

# PCB Class

This is the location for the class materials for the Surface Mount Technology Printed Circuit Board (SMT/PCB) "mini-class" presented at Hacker Dojo during the summer of 2013.

## Goal

The goal of the class is that each participant will design, send out for manufacture, order parts, solder and test out a surface mount technology printed circuit board by the end of the class.

## Administration

The class meets from 12 noon to 1:15pm in the Hacker Dojo "Maker Space" room on Saturdays.

The class is broken into a "lecture" followed by question and answers (Q&A). The goal will be to keep each "lecture" approximately 15 minutes long. Each Q&A session will be as long as there are questions. Both the "lecture" and the Q&A will be video recorded and uploaded to YouTube so that people who can not make a particular session can catch up.

The class is free in the sense that anybody can attend. The class is not free in the sense that if you build a PCB you will have at least pay to get the PCB manufactured and purchase all the required parts that are needed to construct and test the board. If the participant decides to set up the ability to work on surface mount parts outside of Hacker Dojo, there will be some significant expenses for devices like soldering irons, microscopes, PCB shears, etc.

KiCAD is the presented PCB design tool. If a participant is more comfortable with an alternative like Eagle, that is also quite acceptable.

## Basic Class Outline

The basic class outline is:

- Introduction
- PCB's and surface mount technology
- Design, Manufacture, and Test work flow
- Finding parts
- KiCAD
  - ◆ Schematic Capture
  - ◆ Library management
  - ◆ Footprints (IPC-7351)
  - ◆ Footprint editor
  - ◆ PCB design and design rule checking
  - ◆ Gerber generation and inspection
- Documentation
- Panelization and board ordering
- SMT tools
- Soldering PCB's by hand
- Soldering PCB's using stencils + oven

- Testing
  - ◆ Download Blinky program
  - ◆ JTAG debugging
- More advanced topics (TDB)

## Lectures

- Session 1. (15Jun2013) Introduction to class and surface mount PCB's.

- ◆ Lecture 1 -- <http://youtu.be/nnQK5tNLf6A> ; Introductory Material
- ◆ Discussion 1 -- <http://youtu.be/sq9nqoJVbX0>
- ◆ Lecture 2 -- <http://youtu.be/r8KasOYSXJQ> ; Introductory Matirial
- ◆ Discussion 2 -- <http://youtu.be/e9vZ7QtEE6A>
- ◆ Danny's Session 1 Notes from the class are available.
- ◆ Homework. Download KiCAD

An alternative to the videos above is Wayne's SMT for Roboticists HBRC (Home Brew Robotics Club) presentation.

- Session 2. (22Jun2013) Finding chips and Initial Schematic Capture.

- ◆ Video Playlist -- <http://www.youtube.com/playlist?list=PLKMsjYjvN5QFPR9ERFqPuN4P1v2LBj4ki>
- ◆ Lecture 1a -- <http://youtu.be/sjvGTyjGPV8>
- ◆ Lecture 1b -- <http://youtu.be/fAroFyO0bVE> ; Finding parts on the web.
- ◆ Discussion 1 -- <http://youtu.be/xN12sepG-w8>
- ◆ Lecture 2a -- <http://youtu.be/vI0RpbtnBis>
- ◆ Lecture 2b -- <http://youtu.be/wv9gGwcbef4> ; Introduction to KiCAD schematic capture.
- ◆ Discussion 2 -- <http://youtu.be/dPedjUvY7oU>
- ◆ Danny's Session 2 Notes from the class are available.
- ◆ Homework: Do the following:

- Finish downloading KiCad.
- Download LPC175X Template from [github.com](http://github.com). If you are having problems with git, go to Danny's git class on Sunday.
- Fire up KiCAD the lpc175x\_template.sch schematic.
- Add an LED and a resistor in series between an open processor pin and ground.
- Use DigiKey to search for a surface mount LED that is in an 0603 package.
- Use both Findchips.com and Octopart to search for the manufacturer part number to check for price and availability.

- Session 3 (29Jun2013) KiCAD Libraries and Part/Footprint Mapping

- ◆ Lecture 3a -- [http://youtu.be/x8\\_4-4cUuaE](http://youtu.be/x8_4-4cUuaE) ; Using the KiCAD Library Editor
- ◆ Discussion 3a -- <http://youtu.be/fR7kXhqNIbM>
- ◆ Lecture 3bp -- <http://youtu.be/VPfCxGzimtY> ; Design check, Net Generate, and Footprint Binding
- ◆ Discussion 3b -- <http://youtu.be/H-L1BrQR0ak>
- ◆ Homework: Do the following:

- Read KiCAD Library Notes

- Create a bit GitHub or BitBucket account.
- Create a new repository named led64\_XXX where XXX are your initials.
- Download the new repository to your machine (i.e. "git clone https:...")
- Copy and rename all the lpc175x\_template.\* files over to your new repository.
- Rename all the lpc175x\_template.\* files to led64\_XXX.\* files where XXX are your initials.
- Upload the files to the repository (i.e. "git add led64\_XXX.\* ; git commit ; git push")
- Build a schematic library for the ELD207.
- Build a schematic library for the L78M05CDT.
- Modify the schematic to have an 8 x 8 LED array.
- Session 4 (6Jul2013) LPC-7351 Footprints and KiCAD PCB editor introduction.
  - ◆ Lecture 4a -- <http://youtu.be/QRM1CH8v9gw> ; IPC-7351 Footprints
  - ◆ Discussion 4a -- <http://youtu.be/5AHJ811C44w>
  - ◆ Lecture 4b -- <http://youtu.be/oc74sKu4OFs> ; Introduction to *pcbnew* (the KiCAD PCB editor)
  - ◆ Discussion 4b -- <http://youtu.be/3MpBR7ONrCk>
  - ◆ Homework
    - Generate a net list from the schematic editor.
    - Use git to make a clone of Wayne's PCB Footprints
    - Read PCB Footprints
    - Download the IPC7351 Naming conventions.
    - Fire up *cvpcb*:
      - In Library preferences Add ../pcb\_footprints to the search path.
      - Add LPC7351-Nominal.mod to libraries.
      - Bind RESC1809H50N to your current limit resistors
      - Bind LEDC1608X80N to your LED parts.
      - Click on [File]=>[Save]
    - Fire up *pcbnew*:
      - Bring up preferences and make sure that ../pcb\_footprints is in the search path and LPC7351-Nominal.mod is loaded.
      - Import the net list (look for icon with wrench)
      - Place all the components onto your board.
      - Use trace mode to connect the components.
      - Save the result.
      - Homework for Wayne: Research KiCAD autorouting...
- Session5 (13Jul2013) KiCAD Lab.
  - ◆ Biggest issue: Everybody was running different versions of KiCAD. That caused some serious problems. We are going to try to compile a version of KiCAD from source and release it to run under virtual box. That way everybody will be running the same code regardless of platform (Windows/MacOS/Linux.)
  - ◆ Please re-download Wayne's PCB Footprints, the Reference and Value text was too small to be seen.
  - ◆ Most people were having a reasonable time with generating schematics.

- ◆ Most people were struggling the schematic (i.e. .lib) and footprint (i.e. .mod) search paths. We'll just have to keep working on that.
- ◆ On my version of KiCAD [Edit] => [Reset Module Field Sizes] brought up a pop-up window that lets you set the size of all Reference/Value fields on your design. It does so without moving them.
- ◆ In general AutoRouting does work with KiCAD, but it does not currently work with LPC175x*Template/rev*. I'm going to try to see if I can figure out why.
- ◆ You can reload a footprint from the library by using [Mouse over footprint] => [Right click] => [Footprint xx on Front] => [Edit Parameters] . This brings up a pop-up window. Click on [Change Module(s)] . Click on the appropriate radio button and click [OK].
- Session 6 (19Jul2013) KiCAD Lab.
- Session 7 (27Jul2013) KiCAD Lab.
  
- ◆ Many people asked for a more Arduino like initial project to use as a starting point. MBino is a version of the "Arduino" that can be plugged into a bus called MakerBus. Please note, the board outline is the same as the "Arduino". If you need more board space, just add it as needed. Try to keep your board size inside a 10 cm square.

## Links

The links are in "alphabetical" order:

- Class Home Page -- [http://gramlich.net/projects/pcb\\_class/index.html](http://gramlich.net/projects/pcb_class/index.html)
  - ◆ Class Home Page .pdf -- [http://gramlich.net/projects/pcb\\_class/README.pdf](http://gramlich.net/projects/pcb_class/README.pdf)
  - ◆ Class LPC175X Tempate -- [http://github.com/waynegramlich/LPC175X\\_Template](http://github.com/waynegramlich/LPC175X_Template)
  - ◆ Class Materials Git Repository -- [http://github.com/waynegramlich/pcb\\_class](http://github.com/waynegramlich/pcb_class)
- FindChips.Com -- <http://findchips.com/>
- Jameco.Com -- <http://jameco.com/>
- KiCAD Home Page -- <http://www.kicad-pcb.org/>
  - ◆ KiCAD Documentation -- <http://www.kicad-pcb.org/display/KICAD/KiCad+Documentation>
  - ◆ KiCAD Library Notes -- [http://gramlich.net/projects/embeddedsystemnotes/kicad\\_notes.html](http://gramlich.net/projects/embeddedsystemnotes/kicad_notes.html)
  - ◆ KiCAD Download Page -- <http://www.kicad-pcb.org/display/KICAD/Download+Kicad>
  - ◆ KiCAD Tutorial -- <http://teholabs.com/knowledge/kicad.html>
- Mouser.Com -- <http://mouser.com/>
- OctoPart.Com -- <http://octopart.com/>
- SeeedStudio Home Page -- <http://www.seeedstudio.com/>
  - ◆ SeeedStudio PCB Service -- [http://www.seeedstudio.com/depot/services-c-70\\_71/?ref=side](http://www.seeedstudio.com/depot/services-c-70_71/?ref=side)
- Wayne's Nov2012 HBRC SMT Talk -- <http://www.youtube.com/watch?v=GeJ47UOX1hc>
  - ◆ SMT Talk Slides -- <http://gramlich.net/projects/hbrobotics/talks/smt/index.html>
- Wikipedia SMT Page -- [https://en.wikipedia.org/wiki/Surface-mount\\_technology](https://en.wikipedia.org/wiki/Surface-mount_technology)