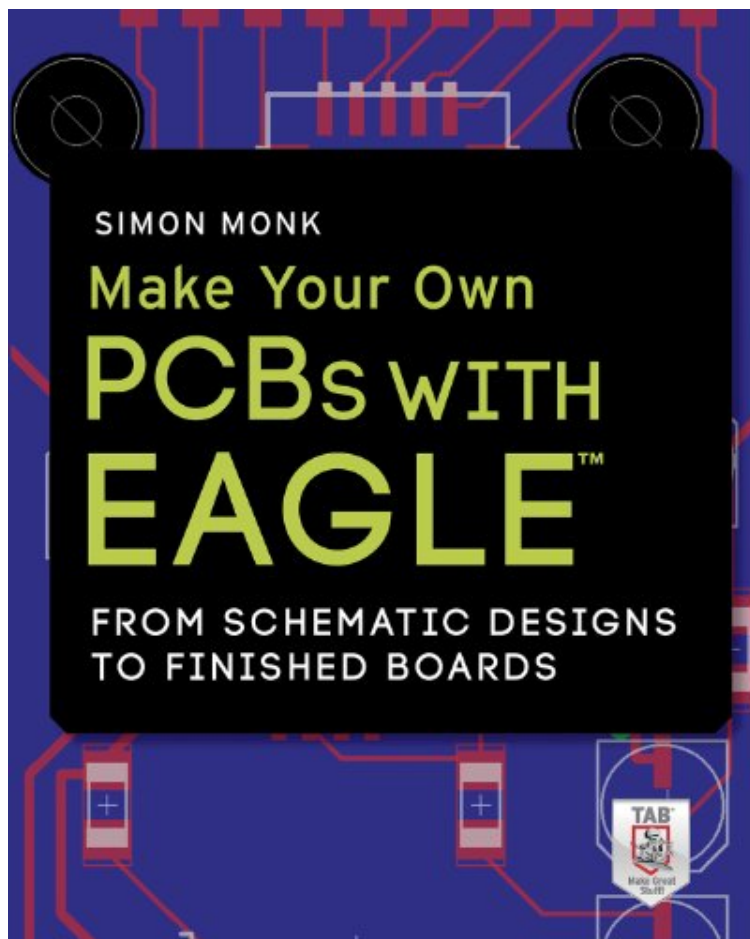


(Online library) File size: 18.Mb

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



*Par Simon Monk*

*DOC / \*audiobook / ebooks / Download  
PDF / ePub*

Dtails sur le produit Rang parmi les ventes : #178811 dans eBooksPubli le: 2014-08-05Sorti le: 2014-08-05Format: Ebook Kindle

(Online library) Make Your Own PCBs with EAGLE: From Schematic Designs to Finished Boards

**Par Simon Monk : Make Your Own PCBs with EAGLE: From Schematic Designs to Finished Boards** before purchasing it in order to gage whether or not it would be worth my time, and all praised Make Your Own PCBs with EAGLE: From Schematic Designs to Finished Boards:

 [Download](#)

 [Read Online](#)

## Description :

Prsentation de l'diteurDesign custom printed circuit boards with EAGLELearn 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 projectCreate the schematic and board files for a simple LED projectFind the right components and libraries for your projectsWork with the Schematic EditorLay out PCBs with through-hole components and with surface mount technologyBuild a sound level meter with a small amplifier and ten LEDsGenerate Gerber design files to submit for fabricationSolder through-hole PCBs and SMD boardsDesign a plug-in Arduino shieldBuild a Raspberry Pi expansion boardAutomate repetitive tasks using scripts and User Language ProgramsCreate your own

libraries and parts and modify existing components  
Prsentation de l'diteur  
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