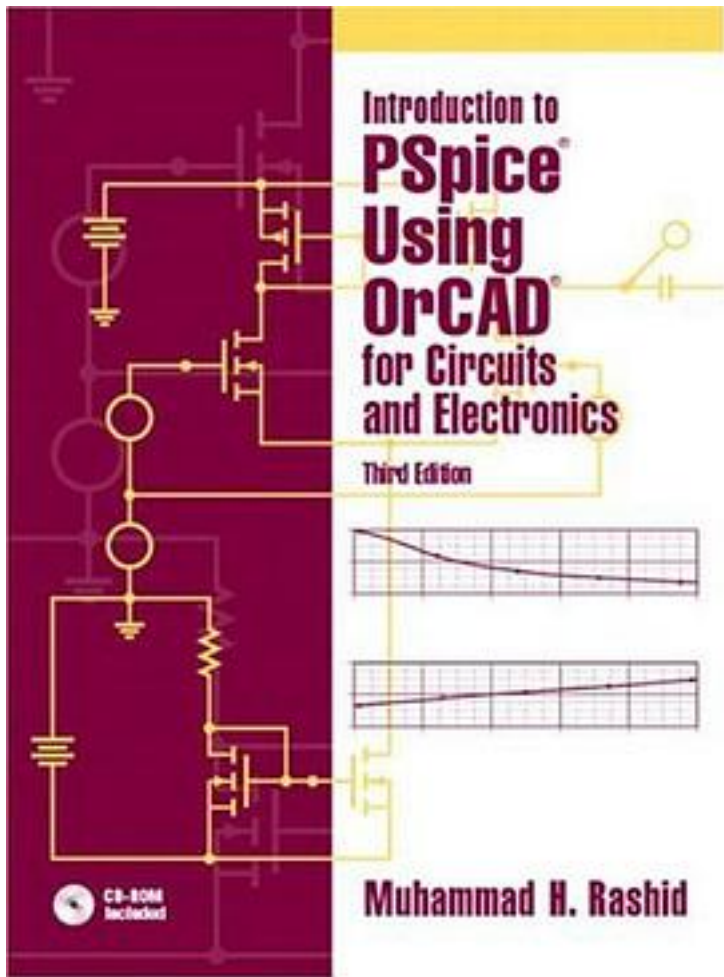# Introduction to PSpice Using OrCAD for Circuits and Electronics

[Introduction to PSpice Using OrCAD for Circuits and Electronics_下载链接1_](#)

著者:Rashid, M.H.

出版者:Prentice Hall

出版时间:2003-8

装帧:Pap

isbn:9780131019881

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.

作者介绍:

目录:

[Introduction to PSpice Using OrCAD for Circuits and Electronics_下载链接1_](#)

标签

评论

[Introduction to PSpice Using OrCAD for Circuits and Electronics_下载链接1_](#)

书评

[Introduction to PSpice Using OrCAD for Circuits and Electronics_下载链接1_](#)