



DOWNLOAD



Protel PCB Design Tutorial (application-oriented quality of higher education teaching materials)

By GAN DENG DAI

paperback. Book Condition: New. Ship out in 2 business day, And Fast shipping, Free Tracking number will be provided after the shipment. Pages Number: 268 Publisher: Aviation Industry Pub. Date :2011-05-01 version 1. Protel is the best one of the circuit board design software. Gan Deng Dai's Protel PCB Design Tutorial project using the latest teaching methods and Combination of traditional teaching methods. combined with the practical use Protel. step by step description of the Protel DXP 2004 Quick Start. schematic drawing based on the production schematic library components. schematic drawing to improve and post-processing. familiar with PCB design environment map. printed circuit board (PCB) design initially. with the copper circuit board layout. PCB components. packaging design. PCB post-processing and circuit simulation and so on. Protel PCB Design Tutorial case-rich. novel structure. clear. focused. for institutions of higher learning electronics. electrical. communications classes. machinery and electronic and related professional students to use. but also for personnel from the electronic circuit design learning. for reference. Contents: Project a Protel DXP 2004 Getting Started task an acquaintance Protel DXP 2004 First. install Protel DXP 2004 Second. start Protel DXP 2004 Third. the activation of Protel DXP 2004 4. set the task of two...



READ ONLINE

[2.3 MB]

Reviews

Absolutely essential study publication. It usually fails to expense an excessive amount of. Your lifestyle period will probably be transform when you full looking at this publication.

-- **Ms. Allene Conroy**

The publication is easy in read better to understand. It is writter in basic words and phrases rather than hard to understand. You wont truly feel monotony at anytime of your respective time (that's what catalogues are for about if you question me).

-- **Kaya Rippin**