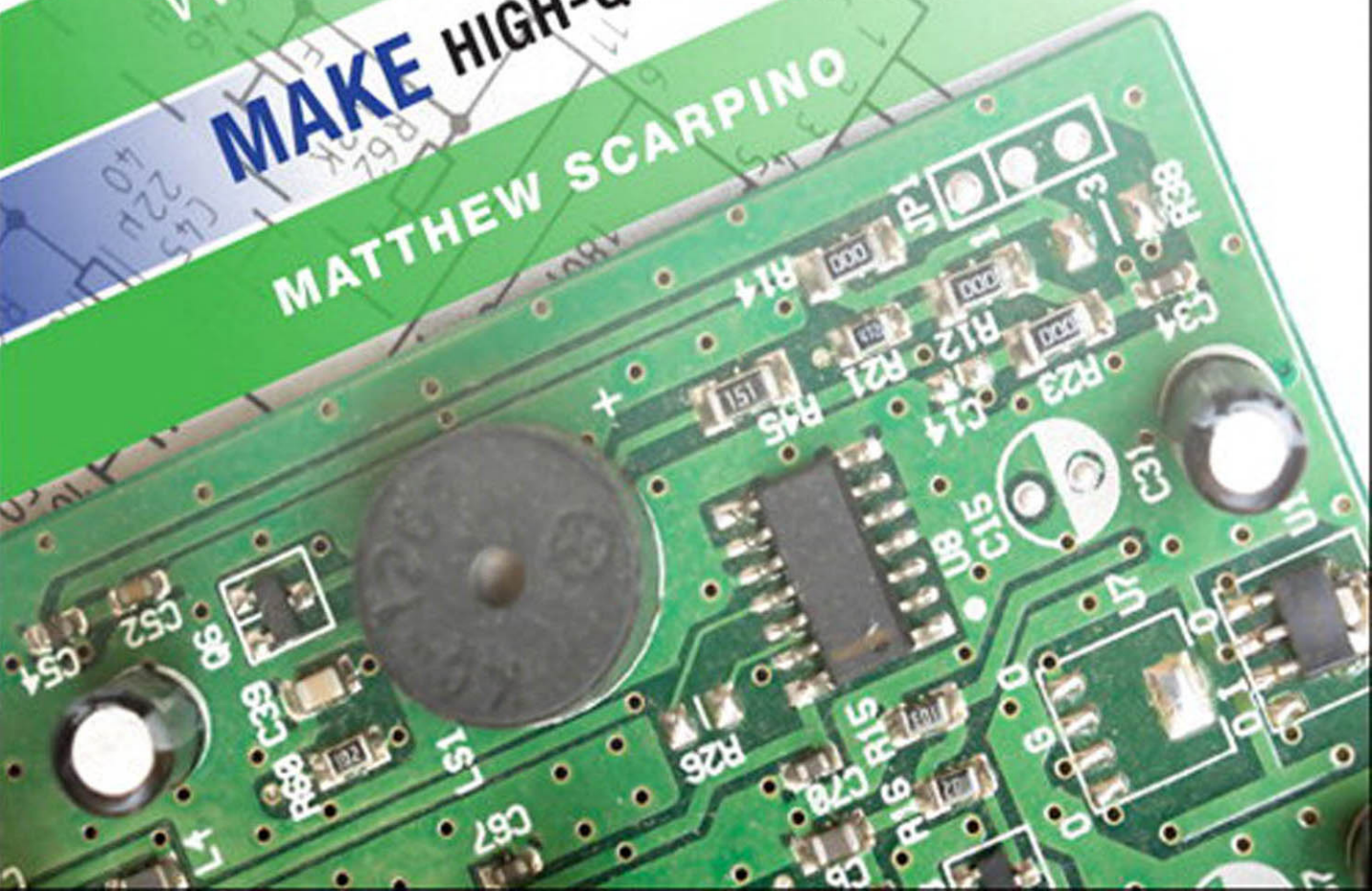


# DESIGNING CIRCUIT EAGLE BOARDS

WITH  
**MAKE** HIGH-QUALITY PCBs AT LOW COST

MATTHEW SCARPINO



## Praise for *Designing Circuit Boards with EAGLE*

“Matt Scarpino has succeeded where scores of others have failed—he’s managed to make the formidable EAGLE software understandable and, more importantly, useable. His presentation is not only approachable and logical, but it’s complete. When you’ve finished his book, you’ll be able to do something meaningful with EAGLE. This book belongs on every engineer’s bookshelf or tablet.”

—**Bryan Bergeron**, Editor, *Nuts & Volts Magazine*

“Matt Scarpino’s *Designing Circuit Boards with EAGLE* is a great resource for electronics enthusiasts who are ready to get serious and produce their own circuit boards. Matt’s sensible instructions take readers through the steps to design simple and not-so-simple circuit boards, and you can really tell that he’s been using EAGLE for 10 years and loves it. I’m recommending this book to all my maker friends.”

—**John Baichtal**, Author of *Arduino for Beginners: Essential Skills Every Maker Needs*

“With the rising popularity of open source hardware projects, the EAGLE circuit board software has become a vital tool for both hobbyists and professional engineers alike. *Designing Circuit Boards with EAGLE* provides all the information you’ll need to get up to speed with the EAGLE software, and to start creating your own circuit board designs. Matt Scarpino has provided a great tool for the hobbyist starting out in the circuit board design world, demonstrating all of the features you’ll need to know to create your own circuit board projects. However, the experienced engineer will also benefit from the book, as it also serves as a complete reference guide to all the EAGLE software configuration settings and features. His insightful guidance helps simplify difficult tasks in the EAGLE software, and his handy tips will help save you hours of trial-and-error experimenting in your circuit board designs.”

—**Rich Blum**, Author of *Sam’s Teach Yourself Arduino Programming in 24 Hours* and *Sam’s Teach Yourself Python Programming for Raspberry Pi in 24 Hours*

# Designing Circuit Boards with EAGLE: Make High-Quality PCBs at Low Cost

## Table of Contents

Cover

Half Title

Title Page

Copyright Page

Contents

Preface

Acknowledgments

About the Author

Part I: Preliminary Introduction

Chapter 1: Introducing EAGLE

1.1 A Whirlwind Tour of EAGLE

1.2 Obtaining EAGLE

1.3 Licensing

1.4 Organization of This Book

1.5 More Information

1.6 Conclusion

Chapter 2: An Overview of Circuit Boards and EAGLE Design

2.1 Anatomy of a Printed Circuit Board

2.2 Overview of Circuit Design with EAGLE

2.3 Conclusion

Chapter 3: Designing a Simple Circuit

3.1 An Inverting Amplifier

3.2 Initial Steps

3.3 The Inverting Amplifier Schematic

# Table of Contents

- 3.4 Board Layout
- 3.5 Routing
- 3.6 CAM Processor
- 3.7 Conclusion

## Part II: Designing the Arduino Femtoduino

### Chapter 4: Designing the Femtoduino Schematic

- 4.1 Initial Steps
- 4.2 The Reset Switch
- 4.3 Voltage Regulation
- 4.4 The ATmega328P Microcontroller
- 4.5 Header Connections
- 4.6 Net Classes
- 4.7 Electrical Rule Check
- 4.8 Generating the Board Design
- 4.9 Framing the Schematic
- 4.10 Attributes and Assembly Variants
- 4.11 Conclusion

### Chapter 5: Layout and Design Rules

- 5.1 Layers
- 5.2 Board Layout
- 5.3 Design Rule Check
- 5.4 Conclusion

### Chapter 6: Routing

- 6.1 An Overview of Routing
- 6.2 Manual Routing
- 6.3 Follow-Me Routing
- 6.4 The Autorouter
- 6.5 Home PCB Fabrication
- 6.6 Conclusion

### Chapter 7: Generating and Submitting Output Files

- 7.1 Jobs and the CAM Processor
- 7.2 Viewing Gerber Files

# Table of Contents

7.3 Drill Files

7.4 Submitting Design Files

7.5 Conclusion

## Part III: Advanced Capabilities

### Chapter 8: Creating Libraries and Components

8.1 Creating the Library

8.2 Creating the SIMPLE-TQFP16

8.3 Creating the VACUUM-TH

8.4 Creating the TW9920

8.5 Conclusion

### Chapter 9: Simulating Circuits with LTspice

9.1 Introducing LTspice

9.2 Designing a Schematic

9.3 Simulating the Circuit

9.4 Exchanging Designs with EAGLE

9.5 Conclusion

## Part IV: Automating EAGLE

### Chapter 10: Editor Commands

10.1 Introducing Editor Commands

10.2 Schematic Editor Commands

10.3 Commands for Board Designs

10.4 Commands for Library Interface

10.5 The Assign, Change, and Set Commands

10.6 Configuration Scripts

10.7 Conclusion

### Chapter 11: Introduction to the User Language (UL)

11.1 Overview of UL

11.2 Simple Data Types and Functions

11.3 Builtins

11.4 Control Structures

11.5 The exit Statement

# Table of Contents

11.6 Conclusion

## Chapter 12: Examining Designs with the User Language

12.1 UL-Specific Data Types

12.2 Schematic Designs (UL\_SCHEMATIC)

12.3 Board Designs (UL\_BOARD)

12.4 Conclusion

## Chapter 13: Creating Dialogs and Menu Items

13.1 Predefined Dialogs

13.2 Custom Dialogs and Widgets

13.3 Dialog Layouts

13.4 The Menu Command

13.5 Conclusion

## Part V: The BeagleBone Black

### Chapter 14: Schematic Design for the BeagleBone Black

14.1 Overview of the BeagleBone Black

14.2 Advanced EAGLE Schematics

14.3 AM3359 Memory/JTAG Connections

14.4 AM3359 I/O Connections

14.5 System Memory

14.6 Power Management

14.7 Ethernet and the Universal Serial Bus

14.8 Graphics Display

14.9 Conclusion

### Chapter 15: Board Design for the BeagleBone Black

15.1 Configuring the Stackup

15.2 Creating and Routing Ball Grid Arrays

15.3 Trace Length and Meander

15.4 The BBB Board Design

15.5 Conclusion

## Appendix A: EAGLE Library Files

A.1 Anatomy of a Library File

# Table of Contents

A.2 Settings, Grid, and Layers

A.3 Overview of Library Elements

A.4 Symbols

A.5 Packages

A.6 Devicesets

A.7 Conclusion

## Appendix B: The Gerber File Format

B.1 Introducing the Gerber Format

B.2 Setting Global Properties

B.3 Aperture Definitions

B.4 Drawing Shapes

B.5 Custom Apertures

B.6 Conclusion

Index