

4th edition 2024
covering KiCad 8

e^lektor books

LIKE
A PRO

KiCad

Advanced Projects and Recipes

Mastering PCB design
with real-world projects



 **HQNextPCB**
www.nextpcb.com

Peter Dalmaris, PhD

 **elektor**
design > share > learn

KiCad Like A Pro

Advanced Projects and Recipes

Mastering PCB design with real-world projects

4th edition covering KiCad 8



Peter Dalmaris, PhD

-
- This is an Elektor Publication. Elektor is the media brand of

Elektor International Media B.V.

PO Box 11, NL-6114-ZG Susteren, The Netherlands

Phone: +31 46 4389444

● All rights reserved. No part of this book may be reproduced in any material form, including photocopying, or storing in any medium by electronic means and whether or not transiently or incidentally to some other use of this publication, without the written permission of the copyright holder except in accordance with the provisions of the Copyright Designs and Patents Act 1988 or under the terms of a licence issued by the Copyright Licensing Agency Ltd., 90 Tottenham Court Road, London, England W1P 9HE. Applications for the copyright holder's permission to reproduce any part of the publication should be addressed to the publishers.

● Declaration

The Author and Publisher have used their best efforts in ensuring the correctness of the information contained in this book. They do not assume, and hereby disclaim, any liability to any party for any loss or damage caused by errors or omissions in this book, whether such errors or omissions result from negligence, accident, or any other cause.

All the programs given in the book are Copyright of the Author and Elektor International Media. These programs may only be used for educational purposes. Written permission from the Author or Elektor must be obtained before any of these programs can be used for commercial purposes.

- British Library Cataloguing in Publication Data

A catalogue record for this book is available from the British Library

- This third edition of **KiCad Like a Pro** consists of two books:

- **KiCad Like A Pro | Fundamentals and Projects**

Getting started with the world's best open-source PCB tool

590 pages

ISBN: 978-3-89576-626-8 print

ISBN: 978-3-89576-627-5 ebook

- **KiCad Like A Pro | Advanced Projects and Recipes**

Mastering PCB design with real-world projects

492 pages

ISBN: 978-3-89576-628-2 print

ISBN: 978-3-89576-629-9 ebook

- © Copyright 2024: Elektor International Media B.V.

Prepress Production: D-Vision, Julian van den Berg

Printed in the Netherlands by Ipskamp Printing, Enschede

Elektor is the world's leading source of essential technical information and electronics products for pro engineers, electronics designers, and the companies seeking to engage them. Each day, our international team develops and delivers high-quality content - via a variety of media channels (including magazines, video, digital media, and social media) in several languages - relating to electronics design and DIY electronics. www.elektormagazine.com

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 templates452
 - 5.22.1. Using a system project template453
 - 5.22.2. Create a user project template456
- 5.23. Share projects and configurations with others458
- 5.24. Calculate the width of a trace.463
- 5.25. Design a custom schematic sheet.464
- 5.26. Python scripting API472
 - 5.26.1. A simple example473
 - 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
- Index488**