

Contents

Did you find an error?11

About the author11

How to read Kicad 8 Like A Pro12

What will you need.....13

The book web page14

KiCad 815

Foreword by Wayne Stambaugh.....16

An introduction: Why KiCad?17

Part 1: Introduction21

 1.1. What is a PCB?.....21

 1.2. The PCB design process.....26

 1.3. Fabrication.....31

 1.4. Get KiCad for your operating system.....32

 1.5. Example KiCad projects.....36

Part 2: Getting started with KiCad45

 2.1. Introduction.....45

 2.2. KiCad Project Manager (main window)45

 2.3. Overview of the individual KiCad apps.....52

 2.4. Paths and Libraries60

 2.5. Create a new project from scratch64

 2.6. Create a new project from a template.....66

 2.7. KiCad on Mac OS, Linux, Windows70

 2.8. KiCad 875

Part 3: Project - A hands-on tour of KiCad - Schematic Design78

 3.1. Introduction to schematic design and objective of this section78

 3.2. Design workflows summary80

 3.3. The finished KiCad project and directory82

 3.4. Start KiCad and create a new project83

 3.5. 1 - Start Eeschema, setup Sheet88

 3.6. 2 - Add symbols95

3.7. 3 - Arrange, annotate, associate	98
3.8. 4 - Wiring	105
3.9. 5 - Nets.	106
3.10. 6 - The Electrical Rules Check	108
3.11. 7 - Comments with text and graphics	111
Part 4: Project- A hands-on tour of KiCad - Layout	115
4.1. Introduction to layout design and objective of this section.	115
4.2. 1 - Start Pcbnew, import footprints.	116
4.3. 2 - Outline and constraints (edge cut).	122
4.4. 3 - Move footprints in place	126
4.5. 4 - Route (add tracks).	129
4.6. 5 - Refine the outline	133
4.7. 6 - Silkscreen (text and graphics).	143
4.8. 7 - Design rules check.	151
4.9. 8 - Export Gerbers and order	154
4.10. The manufactured PCB	161
Part 5: Design principles and PCB terms	164
5.1. Introduction.	164
5.2. Schematic symbols.	164
5.3. PCB key terms	165
5.3.1. FR4	165
5.3.2. Traces	166
5.3.3. Pads and holes	167
5.3.4. Via	169
5.3.5. Annular ring	170
5.3.6. Soldermask	171
5.3.7. Silkscreen	171
5.3.8. Drill bit and drill hit	172
5.3.9. Surface mounted devices	173
5.3.10. Gold Fingers	173
5.3.11. Keep-out areas	174

5.3.12. Panel	175
5.3.13. Solder paste and paste stencil.	176
5.3.14. Pick-and-place	178
Part 6: PCB design workflows	180
6.1. The KiCad Schematic Design Workflow	180
6.1.1. Schematic Design Step 1: Setup	180
6.1.2. Schematic Design Step 2: Symbols	182
6.1.3. Schematic Design Step 3: AAA (Arrange, Annotate, Associate).	184
6.1.4. Schematic Design Step 4: Wire.	186
6.1.5. Schematic Design Step 5: Nets.	187
6.1.6. Schematic Design Step 6: Electrical Rules Check.	187
6.1.7. Schematic Design Step 7: Comments and Graphics.	188
6.2. The KiCad Layout Design Workflow	189
6.2.1. Layout Design Step 1: Setup	190
6.2.2. Layout Design Step 2: Outline and constraints	194
6.2.3. Layout Design Step 3: Place footprints.	195
6.2.4. Layout Design Step 4a: Route.	197
6.2.5. Layout Design Step 4b: Copper fills.	199
6.2.6. Layout Design Step 5: Silkscreen	200
6.2.7. Layout Design Step 6: Design rules check	202
6.2.8. Layout Design Step 7: Export & Manufacture	203
Part 7: Fundamental KiCad how-to: Symbols and Schematic Editor	205
7.1. Introduction.	205
7.2. Left toolbar overview	205
7.3. Top toolbar overview.	212
7.4. Right toolbar overview	226
7.5. Schematic editor preferences.	235
7.6. How to find a symbol with the Chooser	242
7.7. How to find schematic symbols on the Internet	247
7.8. How to install symbol libraries in bulk	253
7.9. How to create a custom symbol	258

7.10. How to associate a symbol with a footprint	269
7.11. Net labels	275
7.12. Net classes	279
7.13. Hierarchical sheets	285
7.14. Global labels	287
7.15. Hierarchical labels and import sheet pin	290
7.16. Electrical rules and customization	293
7.17. Bulk editing of schematic elements	298
Part 8: Fundamental KiCad how-to: Footprints and PCB Editor	304
8.1. Introduction	304
8.2. Left toolbar	305
8.3. Top toolbar	314
8.3.1. Top toolbar Row 1	314
8.3.2. Top toolbar Row 2	326
8.4. Right toolbar	329
8.4.1. Right toolbar main buttons	330
8.4.2. Right toolbar - Appearance	335
8.5. Layout editor preferences	339
8.6. Board Setup	343
8.6.1. Board Setup - Board Stackup	344
8.6.2. Board Setup - Text & Graphics	348
8.6.3. Board Setup - Design Rules and net classes	351
8.6.4. Board Setup - Design Rules - Custom Rules and violation severity	354
8.7. How to find and use a footprint	358
8.8. Footprint sources on the Internet	361
8.9. How to install footprint libraries	362
8.10. Filled zones	369
8.11. Keep-out zones	374
8.12. Interactive router	376
8.13. Length measuring tools	380
8.14. Bulk editing	382

8.15. Create a custom footprint, introduction	386
8.15.1. Create a new library and footprint	389
8.15.2. Create a footprint, 1, Fabrication layer	392
8.15.3. Create a footprint, 2, Pads	393
8.15.4. Create a footprint, 3, Courtyard layer	397
8.15.5. Create a footprint, 4, Silkscreen layer	397
8.15.6. Use the new footprint	399
8.16. Finding and using a 3D shape for a footprint	401
8.17. How to export and test Gerber files.	409
Part 9: Project - Design a simple breadboard power supply PCB	417
9.1. Introduction.	417
9.2. Schematic	422
9.2.1. 1 - Setup	422
9.2.2. 2 - Symbols	424
9.2.3. 2 - Edit Component values	426
9.2.4. 3 - Arrange, Annotate	428
9.2.5. 3 - Associate	432
9.2.6. 4 - Wiring.	433
9.2.7. 5 & 6 - Nets and Electrical Rules Check	437
9.2.8. 7 - Comments.	440
9.3. Layout.	443
9.3.1. 1 - Setup	447
9.3.2. 2 - Outline and constraints	450
9.3.3. 3 - Place footprints	455
9.3.4. 2 - Refine the outline.	459
9.3.5. 4 - Route	465
9.3.6. 5 - Copper fills	468
9.3.7. 6 - Silkscreen	470
9.3.8. 7 - Design Rules Check	476
9.3.9. 8 - Export and Manufacture	478
9.4. Finding and correcting a design defect	483

9.4.1. Fix the schematic	484
9.4.2. Fix the layout	486
Part 10: Recipes	489
10.1. Edit Text & Graphics Properties.	489
10.2. Pack & Move footprints	492
10.3. Interactive router modes	495
10.4. Create a custom silkscreen or copper graphic.	499
10.5. Bill of Materials.	508
10.5.1. Build-in BOM in the PCB editor	508
10.5.2. Build-in BOM in schematic editor.	512
10.5.3. A plug-in for BOM	514
10.6. Grid Overrides	521
10.7. Edit Teardrops	523
10.8. Buses	527
10.9. Field name templates	531
10.10. Plugin and Content Manager: Plugins	536
10.11. Plugin and Content Manager: Libraries	541
10.12. Plugin and Content Manager: Themes	544
10.13. Pcbnew Origins	547
10.14. Edit Track & Via Properties (PCB editor).	552
10.15.1. Change a symbol in bulk	555
10.16.2. Change a footprint in bulk	560
Part 11: PCB manufacturing and tools	565
11.1. NextPCB KiCad Quote and Order plugin	565
11.2. NextPCB Gerber Viewer and Analysis tool	568
11.3. NextPCB PCB Design Analysis Software for Manufacturability	572
11.4. NextPCB KiCad DFM analysis plugin	579
Index	585