



Kicad - PL Editor Reference Manual (Paperback)

By Jean-Pierre Charras

Samurai Media Limited, United States, 2015. Paperback. Condition: New. Language: English . Brand New Book ***** Print on Demand *****.KiCad is an open source software suite for electronic design automation (EDA). It facilitates the design of schematics for electronic circuits and their conversion to PCBs (printed circuit board) design. KiCad was originally developed by Jean-Pierre Charras, and features an integrated environment for schematic capture and PCB layout design. This Reference Manual focuses on PL Editor, the Page Layout Editor. There are other books in this series for tools like cvpcb, pcbnew, gerbview and more.



READ ONLINE

[7.17 MB]

DOWNLOAD



Reviews

It is one of the most popular publications. It really is filled with knowledge and wisdom. It has been designed in an exceedingly straightforward way and it is merely soon after I finished reading this pdf by which it actually transformed me, affecting the way in my opinion.

-- **Gerardo Rath**

This publication is fantastic. It really is full of knowledge and wisdom. You are going to like just how the author wrote this publication.

-- **Harmon Watsica II**