


[DOWNLOAD](#)


Introduction to PSpice® Using OrCAD® for Circuits and Electronics, Third Edition

By Muhammad H. Rashid

PHI Learning 0. Softcover. Book Condition: New. 3rd edition. The aim of this widely used book, now in its third edition, is to introduce readers to the PSpice version of the SPICE simulator. 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). This book is primarily intended as a text for the undergraduate core course on environmental science and for undergraduate honours courses in ecology, environmental biology as well as for postgraduate courses in environmental biology, ecology, and pollution ecology. In addition, the book would also be useful for students appearing in different competitive examinations. This book is the outcome of the author`s experience in integrating SPICE in circuits and electronic courses for undergraduate electrical, electronics and computer engineering students. It can be used either as a textbook on SPICE or as a supplement to any standard textbook on basic circuits or electronics. The book begins by describing techniques for simulating circuits, then presents the various SPICE and OrCAD commands and their applications to electrical and electronic circuits. The third edition includes more hands-on exercises, suggestions, sample problems, and circuit models of actual...



[READ ONLINE](#)
[1.1 MB]

Reviews

It becomes an incredible book that we actually have possibly study. It really is rally exciting through studying period of time. I am very easily could get a satisfaction of reading through a written book.

-- Gianni Hoppe

A really awesome pdf with perfect and lucid reasons. It is actually rally fascinating through reading period of time. Your lifestyle period will probably be transform as soon as you total looking over this ebook.

-- Alford Kihn