Design your own PCB!

(with expert support and tutelage)

Join us in Northern Virginia for a 2 ½ day introductory workshop and design your own 100% custom PCB. The last time Anool ran this workshop, EVERY student successfully sent a design out for manufacture.

DATES: September 13, 14 and 15, 2013

COST: $300

What is the money used for? The tution covers coffee, lunches and snacks. And also helps to offset Anool's (the instructor) travel expenses. A portion of the proceeds will be donated to support KiCAD development (we donated $500 last time).

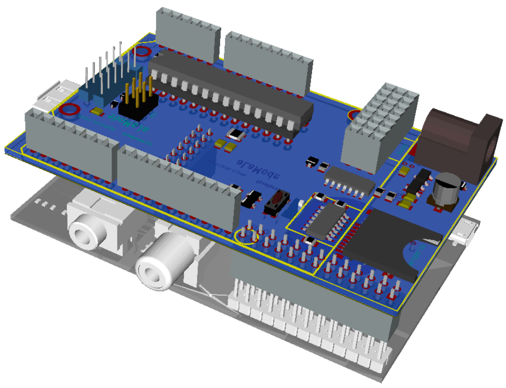
What software will we use? KiCAD. It is free and Open Source and amazing.

About the Instructor:

ANOOL J M (MUMBAI, INDIA)

Engineer by profession (test & measurement + Video Conferencing), dabbler in Astronomy, Origami, Photography, General Tinkering, DiY, and of course Cycling. He braves the roads and streets daily on his bicycle commute to work, which is quite rare (practically unheard of) in Mumbai. You can witness his harrowing commute. Projects: Somehow, between all of his hobbies and cycling, he manages to discuss and create original Open Source circuit boards and projects. As a master of digital design, Anool is the driving force behind WyoLum. Anool is also a regular blogger for Hack-a-Day.

Who this workshop is for?



This workshop is tailored for the electronic enthusiast who has been breadboarding circuits but has never used an Electronic Design Automation (EDA) / computer aided design (CAD) program.
Looking to for the freedom to design any board size you wish and shed the design restrictions in other EDA/CAD programs like Eagle. Last class two middle school students were top of the class.

Why KiCad?

KiCad is an amazing Open Source (free as in freedom!) program that allows you to design your own printed circuit board without arbitrary limitations on board size. KiCad offers an all-in-one solution:

- Full featured Electronic Design Automation (EDA) suite, with a free and open source GNU GPL license
- Cross-platform - Linux, Windows, Mac OSX
- Initial release 1992 :-) extremely stable and mature
- File formats - exports to many common formats like Postscript, Scalable Vector Graphics, Gerber RS274D/X, HPGL, DXF.
- Large, dedicated support community.
- Extensive Documentation, FAQ's, Mailing List, Tutorials, IRC
- Coders can contribute to development, localization and bug testing
- 0.6M lines of code, 150 man-years of devp., almost $9M

What should I bring?
Please bring
  - a laptop we will get KiCAD installed on your machinee on Friday night.
  - A schematic of the circuit you'd like to turn into a PCB. This can be a hand drawn diagram.

Planned KiCad Workshop at NovaLabs

2 ½ Days - decide on a project to be completed during the Workshop (Power Supply, Shield etc)

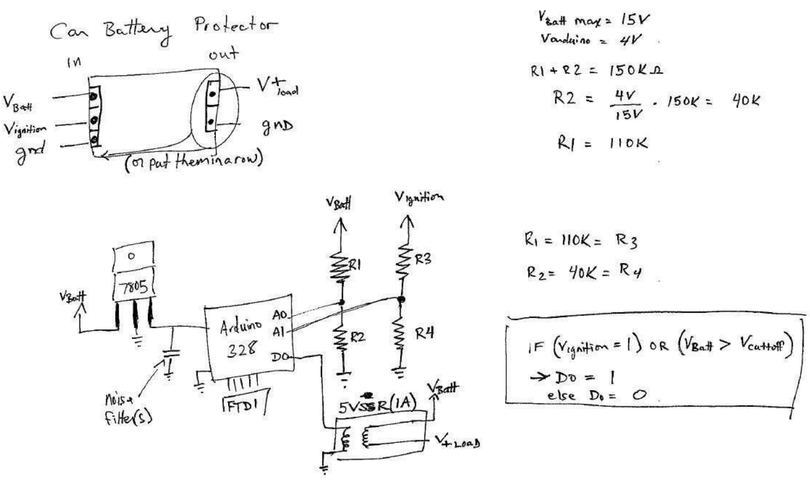
Day 1 : Friday Evening : 3 Hours
- Kicad Introduction and Basics
- KiCad Gallery - Pictures of some Projects
- Download and Install KiCad on all computers

Day 2 and Day 3 : Saturday / Sunday : 4 Hours + 4 Hours
- Recap - KiCad Introduction
- Workflow
- Schematic
  - Initial setup - Page, Grid, ..
  - Symbol Library
  - Annotation
  - DRC
  - Power Flags
  - Labels
  - Sheets - Simple and Hierarchical
  - Netlist
  - BoM

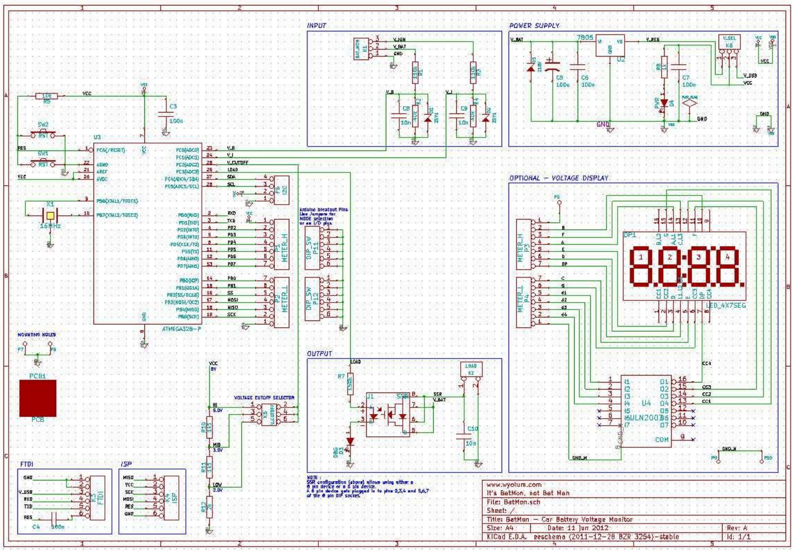
  - Symbol <> Footprint Association
- Board Layout
  - Initial setup - Page, Grid, ...
  - Footprint Library
  - Positioning - Auto vs Manual
  - Board Outline
  - Design Rules - track widths etc
  - Routing - Auto vs Manual
  - DRC
  - Zones
  - Cleaning up the Silk Legend
  - 3D - using Wings to create 3D Parts
- Resources
  - Online Library search - Symbols, Footprints, 3D
  - Help forum
  - Online schematic symbol and footprint generator : http://kicad.rohrbacher.net/quicklib.php
- Plotting
  - Gerber plots
  - Excellon DRL File
  - Prototypes - OSHPark , seeed
- Other Topics (depending on time available)
  - What's New in KiCad -http://wiki.xtronics.com/index.php/Kicad#New_under_development_Distributed_Library_System_for_Kicad.27s_EESCHEMA
  - Refer to User Guides available online.
    - http://www.kicad-pcb.org/download/attachments/1212538/Getting_Started_in_KiCad.pdf
    - http://www.kicad-pcb.org/display/KICAD/KiCad+Documentation
  - How to create mounting holes
  - Creating a PCB Board component for better 3D views

WyoLum @ NOVA Labs

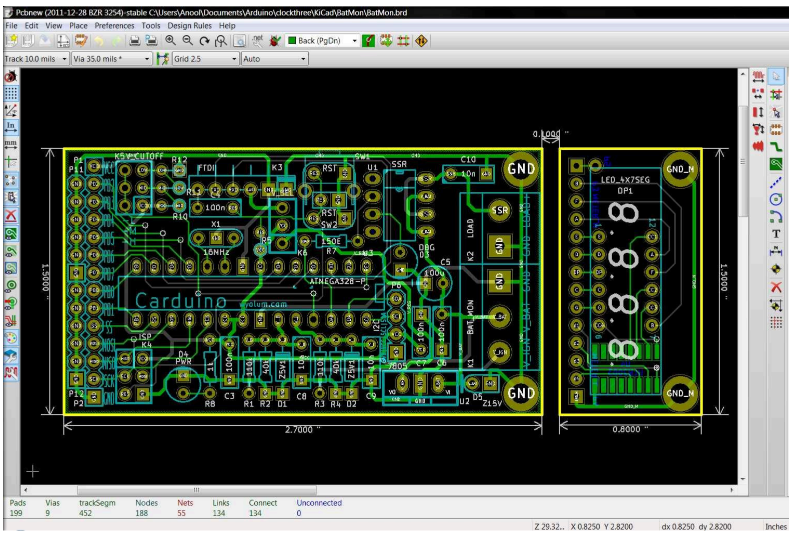
Rediscover the joy of making things



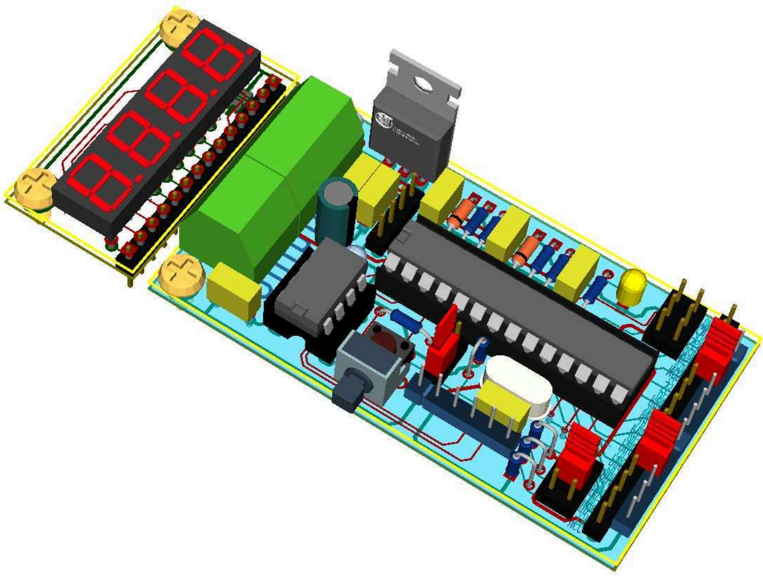
from sketch....

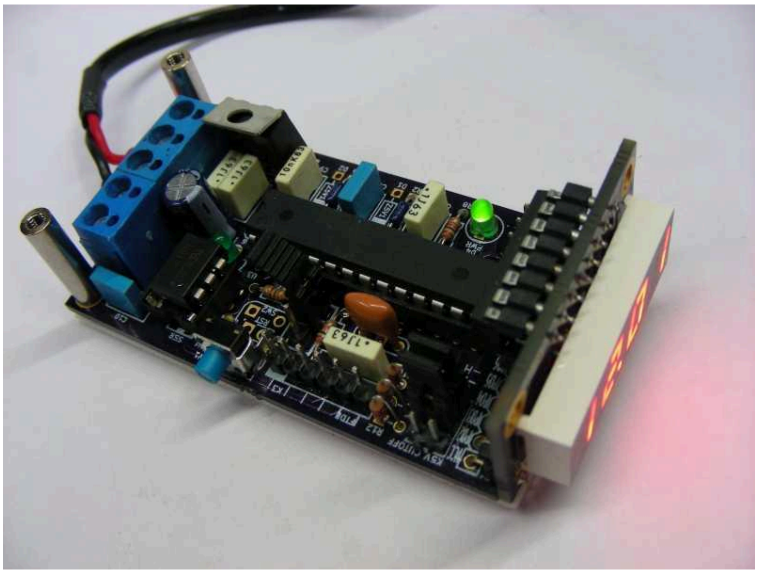


to schematic....



to PCB layout.....


to 3D View....


to final Board !!!!