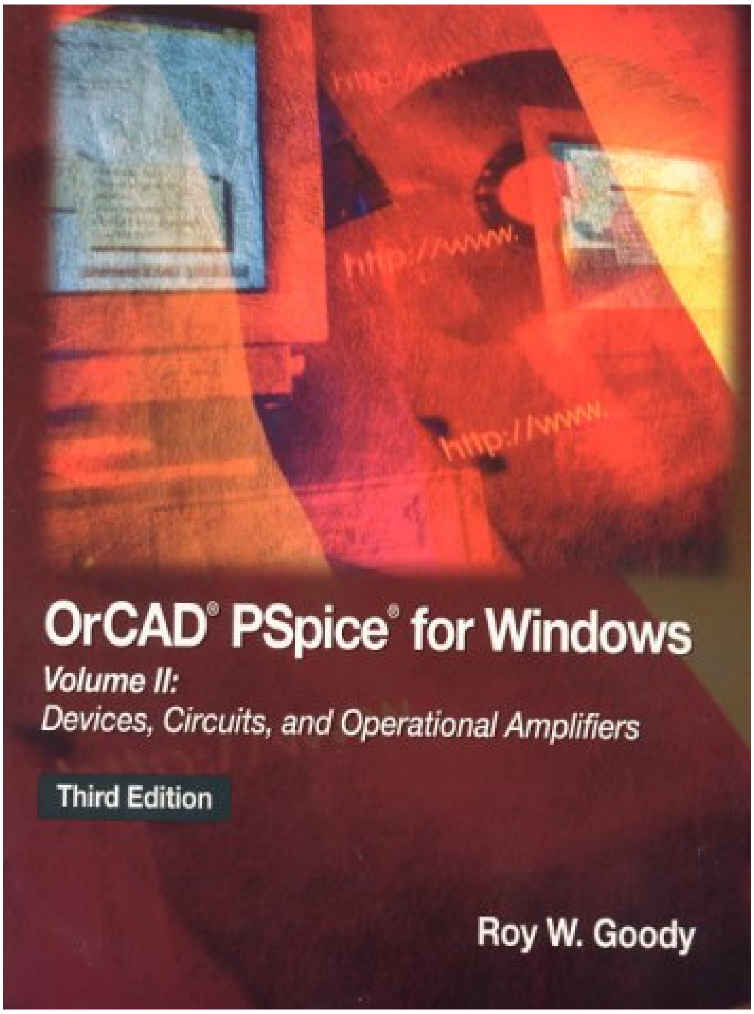
OrCAD PSpice for Windows Volume II: Devices, Circuits, and Operational Amplifiers (3rd Edition)

Roy W. Goody
*audiobook | *ebooks | Download PDF | ePub | DOC*



#1595960 in Books 2000-09-10Ingredients: Example IngredientsOriginal language:EnglishPDF # 1 10.43 x .88 x 8.24l, 2.13 #File Name: 013015797X415 pages | File size: 30.Mb

Roy W. Goody : OrCAD PSpice for Windows Volume II: Devices, Circuits, and Operational Amplifiers (3rd Edition) before purchasing it in order to gage whether or not it would be worth my time, and all praised OrCAD PSpice for Windows Volume II: Devices, Circuits, and Operational Amplifiers (3rd Edition):

0 of 0 people found the following review helpful. Five StarsBy JOE FROM CTGood book.

Assuming a working knowledge of basic PSpice techniques, this book explores simulation studies with more advanced topics such as operational amplifiers, digital, and filter design. It shows users how to use the program to draw circuits directly on the screen, analyze the circuit in seconds using PSpice, and display the results using sophisticated techniques that go far beyond those possible with conventional instruments. Chapter topics include analog/digital conversionsresolution; random-access memorydata acquisition system; and printed circuit boardlayouts. For instruction in DC/AC and devices, operational amplifiers, digital, analog, and filter design.

From the Back CoverAssuming a working knowledge of basic PSpice techniques, this book explores simulation studies with more advanced topics such as operational amplifiers, digital, and filter design. It shows users how to use the program to draw circuits directly on the screen, analyze the circuit in seconds using PSpice, and display the results using sophisticated techniques that go far beyond those possible with conventional instruments. Chapter topics include analog/digital conversionsresolution; random-access memorydata acquisition system; and printed circuit boardlayouts. For instruction in DC/AC and devices, operational amplifiers, digital, analog, and filter design.Excerpt. Reprinted by permission. All rights reserved. Preface What's past is prologue. WILLIAM SHAKESPEARE, THE TEMPEST (1600) This is the second in a three-volume series on PSpice circuit simulation and covers Devices, Circuits, and Operational Amplifiers. (Volume I covers DC and AC Circuits and Volume III covers Digital and Data Communications.) If you are already familiar with Volume I, you will find few surprises in Volume II. If not, a brief overview is provided here. The material is based on OrCAD Lite 9.2, the latest free evaluation version of the most popular simulation software on the market today: OrCAD corporation's PSpice. It is introductory in nature and is appropriate for those with little or no experience in circuit simulation. The level of difficulty is tailored to the technology student, but it offers enough "gentle" material for the technician and enough challenging material for the engineer. It covers both PSpice techniques and analog theory and applications, and the choice and sequence of material closely follows that of a conventional theory text. Since most activities can be done by either PSpice simulation or hands-on construction, it is designed to replace a conventional laboratory manual that covers devices, circuits, and operational amplifiers. Why PSpice? As a dedicated student or educator, you may strongly believe that circuit simulation must be a part of your classroom experience before applying your knowledge and skills on the job. It would be reasonable, therefore, to seek out the most popular circuit simulation software on the market today. It also would be beneficial to use the software package that is used by engineers and technicians on the jobcomplete with professional-level advanced techniques and restricted only by circuit size. Further it would be sensible to reduce your costs to zero and distribute the free software without the need to worry about licenses or copyright restrictions. The solution, then, is PSpice. Volume II We assume that the majority of students reading this preface have at least a passing familiarity with OrCAD PSpice for Windows, Volume I, covering DC and AC circuits. Volume I also covers the most fundamental PSpice techniques and processes, and there is not sufficient room in Volumes II or III to repeat this introductory material. We can, however, present short review inserts where appropriate and reprint several of the special Simulation Notes in Appendix A. Therefore, we recommend that you keep a copy of Volume I available for reference and review. Volume II is divided into six parts: Diode Circuits, Bipolar Transistor Circuits, Field-Effect Transistor Circuits, Special Solid State Studies, Operational Amplifiers, and Special Processes. This is the same order and mix of subjects normally found in a typical devices and circuits and analog course. The special processes of Part 6 illustrate the benefits of using circuit simulation over conventional prototyping. For the most part it is not necessary to complete the first five parts before turning to and dabbling in these powerful techniques. OrCAD's Total Solution For designing electronic circuits, OrCAD offers a total solution package, including schematic entry, FPGA synthesis, digital, analog, mixed-signal simulation, and printed circuit board layouteverything from start to finish. All software components are fully integrated and are designed to follow an engineer's natural design flow. This text is based exclusively on just one part of the complete package: PSpice AID. Fortunately, PSpice A/D is precisely what we need to support a college-level technology class, for this software component simulates nearly any mix of analog and digital circuits and conveniently displays the results in graphical form. It is incredibly powerful, easy to learn, and simple to use. Quite simply, OrCAD's PSpice A/D is one of the best learning tools available. OrCAD Lite Fortunately, for those of us in education, OrCAD Corporation has made PSpice evaluation software available at no cost. All the activities in this book are based on OrCAD Lite version 9.2. Its only major limitation is the number of symbols and components that can be placed on the schematic. Fortunately, we can adjust easily to these limitations, and for the most part they will be completely invisible. Third Edition The major improvement of this third edition is the move up to version 9.2. Although many of the circuits and components are unchanged from the second edition, a greatly improved project manager window facilitates schematic organization, assignment of simulation type, and analysis and display of the circuit response. For this volume II, we now include operational amplifiers. Suggestion Although circuit simulation is the major design and development tool of the future, we recommend that the reader also receive hands-on experience by prototyping actual circuits and troubleshooting with conventional instruments. One computer-saving approach is to divide a class into two or more groups and switch between PSpice and hands-on techniques. It is especially instructive to perform the same activity using both PSpice and hands-on techniques, and to compare the two approaches. In this regard, most of the experimental activities outlined in this text can be performed using either PSpice or hands-on techniques. Further Study If you order the complete set of manuals that comes with PSpice, you would be confronted with more than one thousand pages of data, instructions, and reference material. Clearly, all the information contained within those pages cannot be placed into this introductory text series. Instead, we have included only the most vital and commonly used features of PSpice. For a comprehensive description of all the features of PSpice, refer to the complete set of manuals from OrCAD. Product manuals and many other useful items and features, including technical data, articles, techtips, and university support, can be obtained from OrCAD's website (www.orcad.com). Mouse Conventions Throughout this text, we will adopt the following mouse convention: CLICKL or BOLD PRINT (click left once) to select an item. DCLICKL (double click left) to perform an action. CLICKR (click right once) to open a menu. CLICKLH (click left, hold down, and move mouse) to drag a selected item. Release left button when placed. Acknowledgments I wish to express my sincere gratitude to production editor Rex Davidson and acquisitions editor Scott Sambucci of Prentice Hall. Under their careful guidance, the project steadily moved forward and was released on time. I also thank the reviewers of the manuscript: Ed Bertnolli, University of North Florida; John Brews, University of Arizona; and Ronald Rockland, New Jersey Institute of Technology. Of course, OrCAD Corporation deserves special credit for making the OrCAD Lite evaluation disk available at no cost. Their foresight makes it possible for colleges and universities to teach circuit simulation at the professional level without breaking the ever-shrinking budget. Thank you for adopting OrCAD PSpice for Windows; May you have good luck and success. Ray W. Goody