



Kicad Reference Manual

How to use Kicad

Copyright

This document is Copyright © 2010–2011 by its contributors as listed below. You may distribute it and/or modify it under the terms of either the GNU General Public License (<http://www.gnu.org/licenses/gpl.html>), version 3 or later, or the Creative Commons Attribution License (<http://creativecommons.org/licenses/by/3.0/>), version 3.0 or later.

All trademarks within this guide belong to their legitimate owners.

Contributors

Jean-Pierre Charras, Fabrizio Tappero.

Feedback

Please direct any comments or suggestions about this document to the KiCad mailing list:
<https://launchpad.net/~kicad-developers>

Acknowledgments

None

Publication date and software version

Published 22 September 2011. Based on LibreOffice 3.3.2.

Note for Mac users

None

Table of Contents

1 - Introduction.....	3
Description.....	3
2 - Installation and configuration.....	3
Display options.....	3
Initialisation of the default configuration.....	3
Kicad: principles of use.....	4
3 - Use Kicad.....	4
Main window.....	4
Utility launch pane.....	5
Project tree view.....	5
Top toolbar.....	5

1 - Introduction

Description

The Kicad software suite is a group of programs for schematics and PCBs that is available for the following operating systems:

- LINUX
- Windows XP/2000
- Apple OS X

The kicad utility program is a project manager that facilitates the use of the various programs required to draw schematics, layout PCBs, generate and check the production files.

Software tools included in kicad are:

- Eeschema: Schematic editor.
- Pcbnew: PCB editor.
- Cvpccb: enables the association of schematic components with physical modules (packages) for placement on the PCB.
- Gerbview: Gerber files viewer

2 - Installation and configuration

Display options

It is recommended to set your display/graphics card to use 24 or 32 bits per pixel.

The 16-bit mode will work for Eeschema, but in Pcbnew the display will not function correctly under Linux.

Initialisation of the default configuration

A default configuration file (kicad.pro) is supplied in kicad/template. It serves as the template for each new project. It can be modified or added to if necessary, usually for the list of libraries to load. Run Eeschema via kicad or directly (Linux command; /usr/local/kicad/bin/eeschema). Update the configuration and then save it in /usr/local/kicad/template/kicad.pro

Kicad: principles of use

In order to manage a project, i.e. all the files it constitutes (representing schematics, printed circuit boards, supplementary libraries, manufacturing files for phototracing, drilling and automatic component placement), it is recommended to create a project:

- Create a working directory for the project (using kicad or by other means).
- In this directory, use kicad to create the project file (file .pro) via the “Load existing project” icon.

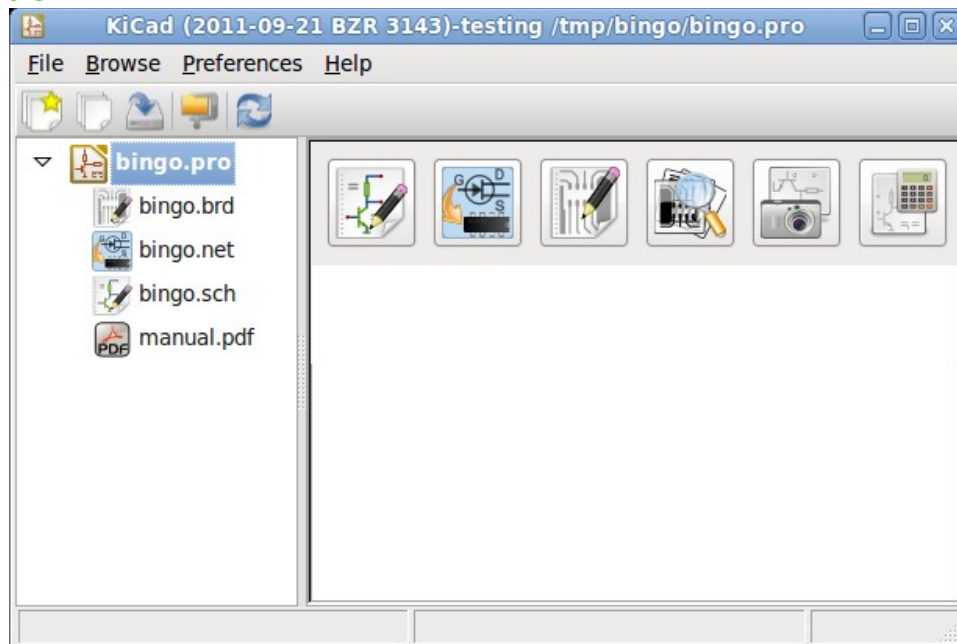
It is strongly recommended to use the same name for the project and its directory.

Kicad creates a file with a .pro extension that maintains a number of parameters relating to project management (such as the filename of the principal schematic, list of libraries used in the schematics and PCBs). The default names of both the principal schematic and the printed circuit board are based upon the name of the project. Thus, if a project called example was created in a directory called example, the default files created would be:

example.pro	project management file.
example.sch	principal schematic file.
example.brd	printed circuit board file.
example.net	netlist file.
example.xxx	various files created by the other utility programs.
example.cache.lib	cache file of the libraries used in the schematic (backup of the components used)

3 - Use Kicad

Main window



The main Kicad window is composed of a project tree view, a pane containing buttons used to run the various utilities, and a message window. The menu and the toolbar can be used to create, read and save project files.

Utility launch pane



These buttons correspond to the following commands (from left to right):

- 1 Run Eeschema
- 2 Run Cvp pcb
- 3 Run Pcbnew
- 4 Run Gerbview
- 5 Run Bitmap2component
- 6 Pcb Calculator

Project tree view



Double-clicking on the Eeschema icons runs the schematic editor which in this case will open the file bingo.sch.

Double-clicking on the Pcbnew icon runs the layout editor, in this case opening the file bingo.brd.

Right clicking on any of the files in the project tree allows generic files manipulation.

Top toolbar



From left to right.

- 1 Create a configuration file for a new project. If the template kicad.pro is found in kicad/template, it is copied into the working directory.
- 2 Open an existing project.
- 3 Update (save) the current configuration.
- 4 Create a zip archive of the whole project (schematic files, libraries, pcb, etc).
- 5 Redraw the tree list (useful after a tree change)