Le Simulateur Ltspice Iv

Loop control is an essential area of electronics engineering that today's professionals need to master. Rather than delving into extensive theory, this practical book focuses on what you really need to know for compensating or stabilizing a given control system. You can turn instantly to practical sections with numerous design examples and ready-made formulas to help you with your projects in the field. You also find coverage of the underpinnings and principles of control loops so you can gain a more complete understanding of the material. This authoritative volume explains how to conduct analysis of control systems and provides extensive details on practical compensators. It helps you measure your system, showing how to verify if a prototype is stable and features enough design margin. Moreover, you learn how to secure high-volume production by bench-verified safety margins.

An up-to-date, practical guide on upgrading from silicon to GaN, and how to use GaN transistors in power conversion systems design This updated, third edition of a popular book on GaN transistors for efficient power conversion has been substantially expanded to keep students and practicing power conversion engineers ahead of the learning curve in GaN technology advancements. Acknowledging that GaN transistors are not one-to-one replacements for the current MOSFET technology, this book serves as a practical guide for understanding basic GaN transistor construction, characteristics, and applications. Included are discussions on the fundamental physics of these power semiconductors, layout, and other circuit design considerations, as well as specific application examples demonstrating design techniques when employing GaN devices. GaN Transistors for Efficient Power Conversion, 3rd Edition brings key updates to the chapters of Driving GaN Transistors; Modeling, Simulation, and Measurement of GaN Transistors; DC-DC Power Conversion; Envelope Tracking; and Highly Resonant Wireless Energy Transfer. It also offers new chapters on Thermal Management, Multilevel Converters, and Lidar, and revises many others throughout. Written by leaders in the power semiconductor field and industry pioneers in GaN power transistor technology and applications Updated with 35% new material, including three new chapters on Thermal Management, Multilevel Converters, Wireless Power, and Lidar Features practical guidance on formulating specific circuit designs when constructing power conversion systems using GaN transistors A valuable resource for professional engineers, systems designers, and electrical engineering students who need to fully understand the state-of-the-art GaN Transistors for Efficient Power Conversion, 3rd Edition is an essential learning tool and reference guide that enables power conversion engineers to design energy-efficient, smaller, and more cost-effective products using GaN transistors.

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.

The book includes the best extended papers which were selected from the 3rd International Conference of Electrical and Information Technologies (ICEIT 2017, Morocco). The book spans two inter-related research domains which shaped modern societies, solved many of their development problems, and contributed to their unprecedented economic growth and social welfare. Selected papers are based on original and high quality research. They were peer reviewed by experts in the field. They are grouped into five parts. Part I deals with Power System and Electronics topics that include Power Electronics & Energy Conversion, Actuators & Micro/Nanotechnology, etc. Part II relates to Control Systems and their applications. Part III concerns the topic of Information Technology that basically includes Smart Grid, Information Security, Cloud Computing Distributed, Big Data, etc. Part IV discusses Telecommunications and Vehicular Technologies topics that include, Green Networking and Communications, Wireless Ad-hoc and Sensor Networks, etc. Part V covers Green Applications and Interdisciplinary topics, that include intelligent and Green Technologies for Transportation Systems, Smart Cities, etc. This book offers a good opportunity for young researchers, novice scholars and whole academic sphere to explore new trends in Electrical and information Technologies.

Praise for CMOS: Circuit Design, Layout, and SimulationRevised Second Edition from the Technical Reviewers "A refreshing industrial flavor. Design concepts are presented as they are needed for 'just-in-time' learning. Simulating and designing circuits using SPICE is emphasized with literally hundreds of examples. Very few textbooks contain as much detail as this one. Highly recommended!" --Paul M. Furth, New Mexico State University "This book builds a solid knowledge of CMOS circuit design from the ground up. With coverage of process integration, layout, analog and digital models, noise mechanisms, memory circuits, references, amplifiers, PLLs/DLLs, dynamic circuits, and data converters, the text is an excellent reference for both experienced and novice designers alike." --Tyler J. Gomm, Design Engineer, Micron Technology, Inc. "The Second Edition builds upon the success of the first with new chapters that cover additional material such as oversampled converters and non-volatile memories. This is becoming the de facto standard textbook to have on every analog and mixed-signal designer's bookshelf." --Joe Walsh, Design Engineer, AMI Semiconductor CMOS circuits from design to implementation CMOS: Circuit Design, Layout, and Simulation, Revised Second Edition covers the practical design of both analog and digital integrated circuits, offering a vital, contemporary view of a wide range of analog/digital circuit blocks, the BSIM model, data converter architectures, and much more. This edition takes a two-path approach to the topics: design techniques are developed for both long- and short-channel CMOS technologies and then compared. The results are multidimensional explanations that allow readers to gain deep insight into the design process. Features include: Updated materials to reflect CMOS technology's movement into nanometer sizes Discussions on phase- and delay-locked loops, mixed-signal circuits, data converters, and circuit noise More than 1,000 figures, 200 examples, and over 500 end-of-chapter problems In-depth coverage of both analog and digital circuit-level design techniques Real-world process parameters and design rules The book's Web site, CMOSedu.com, provides: solutions to the book's problems; additional homework problems without solutions; SPICE simulation examples using HSPICE, LTspice, and WinSpice; layout tools and examples for actually fabricating a chip; and videos to aid learning

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

Cet ouvrage est conçu pour ceux qui souhaitent se perfectionner dans la connaissance de LTspice, découvrir les nouvelles commandes apparues récemment et tirer le meilleur parti des évolutions apportées aux commandes existantes. Il s’adresse aux utilisateurs de LTspice, aux designers, ingénieurs ou techniciens, ainsi qu’aux élèves ingénieurs et étudiants en électronique. Il complète un premier volume du même auteur paru en 2011 sous le titre Le simulateur LTspice IV. Avec, 3,6 millions d’utilisateurs dans le monde, LTspice, est aujourd'hui le simulateur professionnel le plus utilisé. Points forts Les commandes cachées, améliorées ou nouvelles. Les nouvelles astuces et les méthodes statistiques. Une lecture facilitée, illustrée de 540 figures et 40 tableaux synthétiques. Des réponses détaillées aux questions recueillies au cours des sessions de formation LTspice. Un index exhaustif de 1 500 entrées. Sur www.dunod.com/contenus-complementaires/9782100743193 et sur le site de l'auteur www.LTspice.fr de nombreux compléments dont l'ensemble des schémas et des illustrations du livre.

Certains convertisseurs électroniques de puissance sont spécifiquement conçus pour alimenter des équipements sous une tension continue lissée. Par conséquent, l'aspect filtrage implique nécessairement l'usage de composants passifs auxiliaires (inductances et condensateurs). Cet ouvrage traite des aspects techniques tels que la séparation classique entre alimentations isolées et non isolées, et la commutation douce au travers d'un convertisseur particulier. Il répond au problème de la régulation de la tension de sortie des alimentations à découpage sous l'angle de la modélisation et de l'obtention de fonctions de transfert des alimentations à découpage. Electronique de puissance pour l'industrie et les transports 3 propose une étude de cas d'une alimentation isolée Flyback dont la conception complète est présentée : les composants actifs et passifs sont dimensionnés sur la base du cahier des charges fixé initialement. Une attention particulière est portée aux condensateurs de sortie du convertisseur et à l'ensemble des organes environnants.

Compact Models for Integrated Circuit Design: Conventional Transistors and Beyond provides a modern treatise on compact models for circuit computer-aided design (CAD). Written by an author with more than 25 years of industry experience in semiconductor processes, devices, and circuit CAD, and more than 10 years of academic experience in teaching compact modeling courses, this first-of-its-kind book on compact SPICE models for very-large-scale-integrated (VLSI) chip design offers a balanced presentation of compact modeling crucial for addressing current modeling challenges and understanding new models for emerging devices. Starting from basic semiconductor physics and covering state-of-the-art device regimes from conventional micron to nanometer, this text: Presents industry standard models for bipolar-junction transistors (BJTs), metal-oxide-semiconductor (MOS) field-effect-transistors (FETs), FinFETs, and tunnel field-effect transistors (TFETs), along with statistical MOS models Discusses the major issue of process variability, which severely impacts device and circuit performance in advanced technologies and requires statistical compact models Promotes further research of the evolution and development of compact models for VLSI circuit design and analysis Supplies fundamental and practical knowledge necessary for efficient integrated circuit (IC) design using nanoscale devices Includes exercise problems at the end of each chapter and extensive references at the end of the book Compact Models for Integrated Circuit Design: Conventional Transistors and Beyond is intended for senior undergraduate and graduate courses in electrical and electronics engineering as well as for researchers and practitioners working in the area of electron devices. However, even those unfamiliar with semiconductor physics gain a solid grasp of compact modeling concepts from this book. The Open Access version of this book, available at https://doi.org/10.1201/b19117, has been made available under a Creative Commons Attribution-Non Commercial-No Derivatives 4.0 license.

Easily design today's wireless systems and circuits Design an entire radio system from the ground up instead of relying on a simple plug-in selection of circuits to be modified. Avoid an arduous trek through theory and mathematical derivations. Cotter Sayre's Complete Wireless Design covers wireless hardware design more thoroughly than any other handbook —and does it without burying you in math. This new guide from today's bestselling wireless author gives you all the skills you need to design wireless systems and circuits. If you want to climb the learning curve with grace, and start designing what you need immediately, this reasonably priced resource is your best choice. It's certain to be the most-used reference in your wireless arsenal for designing cutting-edge filters, amplifiers, RF switches, oscillators, and more. You get: Simplified calculations for impedance matching, analysis of wireless links, and completing a frequency plan Real-world examples of designing with RFIC's and MMIC's Full circuit and electromagnetic software simulations More

This volume concentrates on three topics: mixed analog--digital circuit design, sensor interface circuits and communication circuits. The book comprises six papers on each topic of a tutorial nature aimed at improving the design of analog circuits. The book is divided into three parts. Part I: Mixed Analog--Digital Circuit Design considers the largest growth area in microelectronics. Both standard designs and ASICs have begun integrating analog cells and digital sections on the same chip. The papers cover topics such as groundbounce and supply-line spikes, design methodologies for high-level design and actual mixed analog--digital designs. Part II: Sensor Interface Circuits describes various types of signal conditioning circuits and interfaces for sensors. These include interface solutions for capacitive sensors, sigma--delta modulation used to combine a microprocessor compatible interface with on chip CMOS sensors, injectable sensors and responders, signal conditioning circuits and sensors combined with indirect converters. Part III: Communication Circuits concentrates on systems and implemented circuits for use in personal communication systems. These have applications in cordless telephones and mobile telephone systems for use in cellular networks. A major requirement for these systems is low power consumption, especially when operating in standby mode, so as to maximise the time between battery recharges.

Provides an introduction to modern object-oriented design principles and applications for the fast-growing area of modeling and simulation Covers the topic of multi-domain system modeling and design with applications that have components from several areas Serves as a reference for the Modelica language as well as a comprehensive overview of application model libraries for a number of application domains

New advanced modeling methods for simulating the electromagnetic properties of complex three-dimensional electronic systems Based on the author's extensive research, this book sets forth tested and proven electromagnetic modeling and simulation methods for analyzing signal and power integrity as well as electromagnetic interference in large complex electronic interconnects, multilayered package structures, integrated circuits, and printed circuit boards. Readers will discover the state of the technology in electronic package integration and printed circuit board simulation and modeling. In addition to popular full-wave electromagnetic computational methods, the book presents new, more sophisticated modeling methods, offering readers the most advanced tools for analyzing and designing large complex electronic structures. Electrical Modeling and Design for 3D System Integration begins with a comprehensive review of current modeling and simulation methods for signal integrity, power integrity, and electromagnetic compatibility. Next, the book guides readers through: The macromodeling technique used in the electrical and electromagnetic modeling and simulation of complex interconnects in three-dimensional integrated systems The semi-analytical scattering matrix method based on the N-body scattering theory for modeling of three-dimensional electronic package and multilayered printed circuit boards with multiple vias Two- and three-dimensional integral equation methods for the analysis of power distribution networks in three-dimensional package integrations The physics-based algorithm for extracting the equivalent circuit of a complex power distribution network in three-dimensional integrated systems and printed circuit boards An equivalent circuit model of through-silicon vias Metal-oxide-semiconductor capacitance effects of through-silicon vias Engineers, researchers, and students can turn to this book for the latest techniques and methods for the electrical modeling and design of electronic packaging, three-dimensional electronic integration, integrated circuits, and printed circuit boards.

The use of MATLAB is ubiquitous in the scientific and engineering communities today, and justifiably so. Simple programming, rich graphic facilities, built-in functions, and extensive toolboxes offer users the power and flexibility they need to solve the complex analytical problems inherent in modern technologies. The ability to use MATLAB effectively has become practically a prerequisite to success for engineering professionals. Like its best-selling predecessor, Electronics and Circuit Analysis Using MATLAB, Second Edition helps build that proficiency. It provides an easy, practical introduction to MATLAB and clearly demonstrates its use in solving a wide range of electronics and circuit analysis problems. This edition reflects recent MATLAB enhancements, includes new material, and provides even more examples and exercises. New in the Second Edition: Thorough revisions to the first three chapters that incorporate additional MATLAB functions and bring the material up to date with recent changes to MATLAB A new chapter on electronic data analysis Many more exercises and solved examples New sections added to the chapters on two-port networks, Fourier analysis, and semiconductor physics MATLAB m-files available for download Whether you are a student or professional engineer or technician, Electronics and Circuit Analysis Using MATLAB, Second Edition will serve you well. It offers not only an outstanding introduction to MATLAB, but also forms a guide to using MATLAB for your specific purposes: to explore the characteristics of semiconductor devices and to design and analyze electrical and electronic circuits and systems.

Circuit simulation is essential in integrated circuit design, and the accuracy of circuit simulation depends on the accuracy of the transistor model. BSIM3v3 (BSIM for Berkeley Short-channel IGFET Model) has been selected as the first MOSFET model for standardization by the Compact Model Council, a consortium of leading companies in semiconductor and design tools. In the next few years, many fabless and integrated semiconductor companies are expected to switch from dozens of other MOSFET models to BSIM3. This will require many device engineers and most circuit designers to learn the basics of BSIM3. MOSFET Modeling & BSIM3 User's Guide explains the detailed physical effects that are important in modeling MOSFETs, and presents the derivations of compact model expressions so that users can understand the physical meaning of the model equations and parameters. It is the first book devoted to BSIM3. It treats the BSIM3 model in detail as used in digital, analog and RF circuit design. It covers the complete set of models, i.e., I-V model, capacitance model, noise model, parasitics model, substrate current model, temperature effect model and non quasi-static model. MOSFET Modeling & BSIM3 User's Guide not only addresses the device modeling issues but also provides a user's guide to the device or circuit design engineers who use the BSIM3 model in digital/analog circuit design, RF modeling, statistical modeling, and technology prediction. This book is written for circuit designers and device engineers, as well as device scientists worldwide. It is also suitable as a reference for graduate courses and courses in circuit design or device modelling. Furthermore, it can be used as a textbook for industry courses devoted to BSIM3. MOSFET Modeling & BSIM3 User's Guide is comprehensive and practical. It is balanced between the background information and advanced discussion of BSIM3. It is helpful to experts and students alike.

Since the first edition of this book was published seven years ago, the field of modeling and simulation of communication systems has grown and matured in many ways, and the use of simulation as a day-to-day tool is now even more common practice. With the current interest in digital mobile communications, a primary area of application of modeling and simulation is now in wireless systems of a different flavor from the `traditional' ones. This second edition represents a substantial revision of the first, partly to accommodate the new applications that have arisen. New chapters include material on modeling and simulation of nonlinear systems, with a complementary section on related measurement techniques, channel modeling and three new case studies; a consolidated set of problems is provided at the end of the book.

Electronic Workbench (EWB) software has forever changed the face of electronics. Including mixed-mode circuit simulation, schematic capture and PCB layout software, it provides a virtual bench for learning, experimenting with, and simulating electronics, including mixed-mode circuit simulation, schematic capture and PCB layout software. Mastering Electronics Workbench, by John Adams, is your guide to successfully using Electronics Workbench. You get detailed explanations of each component, instrument, and function. You learn how to install the program, how to use it to create circuit simulations and analysis models, and how to make complex designs. This guide is also packed with complete projects for hobbyists, technicians and engineers, each designed to help you learn the complexities of the program. The book covers menu options; creating a circuit - the drag and drop interface; the 2 minute circuit - making a simple circuit; advanced circuit simulations; practical uses For EWB; EWB layout software; and much more.

This new book, written by Andre Vladimirescu, who was instrumental in the development of SPICE at the University of California Berkeley, introduces computer simulation of electrical and electronics circuits based on the SPICE standard. Relying on the functionality first supported in SPICE2 that is now supported in all SPICE programs, this text is addressed to all users of electrical simulation. The approach to learning circuit simulation is to interpret simulation results in relation to electrical engineering fundamentals; the book asks the student to solve most circuit examples by hand before verifying the results with SPICE. Addressed to both the SPICE novice and the experienced user, the first six chapters provide the relevant information on SPICE functionality for the analysis of linear as well as nonlinear circuits. Each of these chapters starts out with a linear example accessible to any new user of SPICE and proceeds with nonlinear transistor circuits. The latter part of the book goes into more detail on such issues as functional and hierarchical models, distortion analysis, basic algorithms in SPICE and related options parameters, and, how to direct SPICE to find a solution when it does not converge to a solution. The approach emphasizes that SPICE is not a substitute for knowledge of circuit operation but a complement. The SPICE Book is different from previously published books in the approach of solving circuit problems with a computer. The solution to most circuit examples is sketched out by hand first and followed by a SPICE verification. For more complex circuits it is not feasible to find the solution by hand but the approach stresses the need for the SPICE user tounderstand the results. Readers gain a better comprehension of SPICE thanks to the importance placed on the relation between EE fundamentals and computer simulation. The tutorial approach advances from the hand solution of a circuit to SPICE verification and simulation results interpretation. This book teaches the approach to electrical circuit simulation rather than a specific simulation program. Examples are simulated alternatively with SPICE2, SPICE3 or PSPICE. Accurate descriptions, simulation rationale and cogent explanations make this an invaluable reference.

Operational amplifiers have a very broad range of application. This book focuses on the fundamentals which are applicable to many applications. All of the simulations and experiments demonstrate basic operational amplifier principles. The experiments may be easily modified and may serve as the basis for other applications. This book may be used as a circuit design and application reference for hobbyists, experimenters, and students. It may also be used as a supplement to a college level operational amplifier course and laboratory. An understanding of electric circuit analysis, semiconductor devices, and college level algebra are pre-requisites for this book. Simulation examples are presented using LTspice, a simulation program available as a free download from Linear Technology. TINA-TI, a simulation program available as a free download from Texas Instruments, is also introduced. Experiments provided may be performed using a solder-less breadboard, inexpensive parts, a small power supply, and a digital or USB oscilloscope. Some experiments also require a function generator. The circuits are provided in their basic and simplest forms. The experimenter may modify and augment the circuits as needed for particular applications.

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

This book presents new concepts for a next generation of PV. Among these concepts are: Multijunction solar cells, multiple excitation solar cells (or how to take benefit of high energy photons for the creation of more than one electron hole-pair), intermediate band solar cells (or how to take advantage of below band-gap energy photons) and related technologies (for quantum dots, nitrides, thin films), advanced light management approaches (plasmonics). Written by world-class experts in next generation photovoltaics this book is an essential reference guide accessible to both beginners and experts working with solar cell technology. The book deeply analyzes the current state-of-the-art of the new photovoltaic approaches and outlines the implementation paths of these advanced devices. Topics addressed range from the fundamentals to the description of state-of-the-art of the new types of solar cells.

A step-by-step guide to the design and analysis of CMOS operational amplifiers and comparators This volume is a comprehensive text that offers a detailed treatment of the analysis and design principles of two of the most important components of analog metal oxide semiconductor (MOS) circuits, namely operational amplifiers (op-amps) and comparators. The book covers the physical operation of these components, their design procedures, and applications to analog MOS circuits-particularly those involving switched-capacitor circuits, and analog-to-digital (A/D) and digital-to-analog (D/A) converters. Roubik Gregorian, a leading authority in the field, gives circuit designers the technical knowledge they need to design high-performance op-amps and comparators suitable for most analog circuit applications. In this self-contained treatment, which is loosely based on his well-received 1986 book, Analog MOS Integrated Circuits for Signal Processing (coauthored with Gabor C. Temes), Gregorian reviews the required basics before advancing to state-of-the-art topics and problem-solving techniques. This valuable guide: * Clearly explains configuration and performance limitation issues affecting the operation of CMOS op-amps and comparators * Details advanced design procedures to improve performance * Provides practical design examples suitable for a broad range of analog circuit applications * Incorporates hundreds of illustrations into the text * Concludes each chapter with problems and references to advanced topics, useful in textbook adoptions Introduction to CMOS Op-Amps and Comparators is invaluable for analog and mixed-signal designers, for senior and graduate students in electrical engineering, and for anyone who would like to keep up with this essential technology.

This book shows readers how to learn analog electronics by simulating circuits. Readers will be enabled to master basic electric circuit analysis, as an essential component of their professional education. The author’s approach enables readers to learn theory as needed, then immediately apply it to the simulation of circuits based on that theory, while using the resulting tables, graphs and waveforms to gain a deeper insight into the theory, as well as where theory and practice diverge!

Discover a fresh approach to efficient and insight-driven analog integrated circuit design in nanoscale-CMOS with this hands-on guide. Expert authors present a sizing methodology that employs SPICE-generated lookup tables, enabling close agreement between hand analysis and simulation. This enables the exploration of analog circuit tradeoffs using the gm/ID ratio as a central variable in script-based design flows, and eliminates time-consuming iterations in a circuit simulator. Supported by downloadable MATLAB code, and including over forty detailed worked examples, this book will provide professional analog circuit designers, researchers, and graduate students with the theoretical know-how and practical tools needed to acquire a systematic and re-use oriented design style for analog integrated circuits in modern CMOS.

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

Analog circuit and system design today is more essential than ever before. With the growth of digital systems, wireless communications, complex industrial and automotive systems, designers are challenged to develop sophisticated analog solutions. This comprehensive source book of circuit design solutions will aid systems designers with elegant and practical design techniques that focus on common circuit design challenges. The book’s in-depth application examples provide insight into circuit design and application solutions that you can apply in today’s demanding designs. Covers the fundamentals of linear/analog circuit and system design to guide engineers with their design challenges Based on the Application Notes of Linear Technology, the foremost designer of high performance analog products, readers will gain practical insights into design techniques and practice Broad range of topics, including power management tutorials, switching regulator design, linear regulator design, data conversion, signal conditioning, and high frequency/RF design Contributors include the leading lights in analog design, Robert Dobkin, Jim Williams and Carl Nelson, among others

LTspice est un logiciel de simulation electronique qui permet d'anticiper les caracteristiques et les performances d'un circuit electronique en assemblant a l'ecran des composants virtuels. A partir du noyau spice developpe a l'universite Berkeley mais tres peu convivial, la societe Linar Technology (LT) a developpe une version plus visuelle, plus facile d'emploi, et gratuite.Cet ouvrage est a la fois un manuel utilisateur qui va de la prise en main a une utilisation tres poussee de LTspice IV, et un recueil d'exemples et de procedures avec plus de470 illustrations. Toutes les commandes et les definitions sont expliquees et classees par theme.Cette deuxieme edition integre les dernieres generations de circuits integres produits par Linear Technology."

Le simulateur LTspice IVmanuel, méthodes et applications

This book is concerned with circuit simulation using National Instruments Multisim. It focuses on the use and comprehension of the working techniques for electrical and electronic circuit simulation. The first chapters are devoted to basic circuit analysis. It starts by describing in detail how to perform a DC analysis using only resistors and independent and controlled sources. Then, it introduces capacitors and inductors to make a transient analysis. In the case of transient analysis, it is possible to have an initial condition either in the capacitor voltage or in the inductor current, or both. Fourier analysis is discussed in the context of transient analysis. Next, we make a treatment of AC analysis to simulate the frequency response of a circuit. Then, we introduce diodes, transistors, and circuits composed by them and perform DC, transient, and AC analyses. The book ends with simulation of digital circuits. A practical approach is followed through the chapters, using step-by-step examples to introduce new Multisim circuit elements, tools, analyses, and virtual instruments for measurement. The examples are clearly commented and illustrated. The different tools available on Multisim are used when appropriate so readers learn which analyses are available to them. This is part of the learning outcomes that should result after each set of end-of-chapter exercises is worked out. Table of Contents: Introduction to Circuit Simulation / Resistive Circuits / Time Domain Analysis -- Transient Analysis / Frequency Domain Analysis -- AC Analysis / Semiconductor Devices / Digital Circuits

This introduction to circuit design is unusual in several respects. First, it offers not just explanations, but a full course. Each of the twenty-five sessions begins with a discussion of a particular sort of circuit followed by the chance to try it out and see how it actually behaves. Accordingly, students understand the circuit's operation in a way that is deeper and much more satisfying than the manipulation of formulas. Second, it describes circuits that more traditional engineering introductions would postpone: on the third day, we build a radio receiver; on the fifth day, we build an operational amplifier from an array of transistors. The digital half of the course centers on applying microcontrollers, but gives exposure to Verilog, a powerful Hardware Description Language. Third, it proceeds at a rapid pace but requires no prior knowledge of electronics. Students gain intuitive understanding through immersion in good circuit design.

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

Whetted to the design needs of engineers of the '90s, this reworking of the classic industry text offers a practical, concrete look at designing low-noise electronic systems with the technological tools of the future. Published originally in 1973 as Low-Noise Electronic Design, the first edition was a practical primer for circuit design and system engineers on designing low-level electronic circuits as well as analyzing low-level sensing and measurement systems. Now newly revised as Low-Noise Electronic System Design, this new edition unfolds the technological hardware speeding the electronics industry towards a new century.

This book enables design engineers to be more effective in designing discrete and integrated circuits by helping them understand the role of analog devices in their circuit design. Analog elements are at the heart of many important functions in both discrete and integrated circuits, but from a design perspective the analog components are often the most difficult to understand. Examples include operational amplifiers, D/A and A/D converters and active filters. Effective circuit design requires a strong understanding of the operation of these analog devices and how they affect circuit design. Comprehensive coverage of analog circuit components for the practicing engineer Market-validated design information for all major types of linear circuits Includes practical advice on how to read op amp data sheets and how to choose off-the-shelf op amps Full chapter covering printed circuit board design issues

Microelectronic Circuit Designis known for being a technically excellent text. The new edition has been revised to make the material more motivating and accessible to students while retaining a student-friendly approach.Jaeger has added more pedagogy and an emphaisis on design through the use of design examples and design notes. Some pedagogical elements include chapter opening vignettes, chapter objectives, "Electronics in Action" boxes, a problem solving methodology, and "design note" boxes. The number of examples, including new design examples, has been increased, giving students more opportunity to see problems worked out. Additionally,some of the less fundamental mathematical material has been moved to the ARIS website. In addition this edition comes with aHomework Management System called ARIS, which includes 450 static problems.

Ho il piacere di pubblicare questo nuovo libro dedicato ai principianti che vogliono incominciare ad imparare le basi dell'elettronica. Questo libro riprende la corrispettiva serie di articoli pubblicata sul mio blog, www.mcmajan.com, con le dovute integrazioni, correzioni ed espansioni al fine di poter avere tutto il contenuto in un unico testo facilmente fruibile anche offline. Come molti sapranno, nel blog mi occupo in gran parte di Arduino con il quale ho affrontato tutta una serie di argomenti, ma ci sono alcune nozioni che fanno parte dell'elettronica generale e che è utile imparare prima di cominciare a fare i primi esperimenti. E' difficile comprendere certi progetti se non si è in grado di dimensionare un partitore di tensione o non si sa a cosa serva un diodo. Sono nozioni di base che cercherò di spiegare nel modo più semplice possibile a tutte quelle persone che non sanno assolutamente nulla di elettronica ma vogliono avvicinarsi a questo mondo affascinante. Dopo una parte molto semplice ci saranno argomenti un po' più impegnativi come i transistor. Essendo un libro digitale ho la piccola grande opportunità di espanderlo e correggerlo nel tempo per cui è da considerarsi un lavoro attualmente non ancora concluso. Nella prima versione non erano ad esempio presenti gli operazionali, ne il terzo capitolo su LTSpice. Nella seconda edizione ho aggiunto due capitoli sugli operazionali.