Make Your Own PCBs with Eagle: From Schematic Designs to Finished Boards

By Simon Monk

McGraw-Hill/TAB Electronics. Paperback. Book Condition: New. Paperback. 304 pages. Dimensions: 9.2in. x 7.3in. x 0.6in.Design custom printed circuit boards with EAGLE Learn how to make double-sided professional-quality PCBs from the ground up using EAGLE--the powerful, flexible design software. In this step-by-step guide, electronics guru Simon Monk leads you through the process of designing a schematic, transforming it into a PCB layout, and submitting standard Gerber files to a manufacturing service to create your finished board. Filled with detailed illustrations, photos, and screenshots, Make Your Own PCBs with EAGLE features downloadable example projects so you can get started right away. Install EAGLE Light Edition and discover the views and screens that make up an EAGLE project Create the schematic and board files for a simple LED project Find the right components and libraries for your projects Work with the Schematic Editor Lay out PCBs with through-hole components and with surface mount technology Build a sound level meter with a small amplifier and ten LEDs Generate Gerber design files to submit for fabrication Solder through-hole PCBs and SMD boards Design a plug-in Arduino shield Build a Raspberry Pi expansion board Automate repetitive tasks using scripts and User Language Programs Create your own...

READ ONLINE
[6.09 MB]

Reviews

This pdf can be worthy of a read, and much better than other. I am quite late in start reading this one, but better then never. Its been printed in an remarkably easy way which is merely following i finished reading this book by which basically changed me, alter the way i think.
-- Nedra Kiehn

Without doubt, this is the very best work by any writer. Indeed, it can be play, still an amazing and interesting literature. I am just very easily can get a pleasure of reading through a written pdf.
-- Alda Barton