Headings:

1 - Introduction

1.1 - Description

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

- LINUX
- Windows XP / 2000 / Seven
- In progress: MacOSX

The *kicad* utility program is a project manager that facilitates the use of the various programs required to draw schematics, lay out PCBs, generate and check the production files.
The programs included are:

- *Eeschema*: the schematic editor.
- *Pcbnew*: the PCB editor.
- *Cvpcb*: enables the association of schematic components with physical modules (packages) for placement on the PCB.
- *Gerbview*: is used to visualize Gerber files.

Also two tools are include:

- Bitmap2component that can create a logo from a bitmap picture (this logo is a schematic component or a footprint)
- PcbCalculator. This tool is usefull to calculate resistors for regulators, track widths versus currents, track witdths in transmission lines ...

2 - Installation and configuration

2.1 - 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**

2.2 - Kicad: principles of use

In order to manage simply 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 icon.

> **It is strongly recommended to use the same name for the project and its directory**.
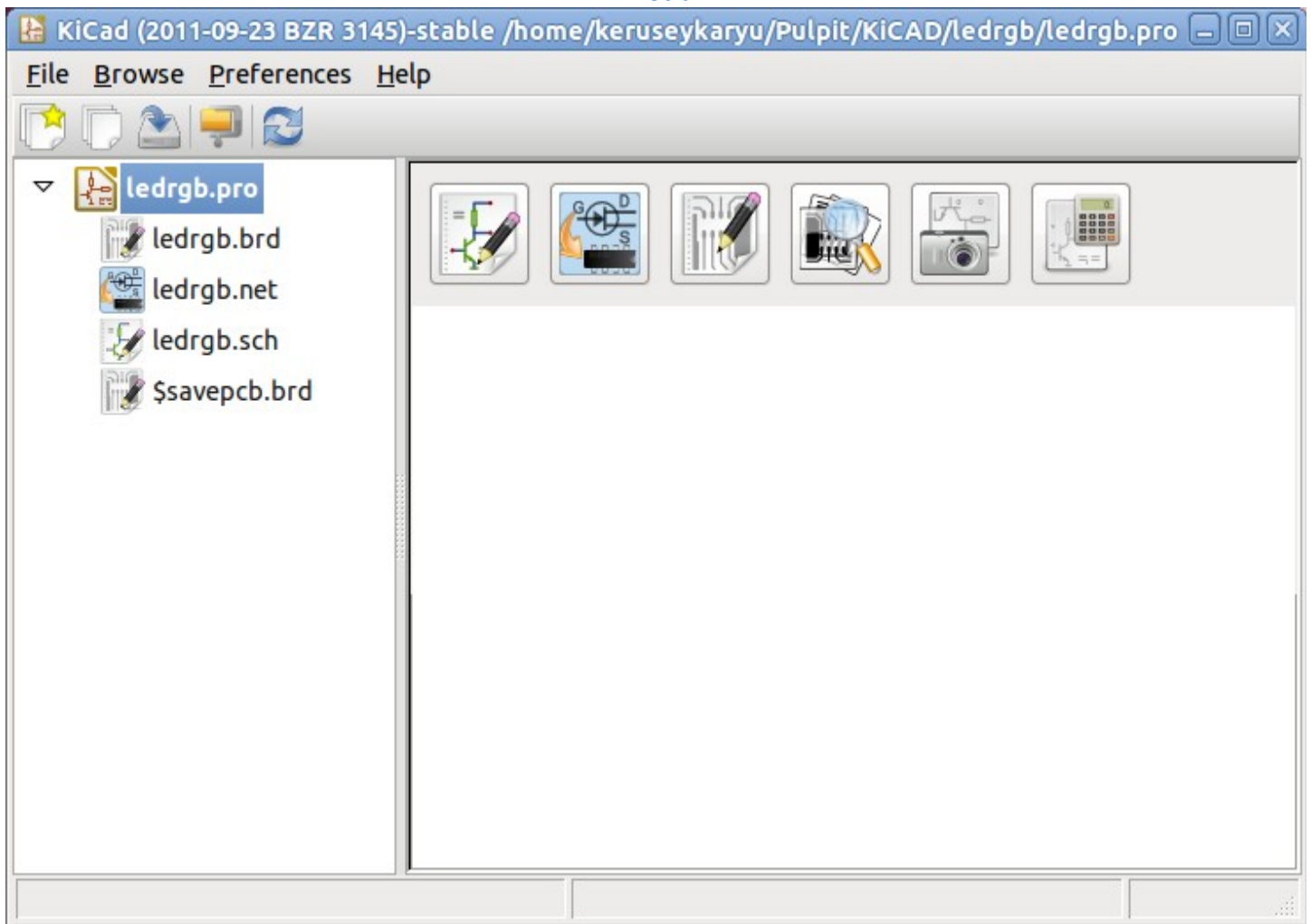
Kicad creates a file with a .pro extension that maintains a number of parameters relating to project management (such as the filename of the principal schematic, list of libraries used in the schematics and PCBs). The default names of both the principal schematic and the printed circuit board are based 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:

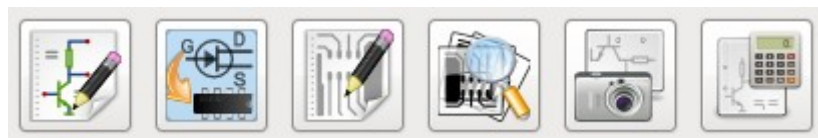|	
example.pro	project management file.
example.sch	principal schematic file.
example.brd	printed circuit board file.
example.net	netlist file.
example.xxx	various files created by the other utility programs.
example-cache.lib	cache file of the libraries used in the schematic (backup of the components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

3 - Use

3.1 - Main window

The main window is composed of a tree view of the project, 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 (*.pro).
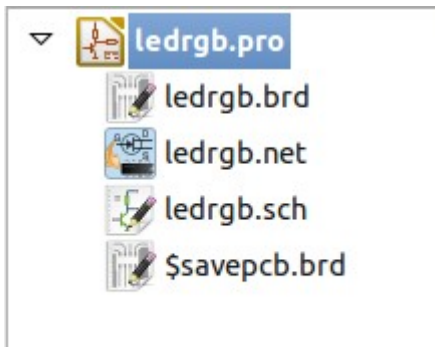

KiCad (2011-09-23 BZR 3145)-stable /home/keruseykaryu/Pulpit/KiCAD/ledrgb/ledrgb.pro

File   Browse   Preferences   Help

- ▽ ledrgb.pro
  - ledrgb.brd
  - ledrgb.net
  - ledrgb.sch
  - $savepcb.brd

3.2 - Utility launch pane

The buttons correspond to the following commands:

	Run **EEschema**		Run **Gerbview**
	Run **CvPcb**		Run **Bitmap2Component**
	Run **PCBnew**		Run **PCB Calculator**

3.3 - Project tree view

Double-clicking on ![icon] runs the schematic editor, in this case opening the file interf_u.sch.

Double-clicking on ![icon] runs the layout editor, in this case opening the file interf_u.brd.

Right clicking allows files operations

3.4 - Toolbar:

	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.
	Open an existing project.
	Update (save) the current configuration.
	Create a zip archive of the whole project (schematic files, libraries, pcb, etc).
	Redraw the tree list (useful after a tree change)