EAGLE Tutorial

Introduction

Eagle is a PCB CAD program for designing and laying out printed circuit board (PCB) designs. A freeware version is available as a download, which is what we will use in this Step-by-step tutorial.

http://www.cadsoftusa.com/download-eagle/

EAGLE works in two stages: Schematic capture and PCB design. In the first, the circuit connectivity is designed using the standard component symbols with which we are all familiar. In the second stage, the components are positioned on the printed circuit board and then traces between connected components are laid out. EAGLE makes this easier by preserving the connections between components and also applying various electrical and design rule checks in order to make sure the board will function as expected!

In this tutorial we are going to begin by designing a PCB board for the active low pass filter below, and then in groups of two you will design a PCB for the potentiostat circuit provided in this document. The low pass filter will have the following component values, and will run off of a ± 15 V supply.

$R_1 = 1\,\text{k}\Omega$
$R_2 = 10\,\text{k}\Omega$
$C = 2\,\text{uF}$

This gives an approximate cut-off frequency f_c of 7.95 Hz and a passband gain of -10.

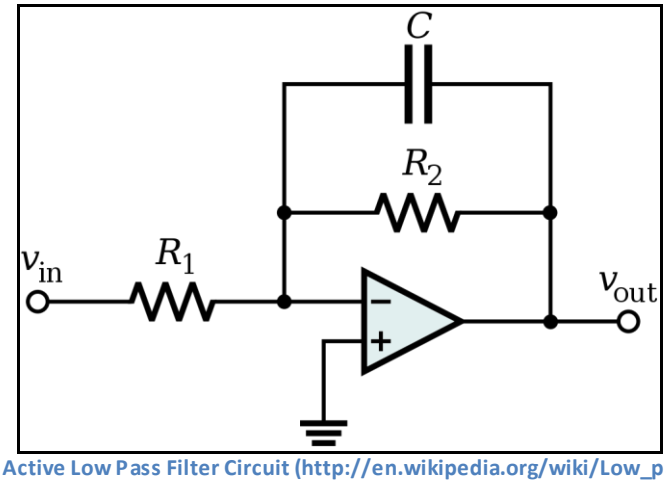

Figure 1: Active Low Pass Filter Circuit (http://en.wikipedia.org/wiki/Low_pass_filter)

Part 1: Low Pass Filter

Getting Started

1. Start Cadsoft EAGLE from the windows start menu (**Start → Programs → EAGLE Layout Editor 5.x.x → EAGLE 5.x.x**)
2. The first time that you run EAGLE, it will ask whether you would like to create a directory, click yes.
3. Expand the '**Projects**' node of the tree on the left of the EAGLE menu.
4. Right click on the '**eagle**' folder and select '**New Project**'. Name it '**LPF**' (for low pass filter).
5. Right click on the LPF project folder (coloured in red) and select **New → Schematic**.

Creating the Schematic

1. Select **File → Save** and save the schematic as '**LPF Schematic.sch**'.
2. First of all, add all of the parts we will need, in this case that is two resistors, a capacitor, and an opamp. We are going to use 0603 sized resistors and capacitors, and a through-hold DIP (Dual Inline Package) opamp.
    a. Click on the '**Add**' icon on the left-side toolbox – .
    b. Start by adding the opamp, type '***LM*741***' into the search box and press **Enter**.
    c. Select the '**DIL08**' version of the component and click '**OK**'. Click on the schematic to place the part. Press the right mouse button twice to rotate the part and then click the center mouse button once to mirror the part. This will orient the positive and negative terminals of the gate properly.
    d. Press **ESC** to return to the ADD dialog.
    e. In the ADD dialog, change the search term to be blank and press **Enter**.
    f. Scroll down until '**rcl**' appears in the list of parts. Expand '**rcl**' and then expand the '**R-US_**'. Scroll down to '**R-US_R0603**', select it, and press '**OK**'. Place to resistors
    g. Press **ESC** again to return to the ADD dialog.
    h. Go back up to the root of the '**rcl**' library and expand the '**C-EU**' node.
    i. Find the '**C-EUC0603**' part and press **OK**. Place the part as per the schematic.
    j. Press **ESC** and then **ESC** again to exit ADD mode.
3. At this point the schematic should look like Figure 2.
4. Now for the connectivity! Select the '**NET**' tool from the toolbox – .

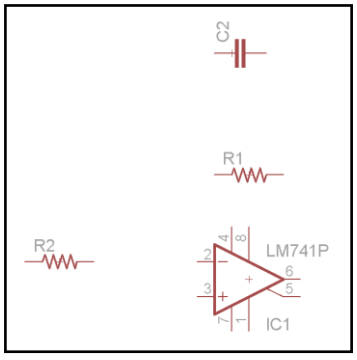

Figure 2: Basic part placement.

5. Draw wires between all of the relevant terminals as per the original schematic. Note: Double click to terminate the current NET. Select the '**Junction**' tool ( ) and click to place junctions between wires (Automatic Junctions will appear on NETs that overlap).
6. Now we need to add the power and data ports:
    a. In the ADD dialog again, place **supply2 → GND** for the positive pin and also in an open area to connect to the power supply.
    b. Place **supply2 → +15V** and **supply2 → -15V** both onto the op-amp and also near the previously placed ground.
    c. Finally, we will add input/output plugs for the various signals needed by the board (+15V, ground, -15V, signal input, and signal output).
    d. Place a **con-lstb → MA05-1** on the schematic near the three floating power symbols.

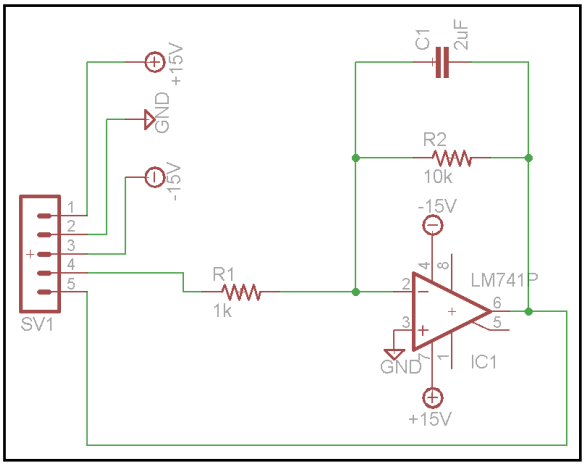

Figure 3: Complete wired schematic.

7. Complete the wiring as per Figure 3.
8. Finally, we will assign component values. Click on the '**Value**' icon – .
9. Click on each of the resistors R1 and R2, and the capacitor C1 and assign them the values of '**1k**', '**10k**', and '**2uF**' respectively.
10. It is a good idea to run an electrical rules check (ERC) to make sure you haven't made any mistakes! Click the '**Electrical Rule Check**' button ( ) and make sure there are no errors or warnings that you are not expecting.
11. Make sure to save the schematic, and then choose **File → Switch to board** to begin creating the associated PCB.

Creating the PCB

1. When asked if you would like to create the '**LPF Schematic.brd**' file, click '**Yes**'.
2. You will now see a blank PCB board, along with the parts from your schematic in the lower left.
    a. The parts don't look the same! These are the *footprints* associated with the components from the schematic. A footprint is the physical set of pin holes and copper areas that components of a particular side need to be soldered/clipped/connected to.
3. We will begin by positioning the footprints on the board through a combination of rotation and movement. Mirroring a footprint is **NOT** allowed unless you are putting it on the opposite side of the board. There are more guidelines below (in the 'tips' section) but the important factors to consider here are:
    a. Make the design compact.
    b. Keep in mind that parts exist in 3 dimensions, and just because the footprints fit near each other, does not mean the parts will.

c. Lay the design out logically to minimize errors now and later on.
d. Try to minimize the number of times that the dark yellow 'connection guidelines' cross. Crossing traces means putting vias through the board which is expensive and takes up space.

4. Component footprints can be positioned using '**Move**' ( ) and '**Rotate**' ( ).
5. Using the '**Delete**' command ( ), delete the existing board outline. Once the traces are laid down a new outline can be created to minimize the necessary surface area of the board. PCBs are priced proportionally to the number of holes in them and their total surface area, so saving space saves money!
6. With your layout completed (Figure 4), it is time to add the traces. Select the '**Route**' tool – .
7. Make sure you are working on the side of the board you want. Layers (top or bottom normally) can be selected in the layer dropdown: 1 Top.
8. To add traces to the board, first click on a connected pad, then click to lay the trace and finally click on the ending pad indicated by the dark yellow line. Remember:
   a. Keep the traces as short as possible! Longer traces add resistance, thermal noise, and can also act as antennae.
   b. Avoid 90 degree corners, 45 degrees is preferred
   c. Avoid running two traces in parallel on top of each other on opposite sides of the board. This can cause capacitive coupling between the traces.
   d. Make sure traces are wide enough, especially power traces.

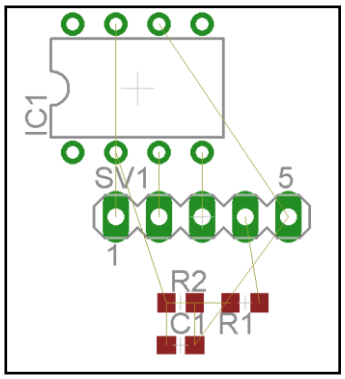

Figure 4: Component positioning.

9. In order to actually know what goes where after the board is manufactured, it is important to make sure that the component labels (e.g. R1, C1, IC1, etc.) are visible on the silkscreen and won't be on top of pads or drill holes or anything like that. In order to move the label of a part around, you have to first '**Smash**' the part ( ) and then select and '**Move**' the label ( ).
10. The final step is to arrive at the final board outline, which in this case will be a simple square. Select the '**Line**' tool ( ) and make sure to select the '**Dimension**' layer. Finally, draw the outline of the PCB board. Make sure to leave a bit of space around the edge of the board.
11. Use the '**Design Rules Check**' ( ) to ensure there are no overlapping traces or other mechanical problems.

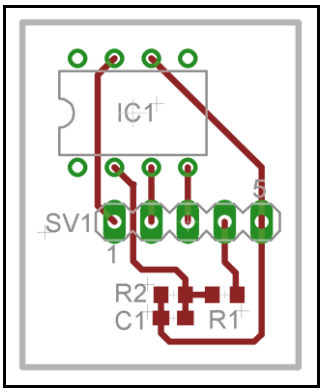

Figure 5: Final board design

Appendix: PCB Design Tips

Component Layout

- Try to lay out the components in a logical way, grouping components and sometimes making the PCB topography reflect the topography of the schematic
- Try to minimize the total surface area required
- Try to minimize the number of crosses in the ratsnest (dark-yellow lines indicating unrouted connections)
- Make sure that not only do the footprints fit around each other, but the 3D structures of the components will not intersect.
- **DO NOT** mirror a component unless you are putting it on the other side of the board.

Trace Routing

- Try to minimize trace lengths. Long traces add more resistive losses, more thermal noise, and can act as an antenna.
- Do not run two parallel traces on opposing sides of the board on top of each other, they can become capacitive coupled.
- Avoid 90° angles and T-intersection. Instead always go around corners and merge traces at 45 degree angles
- Ensure that traces are wide enough to carry their current. Trace width calculators are widely available on the internet. This is especially important for power traces.
- Make sure traces are sufficiently spaced, generally at least one trace-width apart.
- Try to 'balance' traces on differential-type devices as much as possible.
- If you have room on your circuit, use a ground plane for the ground instead of a trace.