

Contents

Did you find an error?	11
About the author	11
How to read Kicad Like A Pro	12
What will you need	13
The book web page	14
KiCad 8	15
Foreword by Wayne Stambaugh.	16
An introduction: Why KiCad?	17
About Tech Explorations.	21
1: Project - Tiny Solar Power Supply	23
1.1. Introduction.	23
1.2. Schematic	26
1.2.1. 1 - Setup	27
1.2.2. 2 - Symbols	28
1.2.3. 2 - Edit Component values	32
1.2.4. 3 - Arrange, Annotate, Associate.	34
1.2.5. 4 - Wiring.	36
1.2.6. 5 & 6 - Nets and Electrical Rules Check	38
1.2.7. 7 - Comments.	39
1.3. Layout.	41
1.3.1. 1 - Setup	42
1.3.2. 2 & 3 - Placement and outline.	44
1.3.3. 4 - Route	48
1.3.4. 2 - Refine the outline.	50
1.3.5. 5 - Copper fills	51
1.3.6. 6 - Silkscreen	56
1.3.7. 7 - Design Rules Check	61
1.3.8. Two more things: 3D models and mounting holes	63
1.3.9. 8 - Export and Manufacture	67
2: Project - A 4 x 8 x 8 LED matrix array	73

2.1. Introduction	73
2.2. Schematic design	78
2.2.1. 1 - Setup	78
2.2.2. 2 - Symbols	81
2.2.3. 3 - Arrange, Annotate	83
2.2.4. 3 - Associate	86
2.2.5. 4 - Wiring	88
2.2.6. 5 - Nets	92
2.2.7. 6 - Electrical Rules Check	94
2.2.8. 7 - Comments	96
2.2.9. Last-minute edits	98
2.3. Layout design editing	100
2.3.1. 1 - Setup	101
2.3.2. 2 - Outline and constraints	103
2.3.3. 3 - Place components	108
2.3.4. 2 - Refine outline	115
2.3.5. 3 - Move footprints	118
2.3.6. 4 - Route	122
2.3.7. 4 - Copper fills	126
2.3.8. 5 - Silkscreen	129
2.3.9. 6 - Design Rules Check	136
2.3.10. 7 - Manufacture	142
2.5. Bonus - 3D shapes	144
2.6. Bonus - Found a bug in the schematic! (and fix).	151
2.7. Assembled PCB	154
3 : Project - MCU datalogger	158
3.1. Project - Introduction	158
3.2. Create the new project and Git repository	162
3.3. Schematic design	165
3.3.1. Schema 1 - Setup	165
3.3.2. Schema 2 - Symbols	167

3.3.3. Schema 2 - Sheet two	170
3.3.4. Schema 3 - Arrange, Annotate	172
3.3.5. Edit component values.	173
3.3.6. Schema 3 - Associate	175
3.3.7. Schema 4 - Wiring of sheet 1	177
3.3.8. Schema 4 - Wiring of sheet 2	179
3.3.9. Schema 5 - Nets	183
3.3.10. Schema 6 - Electrical Rules Check	184
3.3.11. Schema 7 - Comments.	186
3.4. Create the 2-layer branch in Git	188
3.5. Layout design	189
3.5.1. Layout 1 - Setup	189
3.5.2. Layout 2 - Outline and constraints.	191
3.5.3. Layout 3 - Place components	193
3.5.4. Layout 2 - Outline refinement.	194
3.5.5. Layout 4 - Route	196
3.5.6. Layout 4 - Copper fills	201
3.5.7. Layout 4 - Routing improvements	207
3.5.8. Layout 5 - Silkscreen	210
3.5.9. Layout 6 - Design Rules Check	216
3.5.10. Layout 7 - Manufacture	217
3.6. 3D shapes	222
3.7. Merge 2-layer branch to main	225
3.8. Design 4 Layer PCB in new Git branch.	227
3.9. Four-layer PCB routing	228
3.10. Four-layer PCB manufacturing	231
3.11. Updating layout from changes to the schematic with Git	234
3.12. Finding and correcting a design defect.	237
3.12.1. Fix the schematic	239
3.12.2. Fix the 2 layer PCB layout.	246
3.12.3. Fix the 4 layer PCB layout.	249

4 : Project - An ESP32 clone	252
4.1. Project - Introduction	252
4.2. Schematic design	258
4.2.1. Schema 1 - New KiCad project and Schematic Setup.	258
4.2.2. Schema 2 - Symbols	260
4.2.3. Schema 3 - Annotate and set component values.	263
4.2.4. Schema 3 - Arrange	267
4.2.5. Schema 3 - Associate	269
4.2.6. Schema 4 - Wiring	270
4.2.7. Schema 5 - Nets and Net Classes	275
4.2.8. Schema 6 - Electrical Rules Check.	275
4.2.9. Schema 7 - Comments	279
4.3. Layout design	281
4.3.1. Layout 1 - Setup	283
4.3.2. Layout 2 - Outline and constraints.	286
4.3.3. Layout 3 - Place components	291
4.3.4. Layout 2 - refine outline and DRC errors	297
4.3.5. Layout 4 - Route	304
4.3.6. Layout 4 - Copper fills and keep out areas	308
4.3.7. Layout 5 - Silkscreen	312
4.3.8. Layout 4 - Routing improvements	314
4.3.9. Layout 6 - Design Rules Check	315
4.3.10. Layout 7 - Manufacture	316
4.4. 3D shapes	321
4.5. Finding and correcting a design defect	323
4.5.1. Fix the schematic	324
4.5.2. Fix the layout	325
5: Recipes	328
5.1. Interactive delete	328
5.2. Find and Replace (Schematic editor).	330
5.3. Text variables.	332

5.4. Board Setup - pre-defined sizes for tracks and vias	337
5.5. Board Setup - Design rules violation severity	341
5.6. Board Setup - Custom design rules.	344
5.7. Schematic Setup - Electrical Rules and violation severity.	348
5.8. Schematic Setup - Electrical Rules and Pin conflicts map.	351
5.9. Import components from Snapeda	354
5.10. The Freerouting autorouter	362
5.10.1. Freerouting plugin installation and operation.	365
5.10.2. Using Freerouting	366
5.10.3. Autorouting process and customisation	369
5.11. PCB editor Inspection menu	372
5.12. Single track and differential pair routing	381
5.13. Track length tuning.	385
5.14. Differential pair skew tuning.	389
5.15. The footprint wizard	391
5.16. Pin and wire highlighter tool.	396
5.17. KiCad project management with Git	398
5.17.1. Install Git.	403
5.17.2. Git configuration	404
5.17.3. Create a new KiCad project Git repository	405
5.17.4. How to ignore files.	407
5.17.5. Basic Git commands: add, commit.	410
5.17.6. Basic Git commands: branch.	414
5.17.7. Basic Git commands: merge	417
5.18. Sharing your KiCad project on GitHub	419
5.19. Customize the editor color scheme	428
5.20. Import an EAGLE, Altium, or Cadstar schematic	432
5.21. The circuit simulator	435
5.21.1. Prepare the circuit for simulation.	437
5.21.2. Configure the simulator	444
5.21.3. Simulate	446

5.22. KiCad project templates	452
5.22.1. Using a system project template	453
5.22.2. Create a user project template	456
5.23. Share projects and configurations with others	458
5.24. Calculate the width of a trace.	463
5.25. Design a custom schematic sheet.	464
5.26. Python scripting API	472
5.26.1. A simple example	473
5.26.2. Make a change to a PCB.	479
5.26.3. Create a KiCad Action Plugin.	482
5.26.4. Learning More About KiCad Python Scripting API.	487
Index	488