Pspice User S Guide

Practical RF Amplifier Design and Performance Optimization with SPICE and Load- and Source-pull Techniques
The Designer’s Guide to Spice and Spectre®
Introduction to PSpice Manual Using Orcad Release 9.2 for Introductory Circuits for Electrical and Computing Engineering
Introduction to PSpice Manual for Electric Circuits, Using OrCAD Release 9.2
MicroSim PSpice A/d
SPICE
ANALOG ELECTRONICS
The Electronic Design Automation Handbook
Introduction to PSpice Manual, Using ORCad Release 9.2 to Accompany Electric Circuits, Seventh Edition
Control System Design Guide:
Continuous System Modeling
PSpice [4.02]
Spice
Cunningham PSPICE Manual+ibm3
Solutions Manual for PSPICE and MATLAB for Electronics
Introduction to PSpice Manual
MicroSim PSpice AD
Complete PCB Design Using OrCAD Capture and PCB Editor
LABORATORY EXPERIMENTS AND PSPICE SIMULATIONS IN ANALOG ELECTRONICS
Handbook of Memristor Networks
Semiconductor Device Modeling with Spice
PSpice for Basic Circuit Analysis with CD
Pspice
PSpice for Linear Circuits (uses PSpice version 9.2)
Introduction to PSpice Manual Using OrCAD Release 9.2 to Accompany Electric Circuits
MicroSim PSpice A/D
Hands-On PSPICE
PSpice Text-Manual
PSPICE and MATLAB for Electronics
The Illustrated Guide to PSpice
PSpice for Basic Circuit Analysis
Introduction to PSpice for Electric Ciircuits
Fundamentals of Power Integrity for Computer Platforms and Systems
PSpice Simulation of Power Electronics Circuits
Analog Design and Simulation Using OrCAD Capture and PSpice
Basic Circuit Analysis
PSpice Manual for Electric Circuits Fundamentals
Schematic Capture with MicroSim PSpice
Pspice for Basic Microelectronics
Introduction to PSpice Manual, Electric Circuits, Using ORCad Release 10.5

KIERA CHOI

Practical RF Amplifier Design and Performance Optimization with SPICE and Load- and Source-pull Techniques
Prentice Hall
Engineering productivity in integrated circuit product design and - velopment today is limited largely by the effectiveness of the CAD tools used. For those domains of product design that are highly dependent on transistor-level circuit design and optimization, such as high-speed logic and memory, mixed-signal analog-digital int- faces, RF functions, power integrated circuits, and so forth, circuit simulation is perhaps the single most important tool. As the complexity and performance of integrated electronic systems has increased with scaling of technology feature size, the capabilities and sophistication of the underlying circuit simulation tools have correspondingly increased. The absolute size of circuits requiring transistor-level simulation has increased dramatically, creating not only problems of computing power resources but also problems of task organization, complexity management, output representation, initial condition setup, and so forth. Also, as circuits of more c- plexity and mixed types of functionality are attacked with simu- tion, the spread between time constants or event time scales within the circuit has tended to become wider, requiring new strategies in simulators to deal with large time constant spreads.
The Designer’s Guide to Spice and Spectre® Springer Science & Business Media
Computer tools can assist students in the learning process by providing a visual representation of a circuit's behavior, validating a calculated solution, reducing the computational burden of more complex circuits, and iterating toward a desired solution using parameter variation. This computational support is often invaluable in the design process. Updated for PSpice using OrCAD release 10.5, this manual focuses on three things: - Learning to draw and simulate linear circuits using PSpice - Constructing circuit models of basic devices such as op amps - Learning to challenge computer output data as a means of reinforcing confidence in simulation PSpice software may be used to solve many of Nilsson & Riedel's Electric Circuits, 8e Assessment Problems and Chapter Problems but the manual was

designed as a supplement to stand on its own as an instructional unit.
Introduction to PSpice Manual Using Orcad Release 9.2 for Introductory Circuits for Electrical and Computing Engineering Wiley
This book explains and demonstrates with an exhaustive set of design examples, how common types of radio frequency(RF) amplifiers (classes A, B, AB, C, D, E, F, G and H) can be designed, and then have their performance characteristics evaluated and optimized with SPICE. The author demonstrates the transient analysis features of SPICE, along with industry-standard load- and source-pull techniques to simulate the steady-state, long-term time-domain behavior of any test RF amplifier.· Describes methods for designing and evaluating/optimizing the performance characteristics of an RF amplifier that circumvent the issues involved with existing, traditional methods and don’t require expensive, high-end software tools;· Includes C language executables for each RF amplifier type, eliminating errors that might creep in while computing passive component (capacitor, inductor, resistor) values for a given RF amplifier type;· Demonstrates industry-standard load- and source-pull schemes that can be included easily in text SPICE netlists, allowing accurate calculation of impedance matching and impedance values at the input and output ports of the test RF amplifier, eliminating messy, error-prone S parameter based calculations.
Introduction to PSpice Manual for Electric Circuits, Using OrCAD Release 9.2 Elsevier
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
MicroSim PSpice A/d Delmar Pub
The PSpice Manual will be sold as a stand-alone and, also, in packages with Neamen, Electronic Circuit Analysis and Jaeger, Microelectronic Circuit Design. Text introduces readers to the fundamental uses of Pspice in support of Microelectronic circuit analysis. This book goes beyond basic circuit analysis to include analysis of more complex electronic problems. Analysis of diodes, BJTs, JFETs, MOSFETs, and transformers will be included- -all key areas in the Electronics course. Key features include: * Step-by-step instructions to support novice users as they perform schematic capture and circuit simulation. * Detailed explanations and examples of the use of PSpice in typical problem-solving situations. * Explains some of the salient features of PSpice, including information on OrCAD Capture and Probe.
SPICE McGraw-Hill Education
This is a practical approach to control techniques. The author covers background material on analog controllers, digital controllers, and filters. Commonly used controllers are presented. Extended use of PSpice (a popular circuit simulation program) is used in problem solving. The book is also documented with 50 computer programs that circuit designers can use. Explains integration of control systems with a personal computer Compares numerous control algorithms in digital and analog form Details the use of SPICE in problem solving Presents modeling concepts for linear and nonlinear systems Examines commonly used controllers
ANALOG ELECTRONICS Prentice Hall
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 Electronic Design Automation Handbook McGraw Hill Professional
Based on the latest version of the software (DOS/Version 6), this book introduces users to the popular PSpice circuit analysis software program. Each feature of circuit analysis and each feature of the PSpice software is clearly presented and illustrated by an example, putting electronics technicians and professional engineers on the fast -track to increased productivity. An introductory chapter on how to load and configure the PSpice program is also included for those who are new to network analysis or new to using computers.
Introduction to PSpice Manual, Using ORCad Release 9.2 to Accompany Electric Circuits, Seventh Edition McGraw-Hill Europe
This Handbook presents all aspects of memristor networks in an easy to read and tutorial style. Including many colour illustrations, it covers the foundations of memristor theory and applications, the technology of memristive devices, revised models of the Hodgkin-Huxley Equations and ion channels, neuromorphic architectures, and analyses of the dynamic behaviour of memristive networks. It also shows how to realise computing devices, non-von Neumann architectures and provides future building blocks for deep learning hardware. With contributions from leaders in computer science, mathematics, electronics, physics, material science and engineering, the book offers an indispensable source of information and an inspiring reference text for future generations of computer scientists, mathematicians, physicists, material scientists and engineers working in this dynamic field.
Control System Design Guide: CRC Press
Readers benefit because the book is based on these three themes: (1) it builds an understanding of concepts based on information the reader has previously learned; (2) it helps stress the relationship between conceptual understanding and problem-solving approaches; (3) the authors provide numerous examples and problems that use realistic values and situations to give users a strong foundation of engineering practice. The book also includes a PSpice Supplement which contains problems to teach readers how to construct PSpice source files; and this PSpice Version 9.2 can be used to solve many of the exercises and problems found in the book. Topical emphasis is on the basic techniques of circuit analysis -- Illustrated via a Digital-to-Analog Resistive Ladder (Chapter 2); the Flash Converter (Chapter 4); Dual Slope Analog-to-Digital Converter (Chapter 5); Effect of parasite inductance on the step response of a series RLC circuit (Chapter 6); a Two-Stage RC Ladder Network (Chapter 8); and a Switching Surge Voltage (Chapter 9).
Continuous System Modeling McGraw-Hill Science, Engineering & Mathematics
The Electronic Design Automation Handbook carefully details design tools and techniques for high performance ASIC-

design. It shows the best practices for creating reusable designs in an SoC design methodology. The Electronic Design Automation Handbook was developed by colleagues from the Universities of Applied Sciences, Germany, who are engaged in the design of integrated electronics in education and research and which form the MPC Group of the Universities of Applied Sciences of Baden-Württemberg /Germany. MPC works as network of partners to industry and is able, due to the wide varying experiences of the institutes involved, to cover the entire range of the modern day circuit design. Each year more than 600 students are educated in the laboratories of MPC-members. Our personal experience from student and industry-projects ensures authenticity. The practical and theoretical experience from our projects has been used in the basis of this handbook.

PSpice [4.02] Springer Nature
This guide to the PSpice circuit simulator provides a tutorial approach to using PSpice through graduated examples. This edition includes enhanced pedagogical features, and coverage of the newest capabilities of this program. It explains: the use of Monte Carlo methods in PSpice for statistically computing estimates of how circuits will behave with variations in component values and derivation and use of two-port parameters, including s-parameters. It also includes an expanded section on group and time delay, and on noise analysis, as well as fuller descriptions and examples for using parameters, functions and values defined by formulas to generalize circuit blocks and specify component values.

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

Cunningham PSPICE Manual+ibm3 Springer Science & Business Media
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.

Solutions Manual for PSPICE and MATLAB for Electronics Springer Nature
This practical PSpice manual, updated to support the latest release of OrCAD Pspice introduces students to the fundamental uses of this book in support of basic circuit analysis. The organization allows readers to advance quickly to solving a variety of circuit analysis problems. The modular approach allows this hand-on reference to be used with any introductory circuits text.

Introduction to PSpice Manual Newnes
PLEASE PROVIDE COURSE INFORMATION PLEASE PROVIDE

MicroSim PSpice AD PHI Learning Pvt. Ltd.
Modeling and Simulation have become endeavors central to all disciplines of science and engineering. They are used in the analysis of physical systems where they help us gain a better understanding of the functioning of our physical world. They are also important to the design of new engineering systems where they enable us to predict the behavior of a system before it is ever actually built. Modeling and simulation are the only techniques available that allow us to analyze arbitrarily non-linear systems accurately and under varying experimental conditions. Continuous System Modeling introduces the student to an important subclass of these techniques. They deal with the analysis of systems described through a set of ordinary or partial differential equations or through a set of difference equations. This volume introduces concepts of modeling physical systems through a set of differential and/or difference equations. The purpose is twofold: it enhances the scientific understanding of our physical world by codifying (organizing) knowledge about this world, and it supports engineering design by allowing us to assess the consequences of a particular design alternative before it is actually built. This text has a flavor of the mathematical discipline of dynamical systems, and is strongly oriented towards Newtonian physical science.

Complete PCB Design Using OrCAD Capture and PCB Editor Academic Press
Showing students how to use the PSpice circuit simulation program from MicroSim with the schematic capture front end, Schematics, this manual uses examples to provide instructions on creating a circuit, running the different analyses, and obtaining the results. CD-Rom includes Schematics Version 7.1, Version 6.1, Win32s, MicroSim Evaluation Version 5.2, and more. Annotation copyrighted by Book News, Inc., Portland, OR.

LABORATORY EXPERIMENTS AND PSPICE SIMULATIONS IN ANALOG ELECTRONICS PHI Learning Pvt. Ltd.
Chapters in this manual are arranged to match the topics covered in the text. Each chapter introduces device definitions and/or PSpice commands along with examples.

Handbook of Memristor Networks Oxford University Press, USA
This paperback manual clearly explains how PSpice, a circuit design software product, works--enabling students to develop an understanding of how circuits are designed and built. Includes scores of examples and problems and can be used as a supplement to any textbook in either circuits or electronics. A 3.5" data disk containing circuit design problems accompanies the manual.