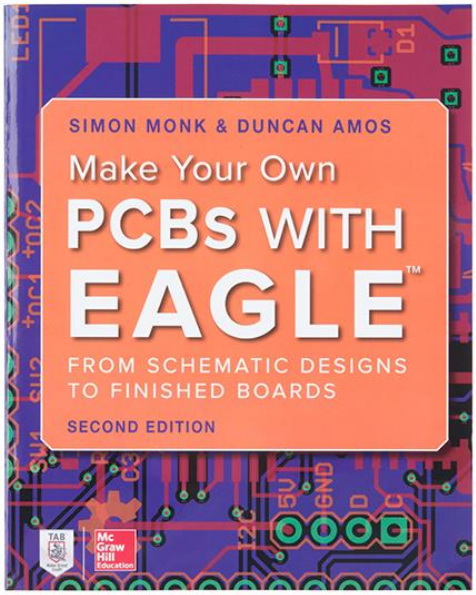
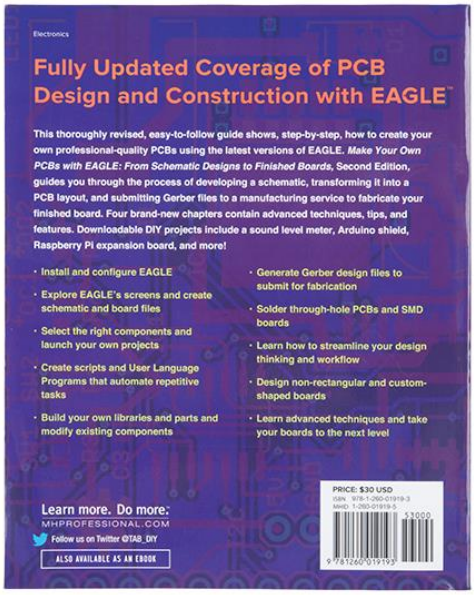




Make Your Own PCBs with Eagle - Second Edition

BOK-14861

his thoroughly revised, easy-to-follow guide shows, step-by-step, how to create your own professional-quality PCBs using the latest versions of EAGLE. *Make Your Own PCBs with Eagle: From Schematic Designs to Finishd Boards*, Second Edition, guides you through the process of developing a schematic , transforming it into a PCB layout, and submitting Gerber files to a manufacturing service to fabricate your finished board. Four brand-new chapters contain advanced techniques, tips, and features. Downloadable DIY projects include a sound level meter, Arduino shield, Raspberry Pi expansion board, and more!

- Install and configure EAGLE.
- Explore EAGLE’s screens and create schematic and board files.
- Select the right components and launch your own projects.
- Create scripts and User Language Programs that automate repetitive tasks.
- Build your own libraries and parts and modify existing components.
- Generate Gerber design files to submit for fabrication.
- Solder through-hole PCBs and SMD boards.
- Learn how to streamline your design thinking and workflow.
- Design non-rectangular and custom-shaped boards.
- Learn advanced techniques and take your boards to the next level.

Info:

- Author: Simon Monk & Duncan Amos
- Publisher: McGraw-Hill / TAB Electronics
- Paperback: 329 pages
- ISBN: 978-1-260-01919-3

