

PCB Workshop.

Imperial College Robotics Society

January 25, 2015

1 Motivation

This workshop aims to arm participants with all the skill necessary to produce PCB designs using the Eagle CAD package. This should be enough for most simple prototype designs for hackers students and research staff alike, those interested in high end PCB development may be more inclined to learn a more robust and intensive package such as Orcad or Altium. We also aim to introduce participants to the process of soldering modern packages onto dense PCB designs using reflow ovens, solder stencils and solder paste. This course of work should be quite intensive, 3 sessions each at 2 hours each, individual experimentation is recommended.

2 Timetable

Sections of work below will each have a short demonstration and discussion of tasks initially followed by a longer time for participants to emulate and experiment on their own machines.

2.1 Session 1: All the basics.

Complete Demonstration (15 min). A complete start to finish demonstration of producing a simple PCB. The aim is to give participants the high level understanding of what the process involves so that discussion can be had in a meaningful way.

Schematic (40 mins). We work through the process of constructing a schematic of a simple op-amp circuit. Discussion of ERC (Electronic Rules Check) and what it is good for.

Layout (60 mins). We layout the design described in the schematic from before. A discussion of DRC (Design Rules Check), copper fill and auto routing.

Audio discussion (Bonus 20 mins). For those who are interested a short presentation of the acoustic design involved in the circuit to be attempted in session 2. This understanding is not critical to the aims of this course. This will allow participants to see a design from its conception in theory, to its manufacture (by the end of the course).

2.2 Session 2: Producing a realistic design.

Re-introduction (10 mins). Reminding participants of teaching points from the last session. Discussion of why we would want to make our own components and the pitfalls that surround this in Eagle.

The audio amplifier: Schematic symbol (15mins). Producing a symbol for a new component.

The audio amplifier: Footprint (30 mins). Producing a footprint for a new component.

Building our design (60 mins). A design for an amplifier circuit will be presented and participants will use their newly learned skills and their new component to produce a PCB design.

Ordering a design (Bonus 20 mins). All participants should hopefully have a design that could be submitted for manufacture. Steps in submitting this so ‘Seed Studio’ will be presented, a Chinese producer of PCB prototypes that are both high quality and very cheap (with long delivery times). This is not necessary to the aims of this course, though will be useful to those who want to use their new skill practically.

2.3 Session 3: Constructing a complete design.

Re-introduction (10 mins). Reminding participants of what they have learned. Distributing PCBs and discussing different soldering processes and their use cases.

Solder masks (20 mins). Demonstration of Solder mask export from Eagle and the steps necessary to make one on the laser cutter.

Soldering workshop (90 mins). Participants will apply solder paste, place components, and bake their board. Due to the likely issues in getting boards constructed correctly and through only one oven, this section will have a freestyle organisation, faster people may go on to test the circuit.

Celebration (Bonus). Sit back and have some reward pizza and reminisce about how many bugs their are in eagle and how damn small those resistors are.