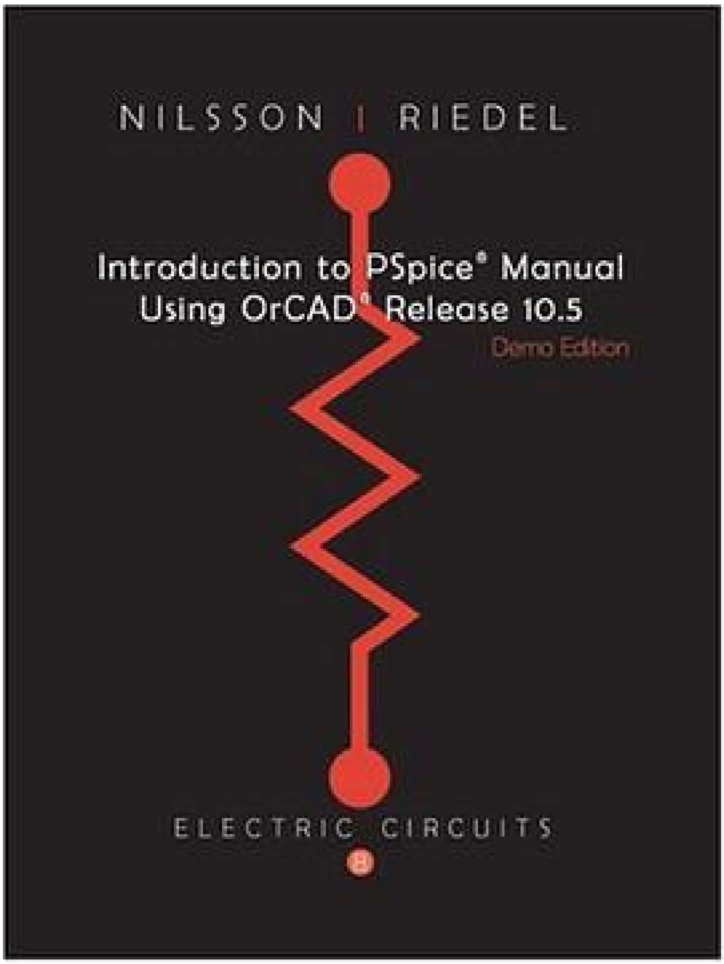
Introduction to PSpice for Electric Circuits



Introduction to PSpice for Electric Circuits_下载链接1_

著者:James W. Nilsson

出版者:Pearson Education, Inc.

出版时间:2007-08-28

装帧:Paperback

isbn:9780132448390

Computer tools can assist students in the learning process by providing a visual representation of a circuit's behavior, validating a calculated solution, reducing the computational burden of more complex circuits, and iterating toward a desired solution using parameter variation. This computational support is often invaluable in the design process. Updated for PSpice using OrCAD release 10.5, this manual focuses on three things: - Learning to draw and simulate linear circuits using PSpice - Constructing circuit models of basic devices such as op amps - Learning to challenge computer output data as a means of reinforcing confidence in simulation PSpice software may be used to solve many of Nilsson & Riedel's Electric Circuits, 8e Assessment Problems and Chapter Problems but the manual was designed as a supplement to stand on its own as an instructional unit.

作者介绍:

目录:

Introduction to PSpice for Electric Circuits_下载链接1_

标签

评论

Introduction to PSpice for Electric Circuits_下载链接1_

书评

Introduction to PSpice for Electric Circuits_下载链接1_