Kicad

Reference manual

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 on September 27, 2011. Based on LibreOffice 3.3.2.

Note for Mac users

The kicad support for the Apple OS X operating system is experimental.

Table of Contents

1 -   Introduction

Kicad

Kicad is an open-source software tool for the creation of electronic schematic diagrams and PCB artwork. Beneath its singular surface, kicad incorporates an elegant ensemble of the following stand-alone software tools:

- Kicad : project manager
- EESchema : schematic editor
- Cvpcb: footprint selector
- PCBnew : circuit board layout editor
- GerbView: Gerber viewer
- Bitmap2Component : component maker

At the time of writing, kicad can be considered mature enough to be used for the successful development and maintenance of complex electronic boards. Kicad does not present any board-size limitation and it can easily handle up to 16 copper layers and up to 12 technical layers. Kicad can create all the files necessary for building printed boards, Gerber files for photo-plotters, drilling files, component location files and a lot more. Being open source (GPL licensed), kicad represents the ideal tool for projects oriented towards the creation of electronic hardware with an open-source flavour.

On the Internet, the home of kicad is:

http://kicad.sourceforge.net/wiki/Main_Page

http://iut-tice.ujf-grenoble.fr/kicad/index.html

http://www.gipsa-lab.inpg.fr/realise_au_lis/kicad/index.html

Kicad is available for Linux, Windows and Apple OS X (experimental).

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 named *kicad.pro* is supplied in kicad/template. It serves as a template for any new project. The default file *kicad.pro* can be freely modified if necessary, for instance to load other libraries files. Run Eeschema via kicad or directly run the 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 kicad project: schematic files, printed circuit board files, supplementary libraries, manufacturing files for photo-tracing, drilling and automatic component placement files, it is recommended to create a project as follows:

• Create a working directory for the project (using kicad or by other means).

• In this directory, use kicad to create a project file (file with extension .pro) via the "Start a new project" icon.

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

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

|	
example.pro	project management file.
example.sch	main 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 libraries used in the schematic (backup of the components used)

3 - Use Kicad

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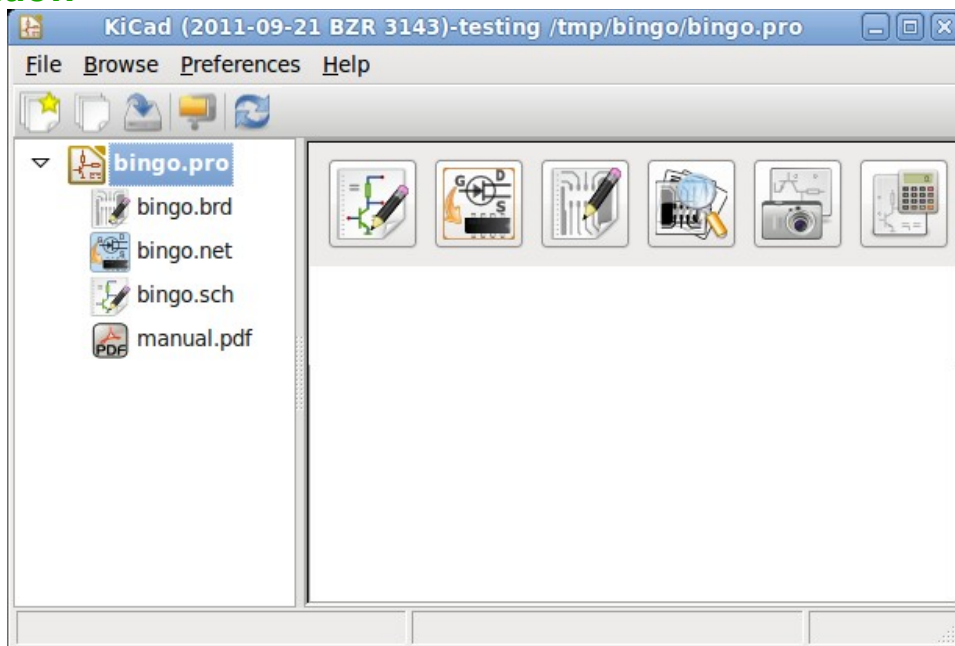
The main kicad window is composed of a project tree view, a launch pane containing buttons used to run the various software tools, and a message window. The menu and the toolbar can be used to create, read and save project files.

Utility launch pane
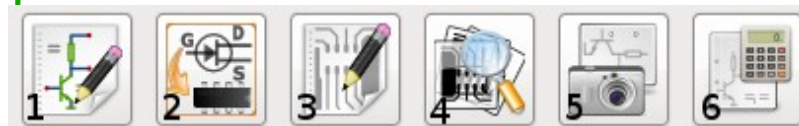
Kicad allows you to run all stand alone software tools that come with it. The launch pane is made of the above 6 buttons that correspond to the following commands (from left to right):

1	Eeschema
2	Cvpcb
3	Pcbnew
4	Gerbview
5	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

Kicad top toolbar allows for some basic files operation (from left to right).

1   Create a project file. If the template kicad.pro is found in kicad/template, it is copied into the working directory.

2   Open an existing project.

3   Update and save the current project tree.

4   Create a zip archive of the whole project. This includes schematic files, libraries, pcb, etc.

5   Redraw the tree list, useful after a tree change.