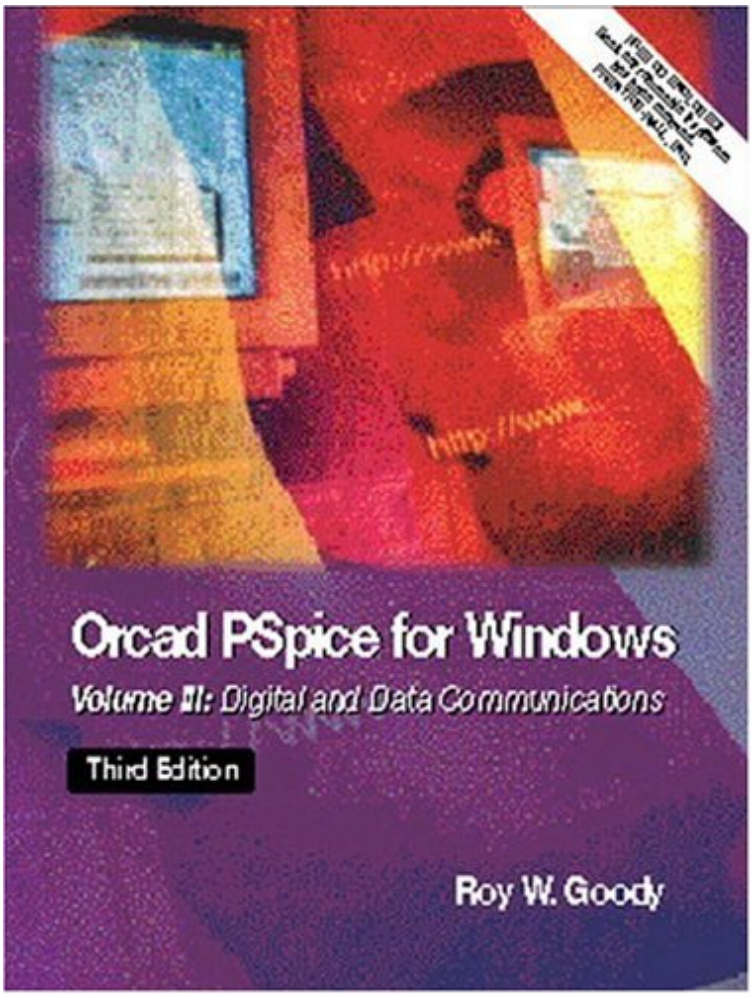
OrCAD PSpice for Windows Volume III: Digital and Data Communications

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



#3352838 in Books Prentice Hall 2000-09-01Ingredients: Example IngredientsOriginal language:EnglishPDF # 1 11.00 x .50 x 8.25l, 1.35 #File Name: 0130311839260 pages | File size: 46.Mb

Roy W. Goody : OrCAD PSpice for Windows Volume III: Digital and Data Communications before purchasing it in order to gage whether or not it would be worth my time, and all praised OrCAD PSpice for Windows Volume III: Digital and Data Communications:

0 of 0 people found the following review helpful. Five StarsBy David Perygood book...ship very quickly

This book takes readers from a simple DC circuit with its customary current and voltage measurements, through a damped resonant circuit requiring rise time characteristics, to a complex digital circuit that demands sophisticated timing and frequency measurements. It provides students with a sophisticated, comprehensive, powerful, user-friendly learning tool. The continuation of simulation studies includes more advanced topics such as operational amplifiers,

digital, and filter design. Comfortable yet challenging, multi-level activities, examples, and exercises show learners 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. A wide range of electronic theory compliments all the major PSpice techniques and processes. For electrical engineers involved with devices and circuits, operational amplifiers, and digital, analog, and filter design.

From the Inside FlapPreface Nothing happens unless first a dream. CARL SANDBURG, Washington Monument by Night, 1970 OrCADPSpicefor Windows, Volume III is the third manual in a three-volume series on Spice circuit simulation. It covers Digital and Data Communications. (Volume I covers DC/AC Circuits and Volume II covers Devices, Circuits, and Operational Amplifiers.) If you are already familiar with Volume I or II, you will find few surprises in Volume III. If not, here is a brief overview: The text is based on the latest free evaluation version 9 of the most popular simulation software on the market today: OrCAD'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 electronic theory and applications, and the choice and sequence of material closely follows that of a conventional text on digital. Since most activities can be done by either PSpice simulation or hands-on construction, it is designed to replace a conventional laboratory manual. Why PSpice for Windows? As a devoted educator, the chances are you strongly believe that your students must have a circuit simulation experience before they apply their knowledge and skills on the job. Would it not be reasonable to seek out the most popular circuit simulation software on the market today? Would it not also be highly beneficial to use the same software package that is used by engineers and technicians on the jobrestricted only by circuit complexity? And would it not be sensible to reduce your costs to zero, and to distribute the software without regard for licenses or copyright restrictions? Volume III We assume that the majority of students reading this preface have at least a passing familiarity with. Volume I or II. The majority of the most fundamental PSpice techniques and processes are covered in Volume I. Since there is insufficient room to repeat all the introductory material here, short review inserts are presented where appropriate and special Simulation Notes are reprinted in Appendix A. Therefore, if this text is being used in a classroom situation, we strongly recommend that several copies of Volume I be available for reference and review. Volume III is divided into six parts: Gates and Flip-Flops; Violations and Hazards; Counters, Shift Registers, Coders, and Timers; Converters and RAM; Data Communications; and Modular Design and Applications. This is generally the same order and mix of subjects found in a typical digital course. The modular techniques of Part 6 can, for the most part, be applied to any of the previous chapters at any time. 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 layout-everything 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 A/D. 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. Newer versions of the OrCAD software are constantly being released and the chances are good that they will work with this manual. In general, you should use the latest version that is available; if any adjustments are necessary, they should be minor. However, to be perfectly safe, you may wish to stay with version 9.2 until the next edition of PSpice for Windows is released. 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 A/D, 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 (orcad). 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. Of course, OrCAD Corporation deserves special credit for making the OrCAD Lite evaluation version 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.Roy W. GoodyFrom the Back CoverThis book takes readers from a simple DC circuit with its customary current and voltage measurements, through a damped resonant circuit requiring rise time characteristics, to a complex digital circuit that demands sophisticated timing and frequency measurements. It provides students with a sophisticated, comprehensive, powerful, user-friendly learning tool. The continuation of simulation studies includes more advanced topics such as operational amplifiers, digital, and filter design. Comfortable yet challenging, multi-level activities, examples, and exercises show learners 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. A wide range of electronic theory compliments all the major PSpice techniques and processes. For electrical engineers involved with devices and circuits, operational amplifiers, and digital, analog, and filter design.Excerpt. Reprinted by permission. All rights reserved. Preface Nothing happens unless first a dream. CARL SANDBURG, Washington Monument by Night, 1970 OrCADPSpicefor Windows, Volume III is the third manual in a three-volume series on Spice circuit simulation. It covers Digital and Data Communications. (Volume I covers DC/AC Circuits and Volume II covers Devices, Circuits, and Operational Amplifiers.) If you are already familiar with Volume I or II, you will find few surprises in Volume III. If not, here is a brief overview: The text is based on the latest free evaluation version 9 of the most popular simulation software on the market today: OrCAD'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 electronic theory and applications, and the choice and sequence of material closely follows that of a conventional text on digital. Since most activities can be done by either PSpice simulation or hands-on construction, it is designed to replace a conventional laboratory manual. Why PSpice for Windows? As a devoted educator, the chances are you strongly believe that your students must have a circuit simulation experience before they apply their knowledge and skills on the job. Would it not be reasonable to seek out the most popular circuit simulation software on the market today? Would it not also be highly beneficial to use the same software package that is used by engineers and technicians on the jobrestricted only by circuit complexity? And would it not be sensible to reduce your costs to zero, and to distribute the software without regard for licenses or copyright restrictions? Volume III We assume that the majority of students reading this preface have at least a passing familiarity with. Volume I or II. The majority of the most fundamental PSpice techniques and processes are covered in Volume I. Since there is insufficient room to repeat all the introductory material here, short review inserts are presented where appropriate and special Simulation Notes are reprinted in Appendix A. Therefore, if this text is being used in a classroom situation, we strongly recommend that several copies of Volume I be available for reference and review. Volume III is divided into six parts: Gates and Flip-Flops; Violations and Hazards; Counters, Shift Registers, Coders, and Timers; Converters and RAM; Data Communications; and Modular Design and Applications. This is generally the same order and mix of subjects found in a typical digital course. The modular techniques of Part 6 can, for the most part, be applied to any of the previous chapters at any time. 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 layout-everything 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 A/D. 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. Newer versions of the OrCAD software are constantly being released and the chances are good that they will work with this manual. In general, you should use the latest version that is available; if any adjustments are necessary, they should be minor. However, to be perfectly safe, you may wish to stay with version 9.2 until the next edition of PSpice for Windows is released. 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 A/D, 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. Of course, OrCAD Corporation deserves special credit for making the OrCAD Lite evaluation version 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. Roy W. Goody