



# KiCad Schematic and PCB layout SW

KIPTON MORAVEC AE5IB

# KiCad

- ▶ Looked at a number of free PCB layout programs that worked on Linux.
- ▶ KiCad met all of the requirements.
  - ▶ Easy to use.
  - ▶ Runs on Linux
  - ▶ A build for the Raspberry Pi
- ▶ Get the program
- ▶ `sudo apt-get install kicad extra-xdg-menus kicad-doc-en`
- ▶ Or build latest one:  
<http://www.kicad-pcb.org/display/DEV/Building+KiCad+on+Linux>
  - ▶ I got an error on the script when I tried to run it.
  - ▶ The source code loads at about 40 kB/s so start it and go to bed.
  - ▶ On raspberry pi edit `/etc/apt/sources.list.s/raspi.list` and remove the comment on deb-src line
    - ▶ Still cannot get it to work

# KiCad

- ▶ With KiCad you can create schematic diagrams and printed circuit board up to 32 copper layers.
- ▶ KiCad comes with a rich set of libraries with 3D models as well.
- ▶ KiCad is a mature EDA software tool under active development by a team of developers and a vibrant user group.
- ▶ KiCad includes a project manager and four main independent software tools:
  - ▶ **Eeschema**, schematic editor.
  - ▶ **Pcbnew**, printed circuit board editor
  - ▶ **Gerbview**, Gerber file viewer.
  - ▶ **Cvpcb**, footprint selector for components association

# KiCad Programs

- ▶ Start Menu -- Electronics -> KiCad
- ▶ Steps:
  - ▶ Build Parts for Schematic
  - ▶ Enter Schematic
  - ▶ Create netlist
  - ▶ Make parts not in layout
  - ▶ Map parts to layout
  - ▶ Place parts & Connect
  - ▶ Create Gerber files

# KiCad in 3 Steps

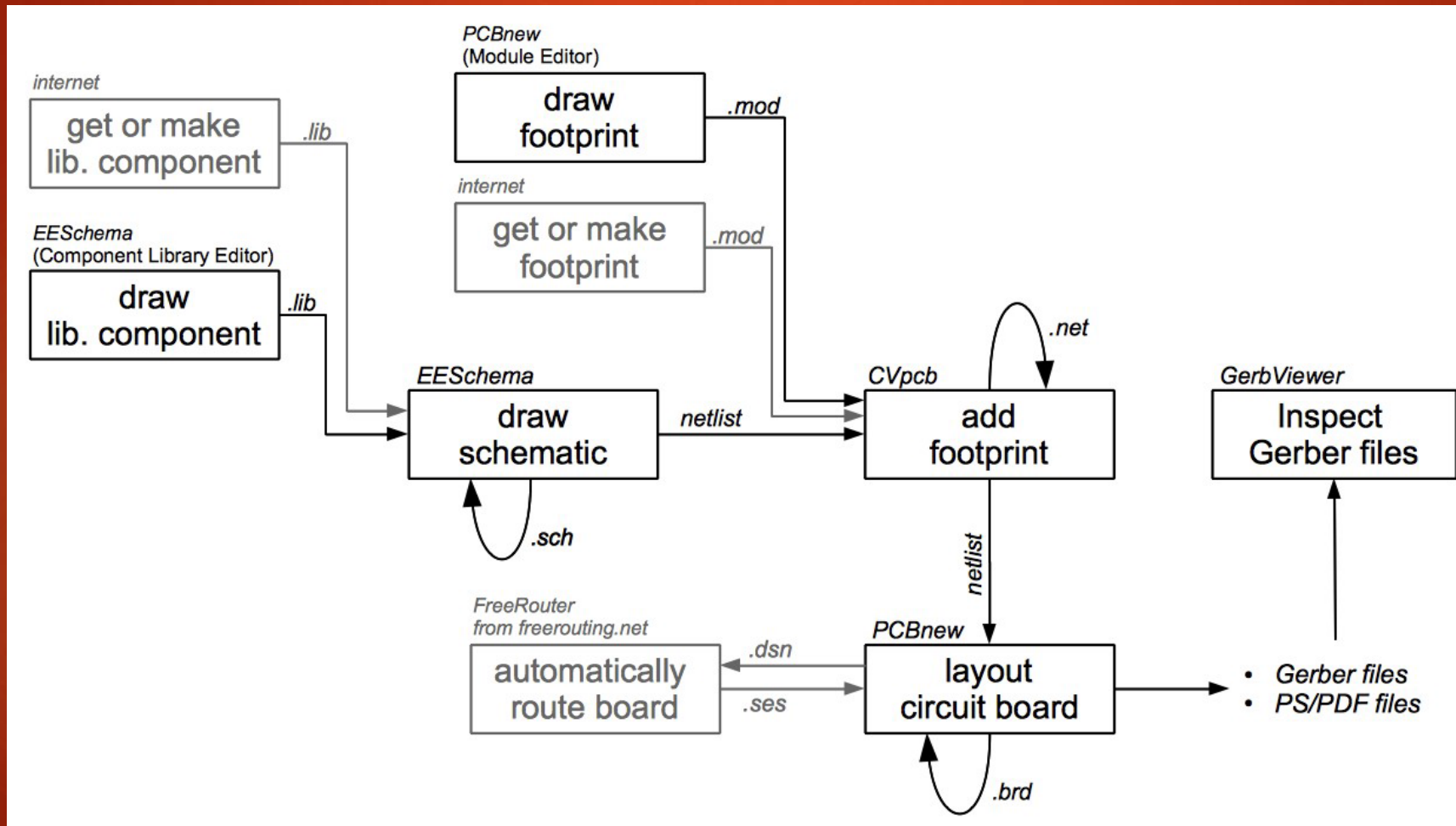
- ▶ **STEP 1, schematic capture.** With the KiCad schematic editor Eeschema, you can create a sophisticated electronic sheet or a group of hierarchical sheets. Several schematic components come with the default KiCad library. A Electrical Rules Check tool (ERC) is available.
- ▶ **STEP 2, component association.** Pcbnew allows you to associate each single schematic component with its footprint component. A very large component footprint library comes with KiCad.
- ▶ **STEP 3, PCB layout.** The Pcbnew board editor can handle up to 16 copper layers plus 12 technical layers (silk screen, solder mask, etc.) and allows you to layout the final printed circuit board.
- ▶ KiCad comes with additional software tools.
  - ▶ Two library component editors allow you to create or modify schematic components and footprint components.
  - ▶ The 3D viewer allow you to render a final 3D model of your PCB.
  - ▶ Pcbnew and Gerbview are used for the generation of the necessary compliant files for manufacturing your printed circuit board (GERBER files for photo-plotters, drilling files and component location files).
  - ▶ Postscript or PDF file generation is also possible.



# KiCad Libraries

- ▶ Kicad comes with a large set of open-source library components. A text-based format is used for both schematic and PCB component. This allow the direct editing of your library files with any text-based software.
- ▶ Both Eeschema and Pcbnew have a library manager as well as a library component editor for modifying and creating components and footprint parts. You can create, edit, delete or exchange easily library items. Documentation files can be associated to components and footprints, and key words, allowing a fast search by function.
- ▶ Very large Libraries, created over many years, are available for schematic components and footprints. Most of printed board modules (footprint) are available with their 3D shape model.
- ▶ The other very exciting aspect pf Kicad is that library components for both PCB and schematic are actually **plain text files**. As natural consequence, you can generate library components in many ways, for instance by using a great [library component generator](#).

# KiCad Parts



# Templates

- ▶ The documentation shows how templates work, with KiCad
  - ▶ [http://docs.kicad-pcb.org/KiCad/KiCad.html#\\_using\\_templates](http://docs.kicad-pcb.org/KiCad/KiCad.html#_using_templates)
- ▶ Looks cool, but could not get it to work
  - ▶ If you figure out the secret let me know
- ▶ Copied template to my directory and wrote over it.



# Getting started -- Preferences

- ▶ In KiCAD IDE they want your default text editor.
- ▶ You have to locate it and tell it where it is.
- ▶ I went with what is there leafpad
  - ▶ `/usr/share/raspi-ui-overrides/applications/leafpad.desktop`
  - ▶ Simple intuitive
- ▶ Asks for favorite pdf reader
- ▶ Pi has `/usr/share/raspi-ui-overrides/applications/xpdf.desktop`

# Schematics

- ▶ Make sure you have enough capacitance on the board
  - ▶ Have a bypass capacitor on every power input line
- ▶ Make sure you allow the ability to break the board in sections on the first prototype.
  - ▶ Especially separate power supplies.
- ▶ Do not be afraid of surface mount
  - ▶ Use SO parts and resistors and capacitors 0805 or larger.
  - ▶ On QFP use biggest package
- ▶ DO NOT USE BGA – hard to design, requires multiple layers, very fine traces, and special equipment for soldering, gold flash

# Parts placement

- ▶ Place parts on the board and determine best orientation for the parts
- ▶ Place parts together that are connected together.
- ▶ Put a capacitor on the power lines for each part.
- ▶ Try to make all parts of the same type have same orientation

# Copper Layers

- ▶ Boards can have 1 copper layer (single sided) or have an even number of layers, 2, 4, 6, 8, ...
  - ▶ A layer has copper and no copper
- ▶ Most common for hobbyists is 2 sided board.
- ▶ Unless you are doing high volume 1 layer is not cheaper than 2 layer.
- ▶ The next step is 4 layers usually with a ground plane and a power plane on the inner layers.
- ▶ The more dense the board the more layers you may need.
  - ▶ Most I have seen is 22 layers.

# Copper

- ▶ Copper is usually measured by oz/ft<sup>2</sup>, they usually just say oz
- ▶ Most common is 1 oz on the outer layers and 0.5 oz on inner layers.
- ▶ Check what your manufacturer does, could be different
- ▶ 2oz, 3 oz, and 4 oz are available for a premium for higher current
  - ▶ Minimum widths are bigger
- ▶ Decide to use minimum width and spacing for the board
  - ▶ 10 mil traces, 10 mil spacing
  - ▶ 8 mil traces and 7 mil spacing
  - ▶ Some will go to 5 mil traces and 5 mil spaces (and smaller for a premium)
- ▶ Decide what layer will be horizontal and what layer will be vertical



# Wider copper means more current

14

Kipton Moravec AE51B  
6/22/2015

- ▶ The more current the thicker the trace needed.
- ▶ Too much current in too thin trace and you make a fuse.
- ▶ You want less than 10° C rise in the traces
  - ▶ Many tables are based on 10° C rise
- ▶ Google search for Trace Width Calculator
- ▶ Also search PCB Via calculator for vias
  - ▶ Similar issues
- ▶ KiCAD has a copper calculator but does not have an oz mode.
- ▶ Example 1 oz outside layer
  - ▶ 8 mil < 750 mA
  - ▶ 10 mil < 880 mA
  - ▶ 12 mil < 1 A
  - ▶ 31 mil < 2 A
  - ▶ 55 mil < 3 A
  - ▶ 109 mil < 5 A
  - ▶ 283 mil < 10 A

# Layout Basics

15

Kipton Moravec AE5IB  
6/22/2015

- ▶ One layer is mostly horizontal traces other mostly vertical traces
- ▶ Vias are used to go between layers.
  - ▶ Plated through hole with connections on each layer
  - ▶ Make sure they are sized for current
- ▶ On a layer make no 90 degree angles with traces. Bevel all corners.
  - ▶ For RF circuits round all corners
- ▶ Layout power and ground first
- ▶ Then work on the most dense areas and critical traces
- ▶ Finally work one side to the other.

# Leaded versus Lead-free

16

- ▶ The parts are all lead free
- ▶ The board will be lead free
- ▶ Use lead solder
  - ▶ Lead free needs hotter soldering iron
  - ▶ Lead free does not solder as easily

# EESchema Schematic Editor

17

Kipton Moravec AE51B  
6/22/2015

- ▶ Draw Schematic
  - ▶ Load parts
  - ▶ Rotate parts
  - ▶ Label parts
  - ▶ Dealing with power
- ▶ Annotate Sheet
- ▶ Checks Schematic
- ▶ Create netlist

# CvPcp

- ▶ Match footprints to the part
  - ▶ The step I do not like prefer the part to have the footprint so I build my library one time.
  - ▶ Does not filter enough
    - ▶ Why offer you a big BGA when the part has 2 pins?



# PCB Layout

- ▶ Loads the parts on the board
- ▶ You have to arrange the parts
- ▶ Decide which layer will be horizontal, which will be vertical.
- ▶ Then start hooking up parts.
  - ▶ I usually start with ground first
  - ▶ Connect the parts with the most current first with traces thick enough for the maximum current
  - ▶ Next rest of the power signals
  - ▶ Next, most dense areas
  - ▶ Finally the rest of the board. I usually start at one corner down to the opposite corner.