PRENTICE HALL

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

Visit us on the Web: informit.com/ph

Library of Congress Control Number: 2013957510

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

Part IV: Automating EAGLE

Part V: The BeagleBone Black

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 desiging 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.

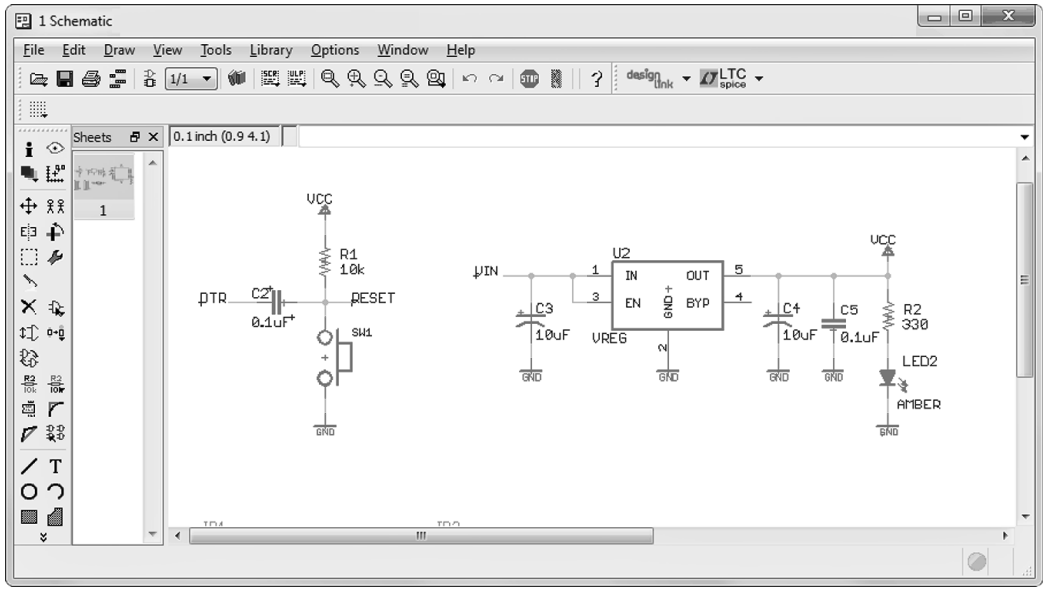

Figure 1.1: The EAGLE Schematic Editor

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.

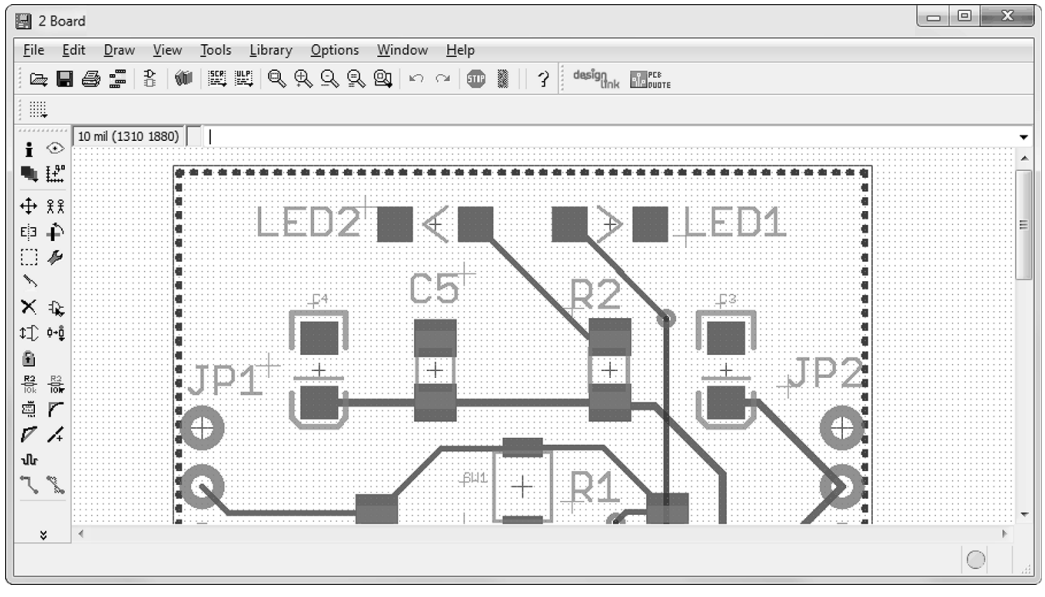

Figure 1.2: The EAGLE Board Editor

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.

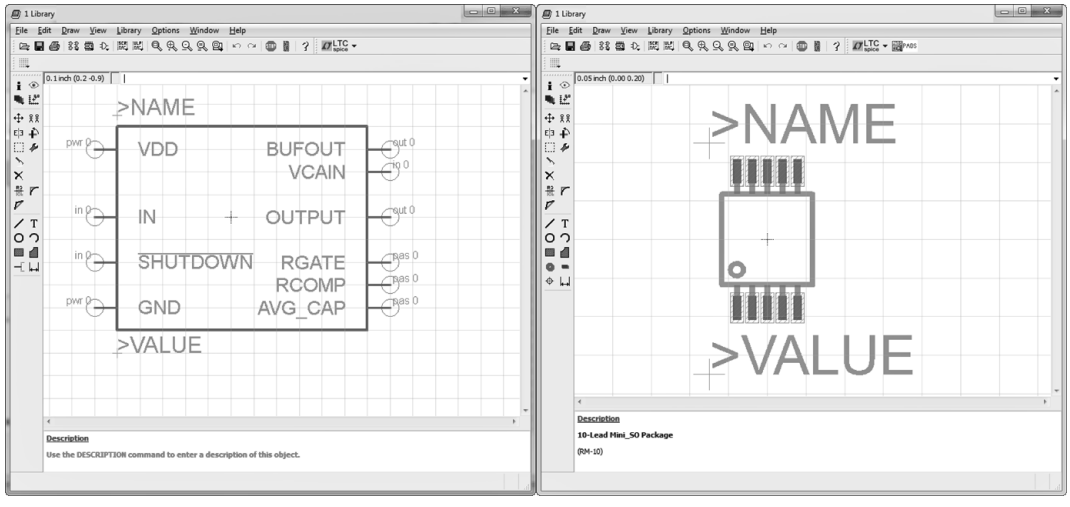

Figure 1.3: The EAGLE Device Editors

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.

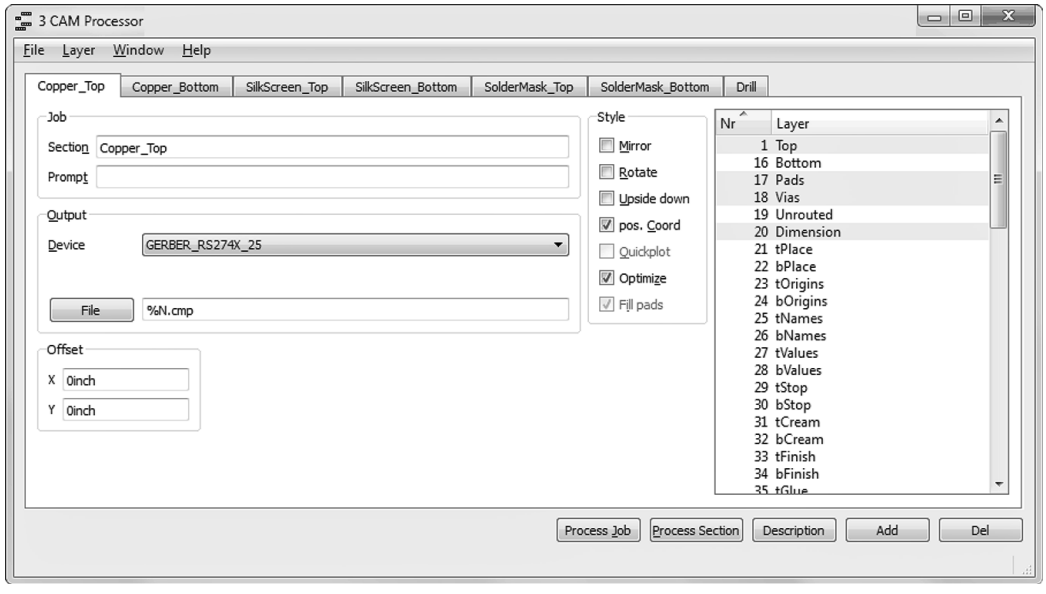

Figure 1.4: The CAM Processor

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 mm²
Light	1	2	100 x 80
Hobbyist	99	6	160 x 100
Standard	99	6	160 x 100
Professional	999	16	4000 x 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 noncommerical 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

A

B

C

D

E

F

G

H

I

J-K

L

M

N

O

P

Q-R

S

T

U

V

W

X-Y-Z