Protel2004 practical tutorial - schematic and PCB design (3rd Edition)(Chinese Edition)

By BEN SHE

DOWNLOAD PDF

paperback. Book Condition: New. Ship out in 2 business day, And Fast shipping, Free Tracking number will be provided after the shipment.Paperback. Pub Date :2012-11-01 Pages: 301 Publisher: Electronic Industry Press Information title: Protel 2004 practical tutorial - schematic and PCB design (3) List Price: $ 39.00 Author: Publisher: Electronic Industrial Press Publication Date :2012-11-1ISBN: 9787121189050 Words: 505.000 yds: 301 Edition: 1 Binding: Paperback: 16 Weight: Editor's executive summary book the main line of a typical application example to introduce Altium company Protel2004 electronic design automation (EDA) software to use. The book gave a detailed schematic design. electronic circuit simulation. and printed circuit board design three sections Protel2004 software. The book is 14 chapters. which Summary Chapter 1 Protel2004 system. Chapter 2 to Chapter 7 for the schematic design part. Chapter 8 presents a schematic level design methods. Chapter 9. for the electronic circuit simulation part 10 part of the Chapter to Chapter 14 of the printed circuit board design. 1.1 protel protel system of Contents Chapter 1. the composition and characteristics 1.1.1 protel characteristics 1.1.2 protel 1.2 protel Runtime Environment 1.3 protel the interface 1.3.1 protel's main menu bar 1.3.2 protel Home 1.3.3 protel work panel 1.4 panel operation...

READ ONLINE
[3.13 MB]

Reviews

A really awesome pdf with lucid and perfect information. It is loaded with wisdom and knowledge I am just effortlessly could get a satisfaction of reading a composed book.
-- Claudine Jerde

A whole new e book with a new perspective. I could comprehended almost everything using this written e ebook. I am very happy to inform you that here is the greatest ebook i have read in my very own life and may be he best publication for ever.
-- Dee Halvorson