Analog Design And Simulation Using Orcad Capture And Pspice

This book provides instruction on

how to use the OrCAD design suite to design and manufacture printed circuit boards. The primary goal is to show the reader how to design a PCB using OrCAD Capture and OrCAD Editor.

Capture is used to build the schematic diagram of the circuit, and Editor is used to design the circuit board so that it can be manufactured. The book is written for both students and practicing engineers who need in-depth instruction on how to use the software, and who need background knowledge of the PCB design process. Beginning to end coverage of the printed circuit board design process. Information is presented in the exact order a circuit and PCB are designed Over 400 full color illustrations, including extensive use of screen shots from the software, allow readers to learn features of the product in the most realistic manner possible Straightforward, realistic examples present the how and why the designs work, providing a comprehensive toolset for understanding the OrCAD software Introduces and follows IEEE, IPC, and JEDEC industry standards for PCB design. Unique chapter on Design for Manufacture covers padstack and footprint design, and component placement, for the design of manufacturable PCB's FREE CD containing the OrCAD demo version and design files 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 Praise for CMOS: Circuit

Design, Layout, and SimulationR evised 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 mul tidimensional 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
The purpose of this book is to

provide a complete working knowledge of the Complementary Metal-Oxide Semiconductor (CMOS) analog and mixed-signal circuit design, which can be applied for System on Chip (SOC) or A pplication-Specific Standard Product (ASSP) development. It begins with an introduction to the CMOS analog and mixed-signal circuit design with further coverage of basic devices, such as the Metal-Oxide Semiconductor Field-Effect Transistor (MOSFET) with both long- and short-channel operations, photo devices, fitting ratio, etc. Seven chapters focus on the CMOS analog and mixed-signal circuit design of amplifiers, low power amplifiers, voltage regulator-reference, data converters, dynamic analog circuits, color and image sensors, and peripheral (oscillators and Input/Output [I/O]) circuits, and Integrated Circuit (IC) layout and packaging. Features: Provides practical knowledge of CMOS analog and mixed-signal circuit design Includes recent research in CMOS color and image sensor

technology
Discusses sub-blocks of typical analog and mixed-signal IC products
Illustrates several design examples of analog circuits together with layout

Describes integrating based CMOS color circuit
Numerical Recipes in Python
Fundamentals of Design and Analysis
Analog Design and Simulation Using OrCAD

Capture and PSpice
Mixed A/D Circuit Design, Sensor Interface Circuits and Communication Circuits Principles, Simulation and Design Theory and

Practice
Intuitive Analog Circuit Design outlines ways of thinking about analog circuits and systems that let you develop a feel for what a good, working analog circuit design should be. This book reflects

author Marc Thompson's 30 years of experience designing analog and power electronics circuits and teaching graduate-level analog circuit design, and is the ideal reference for anyone who needs a straightforward introduction to the subject. In this book, Dr. Thompson describes intuitive and "back-of-the-envelope" techniques for designing and analyzing analog circuits, including transistor amplifiers (CMOS, JFET, and bipolar), transistor switching, noise in analog circuits, thermal circuit design, magnetic circuit design, and control systems. The application of some simple rules of thumb and design techniques is the first step in developing an intuitive understanding of the behavior of complex electrical systems. Introducing analog circuit design with a minimum of mathematics, this book uses numerous real-world examples to help you make the transition to analog design. The second edition is an ideal introductory text for anyone new to the area of analog circuit design. Design examples are used throughout the text, along with end-of-chapter examples Covers real-world parasitic elements in circuit design and their effects Analog Behavioral Modeling With The Verilog-A Language provides the IC designer with an introduction to the methodologies and uses of analog behavioral modeling with the Verilog-A language. In doing so, an overview of Verilog-A language constructs as well as applications using the language are presented. In addition, the book is accompanied by the Verilog-A Explorer IDE (Integrated Development Environment), a limited capability Verilog-A enhanced SPICE simulator for further learning and experimentation with the Verilog-A language. This book assumes a basic level of understanding of the usage of SPICE-based analog simulation and the Verilog HDL language, although any programming language background and a little determination should suffice. From the Foreword: `Verilog-A is a new hardware design language (HDL) for analog circuit and systems design. Since the mid-eighties, Verilog

HDL has been used extensively in the design and verification of digital systems. However, there have been no analogous high-level languages available for analog and mixed-signal circuits and systems. Verilog-A provides a new dimension of design and simulation capability for analog electronic systems. Previously, analog simulation has been based upon the SPICE circuit simulator or some derivative of it.

Digital simulation is primarily performed with a hardware description language such as Verilog, which is popular since it is easy to learn and use. Making Verilog more worthwhile is the fact that several tools exist in the industry that complement and extend Verilog's capabilities ... Behavioral Modeling With the Verilog-A Language provides a good introduction and starting place for students and practicing engineers with interest in understanding this new level of simulation technology. This book contains numerous examples that enhance the text material and provide a helpful learning tool for the reader. The text and the simulation program included can be used for individual study or in a classroom environment ...' Dr. Thomas A. DeMassa, Professor of Engineering,

Arizona State University
Analog Circuit Design is based on the yearly Advances in Analog Circuit Design workshop. The aim of the workshop is to bring together designers of advanced

analogue and RF circuits for the purpose of studying and discussing new possibilities and future developments in this field. Selected topics for AACD 2007 were: (1) Sensors, Actuators and

Power Drivers for the Automotive and Industrial Environment; (2) Integrated PA's from Wireline to RF; (3) Very High Frequency Front Ends.

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
Circuit Design, Layout, and Simulation
CMOS Analog and Mixed-Signal

Circuit Design
Modern Analog Filter Analysis and Design
Analysis and Simulation of Noise in Nonlinear Electronic Circuits and Systems
Analog IC Reliability in Nanometer CMOS
A Framework for

Analog Design Reuse
Analog Integrated Circuits for Communication: Principles, Simulation and Design, Second Edition covers the analysis and design of nonlinear analog integrated circuits that form the basis

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

with basic analysis techniques as well as with the operating principles of semiconductor devices. This book also serves as a useful reference for practicing engineers.
This comprehensive text discusses the fundamentals of

analog electronics applications, design, and analysis. Unlike the physics approach in other analog electronics books, this text focuses on an engineering approach, from the main components of an analog circuit to general analog networks. Concentrating on development of standard formulae for conventional analog systems, the book is filled with practical examples and detailed explanations of procedures to analyze analog circuits. The book covers amplifiers, filters, and op-amps as well as general applications of analog design. This textbook provides a complete introduction to analog filters for senior undergraduate and graduate students.

Coverage includes the synthesis of analog filters and many other filter types including passive filters and filters with distributed elements.

Analog IC design tools have not changed much during the past few decades. While the models and simulation methods have greatly improved in accuracy and performance, analog design still relies on manually constructed schematics and layouts. Each design needs to carry its own test routine, and the quality of the entire system depends on this custom-constructed test routine. It is common for circuits to be tested in an ad-hoc manner through some combination of SPICE decks, MATLAB calculations, and perl scripts. This test collateral is often brittle (i.e., tightly coupled to the process, circuit, and simulation environment) and archived without sufficient documentation. As a result, it is usually easier to recreate the test frame when the circuit is reused in the future. Languages like OCEAN try to address this by providing a common language that both configures the simulation environment and performs calculations. While this reduces the number of files required for a simulation and reduces the need for documentation, it does not solve the problem of tight coupling. Reuse of OCEAN scripts generally means copy-and-paste, and this duplication of code makes the test routines more difficult to debug later. We see an opportunity to improve the productivity of analog design by raising the abstraction level, at least for test construction, used by analog designers. In this work, we present the CircuitBook test framework and repository that complement existing analog design flows. Our test framework is a set of Python libraries that allow high-level specification of analog tests and an associated tool chain that executes these tests. Circuits and tests are defined against an hierarchical tree of interfaces. These common interfaces allow the reuse of tests across different circuits as well as enable faster prototyping of circuits and tests. Tests defined using the CircuitBook test framework separate the simulation directives from the results analysis. This separation of concerns avoids code duplication by separating the reusable parts of the tests from the e nvironment-specific parts. The CircuitBook repository stores circuits, tests, and simulation results to allow designers to leverage existing circuits and tests in new designs. The objects in this repository can browsed via the interface hierarchy or through property tags of the tests. We leverage the high-level nature of the test framework and automatically generate property tags by parsing the test structure. Analog Behavioral

Modeling with the Verilog-A Language
Sensors, Actuators and Power Drivers; Integrated Power Amplifiers from Wireline to RF; Very High Frequency Front Ends
Analog Integrated Circuits for Communication
Analog Filters using

MATLAB
Symbolic Analysis for Automated Design of Analog Integrated Circuits
Design of Analog Circuits Through Symbolic Analysis
Analog Circuit Design – Applicable for bookstore

catalogue
This book focuses on modeling, simulation and analysis of analog circuit aging. First, all important nanometer CMOS physical effects

resulting in circuit unreliability are reviewed. Then, transistor aging compact models for circuit simulation are discussed and several methods for efficient circuit reliability simulation are explained and compared. Ultimately, the impact of transistor aging on analog

circuits is studied. Aging-resilient and aging-immune circuits are identified and the impact of technology scaling is discussed. The models and simulation

techniques described in the book are intended as an aid for device engineers, circuit designers and the EDA community to understand and to mitigate the impact of aging effects on nanometer CMOS ICs. The 2nd Edition of Analog Integrated Circuit Design focuses on more coverage about several types of circuits that have increased in importance in the past decade. Furthermore, the text is enhanced with material on CMOS IC device modeling, updated processing layout and expanded coverage to reflect technical innovations. CMOS devices and circuits have more influence in this edition as well as a reduced amount of text on BiCMOS and bipolar information. New chapters include topics on frequency response of analog ICs and

basic theory of feedback amplifiers.

Art, Science, and Personalities Computer-aided Design of Analog Integrated Circuits and Systems

Complete PCB Design Using OrCAD Capture and PCB Editor
CMOS
Models and CAD Techniques for High-Level Design
VLSI Analog Filters
Analog Circuit Design

contains the contribution of 18 tutorials of the 14th workshop on Advances in Analog Circuit Design. Each part discusses a specific todate topic on new and valuable design ideas in the area of analog circuit design. Each part is presented by six experts in that field and state of the

art information is shared and overviewed. This book is number 14 in this successful series of Analog Circuit Design, providing valuable information and excellent overviews of analog circuit design, CAD and RF systems. Analog Circuit Design is an essential reference source for

analog circuit designers and researchers wishing to keep abreast with the latest development in the field. The tutorial coverage also makes it suitable for use in an advanced design course.

Reliability concerns and the limitations of process technology can sometimes restrict the innovation process involved in designing nano-scale analog circuits. The success of nano-scale analog circuit design requires repeat experimentation, correct analysis of the device physics, process technology, and adequate use of the knowledge database. Starting with the basics, Nano-Scale CMOS Analog Circuits: Models and CAD Techniques for High-Level Design introduces the essential fundamental concepts for designing analog circuits with optimal performances. This book explains the links between the physics and technology of scaled MOS transistors and the design and simulation of nano-scale analog circuits. It also explores the development of structured computer-aided design (CAD) techniques for architecture-level and circuit-level design of analog circuits. The book outlines the general trends of technology scaling with respect to device geometry, process parameters, and supply voltage. It describes models and optimization techniques, as well as the compact modeling of scaled MOS transistors for VLSI circuit simulation. • Includes two learning-based methods: the

artificial neural network (ANN) and the least-squares support vector machine (LS-SVM) method • Provides case studies demonstrating the practical use of these two methods • Explores circuit sizing and specification translation tasks • Introduces the particle

swarm optimization technique and provides examples of sizing analog circuits • Discusses the advanced effects of scaled MOS transistors like narrow width effects, and vertical and lateral channel engineering Nano-Scale CMOS Analog Circuits: Models and CAD Techniques for High-

Level Design describes the models and CAD techniques, explores the physics of MOS transistors, and considers the design challenges involving statistical variations of process technology parameters and reliability constraints related to circuit design.
New to this edition:

Updated to using OrCAD Release 17.2 and its new features; Coverage of PSPICE extra features: PSpice Designer, PSpice Designer Plus, Modelling Application, PSpice Part Search Symbol Viewer, PSpice Report, Associate PSpice model, New delay functions for Behavioural

Simulation Models, New Models, Support for negative values in hysteresis voltage and threshold voltage; A new chapter on PSpice Advanced Analysis Analog Design and Simulation Using OrCAD Capture and PSpice, Second Edition provides step-by-step instructions on how to use the

Cadence/OrCAD family of Electronic Design Automation software for analog design and simulation. The book explains how to enter schematics in Capture, set up project types, project libraries and prepare circuits for PSpice simulation. There are chapters on the different analysis types for DC Bias

point, DC sweep, AC frequency sweep, Parametric analysis, Temperature analysis, Performance Analysis, Noise analysis, Sensitivity and Monte Carlo simulation. Subsequent chapters explain how the Stimulus Editor is used to define custom analog and digital signals, how the Model

Editor is used to view and create new PSpice models and Capture parts and how the Magnetic Parts Editor is used to design transformers and inductors. Other chapters include Analog Behaviorial models, Test Benches as well as how to create hierarchical designs. The book

includes the latest features in the OrCAD 17.2 release and there are exercises with step by step instructions at the end of each chapter that enables the reader to progress based upon their experience and knowledge gained from previous chapters. In addition, there are new chapters

on the PSpice Advanced Analysis suite of tools: Sensitivity Analysis, Optimizer, Monte Carlo, and Smoke Analysis.The chapters show how circuit performance can effectively be maximised and optimised for variations in component tolerances,

temperature effects, manufacturing yields and component stress. Provides both a comprehensive user guide and a detailed overview of simulation using OrCAD Capture and PSpice Includes worked and ready to try sample designs and a wide range of to-do exercises Covers Capture and PSpice

together
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.
Analog Circuit Design
ESD Design for Analog Circuits
Active RC, OTA-C, and SC
A Practical Approach
Analog Design and Simulation using OrCAD Capture and

PSpice

Analog CMOS integrated circuits are in widespread use for communications, entertainment, multimedia, biomedical, and many other applications that interface with the physical world. Although analog CMOS design is greatly complicated by the

design choices of drain current, channel width, and channel length present for every MOS device in a circuit, these design choices afford significant opportunities for optimizing circuit performance. This book addresses tradeoffs and optimization of device and circuit performance for *selections of the drain current, inversion coefficient, and channel length, where channel width is implicitly considered. The inversion coefficient is used as a technology independent measure of MOS inversion that permits design freely in weak, moderate, and strong inversion. This book* *details the significant performance tradeoffs available in analog CMOS design and guides the designer towards optimum design by describing: An interpretation of MOS modeling for the analog designer, motivated by the EKV MOS model, using tabulated hand expressions and figures* *that give performance and tradeoffs for the design choices of drain current, inversion coefficient, and channel length; performance includes effective gate-source bias and drain-source saturation voltages, transconductance efficiency, transconductance distortion, normalized*

drain-source conductance, capacitances, gain and bandwidth measures, thermal and flicker noise, mismatch, and gate and drain leakage current Measured data that validates the inclusion of important small-geometry effects like velocity saturation, vertical-field mobility reduction, drain-

induced barrier lowering, and inversion-level increases in gate-referred, flicker noise voltage In-depth treatment of moderate inversion, which offers low bias compliance voltages, high transconductance efficiency, and good immunity to velocity saturation effects for circuits designed in *modern, low-voltage processes Fabricated design examples that include operational transconductance amplifiers optimized for various tradeoffs in DC and AC performance, and micropower, low-noise preamplifiers optimized for minimum thermal and flicker noise A design spreadsheet,* *available at the book web site, that facilitates rapid, optimum design of MOS devices and circuits Tradeoffs and Optimization in Analog CMOS Design is the first book dedicated to this important topic. It will help practicing analog circuit designers and advanced students of electrical engineering build*

design intuition, rapidly optimize circuit performance during initial design, and minimize trial-and-error circuit simulations.
Bridges the gap between device modelling and analog circuit design.
Includes dedicated software enabling actual circuit design.
Covers the three

significant models: BSIM3, Model 9 &, and EKV. Presents practical guidance on device development and circuit implementation. The authors offer a combination of extensive academic and industrial experience. This book covers active R filters, OTA-C filters, and switched-capacitor filters, including topics

such as differential output opamps, sensitivity analysis for passive components, multiple-feedback techniques, double-sampling, and N-path filters.
Learn how analog circuit simulators work with these easy to use numerical recipes implemented in the popular Python

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

Mixed-Signal Methodology Guide

Analog Integrated Circuit Design

Analog Design for

CMOS VLSI Systems
Transistor Level Modeling for Analog/RF IC Design
A Tutorial Guide to Applications and Solutions
Practices and Innovations
In electronic circuit and system design, the word noise is used to refer to any undesired excitation

on the system. In other contexts, noise is also used to refer to signals or excitations which exhibit chaotic or random behavior. The source of noise can be either internal or external to the system. For instance, the thermal and shot noise generated *within integrated circuit devices are in ternal noise sources, and the noise picked up from the environment through electromagnetic interference is an external one. Electromagnetic interference can also occur between* *different components of the same system. In integrated circuits (les), signals in one part of the system can propagate to the other parts of the same system through electromagnetic coupling, power supply lines and the le substrate. For* *instance, in a mixed-signal le, the switching activity in the digital parts of the circuit can adversely affect the performance of the analog section of the circuit by traveling through the power supply lines and the substrate. Prediction of the* *effect of these noise sources on the performance of an electronic system is called noise analysis or noise simulation. A methodology for the noise analysis or simulation of an electronic system usually has the following four components: 2*

NOISE IN NONLINEAR ELECTRONIC CIRCUITS • Mathematical representations or models for the noise sources. • Mathematical model or representation for the system that is under the in fluence of the noise sources.

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. "Symbolic analyzers have the potential to* *offer knowledge to sophomores as well as practitioners of analog circuit design. Actually, they are an essential complement to numerical simulators, since they provide insight into circuit behavior which numerical " The essentials of analog circuit*

design with a unique all-region MOSFET modeling approach.
CMOS Analog Design Using All-Region MOSFET Modeling
Analog Electronics Applications
Systematic Design of Analog CMOS Circuits
Nano-scale CMOS Analog Circuits

Tradeoffs and Optimization in Analog CMOS Design

CircuitBook

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 S PICE-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.
It is a great honor to provide a few words of introduction for Dr. Georges Gielen's and

Prof. Willy Sansen's book "Symbolic analysis for automated design of analog integrated circuits". The symbolic analysis method

presented in this book represents a significant step forward in the area of analog circuit design. As demonstrated in this book, symbolic analysis opens

up new possibilities for the development of computer-aided design (CAD) tools that can analyze an analog circuit topology and automatically *size the components for a given set of specifications. Symbolic analysis even has the potential to improve the training of young analog* *circuit designers and to guide more experienced designers through second-order phenomena such as distortion. This book can also serve as*

an excellent reference for researchers in the analog circuit design area and creators of CAD tools, as it provides a comprehensive overview and comparison of

various approaches for analog circuit design automation and an extensive bibliography. The world is essentially analog in nature, hence

most electronic systems involve both analog and digital circuitry. As the number of transistors that can be integrated on a single

integrated circuit (IC) substrate steadily increases over time, an ever increasing number of systems will be implemented with one, or a few, very

complex ICs because of their lower production costs. The editors and authors present a wealth of knowledge regarding the most relevant *aspects in the field of MOS transistor modeling. The variety of subjects and the high quality of content of this volume make it a reference document for*

researchers and users of MOSFET devices and models. The book can be recommended to everyone who is involved in compact model

developments, numerical TCAD modeling, parameter extraction, space-level simulation or model standar dization. The book will appeal equally *to PhD students who want to understand the ins and outs of MOSFETs as well as to modeling designers working in the analog and*

high-frequency areas.

This Book and Simulation Software Bundle Project Dear Reader, this book project brings to you a unique study

tool for ESD protection solutions used in analog-integrated circuit (IC) design. Quick-start learning is combined with in-depth understanding for the whole

spectrum of cro-disciplinary knowledge required to excel in the ESD ?eld. The chapters cover technical material from elementary semiconductor

structure and device levels up to complex analog circuit design examples and case studies. The book project provides two different options for *learning the material. The printed material can be studied as any regular technical textbook. At the same time, another option adds parallel exercise using* *the trial version of a complementary commercial simulation tool with prepared simulation examples. Combination of the textbook material with* *numerical simulation experience presents a unique opportunity to gain a level of expertise that is hard to achieve otherwise. The book is* *bundled with simpli?ed trial version of commercial mixed- TM mode simulation software from Angstrom Design Automation. The DECIMM*

(Device Circuit Mixed-Mode) simulator tool and complementary to the book s- ulation examples can be downloaded from www.analogesd.com. The simulation

examples prepared by the authors support the speci?c examples discussed across the book chapters. A key idea behind this project is to

provide an opportunity to not only study the book material but also gain a much deeper understanding of the subject by direct experience through

practical simulation examples. Advanced VLSI Design and Testability Issues Circuit Simulation with SPICE OPUS Intuitive

Analog Circuit Design
Device Modeling for Analog and RF CMOS Circuit Design
The Art and Science of Analog Circuit Design
RF Circuits:

Wide band, Front-Ends, DAC's, Design Methodology and Verification for RF and Mixed-Signal Systems, Low Power and Low Voltage

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
Analog Design and Simulation Using

OrCAD Capture and PSpiceNewnes Starting from the fundamentals, the present book describes methods of designing analog electronic filters and illustrates these methods by providing numerical and circuit simulation programs. The *subject matters comprise many concepts and techniques that are not available in other text books on the market. To name a few - principle of transposition and its application in directly realizing current mode filters from well known voltage mode filters;*

an insight into the technological aspect of integrated circuit components used to implement an integrated circuit filter; a careful blending of basic theory, numerical verification (using MATLAB) and illustration of the actual circuit behaviour using *circuit simulation program (SPICE); illustration of few design cases using CMOS and BiCMOS technological processes. 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. Learn the principles and practices of simulation-based analog IC design This comprehensive textbook and on-the-job reference offers clear instruction on analog integrated circuit design using* *the latest simulation techniques. Ideal for graduate students and professionals alike, the book shows, step by step, how to develop and deploy integrated circuits for cutting-edge Internet of Things (IoT) and other applications. Analog Integrated Circuit Design by*

Simulation: Techniques, Tools, and Methods lays out practical, ready-to-apply engineering strategies. Application layer, device layer, and circuit layer IC design are covered in complete detail. You will learn how to tackle real-world design problems *and avoid long cycles of trial and error. Coverage includes: •First-order DC response•Unified closed-loop model•Accurate modeling of DC response•Frequency and step response•Multi-pole dynamic response and stability•Effect*

of external network on differential gain• Continuous-time and discrete-time amplifiers•MOSFET, NMOS, and PMOS c haracteristics•Small-signal modeling and circuit analysis•Resistor and capacitor design•Current sources, sinks, and mirrors•Basic,

symmetrical, folded-cascode, and Miller OTAs•Opamps with source-follower and common-source output stages•Fully differential OTAs and opamps
Analog Integrated Circuit Design by Simulation: Techniques, Tools, and Methods
Using Pre-

Computed Lookup Tables

Analog Circuit Simulators for Integrated Circuit Designers

CMOS analog circuit design

In this companion text to Analog Circuit Design: Art, Science, and Personalities, seventeen contributors present more tutorial, historical, and editorial viewpoints on subjects related to analog circuit design. By presenting divergent methods and views of people who have achieved some measure of success in their field, the book encourages readers to develop their own approach to design. In addition, the essays and anecdotes give some constructive guidance in areas not usually covered in engineering courses, such as marketing and career development. *Includes visualizing operation of analog circuits *Describes troubleshooting for

optimum circuit performance
*Demonstrates how to produce a saleable product
This book facilitates the VLSI-interested individuals with not only in-depth knowledge, but also the broad aspects of it by explaining its applications in different fields,

including image processing and biomedical. The deep understanding of basic concepts gives you the power to develop a new application aspect, which is very well taken care of in this book by using simple language in explaining the concepts. In the VLSI world, the importance of hardware description languages cannot be ignored, as the designing of such dense and complex circuits is not possible without them. Both Verilog and VHDL languages are used here for designing. The current needs of high-performance integrated circuits

(ICs) including low power devices and new emerging materials, which can play a very important role in achieving new functionalities, are the most interesting part of the book. The testing of VLSI circuits becomes more crucial than the designing of the circuits in this nanometer technology era. The role of fault simulation algorithms is very well explained, and its implementation using Verilog is the key aspect of this book. This book is well organized into 20 chapters. Chapter 1 emphasizes on uses of FPGA on various image processing and biomedical applications. Then, the descriptions enlighten the basic understanding of digital design from the perspective of HDL in Chapters 2–5. The performance enhancement with alternate material or geometry for silicon-based FET designs is focused in Chapters 6 and 7. Chapters 8 and

9 describe the study of bimolecular interactions with biosensing FETs. Chapters 10–13 deal with advanced FET structures available in various shapes, materials such as nanowire, HFET, and their comparison in terms of device performance metrics calculation. Chapters

14–18 describe different application-specific VLSI design techniques and challenges for analog and digital circuit designs. Chapter 19 explains the VLSI testability issues with the description of simulation and its categorization into logic and fault simulation for test pattern generation using Verilog HDL. Chapter 20 deals with a secured VLSI design with hardware obfuscation by hiding the IC's structure and function, which makes it much more difficult to reverse engineer.