benefit graduate engineering students with a strong interest in electric machines and drives.*

Building on solid state device and electromagnetic contributions to the series, this text book introduces modern power electronics, that is the application of semiconductor devices to the control and conversion of electrical power. The increased availability of solid state power switches has created a very rapid expansion in applications, from the relatively low power control of domestic equipment, to high power control of industrial processes and very high power control along transmission lines. This text provides a comprehensive introduction to the entire range of devices and examines their applications, assuming only the minimum mathematical and electronic background. It covers a full year's course in power electronics. Numerous exercises, worked examples and self assessments are included to facilitate self study and distance learning.

PSpice Power Electronic and Power Circuit Simulation

Devices, Circuits and Applications

Spice Simulations of Power Electronics

Introduction to PSpice Using OrCAD for Circuits and Electronics

Harness Powerful SPICE Simulation and Design Tools to Develop Cutting-Edge Switch-Mode Power Supplies Switch-Mode Power Supplies: SPICE

Simulations and Practical Designs is a comprehensive resource on using SPICE as a power conversion design companion. This book uniquely bridges analysis and market reality to teach the development and marketing of state-of-the art switching converters. Invaluable to both the graduating student and the experienced design engineer, this guide explains how to derive founding equations of the most popular converters...design safe, reliable converters through numerous practical examples...and utilize SPICE simulations to virtually breadboard a converter on the PC before using the soldering iron. Filled with more than 600 illustrations, Switch-Mode Power Supplies: SPICE Simulations and Practical Designs enables you to: Derive founding equations of popular converters Understand and implement loop control via the book-exclusive small-signal models Design safe, reliable converters through practical examples Use SPICE simulations to virtually breadboard a converter on the PC Access design spreadsheets and simulation templates on the accompanying CD-ROM, with numerous examples running on OrCADË, ICAPSË, µCapË, TINAË, and more Inside This Powerful SPICE Simulation and Design Resource • Introduction to Power Conversion • Small-Signal Modeling • Feedback and Control Loops • Basic Blocks and Generic Models • Simulation and Design of Nonisolated Converters • Simulation and Design of Isolated Converters-Front-End Rectification and Power Factor Correction •

Simulation and Design of Isolated Converters-The Flyback • Simulation and Design of Isolated Converters-The Forward

This updated edition of this book provides comprehensive coverage of modern power electronics, addressing all the latest trends and hot-button issues—from PWM rectifiers to renewable energy systems to electromagnetic interference. It features an overview of advanced control methods used in today's power electronic converters, numerous SPICE files of typical power conversion circuits, and an Instructor's Manual with solutions to all problems. An extensive body of examples, exercises, computer assignments, and simulations make it highly suitable as a textbook for undergraduate/graduate students of engineering in electrical engineering, industrial engineering or renewable energy, and practicing engineers.

PSpice Simulation of Power Electronics CircuitsAn Introductory GuideSpringer

The Essence of Power Electronics

Spice for Power Electronics and Electric Power

Modern Electrical Drives

Advances in Energy Technology

Selected Readings