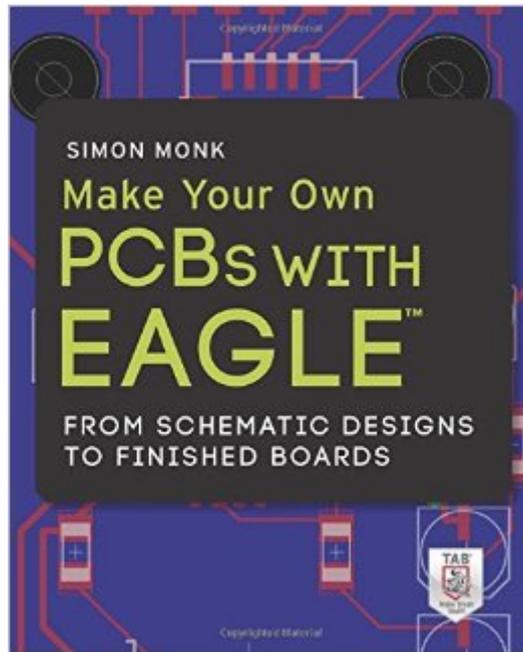


The book was found

# Make Your Own PCBs With EAGLE: From Schematic Designs To Finished Boards



## Synopsis

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 libraries and parts and modify existing components

## Book Information

Paperback: 272 pages

Publisher: McGraw-Hill Education TAB; 1 edition (June 12, 2014)

Language: English

ISBN-10: 0071819258

ISBN-13: 978-0071819251

Product Dimensions: 7.4 x 0.6 x 9.2 inches

Shipping Weight: 1.2 pounds (View shipping rates and policies)

Average Customer Review: 4.5 out of 5 stars [See all reviews](#) (58 customer reviews)

Best Sellers Rank: #36,850 in Books (See Top 100 in Books) #7 in [Books > Engineering & Transportation > Engineering > Electrical & Electronics > Circuits > Design](#) #13 in [Books > Science & Math > Experiments, Instruments & Measurement > Experiments & Projects](#) #15 in [Books > Engineering & Transportation > Engineering > Electrical & Electronics > Electronics > Microelectronics](#)

## Customer Reviews

Wow! I've learned to do something that I've wanted to do for a long time - make my own PC boards. I've been an electronics experimenter and radio amateur for decades and never made my own

boards. I actually bought a set of chemicals once and did nothing with them. Even with programs like Eagle, I was still unsure, and the instructions were sometimes ambiguous. So, to me, Dr. Monk has again done what he does so well. He's taken a complex subject and clarified it to the point where an amateur like me feels perfectly comfortable in giving it a try. That's my favorite thing about this book: It's clear step-by-step instructions and illustrations of how to do something new for the first time. Bottom Line: If you've thought about making your own PC boards and had any hesitancy at all, you can now dive in. Get a copy of this book and go for it!

Eagle is a wonder very powerful program for making electronic schematics and transforming that information into a finished printed circuit board design. Subsequently, that design can be sent to a vendor who specializes in manufacturing the finished PCB. The whole process is much less expensive than producing your own PCB due to the large amount of equipment and expertise required. The problem with Eagle for me in the past is that the documentation that goes with the program is difficult to understand. I was going around in circles for awhile until I read Simon Monk's great new book on the subject. He makes it very simple to use Eagle, either to make schematics or to make PCB designs. The writing and graphics are wonderfully clear and shed light on the whole process. My questions have been answered and at last I am getting somewhere with Eagle. I recommend the book highly for beginners and advanced users alike.

Simon Monk has written several great books that teach about electrical components, simple circuits, and building your own projects. This new book takes that training to the next level. Chapter one begins with an intro to PCB's and helps you download EAGLE for the operating system of your choice. Dr. Monk then takes you through the entire process of making schematics, board layouts, and Gerber files. He explains each step of the process in detail and leads you through examples that are easy to understand but quite challenging as well. By the end of the book, you will have a very good understanding of the power of the EAGLE program and be able to create your own designs to submit for fabrication through a PCB service. He finishes the book with an example of an Arduino shield and a Raspberry Pi expansion board. For programmers, he introduces how to use scripts and user language programs with EAGLE. This book is a great resource!

It's ok if your just getting started with Eaglecad (like me). But here's the deal. At first Eaglecad seems pretty intimidating as it's loaded with stuff - easy to get overwhelmed. But once you get going (after a good say 20 hrs with it), it's actually easy to use albeit the user interface is a wonderful case

study on how not to design software. In a word just crap, it's obvious the designers have never studied Human Factors, or just ignored them and did their own ungodly mess of an interface, but I digress. Humans are an amazing bunch, even with piss poor user interfaces, we adapt and get with the program. Now back to the book: it's well written, easily understandable and will definitely get the new person up and running. My only complaint - and it's minor - is that it just doesn't cover in detail other ways of doing tasks, only one way. But like I said in 20 hrs you will need to learn other techniques. For example, Mr. Monk does a nice job of explaining ground pours. However, what if I have a mixed dual layer board with components on both sides of the board. You don't need a PhD in Physics to see the described technique isn't going to work. So, what's the work around? Nice follow up with be "Advanced EagleCad....". I would recommend the book for it's cost clarity of writing, worth a read.

I bought this book along with "Eagle V6" by Mitchell Duncan. You can read my review on that book as it also applies to this book. Basically, Simon Monk's book, along with Eagle V6 by Mitchell Duncan are the only two books you need to learn and use Eagle. Each book separately lacks certain information, but together they include everything you'll need for tutorial and reference purposes.

I own most of Dr. Monk's books and appreciate his clear writing style and depth. I worked through the bulk of the book over a couple of days after receiving my order and am happy to report that I am now reasonably fluent with Eagle. 20 bucks is a modest price to pay for the many hours I would have lost mucking around on the internet to try to develop a similar level of fluency. I only have two small quibbles: 1. There seems to be a step missing in the section on importing a new libraries. Unlike the Arduino environment, you evidently need to visit the Library | Use menu for the parts to show up in the schematic view after copying them to the appropriate directory. I assume this may be a change with the newest version of Eagle. 2. Given the highly-detailed information on using Eagle, it would have been nice to have more detail relating to making PCBs at home. I've made a few PCBs and would have appreciated following a more detailed walk-through involving the two DIY approaches he discusses in Chapter 6. There are only three sentences devoted to toner transfer and just a couple of paragraphs relating to photo etching. Even a list of vendors and products would be a good start in learning how to do an occasional one off at home. Overall a great book and well worth the modest price if you want to get up to speed with Eagle. Can't wait to order my first PCBs!

I started reading this book earlier this week and have already been able to design my own professional looking (to me, at least!) printed circuit board. It isn't as hard as I thought it would be and this is going to be a very helpful new skill for me to help with my electronics/robotics hobby. Also, having the PCBs printed is actually very cheap too. The book is full of useful resources about where and how to have them made.

[Download to continue reading...](#)

Make Your Own PCBs with EAGLE: From Schematic Designs to Finished Boards Inside a Bald Eagle's Nest: A Photographic Journey Through the American Bald Eagle Nesting Season Rich Dad Advisor's Series: Own Your Own Corporation: Why the Rich Own Their Own Companies and Everyone Else Works for Them (Rich Dad's Advisors) 9-Patch Pizzazz: Fast, Fun & Finished in a Day Pine Needle Basketry: From Forest Floor to Finished Project The Whole Craft of Spinning: From the Raw Material to the Finished Yarn Light It, Shoot It, Retouch It: Learn Step by Step How to Go from Empty Studio to Finished Image (Voices That Matter) Encyclopedia Of Mosaic Techniques: A Step-by-step Visual Directory, With An Inspirational Gallery Of Finished Works (Encyclopedia of Art Techniques) Tropical Insects: 50 Incredible Insect and Beautiful Butterflies Designs to Raise up Your Creativity (butterflies and flowers, insect designs, butterfly designs) Boards That Make a Difference: A New Design for Leadership in Nonprofit and Public Organizations Event Planning: Plan Events Like a Professional, Impress Your Clients and be Your Own Boss in 12 Simple Steps (event planning, experience, organise, manage, ... be your own boss, work from home Book 4) The Fundraising Habits of Supremely Successful Boards: A 59-Minute Guide to Assuring Your Organization's Future Make Your Own Movie Machine: Build a Paper Zoetrope and Learn to Make Animation Cells Start and Run Your Own Record Label, Third Edition (Start & Run Your Own Record Label) Kidz Bop: Be a Pop Star!: Start Your Own Band, Book Your Own Gigs, and Become a Rock and Roll Phenom! (Kidz Bop (Paperback)) Build Your Own Wi-Fi Network (Build Your Own...(McGraw)) Start Your Own Law Practice: A Guide to All the Things They Don't Teach in Law School about Starting Your Own Firm (Open for Business) How to Start Your Own Real Estate Photography Business!: A Step-by-Step Guide to Show You How to Begin Your Own Real Estate Photography Business in 14 ... for real estate, photographing houses) Food Truck Business: How to Start Your Own Food Truck While Growing & Succeeding as Your Own Boss More for Eagle Eyes

[Dmca](#)