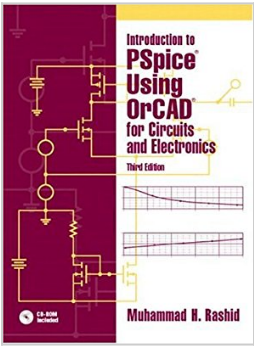

Introduction to PSpice Using OrCAD for Circuits and Electronics (3rd Edition)

By Muhammad H. Rashid

Introduction to PSpice Using OrCAD for Circuits and Electronics (3rd Edition) By Muhammad H. Rashid

For second and third year Electrical Engineering courses in Electronics, Circuit Analysis, and Circuit Simulation. Implementing the industry-standard software, this book can be used as a textbook for teaching the simulation of electronics and electrical circuits through SPICE, PSpice A_D, Windows-based PSpice Schematics, or Orcad Capture. Covering topics in basic circuits and electronics, it could also be used as a supplement to books on basic circuits and/or electronics.

Introduction to PSpice Using OrCAD for Circuits and Electronics (3rd Edition)

By Muhammad H. Rashid

Introduction to PSpice Using OrCAD for Circuits and Electronics (3rd Edition) By Muhammad H. Rashid

For second and third year Electrical Engineering courses in Electronics, Circuit Analysis, and Circuit Simulation. Implementing the industry-standard software, this book can be used as a textbook for teaching the simulation of electronics and electrical circuits through SPICE, PSpice A_D, Windows-based PSpice Schematics, or Orcad Capture. Covering topics in basic circuits and electronics, it could also be used as a supplement to books on basic circuits and/or electronics.

Introduction to PSpice Using OrCAD for Circuits and Electronics (3rd Edition) By Muhammad H. Rashid Bibliography

- Sales Rank: #1342311 in Books
- Published on: 2003-09-07
- Original language: English
- Number of items: 1
- Dimensions: 9.16" h x .74" w x 6.98" l, 1.65 pounds
- Binding: Paperback
- 480 pages

Download and Read Free Online Introduction to PSpice Using OrCAD for Circuits and Electronics (3rd Edition) By Muhammad H. Rashid

Editorial Review

From the Back Cover

This widely used book uses a top-down approach to introduce readers to the SPICE simulator. It begins by describing techniques for simulating circuits, then presents the various SPICE and OrCAD commands and their applications to electrical and electronic circuits. Lavishly illustrated, this new edition includes even more hands-on exercises, suggestions, sample problems, and circuit models of actual devices. It is an ideal supplement for courses in electric or electronic circuitry and is also a solid professional reference.

About the Author

Muhammad H. Rashid received the B.Sc. degree in electrical engineering from the Bangladesh University of Engineering and Technology and the M.Sc. and Ph.D. degrees from the University of Birmingham, UK.

Currently, he is a Professor of electrical engineering with the University of Florida and the Director of the OF/UWF Joint Program in Electrical and Computer Engineering. Previously, he was a Professor of electrical engineering and the Chair of the Engineering Department at Indiana University-Purdue University at Fort Wayne. In addition, he was a Visiting Assistant Professor of electrical engineering at the University of Connecticut, Associate Professor of electrical engineering at Concordia University (Montreal, Canada), Professor of electrical engineering at Purdue University, Calumet, and Visiting Professor of electrical engineering at King Fahd University of Petroleum and Minerals, Saudi Arabia. He has also been employed as a design and development engineer with Brush Electrical Machines Ltd. (UK), as a Research Engineer with Lucas Group Research Centre (UK), and as a Lecturer and Head of Control Engineering Department at the Higher Institute of Electronics (Malta). He is actively involved in teaching, researching, and lecturing in power electronics. He has published 14 books and more than 100 technical papers. His books have been adopted as textbooks all over the world. His book *Power Electronics* has been translated into Spanish, Portuguese, Indonesian, Korean and Persian. His book *Microelectronics* has been translated into Spanish in Mexico and Spain. He has had many invitations from foreign governments and agencies to be a keynote lecturer and consultant, from foreign universities to serve as an external Ph.D. examiner, and from funding agencies to serve as a research proposal reviewer. His contributions in education have been recognized by foreign governments and agencies. He has previously lectured and consulted for NATO for Turkey in 1994, UNDP for Bangladesh in 1989 and 1994, Saudi Arabia in 1993, Pakistan in 1993, Malaysia in 1995 and 2002, and Bangkok in 2002, and has been invited by foreign universities in Australia, Canada, Hong Kong, India, Malaysia, Singapore to serve as an external examiner for undergraduate, master's and Ph.D. degree examinations, by funding agencies in Australia, Canada, United States, and Hong Kong to review research proposals, and by U.S. and foreign universities to evaluate promotion cases for professorship. He has previously authored seven books published by Prentice Hall: *Power Electronics–Circuits, Devices, and Applications* (1988, 2/e 1993), *SPICE For Power Electronics* (1993), *SPICE for Circuits and Electronics Using Pspice* (1990, 2/e 1995), *Electromechanical and Electrical Machinery* (1986), and *Engineering Design for Electrical Engineers* (1990). He has authored five IEEE self-study guides: *Self-Study Guide on Fundamentals of Power Electronics, Power Electronics Laboratory Using PSpice, Selected Readings on SPICE Simulation of Power Electronics,* and *Selected Readings on Power Electronics* (IEEE Press, 1996) and *Microelectronics Laboratory Using Electronics Workbench* (IEEE Press, 2000). He also wrote two books: *Electronic Circuit Design using Electronics Workbench* (January 1998), and *Microelectronic Circuits*

Analysis and Design (April 1999) by PWS Publishing). He is editor of *Power Electronics Handbook* published by Academic Press, 2001.

Dr. Rashid is a registered Professional Engineer in the Province of Ontario (Canada), a registered Chartered Engineer (UK), a Fellow of the Institution of Electrical Engineers (IEE, UK) and a Fellow of the Institute of Electrical and Electronics Engineers (IEEE, USA). He was elected as an IEEE Fellow with the citation "*Leadership in power electronics education and contributions to the analysis and design methodologies of solid-state power converters.*" He was the recipient of the *1991 Outstanding Engineer Award* from The Institute of Electrical and Electronics Engineers (IEEE). He received the 2002 IEEE Educational Activity Award (EAB) Meritorious Achievement Award in Continuing Education with the citation "*for contributions to the design and delivery of continuing education in power electronics and computer-aided-simulation*". He was also an ABET program evaluator for electrical engineering from 1995 to 2000 and he is currently an engineering evaluator for the Southern Association of Colleges and Schools (SACS, USA). He has been elected as an IEEE-Industry Applications Society (IAS) Distinguished Lecturer. He is the Editor-in-Chief of the *Power Electronics and Applications Series*, published by CRC Press.

The Engineering Accreditation Commission of the Accreditation Board for Engineering and Technology (EAC/ ABET) requirements specify the integration of computeraided analysis and design in electrical and computer engineering curricula. SPICE is very popular software for analyzing electrical and electronic circuits. The MicroSim Corporation first introduced the PSpice simulator, which can run on personal computers (PCs). It is similar to the University of California (UC) Berkeley SPICE. The student version of PSpice, which is available free to students, is ideal for classroom use and for assignments requiring computer-aided simulation and analysis. PSpice widens the scope for the integration of computer-aided simulation to circuits and electronics courses for undergraduate and graduate students.

It may not be possible to add a one-credit-hour course on SPICE to integrate computer-aided analysis in circuits and electronics courses. However, students need some basic knowledge of how to use SPICE. They are constantly under pressure with course loads and do not always have the free time to read the details of SPICE, PSpice, or OrCAD from manuals and books of a general nature.

This book is the outcome of the author's experience in integrating SPICE in circuits and electronics courses at the 200-, 300-, or 400-level. The objective is to introduce the SPICE simulator to the electrical and computer engineering curriculum at the sophomore or junior level with a minimum amount of time and effort. This book requires no prior knowledge of the SPICE simulator. A course on basic circuits should be a prerequisite or co-requisite. Once the student develops an interest in and an appreciation for the applications of circuit simulators like SPICE, he or she can move on to more advanced materials for the full utilization of SPICE, PSpice, or OrCAD in solving complex circuits and systems.

This book can be divided into six parts:

(1) introduction to SPICE simulation—Chapters 1 and 2;
(2) DC, transient and AC circuit analysis—Chapters 3, 4, and 5;
(3) advanced SPICE commands and analysis—Chapter 6;
(4) semiconductor devices modeling and circuits—Chapters 7, 8, and 9;
(5) op-amp circuits and differential amplifiers—Chapter 10, and
(6) difficulties—Chapter 11.

Chapters 8, 9, and 10 describe the simple equivalent circuits of transistors and opamps, which are commonly used in analyzing electronic circuits. Although SPICE generates the parameters of complex transistor models, analysis with a simple circuit model exposes the students to the mechanism of computation by,SPICE .MODEL commands. This approach has the advantage that the students can compare the results, which are obtained in a classroom environment with the simple circuit models of devices, to those obtained by using complex SPICE models.

The commands, models, and examples that are described for PSpice are also applicable to UC Berkeley SPICE with minor modifications. The changes for running a PSpice circuit file on SPICE and vice-versa are discussed in Chapter 11. The filenames for the circuit files in this book are named using all uppercase so that the same file can be run on either the PSpice or the SPICE simulator.

Probe is a graphics post-processor and is very useful in plotting the results of simulation, especially with the capability of arithmetic operation. It can be used to plot impedance, power, and so on. Once students have experience programming in PSpice, they will really appreciate the advantages of .Probe command. *Probe* is an option on PSpice, available with the student version. Running *Probe* does not require a math coprocessor. The students can also get the normal printer output or printer plotting. The prints and plots are very helpful to the students in their theoretical understanding and in making judgments on the merits of a circuit and its characteristics.

This book can be used as a textbook on SPICE with a course on basic circuits being the prerequisite or co-requisite. It can also be used as a supplement to any standard textbook on basic circuits or electronics. In the latter case, the following sequence is recommended for the integration of SPICE at the basic circuits level of the curriculum:

1. As a supplement to a basic circuits course with three hours of lectures (or equivalent lab hours) and self-study assignments from Chapters 1 to 6. Starting from Chapter 2, the students should work hands-on with PCs.
2. In an electronics course it should continue to be used, with two hours of lectures (or equivalent lab hours) and self-study assignments from Chapters 6 to 10.

For integrating SPICE at the electronics level, three hours of lectures (or equivalent lab hours) are recommended on Chapters 1 to 6. Chapters 7 to 10 could be left for self-study assignments. From the author's experience in the class, it has been observed that after three lectures of 50 minutes duration, all students could solve assignments independently without any difficulty. The class could progress in a normal manner with one assignment per week on electronic circuits simulation and analysis with SPICE. Although the materials of this book have been tested in a basic circuits course for engineering students and in two electronics courses for electrical and computer engineering students, the book is also recommended for electrical engineering technology students.

Users Review

From reader reviews:

Pamela Dudley:

What do you think about book? It is just for students since they're still students or the idea for all people in the world, exactly what the best subject for that? Just you can be answered for that question above. Every person has various personality and hobby for every single other. Don't to be compelled someone or something that they don't wish do that. You must know how great as well as important the book Introduction to PSpice Using OrCAD for Circuits and Electronics (3rd Edition). All type of book would you see on many methods. You can look for the internet resources or other social media.

Thomas Welty:

Spent a free time and energy to be fun activity to do! A lot of people spent their leisure time with their family, or their very own friends. Usually they doing activity like watching television, gonna beach, or picnic within the park. They actually doing same task every week. Do you feel it? Do you wish to something different to fill your own personal free time/ holiday? May be reading a book could be option to fill your no cost time/ holiday. The first thing you ask may be what kinds of publication that you should read. If you want to test look for book, may be the guide untitled Introduction to PSpice Using OrCAD for Circuits and Electronics (3rd Edition) can be great book to read. May be it can be best activity to you.

Brooke Callender:

Playing with family in a very park, coming to see the coastal world or hanging out with friends is thing that usually you will have done when you have spare time, subsequently why you don't try issue that really opposite from that. One particular activity that make you not experience tired but still relaxing, trilling like on roller coaster you already been ride on and with addition info. Even you love Introduction to PSpice Using OrCAD for Circuits and Electronics (3rd Edition), you could enjoy both. It is very good combination right, you still desire to miss it? What kind of hang type is it? Oh can happen its mind hangout folks. What? Still don't get it, oh come on its known as reading friends.

Juli Gadberry:

What is your hobby? Have you heard which question when you got scholars? We believe that that question was given by teacher for their students. Many kinds of hobby, Every individual has different hobby. Therefore you know that little person just like reading or as studying become their hobby. You have to know that reading is very important and book as to be the point. Book is important thing to incorporate you knowledge, except your personal teacher or lecturer. You find good news or update in relation to something by book. A substantial number of sorts of books that can you choose to use be your object. One of them are these claims Introduction to PSpice Using OrCAD for Circuits and Electronics (3rd Edition).

Read Introduction to PSpice Using OrCAD for Circuits and Electronics (3rd Edition) By Muhammad H. Rashid for online ebook

Online Introduction to PSpice Using OrCAD for Circuits and Electronics (3rd Edition) By Muhammad H. Rashid ebook PDF download

Introduction to PSpice Using OrCAD for Circuits and Electronics (3rd Edition) By Muhammad H. Rashid Doc

Introduction to PSpice Using OrCAD for Circuits and Electronics (3rd Edition) By Muhammad H. Rashid Mobipocket

Introduction to PSpice Using OrCAD for Circuits and Electronics (3rd Edition) By Muhammad H. Rashid EPub

DG7B2N438ST: Introduction to PSpice Using OrCAD for Circuits and Electronics (3rd Edition) By Muhammad H. Rashid