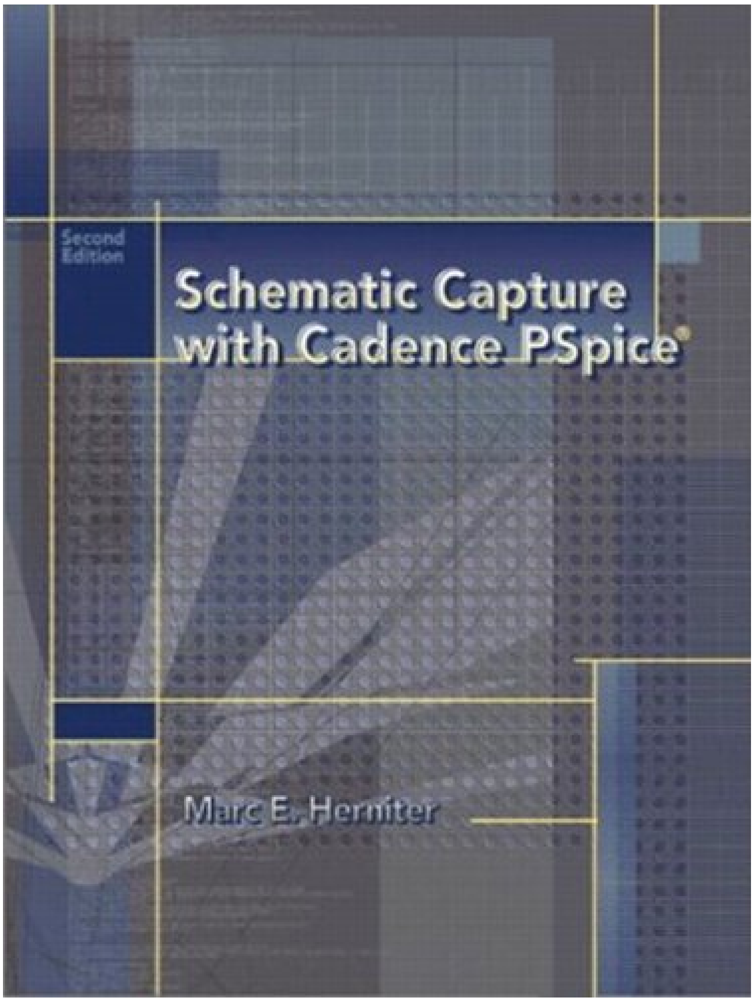
Schematic Capture with Cadence PSpice (2nd Edition)

Marc E. Herniter Ph.D.



#1574509 in Books 2002-05-31Original language:EnglishPDF # 1 10.80 x 1.70 x 8.20l, 3.22 #File Name: 0130484008650 pages | File size: 29.Mb

Marc E. Herniter Ph.D. : Schematic Capture with Cadence PSpice (2nd Edition) before purchasing it in order to gage whether or not it would be worth my time, and all praised Schematic Capture with Cadence PSpice (2nd Edition):

3 of 4 people found the following review helpful. Good and BadBy Michael EdickVery unhappy with this text versus the summary provided on 's site. First, there was no PSPICE-CD included with the text (haven't found the free version from Cadence's website yet either...). Second, text only shows how to operate the Cadence PSPICE program, it isn't a book for anyone who already knows something about SPICE.There is only minimal reference to creating models (assumes all you want to do is download a manufacturer's model), and has no reference to text-editing netlists, models, subcircuits, etc.The book's first-line of the preface says it all, "This manual is designed to show students how to use the PSpice circuit simulation program from Orcad..."I was searching for a book showing how to text-edit .op, .tran, .noise, etc commands, plus I want to mathematically manipulate currents and voltages based on other nets in equation form.

Commands like **bi 2 0 VALUE = {I(V1)*"R(V(2))"}** don't even show up as a possibility in this text.If you're a newbie and want to learn the specific Cadence PSPICE schematic-capture program, then this text is probably adequate (note: I agree with others' complaints about "black-on-black" examples are VERY difficult to read). If you're looking for anything SPICE-related (not program-related), look elsewhere. This will probably be the first thing I've ever had to send back to .0 of 0 people found the following review helpful. Three StarsBy Suhail Hasanout dated book; need a new book to keep up with latest trends0 of 0 people found the following review helpful. Three StarsBy Colin ButlerCan't beat this price!

These manuals show the reader how to use the Cadence/Orcad version of the PSpice circuit simulation program with the Orcad Capture front end. Focusing on a wide range of circuits, they feature a collection of examples that show how to create a circuit, how to run the different analyses, and how to obtain the results from those analyses. Use this URL to access the examples www.pearsoned.com/electronics The topics covered in this book are editing a basic schematic using Orcad/Capture, Probe, DC nodal analysis, DC sweep, AC sweep, transient analysis, creating and modifying models using Orcad/Capture, digital simulations, Monte Carlo analyses, and project management with Orcad/Capture CIS. For electrical engineers, computer engineers, and workers in power electronics, analog electronics, circuit simulation, and project managers.

From the Back CoverThis authoritative book shows students how to use the Pspice circuit simulation program with the schematic capture front end, Capture. It features an abundant collection of examples that demonstrate how to create a circuit, run different analyses, and obtain results from those analyses.Excerpt. Reprinted by permission. All rights reserved.This manual is designed to show students how to use the PSpice circuit simulation program from Orcad with the schematic capture front end, Capture. It is a collection of examples that show students how to create a circuit, how to run the different analyses, and how to obtain the results from those analyses. This manual does not attempt to teach students circuit theory or electronics; that task is left for the main text. Instead, the manual takes the approach of showing students how to simulate many circuits found throughout the engineering curriculum. An example is the DC circuit shown below. It is assumed that the student has been given enough information to completely analyze the circuit. This manual assumes that the student wishes to check his or her answers (or intuition) with this program. The student would construct the circuit as shown in Part 1 and then run either the node voltage analysis in Part 3 or the DC Sweep in Part 4. This circuit is different from the circuits in Parts 3 and 4, but the procedure given in those parts can be applied to the circuit. This manual was designed to be used by students for their entire educational career and beyond. Since the parts are arranged by analysis type, they contain a range of examples from circuits covered in first-semester circuit theory courses to senior-level amplifier and switching circuits. Sections that are too advanced for beginning students may be skipped without loss of continuity. All parts contain both simple circuits and advanced circuits to illustrate the analysis types. Sections do not have to be covered sequentially. Individual examples can be identified that apply to specific courses. However, the following sequence is suggested for first-time users. All beginning users should follow Parts 1 and 2 completely to learn how to draw, print, and save schematics, and how to use Probe, the graphical post-processor, for viewing results. All students should follow some of the examples in Parts 3 and 4 that are relevant to the course and also cover a few of the examples that may apply to earlier courses (if any). The early examples in these parts have the most step-by-step detail of how to use the software. This manual contains examples that apply to courses throughout the engineering curriculum. Introductory circuits classes usually cover DC circuits, AC circuits with phasors, and transient circuits with a single capacitor or inductor and a switch. Examples are given to cover these types of problems. After reviewing the examples in this manual, a student should be able to simulate similar problems. A typical first electronics course may cover transistor biasing, amplifier gain, and amplifier frequency response. Examples of these analyses are also given. Higher-level electronics courses would cover Monte Carlo analysis, Worst Case analysis, and distortion. Examples of these types of analyses are included. Exercises are given at the end of each section. These exercises specify a circuit and give the simulation results. The students are encouraged to work these problems to see if they can obtain the same simulation results. The exercises are intended to give students practice in using the software, not to teach them circuits. Since this software covers such a wide variety of courses, problems are not given. These problems are best left to the instructor or main text. Using PSpice on problems specific to the class material is far more instructional than using it on problems designed to teach PSpice. My philosophy is that PSpice should be used only to verify one's own calculations or intuition. In my classes I assign problems that are worked by hand calculation, simulated with PSpice, and then tested in the lab. The students then compare the measured results to the hand calculations and PSpice simulations. Without hand calculation, it is impossible to know if the PSpice simulations are correct. The book is written as if the instructor were giving a class demonstration on how to use the software. Intermediate windows are shown, and all mouse selections are specified. When I first started teaching the schematic capture version, I brought the students into the computer lab. I gave a lecture using an LCD projection screen and an overhead projector. The students could see the screens projected by the overhead and could follow along using their own computers. It required too much lecture time to cover the wide scope of the Orcad Lite software, so I wrote this manual using the philosophy that the screen captures presented in this manual would be the same as if I were presenting the software in a lecture. The main advantage of the schematic capture front end of PSpice is its ease of use. At first this may not be apparent. When I first started to use Capture, I tested it with a simple three- or four-node circuit. Since I was familiar with writing netlists, it was far easier to write a simple four-line netlist than to search through the many menus of Capture and create a schematic. As I became more proficient at using the program and remembering the standard parts, I could create a schematic faster than I could type a netlist. The schematic version becomes much easier when you use parts with which you are not familiar, such as an . exponential or pulsed voltage source. How many of us can remember the order of the parameters in a pulsed voltage source? Usually you have to look them up in the manual. With Capture, a manual is not necessary. Suppose that you get a part called Vpulse. The parameters of the source are listed in the part's attributes, and the order is not needed since Capture takes care of the order automatically. Another example would be an operational amplifier. If you were describing the circuit using a netlist, you would first have to find the order and number of calling nodes of the op-amp subcircuit. To figure out the calling nodes, you have to look at the library listing that contains the netlist of the op-amp subcircuit. Since the MS-DOS operating system did not allow multitasking, this usually involved exiting PSpice and listing the library. In the schematic capture version, you only need to get the op-amp part. All nodes are shown on the schematic, and ,the correct calling order is not needed. This makes the schematic capture version far simpler to use. Another major advantage of the schematic capture version is that students find drawing circuits .such more interesting than writing netlists. Students tend to explore the schematic capture front end much more than a text-based shell. Since all of the analyses and parts are available on-line as windows, graphics, or help files, students tend to explore the abilities of the program and they don't have to dig through a manual. In-the text-based version of PSpice, students would first come to the instructor rather than look through a manual. There are many other advantages to using the schematic capture version. The enormous popularity of the Microsoft Windows operating system should attest to the ease of use of a graphics-based interface compared to a text-based interface. One such advantage is automatic documentation. When you simulate a circuit, you automatically have a circuit schematic. This schematic can be incorporated into lab notebooks and reports. In a corporate environment, documentation of this type is extremely important. With the Windows operating system, the schematics can be incorporated into written documents using screen captures. This manual is an example of what can be accomplished. The version of Orcad Lite described in this manual is Version 9.2 displayed using Windows 2000. The parts libraries described in this manual have been changed slightly from Orcad's distribution libraries. New parts were added to make the program simpler for beginning students to use. Please note that the libraries contained in this manual are slightly different from the factory distribution libraries.