

DESIGNING CIRCUIT BOARDS WITH EAGLE

MAKE HIGH-QUALITY PCBs AT LOW COST

MATTHEW SCARPINO



FREE SAMPLE CHAPTER



SHARE WITH OTHERS

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 *Sams Teach Yourself Python Programming for Raspberry Pi in 24 Hours*

This page intentionally left blank

Designing Circuit Boards with EAGLE

This page intentionally left blank

Designing Circuit Boards with EAGLE

Make High-Quality PCBs at Low Cost

Matthew Scarpino



Upper Saddle River, NJ • Boston • Indianapolis • San Francisco
New York • Toronto • Montreal • London • Munich • Paris • Madrid
Cape Town • Sydney • Tokyo • Singapore • Mexico City

Many of the designations used by manufacturers and sellers to distinguish their products are claimed as trademarks. Where those designations appear in this book, and the publisher was aware of a trademark claim, the designations have been printed with initial capital letters or in all capitals.

The author and publisher have taken care in the preparation of this book, but make no expressed or implied warranty of any kind and assume no responsibility for errors or omissions. No liability is assumed for incidental or consequential damages in connection with or arising out of the use of the information or programs contained herein.

For information about buying this title in bulk quantities, or for special sales opportunities (which may include electronic versions; custom cover designs; and content particular to your business, training goals, marketing focus, or branding interests), please contact our corporate sales department at corpsales@pearsoned.com or (800) 382-3419.

For government sales inquiries, please contact governmentsales@pearsoned.com.

For questions about sales outside the U.S., please contact international@pearsoned.com.

Visit us on the Web: informit.com/ph

Library of Congress Control Number: 2013957510

Copyright © 2014 Pearson Education, Inc.

All rights reserved. Printed in the United States of America. This publication is protected by copyright, and permission must be obtained from the publisher prior to any prohibited reproduction, storage in a retrieval system, or transmission in any form or by any means, electronic, mechanical, photocopying, recording, or likewise. To obtain permission to use material from this work, please submit a written request to Pearson Education, Inc., Permissions Department, One Lake Street, Upper Saddle River, New Jersey 07458, or you may fax your request to (201) 236-3290.

ISBN-13: 978-0-13-381999-1

ISBN-10: 0-13-381999-X

Text printed in the United States on recycled paper at Edward Brothers Malloy in Ann Arbor, Michigan.

First printing: April 2014

Executive Editor

Bernard Goodwin

Managing Editor

Kristy Hart

Senior Project Editor

Betsy Gratner

Copy Editor

Apostrophe Editing Services

Indexer

Tim Wright

Proofreader

Kathy Ruiz

Technical Reviewers

John Baichtal

Bryan Bergeron

Rich Blum

Editorial Assistant

Michelle Housley

Interior Designer

Matthew Scarpino

Cover Designer

Chuti Prasertsith

Compositor

Matthew Scarpino

Senior Compositor

Gloria Schurick

Contents

| | |
|------------------------|-----|
| Preface | xi |
| Acknowledgments | xiv |
| About the Author | xv |

Part I: Preliminary Introduction

| | |
|--|-----------|
| Chapter 1: Introducing EAGLE. | 1 |
| 1.1 A Whirlwind Tour of EAGLE | 2 |
| 1.2 Obtaining EAGLE | 6 |
| 1.3 Licensing | 7 |
| 1.4 Organization of This Book | 8 |
| 1.5 More Information | 10 |
| 1.6 Conclusion | 11 |
| Chapter 2: An Overview of Circuit Boards and EAGLE Design | 13 |
| 2.1 Anatomy of a Printed Circuit Board | 14 |
| 2.2 Overview of Circuit Design with EAGLE | 22 |
| 2.3 Conclusion | 28 |
| Chapter 3: Designing a Simple Circuit. | 31 |
| 3.1 An Inverting Amplifier | 31 |
| 3.2 Initial Steps | 33 |
| 3.3 The Inverting Amplifier Schematic | 35 |
| 3.4 Board Layout | 44 |
| 3.5 Routing | 48 |
| 3.6 CAM Processor | 52 |
| 3.7 Conclusion | 55 |

Part II: Designing the Arduino Femtoduino

| | |
|--|-----------|
| Chapter 4: Designing the Femtoduino Schematic | 57 |
| 4.1 Initial Steps | 59 |
| 4.2 The Reset Switch | 61 |
| 4.3 Voltage Regulation | 65 |
| 4.4 The ATmega328P Microcontroller | 67 |
| 4.5 Header Connections | 71 |
| 4.6 Net Classes | 73 |
| 4.7 Electrical Rule Check | 74 |
| 4.8 Generating the Board Design | 76 |

| | |
|---|------------|
| 4.9 Framing the Schematic. | 76 |
| 4.10 Attributes and Assembly Variants | 77 |
| 4.11 Conclusion. | 79 |
| Chapter 5: Layout and Design Rules | 81 |
| 5.1 Layers | 81 |
| 5.2 Board Layout | 84 |
| 5.3 Design Rule Check | 90 |
| 5.4 Conclusion. | 100 |
| Chapter 6: Routing | 101 |
| 6.1 An Overview of Routing | 102 |
| 6.2 Manual Routing | 103 |
| 6.3 Follow-Me Routing. | 108 |
| 6.4 The Autorouter | 109 |
| 6.5 Home PCB Fabrication | 113 |
| 6.6 Conclusion. | 118 |
| Chapter 7: Generating and Submitting Output Files. | 119 |
| 7.1 Jobs and the CAM Processor. | 119 |
| 7.2 Viewing Gerber Files | 125 |
| 7.3 Drill Files | 126 |
| 7.4 Submitting Design Files. | 130 |
| 7.5 Conclusion. | 140 |
| Part III: Advanced Capabilities | |
| Chapter 8: Creating Libraries and Components. | 141 |
| 8.1 Creating the Library. | 142 |
| 8.2 Creating the SIMPLE-TQFP16. | 143 |
| 8.3 Creating the VACUUM-TH | 149 |
| 8.4 Creating the TW9920. | 153 |
| 8.5 Conclusion. | 160 |
| Chapter 9: Simulating Circuits with LTspice. | 161 |
| 9.1 Introducing LTspice | 161 |
| 9.2 Designing a Schematic. | 163 |
| 9.3 Simulating the Circuit | 170 |
| 9.4 Exchanging Designs with EAGLE | 174 |
| 9.5 Conclusion. | 176 |

Part IV: Automating EAGLE

| | |
|---|------------|
| Chapter 10: Editor Commands | 177 |
| 10.1 Introducing Editor Commands | 177 |
| 10.2 Schematic Editor Commands | 180 |
| 10.3 Commands for Board Designs | 187 |
| 10.4 Commands for Library Interface | 192 |
| 10.5 The Assign, Change, and Set Commands | 202 |
| 10.6 Configuration Scripts | 207 |
| 10.7 Conclusion | 208 |
| Chapter 11: Introduction to the User Language (UL) | 209 |
| 11.1 Overview of UL | 209 |
| 11.2 Simple Data Types and Functions | 211 |
| 11.3 Builtins | 214 |
| 11.4 Control Structures | 219 |
| 11.5 The exit Statement | 221 |
| 11.6 Conclusion | 222 |
| Chapter 12: Examining Designs with the User Language | 225 |
| 12.1 UL-Specific Data Types | 226 |
| 12.2 Schematic Designs (UL_SCHEMATIC) | 227 |
| 12.3 Board Designs (UL_BOARD) | 243 |
| 12.4 Conclusion | 256 |
| Chapter 13: Creating Dialogs and Menu Items | 257 |
| 13.1 Predefined Dialogs | 257 |
| 13.2 Custom Dialogs and Widgets | 264 |
| 13.3 Dialog Layouts | 275 |
| 13.4 The Menu Command | 278 |
| 13.5 Conclusion | 279 |

Part V: The BeagleBone Black

| | |
|--|------------|
| Chapter 14: Schematic Design for the BeagleBone Black | 281 |
| 14.1 Overview of the BeagleBone Black | 282 |
| 14.2 Advanced EAGLE Schematics | 284 |
| 14.3 AM3359 Memory/JTAG Connections | 286 |
| 14.4 AM3359 I/O Connections | 290 |
| 14.5 System Memory | 293 |
| 14.6 Power Management | 295 |

| | |
|---|------------|
| 14.7 Ethernet and the Universal Serial Bus | 297 |
| 14.8 Graphics Display | 300 |
| 14.9 Conclusion | 301 |
| Chapter 15: Board Design for the BeagleBone Black. | 303 |
| 15.1 Configuring the Stackup | 304 |
| 15.2 Creating and Routing Ball Grid Arrays | 307 |
| 15.3 Trace Length and Meander | 313 |
| 15.4 The BBB Board Design | 316 |
| 15.5 Conclusion | 321 |
| Appendix A: EAGLE Library Files | 323 |
| A.1 Anatomy of a Library File | 323 |
| A.2 Settings, Grid, and Layers | 325 |
| A.3 Overview of Library Elements | 327 |
| A.4 Symbols | 329 |
| A.5 Packages | 336 |
| A.6 Devicesets | 340 |
| A.7 Conclusion | 346 |
| Appendix B: The Gerber File Format. | 347 |
| B.1 Introducing the Gerber Format | 347 |
| B.2 Setting Global Properties | 349 |
| B.3 Aperture Definitions | 353 |
| B.4 Drawing Shapes | 356 |
| B.5 Custom Apertures | 361 |
| B.6 Conclusion | 368 |
| Index | 369 |

Preface

As I write this in late 2013, the Maker Movement has flourished from a tiny group of tinkerers into a passionate community of millions. Hobbyists have become entrepreneurs and entrepreneurs have become large-scale manufacturers. 3-D printers have fallen into the price range of the average consumer, and the printers' capabilities have risen to such an extent that they're being used to fabricate high-precision aircraft parts and medical equipment. With good reason, many economists and journalists have likened the rise of the Maker Movement to a second Industrial Revolution.

Nothing better illustrates the movement's success than the popularity of the Arduino platform. The first Arduino board design, the Arduino USB, was released in 2005, giving students and hobbyists a low-cost means of programming Atmel microcontrollers. Since then, hundreds of thousands of Arduino boards have been sold, and the Arduino family has expanded to include a vast array of boards, shields, kits, and accessories. Arduino boards have found their way into robots, musical instruments, game platforms, and even unmanned aerial vehicles. The boards have become so popular that many hobbyists-turned-entrepreneurs use them to build prototypes of new inventions.

But Makers still demand more: more capability, more affordability, and more customization. This means designing new circuit boards, a task that requires specialized knowledge and software. Most professional design tools are beyond the price range of the average Maker, but not EAGLE. Since its release in 1988, EAGLE has grown steadily in features and stability while maintaining a price that even cash-strapped students can afford. EAGLE wins legions of admirers with every new version, and the analogy couldn't be clearer: What Arduino is to hardware, EAGLE is to software. It's no wonder that all open-source Arduino designs are released in EAGLE's format.

In writing this book, my mission is to show Makers how to take full advantage of EAGLE's capabilities. This requires a basic understanding of circuit theory, including Ohm's law and Kirchoff's laws, but nothing beyond that. You won't find any transistor analysis or differential equations here. Instead, my goal is to provide a practical, hands-on exploration of EAGLE so that readers can design practical circuit boards, thereby bringing exciting new gadgets to the marketplace and continuing the extraordinary momentum of the Maker Movement.

Matthew Scarpino

Structure of This Book

This book presents EAGLE by walking through a series of circuit design projects, starting with a simple inverting amplifier and proceeding to a six-layer, single-board computer. As the circuits grow in complexity, I'll explain more advanced features of EAGLE and show how to automate repetitive tasks. This book also includes a great deal of material to help readers understand the fundamentals of circuit boards and the theory behind the example circuits.

Chapters 1, 2, and 3 introduce the topics of EAGLE and circuit board design. Their primary purpose is to familiarize you with EAGLE's capabilities and present the terminology used throughout the book. Chapter 3 breezes through the complete design of a trivially simple circuit.

Chapters 4 through 7 present the design of a practical circuit board: the Arduino Femtoduino. These chapters take a hands-on approach to explaining the four fundamental steps of circuit board design: drawing a schematic, laying out components, routing connections, and generating Gerber/Excellon files.

Chapters 8 through 13 discuss an assortment of topics related to EAGLE circuit design. These include circuit simulation, the process of creating custom components, and the all-important subject of design automation. Design automation is one of the most powerful aspects of EAGLE, but it's also one of the most overlooked. For this reason, I highly recommend becoming familiar with editor commands and User Language programs.

Chapters 14 and 15 present the book's advanced example design: the BeagleBone Black. The name may sound silly but there's nothing silly about the circuit. It has six board layers, hundreds of components, and thousands upon thousands of routed connections. As I present the design, I'll discuss EAGLE's advanced capabilities and ways to take advantage of design automation.

Example File Archive

To supplement the text, all the circuit designs, programs, and support files in this book are provided in a zip file called eagle-book.zip. This can be freely downloaded from <http://eagle-book.com>. As you follow the discussion, I recommend that you compare the theoretical discussion to the real-world EAGLE designs. In addition, the color figures for this book can be accessed at www.informit.com/title/9780133819991.

Acknowledgments

First and foremost, I'd like to thank Bernard Goodwin of Pearson North America for his support and sage wisdom during the creation of this book. Thanks to his deft handling, the development process was as frictionless as could be asked.

I'd like to express my deep appreciation to San Dee Phillips of Apostrophe Editing Services, who caught so many of my formatting, spelling, and grammar errors. I'd also like to thank Betsy Gratner for her cheerful yet firm masterminding of this book's production, Gloria Schurick for her painstaking efforts in compositing, Kathy Ruiz for her eagle-eyed proofreading, and Laura Robbins for managing this book's images.

Last but not least, I'd like to extend my gratitude to Bryan Bergeron of *Nuts & Volts Magazine*, Richard Blum, author of *Sams Teach Yourself Arduino Programming in 24 Hours*, and John Baichtal, author of *Arduino for Beginners*. These reviewers bravely made their way through the book in its rough state and provided many useful suggestions. Their comments have improved the book's accessibility to newcomers and expanded the number of topics.

About the Author

Matthew Scarpino is an engineer with more than 12 years of experience designing hardware and software. He has a Master's degree in electrical engineering and is an Advanced Certified Interconnect Designer (CID+). He currently resides in Massachusetts where he develops software for embedded systems. In his spare time, he uses EAGLE to design accessories for his Android smartphone and the Google Glass.

This page intentionally left blank

Chapter 1

Introducing EAGLE

Circuit design applications can be divided into two categories: those intended for large design firms and those intended for everyone else. Applications in the first category provide high reliability, a wide range of features, and responsive technical support. But these advantages come with a hefty price tag. A perpetual license for Altium Designer costs more than \$7,000 and Cadence's OrCad suite costs nearly \$10,000.

Applications in the second category are less expensive, and this makes them accessible to students, individuals, and small-to-medium businesses. Unfortunately, they tend to be unreliable and plagued with bugs. Without technical support, there may be no way to work around these difficulties. What's worse, the companies that release these tools tend to be as flaky as their software and may disappear before their support contracts expire.

But not CadSoft's EAGLE. The Easily Applicable Graphical Layout Editor provides the best of both worlds: the quality of a first-tier design application for the price of a second-tier application. EAGLE has been around since 1988, and with each year, it has improved in capability and reliability. It provides a complete set of features for designing circuit boards, and despite thousands of hours of use, it has never crashed on me. If problems arise, users can visit multiple online forums or read through the many online articles.

EAGLE has one major drawback: its user interface. If you're a frequent Windows user, you're accustomed to applications behaving in a certain manner. You're used to a common set of toolbar items and mouse gestures. But EAGLE has its own unique behavior, and it's impossible to simply start the application and figure out how everything works. It takes time to understand the many editors, dialogs, menus, and commands. And because circuit design is such a complex task to begin with, many newcomers to EAGLE give up.

The goal of this book is to ease the process of learning EAGLE. In these chapters, I'm going to walk through the process of designing circuits, starting with a simple circuit (a noninverting amplifier), proceeding to an intermediate circuit (the Arduino Femtoduino), and finally reaching an advanced circuit (the BeagleBone Black). During the course of this presentation, I'll describe both the EAGLE interface and the general process of designing circuit boards.

In addition to point-and-click design, a significant portion of this book is devoted to automation. EAGLE has a rich command language that can be accessed through scripts and User Language programs, or ULPs. When you have a solid grasp of how to create circuit designs in code, you can perform long, repetitive tasks with a single command. With this automation, your errors will decrease and your productivity will skyrocket.

1.1 A Whirlwind Tour of EAGLE

EAGLE is a software application that makes it possible to design circuit boards. Boiled down to its essentials, EAGLE consists of six features:

- **Component library**—The set of devices that can be inserted into a design
- **Schematic editor**—An editor that makes it possible to draw the circuit's preliminary design
- **Board editor**—An editor that defines the circuit board's physical layout and routing
- **Device editors**—Editors used to design new components
- **Autorouter**—A tool that automatically determines how circuit elements can be connected
- **CAM (Computer Aided Manufacturing) processor**—A tool that reads in a board design and produces files for the board's fabrication

This section briefly describes each of these features and how they relate to the overall process of circuit design.

1.1.1 The Component Library

One of the most important features of any circuit design tool is the set of available parts. This set of components is called a *library*, and the larger the library, the less time the designer needs to spend defining new devices.

Thanks to its longevity, EAGLE's set of libraries has expanded to thousands and thousands of components, from vacuum tubes to field programmable gate arrays. No matter how complex the design, the odds are that EAGLE will have most of the

required parts. If it doesn't, the site <http://www.cadsoftusa.com/downloads/libraries> provides more libraries for free download. If a part still can't be found, Chapter 8, "Creating Libraries and Components," explains how to design custom parts.

One new feature of EAGLE 6 is the format used by the library files. Each library is defined within a *.lbr file, and the format for this file is the eXtensible Markup Language (XML), which is popular throughout the world of computing. Appendix A, "EAGLE Library Files," describes the XML schema that defines the structure of EAGLE's library files.

1.1.2 The Schematic Editor

After you verified that your circuit's components are available, you can select and connect them inside a schematic design, as shown in Figure 1.1.

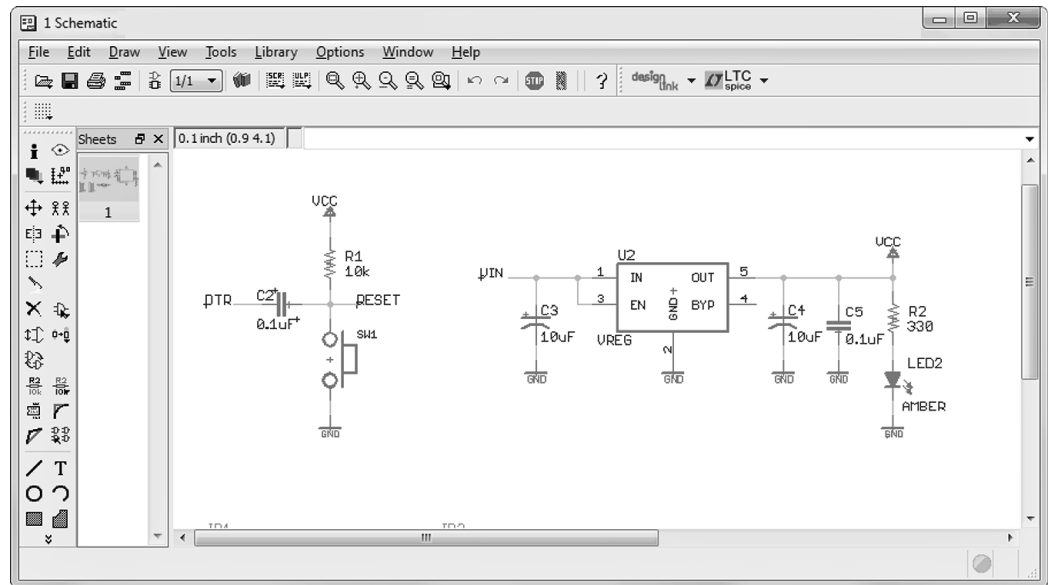


Figure 1.1: The EAGLE Schematic Editor

As with most schematic editors, this keeps track of four important pieces of information:

- Which components are present in the design
- Connections between the components' pins
- Names and values associated with the components
- Properties of the components' connections

EAGLE's schematic editor makes it easy to design a preliminary circuit. Just select a part from the library, move it to a position, and draw connections between it and other components. Afterward, you may assign names and values to the component, such as a resistor's resistance in ohms. Chapter 3, "Designing a Simple Circuit," and Chapter 4, "Designing the Femtoduino Schematic," discuss the schematic editor in detail.

1.1.3 The Board Editor

After a schematic design is complete, EAGLE can generate a board file (*.brd) that defines the layout of the actual circuit board. Board files are modified in EAGLE's board editor, as shown in Figure 1.2.

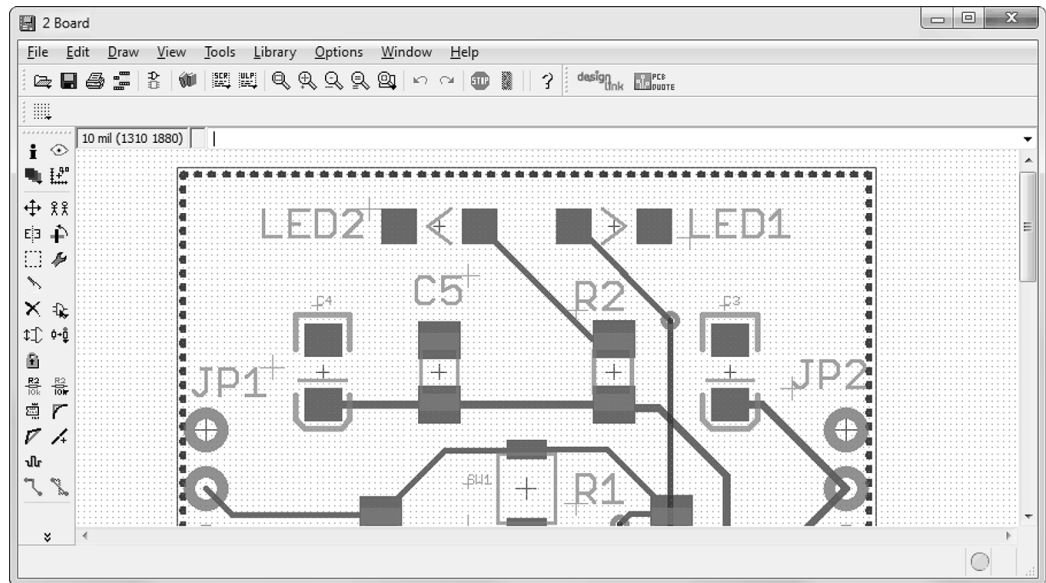


Figure 1.2: The EAGLE Board Editor

In this editor, the designer positions the real-world devices corresponding to the components in the schematic. This position includes not only x and y coordinates, but also whether the components are on the top or bottom layer.

1.1.4 The Device Editors

If the EAGLE library doesn't contain a crucial part, the device editors make it possible to design a new one. This process has three steps:

1. Create a design for the schematic editor. This is called a *symbol*.
2. Create a design for the board editor. This is called a *package*.
3. Create an association between the symbol and its package. This is called a *device*.

EAGLE provides editors for laying out a component's symbol and package. These are collectively called the device editors, as shown in Figure 1.3.

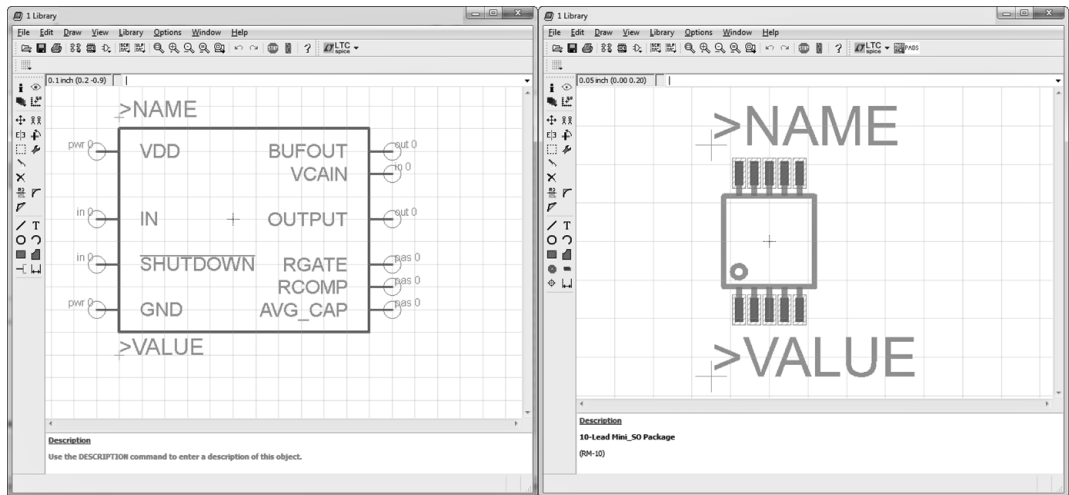


Figure 1.3: The EAGLE Device Editors

The left side of the figure displays the symbol for Analog Devices' SSM2167 component. The right side displays the component's package, which can be used in the board design.

Don't be concerned about terms like symbol, package, and device just yet. These topics will be explored throughout this book, and Chapter 8 presents the entire process of designing new components. Appendix A explains the file format used by EAGLE to store these designs.

1.1.5 The Autorouter

After the boards' devices are in place, the next step is to create the connections between them. This is called *routing*, and even with high-end design tools, this process can be complex and time-consuming.

EAGLE's autorouter simplifies the routing process and provides insight into how circuit components can be connected. But for large-scale circuits, it generally isn't capable of completely routing a board on its own. However, if a designer manually creates initial routes, it will help the autorouter do its job. Chapter 6, "Routing," explains all the different routing methods supported by EAGLE.

1.1.6 The CAM Processor

Most fabrication facilities don't accept EAGLE design files, so EAGLE's CAM (Computer Aided Manufacturing) Processor converts EAGLE designs into different formats. Figure 1.4 shows the processor's dialog.

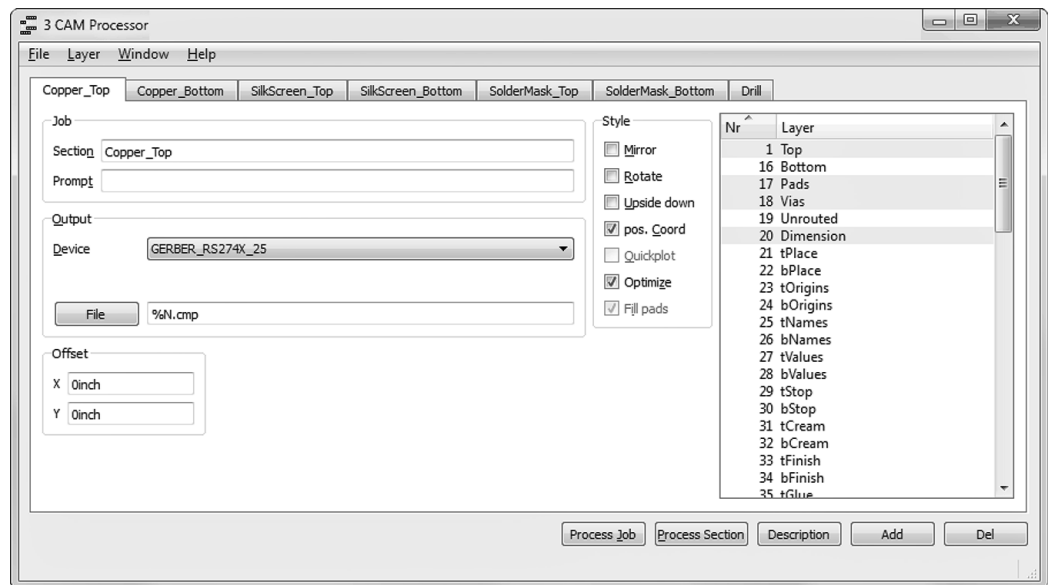


Figure 1.4: The CAM Processor

When the Process Job button is pressed, the processor executes a sequence of tasks called a *job*. A designer can load a job from a file (*.cam) or create a new job from scratch. As a job executes, each of its tasks reads a portion of the board design and creates a file of the selected type.

To fabricate a circuit board, most manufacturers require two types of files. To define a circuit's geometry and connections, the accepted file format is RS-274X, also called the Gerber format. To specify drill diameters and drill locations, the accepted format is the Excellon format. The CAM Processor generates files of both types.

1.2 Obtaining EAGLE

CadSoft makes it easy to start with EAGLE. After you download the executable, you can try it out without registering or paying anything. If you're interested in more features, you can make decisions regarding purchasing, licensing, and registration.

If you run a supported operating system and connect to the Internet, you need to download only a single file. Currently, EAGLE can run on any of the following operating systems:

- Windows 8, Windows 7, Windows Vista, or Windows XP
- Mac OS 10.6, 10.7 on Intel-based processors
- Linux (kernel 2.6, Intel processors, 32-bit runtime environment)

CadSoft's primary web site is <http://www.cadsoft.de> but the company can also be accessed at locale-specific sites such as <http://www.cadsoftusa.com>. At these sites, you can download EAGLE by finding the Downloads link in the upper menu and selecting Download EAGLE. This takes you to a page with download links, and you can choose between the Windows, Linux, and Mac OS offerings.

1.3 Licensing

When you first launch EAGLE, a dialog appears and gives you the option of providing a license key or running the tool as freeware. The Freeware option enables you to run EAGLE in a special configuration called the Freeware version of EAGLE Light. This enables you to access EAGLE's editing and routing for free, but with a limited set of features. In addition, this version can be used only for evaluation or nonprofit purposes. If you intend to make money through your PCB design, CadSoft asks that you purchase a license.

EAGLE provides four types of licenses that appeal to different segments of the PCB design community. Each has a different price and set of features (the higher the price, the more features). Specifically, the license type determines the maximum number of schematic sheets, the maximum number of board layers, and the maximum routing area. Table 1.1 lists the characteristics of the different licenses.

Table 1.1

EAGLE License Features

| License | Number of Sheets | Number of Layers | Routing Area in mm ² |
|--------------|------------------|------------------|---------------------------------|
| Light | 1 | 2 | 100 × 80 |
| Hobbyist | 99 | 6 | 160 × 100 |
| Standard | 99 | 6 | 160 × 100 |
| Professional | 999 | 16 | 4000 × 4000 |

Looking at this table, you may wonder what the difference is between the Hobbyist and Standard licenses. The Hobbyist license is much, much less expensive but carries the requirement that EAGLE can be used only for noncommercial purposes. CadSoft requires a signed statement to this effect.

Table 1.1 doesn't list the prices for these licenses for three reasons:

1. EAGLE's prices change over time and any listed price will prove inaccurate in the near future.
2. For the Standard and Professional licenses, CadSoft doesn't sell EAGLE as an integrated application. Instead, it splits EAGLE into three parts (schematic editor, autorouter, and board editor), and sells them separately.
3. For the Standard and Professional licenses, pricing depends on how many users can use the tool at once.

For a full presentation of the EAGLE pricing structure, visit the CadSoft web site. For prices in American dollars, the link is <http://www.cadsoftusa.com/shop/pricing/?language=en>.

In writing this book, I have made no assumptions regarding which license readers have purchased or if any license has been purchased at all. But this book covers every aspect of EAGLE, so if one or more features are unavailable in your installation, skip over the corresponding material. The first and second example circuits can be designed with any of the licenses, but the final design (discussed in Chapters 14, “Schematic Design for the BeagleBone Black,” and 15, “Board Design for the BeagleBone Black”) requires more advanced capabilities.

1.4 Organization of This Book

This book is structured so that the material proceeds from the simple to the complex and from the fundamentally important to the esoteric. More specifically, the chapters in this book can be divided into five parts, each of which focuses on a different task or aspect of EAGLE.

Part I: Preliminary Introduction

The first part of this book provides essential information for readers new to circuit board design and EAGLE. Chapter 2, “An Overview of Circuit Boards and EAGLE Design,” explains what circuit boards are and how they’re manufactured, thereby establishing the vocabulary that will be used throughout this book. It also explains the overall circuit board design process with EAGLE.

Chapter 3, “Designing a Simple Circuit,” expands on this introduction and walks through the schematic design and board design for a simple amplifier circuit. This circuit isn’t intended to be manufactured, but the design process will be helpful to inexperienced readers.

Part II: Designing the Arduino Femtoduino

The second and largest part of this book centers on designing an Arduino Femtoduino. The Arduino family of circuit boards enjoys a great deal of popularity among amateurs and professionals, and Chapters 4 through 7 explain how to design one for yourself. Chapter 4, “Designing the Femtoduino Schematic,” explains how to create the schematic and Chapter 5, “Layout and Design Rules,” explains how to position the packages in the board editor.

Chapter 6, “Routing,” discusses the process of design rule checking and shows how to route the connections on the Arduino Femtoduino. Lastly, Chapter 7, “Generating and Submitting Output Files,” presents the Computer Aided Manufacturing (CAM) processor and explains how to generate the final artwork files for the Femtoduino.

It also presents five different fabrication services that accept these files and deliver finished circuit boards.

Part III: Advanced Capabilities

The next part of the book covers two topics that go beyond regular schematic/board design. Chapter 8, “Creating Libraries and Components,” explains how to create custom components for EAGLE and walks through two designs. The first creates a symbol and package for a through-hole component and the second creates a symbol and package for a surface-mount component.

Chapter 9, “Simulating Circuits with LTspice,” delves into one of EAGLE’s newest and most interesting features: circuit simulation with LTspice. LTspice is a freely downloadable simulation tool that makes it possible to draw circuits, assign inputs, and simulate the circuit’s operation. By combining EAGLE and LTspice, designers can test a design before sending it out for fabrication.

Part IV: Automating EAGLE

The fourth part of this book focuses on automating EAGLE using scripts and program files. Chapter 10, “Editor Commands,” presents the EAGLE command language, which executes design operations in text. For example, the `add` command adds a new component to a schematic or board design, and the `auto` command launches the autorouter.

Chapters 11 through 13 explain how to write User Language programs (ULPs), which make it possible to examine circuit designs automatically. These chapters provide many useful examples that can simplify EAGLE usage and reduce time associated with the design process.

Part V: The BeagleBone Black

The last two chapters of this book focus on designing the BeagleBone Black. This advanced circuit board has six layers and hundreds of components, some of which have high-density ball grid array (BGA) pins. Though difficult to design, the BeagleBone Black has gained a significant following among programmers because of its extraordinary amount of computing power.

Example File Archive

All the designs, programs, and support files discussed in this book are freely available online. The archive is called `eagle-book.zip` and it can be downloaded from <http://eagle-book.com>.

1.5 More Information

One of EAGLE's greatest advantages is the staggering amount of information available. No matter what problem you face, it's likely that someone has already encountered it and found a solution. In addition to this book, here are four sources of information that I highly recommend.

1.5.1 Element14—www.element14.com

EAGLE is maintained and released by CadSoft, but in 2009, CadSoft was acquired by Premier Farnell PLC, a distributor of electronic components. That same year, Premier Farnell created element14, an online community to provide support for circuit designers. This community has grown significantly over the years, and each day its forum receives hundreds of designers asking and answering questions. In addition, it provides a library of documentation and videos related to electronic design.

EAGLE isn't the only topic discussed at element14, but the subforum devoted to EAGLE support is one of busiest places on the site. Here, users ask questions ranging from routing issues to library entries to converting file formats to those used by other tools. Richard Hammerl, a chief technician at CadSoft, frequently answers questions, which means the subforum is nearly as good as full professional support.

1.5.2 SparkFun—www.sparkfun.com

In 2003, Nathan Seidle founded SparkFun Electronics to “make electronics accessible to the average person.” This site sells development tools and kits, such as Arduino boards, and it also provides articles related to electrical design. The list of tutorials includes SMT soldering, programming, robotics, and of course, EAGLE. Nathan Seidle has written a series of articles that discuss EAGLE, and SparkFun provides its own EAGLE scripts, programs, and CAM Processor jobs.

The SparkFun forum is very active and its subforums discuss topics as diverse as wireless/RF design, GPS projects, and shipping times for fabrication facilities. In the PCB Design Questions subforum, many EAGLE users submit questions and receive answers.

1.5.3 YouTube—www.youtube.com

If you search for **EAGLE** and **PCB** or **CadSoft** on YouTube, you'll find many YouTube videos devoted to explaining EAGLE usage. Some may be out of date, but taken as a whole, they provide a friendly introduction to this complicated topic.

1.5.4 CadSoft—www.cadsoftusa.com/www.cadsoft.de

Last but not least, I recommend CadSoft's main site. CadSoft provides a great deal of documentation on EAGLE, but in general, you can find the same documentation inside EAGLE's top-level doc directory. One major point of interest is the Downloads link, which makes it possible to download additional libraries, ULPs, and actual EAGLE projects.

1.6 Conclusion

I first used EAGLE around 2003, and though it had many of the same capabilities as today, it tended to crash at least three times an hour. On message forums, users railed against EAGLE's instability and exchanged workarounds for dealing with its many bugs. But CadSoft persevered in its work on EAGLE, and nearly 10 years later, the bugs and instability are gone. Instead of complaining, today's users defend the application fiercely.

I'm a devoted EAGLE user, and my goal in writing this chapter is to explain why I think the tool is so wonderful. EAGLE provides a full suite of design features, including a schematic editor, a board editor, device editors, and a CAM processor. Its libraries contain thousands and thousands of electronic components. It's stable, runs at high speed, and if I encounter issues, there are many online resources I can turn to.

One of the reasons I'm so impressed with EAGLE is its generous licensing. Users can try out the tool for free and continue using it indefinitely. If they'd like to take advantage of its advanced features, they can purchase a license without spending great sums of money.

This page intentionally left blank

Index

Symbols & Numerics

? wildcard, 179

[] wildcard, 179

* wildcard, 179

4PCB, 131

FreeDFM, 132

quote generation, 132

12AX7 vacuum tube, 150

A

active components, 14

add command, 181-182

adding

components

to inverting amplifier

design, 39

to schematic, 24

pins to SIMPLE-TQFP16,
144-145

Advanced Circuits, 131

FreeDFM, 132

price matrices, 133

quote generation, 132-134

advanced configuration op-
tions, autorouter, 110-112

airwire, 41

AM3359, 286

architecture

ARM Cortex-A8 proces-
sor core, 287-288

JTAG interface, 289

I/O connections, 290-293

analog-to-digital conver-
sion, 291

I²C communication,
291-292

SPI communication,
292-293

signals, routing for BBB
design, 318-320

system memory, 293-294

analog-to-digital conversion,
AM3359, 291

apertures, 353

circle, 354

custom apertures, 362-368

circle, 363-364

moiré, 365-366

outline, 364

polygon, 365

rectangle, 354, 363-364

thermals, 366

polygon, 355

with multiple primitives,
367-368

arc command, 195

architecture, AM3359

ARM Cortex-A8 processor
core, 287-288

JTAG interface, 289

arcs, 359-360

Arduino, 58

ARM Cortex-A8 processor
core, 287-288

array constants, 215-216

arrays (UL), 212

AS (Axis Select), Gerber files,
351

assembly variants, 78

assign command, 203

ATmega328P microcontroller,
Femtoduino circuit board,
67-69

attribute command, 183-184
 attributes, component-specific, 78
 auto command, 192
 automation, ULPs, 2
 autorouter, 5, 49, 109
 configuring, 109-112
 cost parameters, 111
 follow-me routing, 108
 operation, 112

B

BBB (BeagleBone Black), 282-283, 303
 BGA devices, 307-313
 packages, generating, 308-310
 routing signals from BGA pads, 310-313
 block diagram, 283
 design rules, 316-317
 dog bone, 310
 drawbacks of, 281-282
 graphics display
 HDCP encryption, 300-301
 TMDS, 301
 quadrant dog bone, 311
 routing AM3359's signals, 318-320
 schematic design
 AM3359, 286-293
 buses, 286

Ethernet, 297
 gates, 285-286
 power management, 295-296
 sheets, 284
 system memory, 293-294
 USB, 298-299
 stackup configuration, 304-307
 four-layer stackup, 305-307
 trace length, obtaining, 313-316
 BGA (ball grid array), 14, 153, 156
 BBB board design, 307-313
 packages, generating, 308-310
 routing signals from BGA pads, 310-313
 NFBGA, 318
 blind vias, 21
 block diagram of BBB design, 283
 board design, BBB
 BGA devices, 307-313
 design rules, 316-317
 board editor, 4, 25
 auto command, 192
 Design Rule dialog
 Clearance tab, 94
 Distance tab, 94

File tab, 92
 Layers tab, 92-93
 Masks tab, 98-99
 Misc tab, 99
 Restraining tab, 96
 Shapes tab, 97
 Sizes tab, 95
 Supply tab, 98
 devices, positioning, 87-88
 display command, 82, 189
 DRCs, 91
 editor commands, 187
 grid command, 188
 grids, 45-47
 ground planes, 85
 layer command, 188
 layers, 83-84
 layout, 84
 manual routing, 105
 mirror command, 189
 packages, 4
 moving, 47
 positioning, 88-89
 polygons, 85
 configuring, 86-87
 forming, 86
 ratsnest command, 190
 ripup command, 191
 routing, 102-103
 via command, 191

- board file
 - creating for inverting amplifier, 44-45
 - selecting with CAM Processor, 120
- BOM (bill of materials), creating, 230-232
- breadboard headers, Femtoduino circuit board, 71
- buildup, 138
- builtin constants
 - array constants, 215-216
 - single-valued constants, 214-215
- builtin functions, 216-218
- buried vias, 21
- bus, 41
- buttons
 - cancel buttons, 261-262
 - push buttons, 268-269
 - radio buttons, 269-270

C

- C Primer Plus, 210
- C programming language, 210-211
- CadSoft
 - EAGLE
 - licensing, 7
 - obtaining, 6-7
 - pricing structure, 8
 - web site, 11

- CAM (Computer Aided Manufacturing) Processor, 5-6
 - Gerber files, generating, 53-54
 - jobs, 119, 123-124
 - launching, 52
 - selecting a board file, 120
- cancel buttons, 261-262
- cells, 277-278
- centroid files, 90
- change command, 203
- checkboxes, 274
- circle aperture, 354
- circle command, 196
- circuit board design, 25-26
 - creating a project, 22-23
 - generating design files, 27-28
 - inverting amplifier, 31
 - board file, creating, 44-45
 - components, adding to design, 39
 - packages, moving in board editor, 47
 - resistors, adding to schematic, 39-40
 - routing, 48-52
- inverting amplifier circuits, 32-33
- routing connections, 26
- schematic design, creating, 23-25
- circular interpolation, 358-359
- classes, net classes, 73, 240

- Clearance tab, Design Rule dialog, 94
- combo boxes, 272-273
- commands
 - editor commands, 177
 - add command, 181-182
 - executing, 178-179
 - points, 180-181
 - syntax, 178
 - value command, 182-183
 - wildcards, 179-180
 - keystrokes, assigning to, 203
 - schematic editor, 180-181
- comparing DRCs and ERCs, 91
- component library (EAGLE), 2-3
- components
 - active components, 14
 - adding to schematic, 24
 - electrical components, 14
 - leads, 14
 - SMT, 15
 - through-hole components, 15
 - inverting amplifier, adding to design, 39
 - naming, 167-168
 - pads, 15
 - passive components, 14
- component-specific attributes, 78
- configuration scripts, 207. *See also* editor commands

configuring

- autorouter, 109-112
- Femtoduino circuit board
 - grid, 60
- polygons, 86-87
- properties with set command, 205-206

connect command, 201-202

connections

- creating with LTspice, 167
- routing, 26, 101-102
 - follow-me routing, 108
 - manual routing, 104-105
- traces, 48

connectors for inverting amplifier, adding to schematic, 40

constants, builtin constants

- array constants, 215-216
- single-valued constants, 214-215

contacts, 249-250

control structures

- do..while loops, 220
- if..else statement, 219
- for loops, 220-221
- switch..case statement, 219-220
- while loops, 220

copper

- etching, 117
- thickness, 17

copper pour, 85

core, 16

cost parameters, autorouter, 111

creating

- BOM (bill of materials), 230
- custom dialogs, 264-265
- drill racks, 129-130
- inverting amplifier schematic, 35-36
- jobs, 123-124
- projects, 22-23
- schematic design, 23-25
- SIMPLE symbol, 145-146
- SIMPLE-TQFP16, 143, 148-149
- TH9 package, 151-152
- TQFP16 package, 147-148
- TW9920 device, 159
- TW9920 symbol, 154-155
- VACUUM symbol, 150-151
- VACUUM-TH device, 149, 152
- VFBGA100L-8X8 package, 156-159
- vias, 20, 106-107

cupric chloride, 117

custom apertures, 362-368

- apertures, 361
- circle, 363-364
- moiré, 365-366
- outline, 364-365
- polygon, 365
- rectangle, 363

thermals, 366-367

- with multiple primitives, 367-368

custom dialogs, creating, 264-265

custom versions of SPICE, 162

D

data structures, 225

data types, 211

design files, generating, 27-28

designing circuit boards, 25-26

Design Rule dialog

- Clearance tab, 94

- Distance tab, 94

- File tab, 92

- Layers tab, 92-93

- Masks tab, 98-99

- Misc tab, 99

- Restraining tab, 96

- Shapes tab, 97

- Sizes tab, 95

- Supply tab, 98

design rules for BBB design, 316-317

developer, 18

device editors, 4-5

<device> element, technologies, 346

devices, 4, 40-41, 142

- positioning in board editor, 87-88

- routing, 5

- SIMPLE-TQFP16 device,
 - creating, 148-149
- technologies, 344
- TW9920 device, creating, 159
- VACUUM-TH, creating, 152
- <devices> element, 342-343
- deviceset (library files), 328-346
- <devicesets> element, 340-341
 - <gates> element, 341-342
 - <devices> element, 342-343
- dialogs
 - custom dialogs
 - creating, 264-265
 - icons, 259-261
 - layouts
 - grid layouts, 277-278
 - horizontal layouts, 276
 - vertical layouts, 276
 - predefined dialogs
 - directory dialogs, 262-263
 - file selection dialogs, 263-264
 - message boxes, 258-262
- directory dialogs, 262-263
- display command, 189
- Display command, board editor, 82
- dog bone, 310
- do..while loops, 220
- double-sided boards, 19-20

- drawing
 - nets, 42-43
 - shapes
 - exposure, 356
 - motion, 356
- <drawing> element
 - <grid> tag, 326-327
 - <layers> tag, 326-327
 - <packages> element, 336-337
 - <settings> tag, 325-326
- DRCs (Design Rule Checks), 81, 90-91
- .dri files, 128-130
- drill rack, 129
 - creating, 130
 - generating, 254

E

- EAGLE (Easily Applicable Graphical Layout Editor), 1
 - autorouter, 5
 - board editor, 4, 25
 - CAM processor, 5-6
 - component library, 2-3
 - device editor, 4
 - licensing, 7
 - obtaining, 6-7
 - pricing structure, 8
 - schematic editor, 3-4
 - schematics, exporting to LTspice, 175
 - user interface, 1

- eagle-book.lbr, installing, 33-34
- edit command, 193
- edit widgets, 270-271
 - number entry widgets, 271-272
 - text entry widgets, 270-271
- editor commands, 177
 - attribute command, 183-184
 - board editor, 187
 - auto command, 192
 - display command, 189
 - grid command, 188
 - layer command, 188-189
 - mirror command, 189
 - ratsnest command, 190
 - ripup command, 191
 - via command, 191
 - frame command, 186
 - label command, 186
 - library interface
 - arc command, 195
 - circle command, 196
 - connect command, 201
 - edit command, 193
 - open command, 193
 - package command, 200
 - pad command, 198
 - pin command, 196
 - prefix command, 200
 - smd command, 199
 - technology command, 201

- wire command, 194
- write command, 194
- move command, 184
- name command, 182
- net command, 185
- points, 180
- rotate command, 184
- syntax, 178
- use command, 181
- value command, 182
- wildcards, 179
- electrical components, 14
 - leads, 14
 - SMT, 15
 - through-hole
 - components, 15
- Element14, 10
- elements of library files, 327-346
- ERC (electrical rule check), 74-75, 91
- errors, ERC, 75
- etching copper, 117
- Ethernet, BBB design, 297
- Eurocircuits, 136-138
- example archive,
 - femtoarduino.cam file, 121
- Excellon files, 22
 - header, 126-127
 - program body, 127-128
- executing editor commands, 178

- exit statement (UL), 221-222
- exporting schematics to
 - LTspice, 175
- exposure, 356-357

F

- fabrication companies
 - Advanced Circuits, 131
 - FreeDFM, 132
 - price matrices, 133
 - quote generation, 132-134
- Eurocircuits, 136-137
- OSH Park, 130-131
- Sseed Studio, 138
 - Fusion, 138-139
 - PCBA Prototype, 139
- Sunstone
 - PCBExpress option, 135-136
 - ValueProto option, 134-135
- fanout, 310
- FCBGAs (flip-chip ball grid arrays), 153
- femtoarduino.cam file, 121
 - jobs, loading, 122-123
- Femtoarduino circuit board, 58
 - ATmega328P microcontroller, 67-69
 - board design, generating, 76
 - breadboard headers, 71

- ERC, 74-75
- grid, configuring, 60
- headers, ICSP headers, 72
- manual routing, 103-104
- microcontrollers, connecting to the headers, 105
- net classes, 73
- new project, creating, 59
- reset switch, 61
- routing, 105
- schematic, framing, 76
- voltage regulation, 65
- ferric chloride, 113
- file selection dialogs, 263-264
- files
 - .brd, 4
 - centroid files, 90
 - design files, generating, 27-28
 - .dri files, 128, 130
 - Excellon files, 22
 - header, 126-127
 - program body, 127-128
 - femtoarduino.cam file, 121
 - Gerber files, 21-22, 347
 - global properties, setting, 349
 - viewing, 125-126
 - RS-274X, 6
- File tab (Design Rules dialog), 92
- follow-me routing, 108

- for loops, 220-221
- forming polygons, 86
- four-layer stackup (BBB), 305-307
- FR4, 16
- frame command, 186
- <frame> element, 335-336
- framing Femtoduino circuit
 - board schematic, 76
- Freeware EAGLE option, 7
- FS (Format Statement), Gerber files, 350
- functions, builtin functions, 216-218
- Fusion, 138-139

G

- G codes, 358
- gates, 142, 285-286
- <gates> element, 341-342
- generating
 - BOM, 232
 - design files, 27-28
 - drill racks, 254
 - Femtoduino board design, 76
 - Gerber files with CAM processor, 53-54
- Gerber files, 6, 21-22, 347
 - apertures, 353
 - circle, 354
 - custom apertures, 361-368

- polygon, 355
 - rectangle, 354
- generating with CAM processor, 53-54
- global properties
 - AS, 351
 - FS, 350
 - MI, 352
 - MO, 351
 - OF, 352
 - setting, 349
 - SF, 352
- viewing, 125-126
- global properties, Gerber files, 349
 - AS, 351
 - FS, 350
 - MI, 352
 - MO, 351
 - OF, 352
 - SF, 352
- global variables, 77
- graphics display, BBB design
 - HDCP encryption, 300-301
 - TMDS, 301
- grids
 - configuring for Femtoduino circuit board, 60
 - layouts, 277-278
 - perimeter, setting in board editor, 45-47
- grid command, 188

- <grid> tag, 326-327
- Grid tool (board editor), 45
- ground connection, adding to inverting amplifier schematic, 40
- ground plane, 85, 304-307

H

- Hammerl, Richard, 10
- headers
 - breadboard headers, 71
 - Excellon files, 126-127
 - ICSP headers, 72
- HDCP (high-bandwidth digital content protection), BBB design, 300-301
- holes, 252
- home fabrication of PCBs, 113
 - etching copper, 117
 - toner transfer method, 113-116
- horizontal layouts, 276
- hotkeys
 - assigning to commands, 203
 - schematic editor, 38

I

- I²C communication, AM3359, 291-292
- I/O connections
 - AM3359, 290-293
 - analog-to-digital conversion, 291

- I²C communication, 291-292
- SPI communication, 292-293
- ICSP (In-Circuit Serial Programming) headers, Femtoduino circuit board, 72
- if..else statement, 219
- importing schematics into EAGLE, 174
- installing eagle-book.lbr, 33-34
- interpolation, 359
 - circular interpolation, 358
 - linear interpolation, 358
- inverting amplifier circuit, 31-33
 - board file, creating, 44-45
 - components, adding to design, 39
 - connector, adding to schematic, 40
 - ground connection, adding, 40
 - packages, moving in board editor, 47
 - resistors, adding to schematic, 39-40
 - routing, 48, 52
 - autorouter, 49-50
 - manual routing, 50
 - Route tool, 51
 - schematic, creating, 35-36
- Invoke tool (schematic editor), 285-286

J-K

- jobs, 6, 119-121
 - creating, 123-124
 - loading femtoduino.cam file, 122-123
- JTAG interface (AM3359), 289

- Kirchoff's Current law, 32

L

- label command, 186
- labels, 265-266
- launching CAM processor, 52
- layer command, 188
- layers, 14, 81, 246
 - visibility, 82-84
- Layers tab (Design Rule dialog), 92-93
- <layers> tag, 326-346
- layout, 44
 - board editor, 45-46, 84
 - configuring polygons, 86-87
 - forming polygons, 86
 - ground planes, 85
 - packages, moving, 47
 - polygons, 85
 - grid layouts, 277-278
 - horizontal layouts, 276
 - inverting amplifier, creating board file, 44-45
 - layers, 81-84

- layout blocks, 275-276

- leads, 14-15, 142

- length of traces, obtaining, 313-316

- libraries

- eagle-book.lbr, installing, 33-34

- LTspice-compatible, 175

- library editor, Symbol toolbar item, 143

- <library> element, subelements, 327-328

- library files

- <connect> element, 343

- <devicesets> element, 340

- <devices> element, 342

- <gates> element, 341

- technologies, 344

- <drawing> element

- <grid> tag, 326

- <layers> tag, 326

- <packages> element, 336

- <settings> tag, 325

- <library> element, 327

- <packages> element

- surface mount pads, 339

- through-hole pads, 338

- structure of, 323

- <symbol> element

- <frame> element, 335

- <pins> element, 332

- <polygon> element, 333

- <text> element, 331

- <vertex> element, 333
 - <wire> element, 331
 - library interface
 - arc command, 195
 - circle command, 196
 - connect command, 201
 - edit command, 193
 - open command, 193
 - package command, 200
 - pad command, 198
 - pin command, 196
 - prefix command, 200
 - smd command, 199
 - technology command, 201
 - wire command, 194
 - write command, 194
 - licensing EAGLE, 7
 - linear interpolation, 358
 - list boxes, 273
 - list views, 273-274
 - list widgets
 - combo boxes, 272-273
 - list boxes, 273
 - list views, 273-274
 - LM741, adding to inverting amplifier schematic, 39
 - loading jobs, femtoduino.cam file, 122
 - loop members, 226
 - LTspice, 161
 - designing a schematic, 163
 - adding components, 164-166
 - creating connections, 167
 - named nets, 169
 - naming components, 167-168
 - obtaining, 162-163
 - schematics, importing, 174-175
 - simulating a circuit, 170, 173
 - configuring simulation appearance, 173
 - running the simulation, 172
 - setting simulation parameters, 170-171
- ## M
- Maker Movement, xi
 - manual routing, 50-52, 103, 105
 - Route command, 104
 - vias, creating, 106-107
 - Masks tab (Design Rule dialog), 98-99
 - math functions (UL), 212
 - Meander tool, 313-316
 - members
 - loop members, 226
 - UL, 226
 - UL_ATTRIBUTE, 231
 - UL_BOARD, 243
 - UL_CLASS, 240
 - UL_CONTACT, 250
 - UL_ELEMENT, 244
 - UL_FRAME, 241
 - UL_GATE, 235
 - UL_GRID, 229
 - UL_HOLE, 253
 - UL_INSTANCE, 234
 - UL_LAYER, 246
 - UL_NET, 237
 - UL_PACKAGE, 245
 - UL_PAD, 251
 - UL_PART, 230
 - UL_PIN, 237
 - UL_PINREF, 237
 - UL_POLYGON, 247
 - UL_SCHEMATIC, 228-229
 - UL_SHEET, 241
 - UL_SIGNAL, 248
 - UL_SMD, 251
 - UL_SYMBOL, 235
 - UL_VARIANT, 242
 - UL_VIA, 253
 - menu command, 278-279
 - message boxes, 258-262
 - microcontrollers, drawing
 - ATmega328P schematic, 69
 - microvias, 20
 - mils, 16
 - MI (Mirror Image) statement,
 - Gerber files, 352
 - mirror command, 189
 - Misc tab (Design Rule dialog), 99
 - miter, 104
 - modal tools, 38
 - moiré, 365

MO (Mode), Gerber files, 351
motion, 356
move command, 184
moving packages in board editor, 47
multilayer boards, 20
 stackup, 20-21
 vias, 21
multiquadrant arcs, 360

N

name command, 182
named nets, 169
naming components, 167-168
net, 41
net classes, Femtoduino circuit board, 73
net command, 185
nets, 42-43, 236
NFBGA (New Fine-Pitch Ball Grid Array), 318
number entry widgets, 271-272

O

obtaining
 EAGLE, 6-7
 LTspice, 162-163
 trace length for BBB design, 313-316
offsets, arcs, 359
OF (Offset) statement, Gerber files, 352

Ohm's law, 32
op-amps, 31-33
open command, 193
opening projects, 23
operational codes, 357
organization of this book, 8-9
OSH Park, 131
outlines, 364-368

P

package command, 200
package designation (library files), 328-346
packages, 4, 40, 142, 245
 devices, 40
 moving in board editor, 47
 pads, adding, 147-148
 positioning in board editor, 88-89
 TH9 package, creating, 151-152
 TQFP16 package, creating, 146-148
 VFBGA100L-8X8 package, creating, 157-159
<packages> element, 336-337
 surface mount pads, 339-340
 through-hole pads, 338-339
pad command, 198
pads, 15, 41, 142
 adding to packages, 147-148
 SMDs, 19

 surface mount pads, 339-346
 through-hole pads, 338-346
passive components, 14
PCBA Prototype, 139
PCBExpress option, Sunstone, 135-136
PCBs, 14
 core, 16
 double-sided, 19-20
 home fabrication, 113
 etching copper, 117
 toner transfer method, 113-116
 layers, 14
 multilayer boards
 stackup, 20-21
 vias, 21
 photolithography, 17-18
 post-processing, 18-19
 silk-screening, 19
 single-sided, 16
 copper thickness, 17
 core, 16
 traces, 16
PCBs (printed circuit boards), 13, 115
perimeter of grid, setting in board editor, 46-47
photolithography, 17-18
photoresist, 18
pin command, 196

pins, 41, 142
 adding to SIMPLE-TQFP16, 144-145
 on ATmega328P, 67
 points (editor commands), 180
 polygon apertures, 355
 <polygon> element, 333-334
 polygons, 85, 365-368
 configuring, 86-87
 forming, 86
 positioning
 devices in board editor, 87
 packages in board editor, 88-89
 symbols, 41-42
 post-processing, 18-19
 power management, BBB, 295-296
 power planes, BBB stackup
 configuration, 304-307
 Prata, Stephen, 210
 predefined dialogs
 directory dialogs, 262-263
 file selection dialogs, 263-264
 message boxes, 258-262
 cancel buttons, 261-262
 dialog icons, 259-261
 prefix command, 200
 Premier Farnell PLC, 10
 price matrices, Advanced Circuits, 133
 pricing structure for EAGLE, 8

primary grid, 46
 primitives
 circle, 363
 moiré, 365
 outline, 364
 polygon, 365
 rectangle, 363
 thermals, 366
 program body, Excellon files, 127-128
 projects
 creating, 22-23
 Femtoduino circuit board, 59
 inverting amplifier
 board file, creating, 44-45
 components, adding to design, 39
 connectors, adding, 40
 creating, 34-35
 ground connection, adding, 40
 packages, moving in board editor, 47
 resistors, adding to schematic, 39-40
 routing, 48-52
 schematic, creating, 35
 opening, 23
 properties
 configuring with set command, 205-208
 of grids, configuring, 45

PTHs (plated-through holes), 20
 push buttons, 268-269

Q-R

QFP (quad flat package), 146
 quadrant dog bone, 311
 quote generation, Advanced Circuits, 133-134

 radio buttons, 269-270
 ratsnest command, 190
 Ratsnest tool, 51
 rectangle aperture, 354
 references to contacts, 249
 reset switch, Femtoduino circuit board, 61
 Restraining tab (Design Rule dialog), 96
 ripup command, 191
 Ripup tool, 51
 rotate command, 184
 Route command (board editor), 103-105
 Route tool, 51
 routing, 5, 16, 48, 101-102
 autorouter, 49-50
 configuring, 109-112
 operation, 112
 connections, 26
 Femtoduino circuit, 105
 follow-me routing, 108

manual routing, 50-52, 103, 105

Route command, 104

vias, creating, 106-107

Route tool, 51

RS-274X files. *See* Gerber files

S

schemas, XML, 324-346

schematics

adding components, 24

BBB

buses, 286

Ethernet, 297

gates, 285-286

power management,
295-296

sheets, 284

USB, 298-299

designing with LTspice,
163-164

adding components,
164-165

creating connections, 167

moving components, 166

named nets, 169

naming components,
167-168

exporting to LTspice, 175

Femtoduino circuit board,
framing, 76

gates, 142

importing into EAGLE, 174

inverting amplifier

components, adding, 39

connectors, adding, 40

ground connection, add-
ing, 40

LM741, adding, 39

resistors, adding, 39-40

UL_SCHEMATIC, 227

schematic editor, 3-4, 36

commands

add command, 181

attribute command, 183

frame command, 186

label command, 186

move command, 184

name command, 182

net command, 185

rotate command, 184

use command, 181

hotkeys, 38

nets, drawing, 42-43

signals, 43-44

symbols, 4, 40-42

vertical toolbar, 37-38

scripts

configuration scripts,
207-208

executing, 179

Seeed Studio

Fusion, 138-139

PCBA Prototype, 139

Seidle, Nathan, 10

selecting board file with CAM
Processor, 120

set command, 205-207

<settings> tag, 325-326

SF (Scale Factor) statement,
Gerber files, 352

shapes

apertures, 353, 361-368

drawing, 356

Shapes tab, Design Rule
dialog, 97

sheets, BBB design, 284

shortened command forms, 179

signals, 43-44, 248, 318-320

silk-screening, 19

SIMPLE-TQFP16

adding pins, 144-145

creating, 143

device, creating, 148-149

package, drawing, 146

SIMPLE symbol

creating, 144

symbol body, creating,
145-146

TQFP16 package, creating,
147-148

simulating a circuit, 170, 173

configuring simulation ap-
pearance, 173

running the simulation, 172

setting simulation param-
eters, 170-171

- Simulation Program with Integrated Circuit Emphasis, 162
 - single-quadrant arcs, 360
 - single-sided boards, 16
 - copper thickness, 17
 - photolithography, 17-18
 - post-processing, 18-19
 - single-valued constants, 214-215
 - Sitara AM3359, 286
 - architecture
 - ARM Cortex-A8 processor core, 287-288
 - JTAG interface, 289
 - I/O connections, 290-293
 - Sizes tab (Design Rule dialog), 95
 - smd command, 199
 - SMT (surface mounted technology), 14-15
 - solder mask, 19
 - solder paste, 19
 - solder paste stencils, 123
 - SparkFun, 10
 - SPI (Serial Peripheral Interface) communication, AM3359, 292-293
 - SPICE, 161-162
 - spinboxes, 274-275
 - stackup, 20-21
 - four-layer stackup (BBB), 305-307
 - ground planes (BBB), 304-307
 - stencils, solder paste stencils, 123
 - Stop tool, 38
 - strings (UL), 213-214
 - structure of library files, 323-346
 - structure of this book, xii
 - stub vias, 21
 - subelements of <package> element, 337
 - submitting design files
 - Advanced Circuits, 131-133
 - FreeDFM, 132
 - price matrices, 133
 - quote generation, 132-134
 - Eurocircuits, 136-137
 - OSH Park, 130-131
 - Seed Studio
 - Fusion, 138-139
 - PCBA Prototype, 139
 - Sunstone
 - PCBExpress option, 135-136
 - ValueProto option, 134-135
 - Supply tab (Design Rule dialog), 98
 - surface mount pads, 339-346
 - switch..case statement, 219-220
 - symbols, 4, 40-41, 142
 - devices, 40, 142
 - positioning, 41-42
 - SIMPLE symbol, creating, 145-146
 - TW9920 symbol, creating, 154-155
 - VACUUM symbol, creating, 150-151
 - <symbols> element
 - <frame> element, 335-336
 - <pins> element, 332-333
 - <polygon> element, 333-334
 - <text> element, 331-332
 - <vertex> element, 333-334
 - <wire> element, 331-332
 - Symbol toolbar item (library editor), 143
 - syntax, editor commands, 178
 - system memory, BBB, 293-294
- ## T
- technologies, 345-346
 - technology command, 201
 - <text> element, 331-346
 - text entry widgets, 270-271
 - text views, 266-267
 - TH9 package, creating, 151-152
 - thermals, 366-368
 - through-hole components, 15
 - through-hole leads, 14
 - through-hole pads, 338-346

through vias, 21

TMDS (Transition-Minimized Differential Signaling), 301

toner transfer method, 113-115
 removing copper and toner, 117

toner transfer method of home PCB fabrication, 116

tool information files, 128

tools
 in vertical toolbar (schematic editor), 37-38
 modal, 38
 Stop tool, 38

top-level structures, UL, 227

TQFP (thin quad flat pack), 143
 SIMPLE-TQFP symbol, creating, 143
 SIMPLE-TQFP16
 adding pins, 144-145
 creating, 143
 SIMPLE symbol, creating, 144

TQFP16 package, creating, 146-148

traces, 16, 41, 48, 51-52, 313-316

TW9920 device, creating, 159

TW9920 symbol, creating, 154-155

U

UL

buttons
 push buttons, 268-269
 radio buttons, 269-270

data structures, 225

data type members, 226

dialogs
 custom dialogs, creating, 264-265
 layouts, 276-278
 predefined dialogs, 257-264

members, loop members, 226

top-level structures, 227

widgets
 checkboxes, 274
 edit widgets, 270-271
 labels, 265-266
 list widgets, 272-274
 spinboxes, 274-275

UL_ATTRIBUTE, members, 231

UL_BOARD, members, 243

UL_CLASS, members, 240

UL_CONTACT, members, 250

UL_CONTACTREF, 249

UL_ELEMENT, members, 244

UL_FRAME, members, 241

UL_GATE, 234-235

UL_GRID, 229

UL_HOLE, members, 253

UL_INSTANCE, members, 234

UL_LAYER, members, 246

UL_NET, 236

UL_PACKAGE, members, 245

UL_PAD, members, 251

UL_PART, members, 230

UL_PIN, members, 237

UL_PINREF, members, 237

UL_POLYGON, members, 247

UL_SCHEMATIC, 227-229

UL_SHEET, members, 241

UL_SIGNAL, members, 248

UL_SMD, members, 251

UL_SYMBOL, 234-235

UL_VARIANT, members, 242

UL_VIA, members, 253

ULPs (User Language programs), 2
 arrays, 212
 builtins
 builtin constants, 214-216
 builtin functions, 216-218
 builtin variables, 216
 comparing with C programming language, 211
 control structures
 do..while loops, 220
 if..else statement, 219
 for loops, 220-221

- switch..case statement,
219-220
- while loops, 220
- data types, 211
- executing, 210
- exit statement, 221-222
- math functions, 212
- strings, 213-214
- unit conversion functions, 218
- USB, BBB design, 298-299
- use command, 181
- user interface, EAGLE, 1

V

VACUUM-TH

- creating, 149
- TH9 package, creating,
151-152
- VACUUM symbol, creating,
150-151
- VACUUM-TH device, creat-
ing, 152
- vacuum tubes, 149
- value command, 182
- ValueProto option, Sunstone,
134-135
- Varesano, Fabio, 58
- variables, global variables, 77
- variants, 242
 - assembly variants, 78
- <vertex> element, 333-334

- vertical layouts, 276
- vertical toolbar (schematic
editor), 37-38
- VFBGA100L-8X8 package,
creating, 156-159
- via command, 191
- vias, 20, 252
 - blind vias, 21
 - buried vias, 21
 - creating, 20, 106-107
 - microvias, 20
 - PTHs, 20
 - stub vias, 21
 - through vias, 21
- viewing Gerber files, 125-126
- visibility of layers, 82-84
- voltage rails, 33
- voltage regulation, Femtoduino
circuit board, 65

W

- warnings, ERC, 75
- web sites
 - Advanced Circuits, 131
 - CadSoft, 7, 11
 - Element14, 10
 - OSH Park, 130
 - SparkFun, 10
 - Sunstone, 134
 - YouTube, 10

- while loops, 220
- widgets
 - checkboxes, 274
 - edit widgets, 270-271
 - number entry widgets,
271-272
 - text entry widgets,
270-271
 - labels, 265-266
 - list widgets
 - combo boxes, 272-273
 - list boxes, 273
 - spinboxes, 274-275
- wildcards, editor commands,
179-180
- wire command, 194
- <wire> element, 331-332
- wires, 142
- write command, 194-195

X-Y-Z

- XML, 323-346
 - library files, structure of,
323-346
 - schemas, 324-346

YouTube, 10