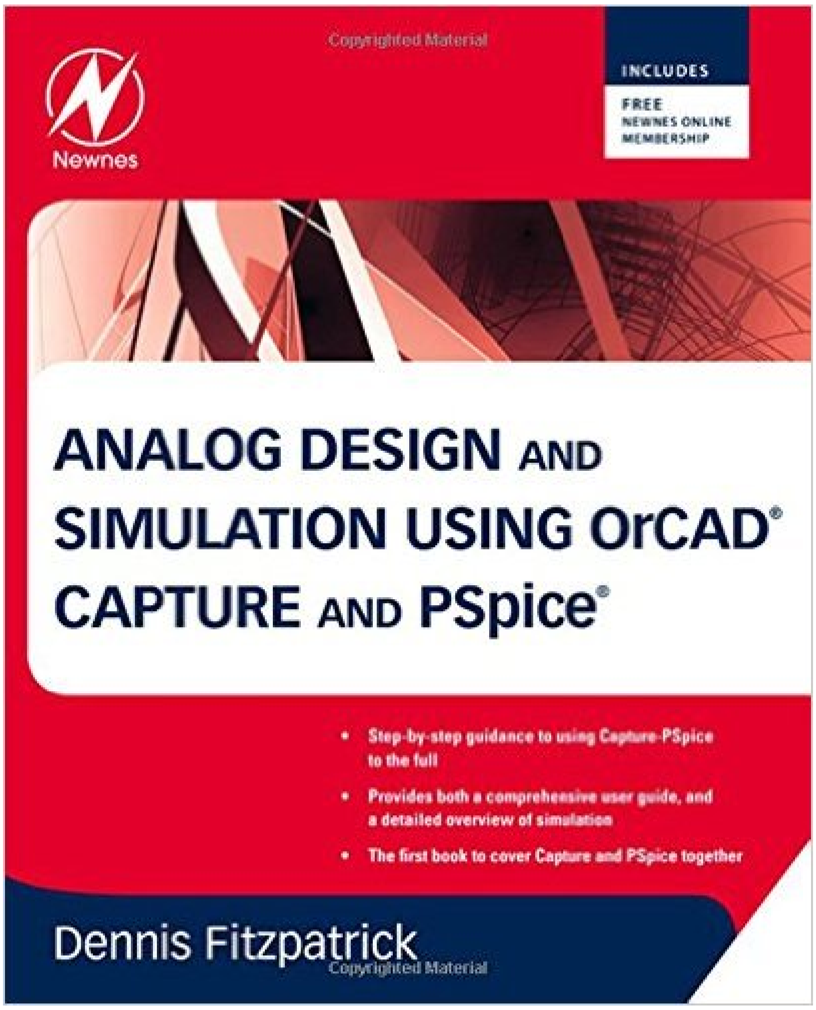
The book was found

Analog Design And Simulation Using OrCAD Capture And PSpice



Synopsis

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 simulationEach chapter has worked and ready to try sample designs and provides a wide range of to-do exercisesCore skills are developed using a running case study circuitCovers Capture and PSpice together for the first time

Book Information

Paperback: 344 pages

Publisher: Newnes; 1 edition (November 30, 2011)

Language: English

ISBN-10: 0080970958

ISBN-13: 978-0080970950

Product Dimensions: 7.5 x 0.8 x 9.2 inches

Shipping Weight: 1.6 pounds (View shipping rates and policies)

Average Customer Review: 4.7 out of 5 starsÂ Â See all reviewsÂ (6 customer reviews)

Best Sellers Rank: #244,525 in Books (See Top 100 in Books)   #36 inÂ Books > Engineering & Transportation > Engineering > Electrical & Electronics > Circuits > Integrated   #44 inÂ Books >

Engineering & Transportation > Engineering > Industrial, Manufacturing & Operational Systems > Industrial Design > Products #77 inÂ Books > Engineering & Transportation > Engineering > Electrical & Electronics > Circuits > Design

Customer Reviews

this is a must-have for anyone learning or using OrCad,i downloaded the free version of OrCad - OrCad Lite from the Cadence websiteand together with this book have successfully completed a range of electronis simulations and builds, some rather simple and some quite complicatited. it is easy to follow and apply as there are examples all the way through.i have also used this book non-stop through my studies and would recomed it to anyone doing an electronics course.

This book is a perfect guide for learning PSpice tool. It explains step-by-step along with OrCAD screen pictures so that we can get ahead on learning the tool, instead of getting confused by those bad programming or computing stuff. What we want is to learn PSpice, not computer.Author uses easy circuits to show great PSpice tool potential. We can draw it in minutes and simulate immediately. That's the shortest path in this learning. Except a couple of larger circuits which takes longer to find and place parts, no CD is required.To learn PSpice 16.5, this is the best book I can find. And, I have searched through more than 10 other related books. Some interested ones I used less than 1 chapter and gave up for reasons of inaccuracy, missing importance step or information etc.The book does have some minor typo's, a couple of missing or wrong component values, and wrong figure numbers. Kind of adding the fun of debugging and learning. Refer to PSpice manual (OrCAD help) if definition on terminology unclear, such as T_REL_LOCAL. These small things help me actually.It's about an useful tool for long. Reading easy tends to forget soon. The book has the real feel and the touch for a person who is really doing it. Taking notes alongside can help in case need to come back later.cheers,Tom

I found the book an excellent guide to use PSpice and OrCAD. The chapter on creating new models is not quite good though.

Download to continue reading...

Analog Design and Simulation using OrCAD Capture and PSpice PSPICE and MATLAB for Electronics: An Integrated Approach, Second Edition (VLSI Circuits) FinFET Modeling for IC Simulation and Design: Using the BSIM-CMG Standard Analog Circuit Design: Art, Science and Personalities (EDN Series for Design Engineers) Process Simulation Using WITNESS Foundations of Analog and Digital Electronic Circuits (The Morgan Kaufmann Series in Computer Architecture and Design) Design With Operational Amplifiers And Analog Integrated Circuits (McGraw-Hill Series in Electrical and Computer Engineering) Mixed-signal and DSP Design Techniques (Analog Devices) Design of Analog Filters 2nd Edition (The Oxford Series in Electrical and Computer Engineering) CMOS Analog Circuit Design (The Oxford Series in Electrical and Computer Engineering) Analysis and Design of Analog Integrated Circuits, 5th Edition Analog Design Essentials (The Springer International Series in Engineering and Computer Science) Analog Filter and Circuit Design Handbook High-Frequency Analog Integrated Circuit Design (Wiley Series in Microwave and Optical Engineering) Design with Operational Amplifiers and Analog Integrated Circuits Analysis and Design of Analog Integrated Circuits (4th Edition) Analog Design for CMOS VLSI Systems (The Springer International Series in Engineering and Computer Science) VLSI Design Techniques for Analog and Digital Circuits (McGraw-Hill Series in Electrical Engineering) Zen of Analog Circuit Design Design of Analog CMOS Integrated Circuits

Dmca