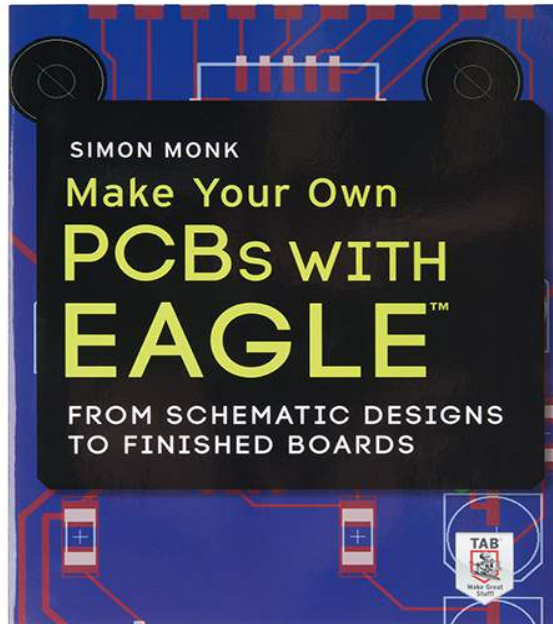# Make Your Own PCBs with Eagle

BOK-13997

Description: 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 10 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

Info:

Author: Simon Monk

Publisher: McGraw-Hill / TAB Electronics

Paperback: 272 pages

SparkFun Electronics ®