“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
Designing Circuit Boards with EAGLE

Make High-Quality PCBs at Low Cost

Matthew Scarpino
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
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
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.
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.

Acknowledgments
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.
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](image)

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 Femtduino 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.

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](image)

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.
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>
<thead>
<tr>
<th>License</th>
<th>Number of Sheets</th>
<th>Number of Layers</th>
<th>Routing Area in mm²</th>
</tr>
</thead>
<tbody>
<tr>
<td>Light</td>
<td>1</td>
<td>2</td>
<td>100 x 80</td>
</tr>
<tr>
<td>Hobbyist</td>
<td>99</td>
<td>6</td>
<td>160 x 100</td>
</tr>
<tr>
<td>Standard</td>
<td>999</td>
<td>16</td>
<td>4000 x 4000</td>
</tr>
</tbody>
</table>

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 website. 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.
Chapter 1: Introducing EAGLE

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
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 options, autorouter, 110-112
airwire, 41
AM3359, 286
architecture
ARM Cortex-A8 processor core, 287-288
JTAG interface, 289
I/O connections, 290-293
analog-to-digital conversion, 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
BGA 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
Restrings tab, 96
Shapes tab, 97
Sizes tab, 95
Supply tab, 98
devices, positioning, 87-88
display command, 82, 189
DRCs, 91
designer 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, Femtduino 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
routong, 26, 101-102
follow-me routing, 108
manual routing, 104-105
traces, 48
connectors for inverting amplifier, adding to schematic, 40
constants, built-in 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
Restring 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, femtoduino.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
Seeed 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
femtoduino.cam file, 121
jios, loading, 122-123
Femtoduino 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
teric 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
femtoduino.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
routings, 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
Kirchhoff’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
Restrict 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, adding, 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 appearance, 173
running the simulation, 172
setting simulation parameters, 170-171
through-hole pads

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
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_SCHEMA, 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, creating, 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