
Did you find an error?	13
About the author	14
From the back cover	15
How to read this book	17
Requirements	18
The book web page	19
An introduction: Why KiCad?	20
Part 1: A quick introduction to PCB design	23
Chapter 1 • What is a PCB?	23
Chapter 2 • The PCB design process	29
Chapter 3 • Fabrication	32
Chapter 4 • Installation	34
Chapter 5 • Examples of KiCad projects	37
Part 2: A hands-on tour of KiCad with a very simple project.	44
Chapter 6 • Introduction to this section	44
Chapter 7 • Start KiCad	45
Chapter 8 • Schematic design in Eeschema	48
8.1. The schematic sheet	48
Page layout description file.	51
8.2. Mouse buttons and hotkeys	51
8.3. Eeschema buttons and menus	55
Left toolbar	55
Right toolbar	57
Component selector	58
Power port selector	59
Wiring and delete tools	61
Junction tool.	63
Text tool	64
Graphics line tool	67
Top toolbar.	68
Quick tour of the top toolbar	68
Library browser	69

The annotator	71
Electrical rules check (ERC)	72
Cvpcb: Associate components and footprints	75
Create the Netlist file.	78
Status bar	80
Menus	82

Chapter 9 • Layout in Pcbnew 86

9.1. The user interface.	87
9.2. The layout sheet.	88
9.3. Mouse buttons and hotkeys	90
9.4. Pcbnew toolbars and menus.	96
Left toolbar	97
Top toolbar.	99
Import the Netlist	99
Move footprints.	101
Design Rules Check (DRC)	103
Plot for Gerbers	108
Layer chooser.	110
Right toolbar	112
Standard mode.	113
Net highlighter	114
Add footprint	115
Wiring	119
Fill zones and keepout areas.	122
Edge cut	123
Graphics	132
Text.	134
Layers manager	138
Status bar	140
Menus	141
File	142
Edit	144

View	147
Setup	149
Design Rules Editor	149
Layers Setup	151
Other setup options.	152
Place	155
Route	155
Interactive Routing	156
Inspect	159
Tools	161
Preferences	164
General	165
Display	165
Language.	166
Hotkeys	167
Part 3: Design principles and basic concepts.	168
Chapter 10 • About this Part	168
Chapter 11 • Schematic symbols	169
Chapter 12 • PCB key terms.	171
12.1. FR4.	171
12.2. Trace.	171
12.3. Pads and holes	173
12.4. Via	174
12.5. Annular ring.	176
12.6. Soldermask	176
12.7. Silkscreen	177
12.8. Drill bit and drill hit.	177
12.9. Surface mounted devices.	178
12.10. Gold Fingers.	179
12.11. Panel.	179
12.12. Solder paste and paste stencil	180
12.13. Pick-and-place	182

- Chapter 13 • Schematic design workflow 184**
 - 13.1. Step 1. Setup 184
 - 13.2. Step 2. Symbols 186
 - 13.3. Step 3. Place and annotate symbols 187
 - 13.4. Step 4. Wire 187
 - 13.5. Step 5. Nets 188
 - 13.6. Step 6. Electrical Rules Check 189
 - 13.7. Step 7. Comments 189
 - 13.8. Step 8. Netlist 190

- Chapter 14 • PCB layout workflow 192**
 - 14.1. Step 1. Setup 192
 - 14.2. Step 2. Outline and mechanical constraints 196
 - 14.3. Step 3. Placement of components 197
 - 14.4. Step 4. Routing 199
 - 14.5. Step 5. Copper fills 200
 - 14.6. Step 6. Silk screen 202
 - 14.7. Step 7. Design Rules Check 204
 - 14.8. Step 8. Manufacturing 206

- Chapter 15 • Additional design considerations 207**
 - 15.1. Shape and size 207
 - 15.2. Layers 209
 - 15.3. Traces 210
 - Length 210
 - Angles 210
 - Weight 210
 - Width 211
 - Proximity 211

- Part 4: Projects 212**
- Chapter 16 • About this Part 212**
- Chapter 17 • Project 1: Design a simple breadboard power supply PCB 213**
 - 17.1. Walk through a simple project 213
 - What you will built and list of parts 213

What you will learn	217
Project repository	218
17.2. Schematic design: Eeschema	218
Step 1: Setup	219
Step 2: Symbols	220
Step 3: Arrange, Annotate, Associate	222
Arrange	222
Annotate	223
Associate	224
Step 4: Wiring	230
Step 5: Nets	233
Step 6: Electrical Rules Check	235
Step 7: Comments	235
Step 8: Netlist	236
17.3. Footprint layout in Pcbnew	237
Step 1: Setup	237
Step 2: Outline and constraints	241
Step 3: Place components	244
Step 4: Route	247
Step 5: Copper fills	250
Step 6: Silkscreen	253
Step 7: Design Rules Check	255
Step 8: Manufacture	256
17.4. Project extensions	256
Chapter 18 • Project 2: Design a small Raspberry Pi HAT	257
18.1. What you will build and list of parts	257
18.2. What you will learn	258
18.3. Project repository	258
18.4. Schematic design in Eeschema	258
Step 1: Setup	258
Step 2: Symbols	259
Step 3: Arrange, Annotate, Associate	261

Step 4: Wiring	264
Step 5: Nets.	265
Step 6: Electrical Rules Check	267
Step 7: Comments	267
Step 8: Netlist	268
18.5. Footprint layout in Pcbnew	268
Step 1: Setup.	268
Step 2: Outline and constraints	270
Step 3: Place components	271
Step 4: Route.	272
Step 5: Copper fills	273
Step 7: Design Rules Check	276
Step 8: Manufacture	277
Chapter 19 • Project 3: Arduino clone with build-in 512K EEPROM and clock . . .	280
19.1. Project details	280
19.2. Project repository	281
19.3. Schematic design in Eeschema	282
Step 1: Setup.	282
Step 2: Symbols.	282
Step 3: Arrange, Annotate, Associate	285
Step 4 and 5: Wiring and Nets	287
Step 6: Electrical Rules Check	290
Step 7: Comments	290
Step 8: Netlist	291
19.4. Footprint layout in Pcbnew	292
Step 1: Setup.	292
Step 2 + 3: Outline, constraints and component placement	293
Step 4: Route.	297
Using the autorouter two layers	297
Using the autorouter four layers	299
Two layers, or more?	301
Step 5: Copper fills	301

Step 6: Silkscreen	302
Step 7: Design Rules Check	303
Step 8: Manufacture	304
Part 5: Recipes	305
Chapter 20 • Adding a schematic symbol library in Eeschema	305
Chapter 21 • Adding a footprint library in Pcbnew	310
Chapter 22 • Using footprints offline	314
Chapter 23 • Using symbols offline	317
Chapter 24 • Create a keep-out zone	319
Chapter 25 • Creating copper fills	320
Chapter 26 • How to calculate the width of a trace	323
Chapter 27 • Custom Global Design Rules and changing the width of a trace	325
Chapter 28 • Create custom net design rules	327
Chapter 29 • How to add silkscreen text and simple graphics	329
Chapter 30 • How to add a custom logo to the silkscreen	335
Chapter 31 • How to manufacture a PCB with Oshpark	339
Chapter 32 • How to make and test Gerber files	342
Chapter 33 • How to manufacture a PCB with PCBWay	346
Chapter 34 • Rounded corners	350
Chapter 35 • Mounting holes and openings	354
Chapter 36 • Creating a new component (symbol)	359
Chapter 37 • Modifying an existing component (symbol)	368
Chapter 38 • Creating a new footprint - manually	373
Front Fabrication layer ('F.Fab')	376
Front Courtyard layer ('F.CrtYd')	382
Front Silkscreen	383
Tidy up	383
Save the footprint	384
Test the footprint	385
Chapter 39 • Creating a new footprint - using the footprint wizard	387
Chapter 40 • Modifying an existing footprint	391

- Chapter 41 • Using an autorouter 394**
- Chapter 42 • How to create a bill of materials (BoM) 398**
- Chapter 43 • How to design a custom page layout 401**
- Chapter 44 • How to use hierarchical sheets. 408**
- Chapter 45 • How to use differential pairs 412**
- Chapter 46 • Interactive router 417**
- Chapter 47 • Creating unique board edge cuts 421**
- Chapter 48 • Using Git for version control. 426**
 - Uploading your repository to Github 432
 - Authentication 435
- Chapter 49 • Creating a multi-layer PCB 437**
- Chapter 50 • How to use buses 442**
- Chapter 51 • How to update your schematic and layout (with Git). 446**
- Chapter 52 • Starting KiCad apps individually. 450**
- Chapter 53 • Creating a new version of a PCB without altering the original. . . . 452**
- Chapter 54 • Making a PCB without a schematic. 459**
- Chapter 55 • How to set a text editor and why 461**
- Chapter 56 • How to install 3D shapes 463**