Analog Electronics Fundamentals 105

-Open Source EagleCAD libraries

-EagleCAD Introduction

-Final Project

hubcitylabs

1   Open Source EagleCAD libraries

- Before we get started, do yourself a favour and download the following libraries:
    - https://github.com/adafruit/Adafruit-Eagle-Library/
    - https://github.com/sparkfun/SparkFun-Eagle-Library
    - http://dangerousprototypes.com/docs/Dangerous_Prototypes_Cadsoft_Eagle_parts_library
    - https://github.com/Seeed-Studio/EagleLib-of-Seeedstudio

- Installation:
    - Open Eagle and select the Control Panel window.
    - Choose Options and from the drop down that appears, Directories
    - Change the Libraries line from: $EAGLEDIR/lbr to something like:
        - $EAGLEDIR/lbr:$HOME/external_lbrs (for OS X/Linux/UNIX)
        - $EAGLEDIR\lbr;$HOME\external_lbrs (for Windows)
    - Click OK to save your changes.
    - Eagle will prompt to create the directory if it does not already exist. Note the location and choose Yes to create the directory.
        - On OS X, $HOME/external_lbrs changes to: /Users/mosfet/external_lbrs/
        - On Windows, $HOME\external_lbrs changes to: C:\Users\Mosfet\Documents\external_lbrs
    - Put all your newly downloaded libraries under the "**external_lbr**" folder.
    - Restart Eagle. The library should be now be usable.

hubcitylabs

2 EagleCAD Introduction

- The "EagleCAD" introduction will be done live.

hubcitylabs

3   Final Project: Variable Supply 1.25v~Xv

PreRequisites:

- Install the free version of EagleCAD (http://www.cadsoftusa.com/download-eagle/?language=en)
- Try to familiarize yourself with EagleCAD as much as possible. Some tutorials:
  - https://www.sparkfun.com/tutorials/108
  - http://www.ianstedman.co.uk/Technical/Starting_with_EagleCAD/starting_with_eaglecad.html

Project:

- Schematic provided with the LM317 datasheet, on the first page. Feel free to use that as your design, or basis for your design.
- You could also add improvements, like adding an LED and switch for ON/OFF.
- You will need to prototype your design on a breadboard, and show it's working.
- Once you have a working prototype, you can put your design's schematic in EagleCAD, and create a PCB (Printed Circuit Board Layout) for it.
- Workshop 105 will be dedicated to EagleCAD circuit and PCB layout making.
- During workshop 106, if you have a layout ready, we will go ahead and etch our own PCB, then drill them and populate the components. You will have made your own portable, variable power supply.
- If by workshop 106 you don't have a PCB layout ready, I will make some of my proto boards available so that you can still build your designed power supply.

hubcitylabs