Printed Circuit Board Layout

PC boards for Lab 3 will be fabricated by ExpressPCB. You can access the layout software on the PCs in 5162 or ECI. Or you can download the software onto your own PC. In either case, visit http://www.expresspcb.com for an orientation on their layout software and board specifications. If you use your own PC, then you must also download the component footprint for the surface mount transistor (either NE85639 or NE34018), and a board-mounted SMA connector (SMApad.p). Copy these files into the following directory:

> Programs\ExpressPCB\PCBComponents_Custom

When your layout is ready to submit, email it to: long@ece.ucsb.edu. I will submit all of the layouts for the class.

Some specific recommendations:

1. Look at the Quick Start Guide in the Help menu.

2. Please put some text on your project so I can clearly identify your board.

3. You will want to implement transmission lines with high resolution. Use the rectangle feature rather than the wire tool to draw transmission lines.

4. The back side of the board will need to be filled in with ground plane anywhere a ground is needed, especially under transmission lines.

5. Note that the leaded components all have plated through holes. So if you do not want them to contact the ground plane on the back side of the board you must remove the metal on the back around the hole. The same is true for any via or hole in the board.

6. The standard component library contains footprints for the components used for this project:

Component	Size	Footprint from library
Chip Capacitor	0805	SMT - 0805 (top surface)
Chip resistor	1206	SMT – 1206 (top surface)
Chip inductor	0603	SMT – 0603 (top surface)
Surface mount transistors		NE85639.p or NE34018.p
Leaded transistor	2N3906	TO-92
Board mounted SMA		SMApad.p (custom footprint)

IMPORTANT!!!!
Instructions on ordering class project boards from ExpressPCB:

1) Each student or group will receive two PC boards made using their design file. Every student should review the manufacturing specification for Standard Service Boards
         Help Menu > Manufacturing Specifications

2) Each design should have the student's name written on the PCB. Using the ExpressPCB program, place this text within the board perimeter. Do so by:
         a) Select the "A" button on the side toolbar.
         b) Select a text size of 0.060" from the top toolbar.
         c) Type in your name in the text field on the top toolbar.
         d) Click inside the layout at the location where the text should be placed.

By adding this text, it will help us identify your design from the single shipment of all the designs that ExpressPCB delivers.

3) The ExpressPCB program can check the layout for certain design problems. This does NOT check for many of the possible problems, so you should very carefully inspect your layout to make sure that it matches your schematic and does not violate the manufacturing specifications.   (One common problem that you should check for manually is to verify that you have a minimum of 0.007" blank space between conductors. The program does not make this check for you.)

To check the layout:
         a) From the "Layout" menu within the ExpressPCB program, run the command
             "Compute board cost".
          b) In the first screen, select the country "USA", enter "CA" in the state field and
         if in California choose "Yes - State sales tax must be paid", then press "Next".
          c) Select the "2 layer boards" option, then press "Next".
          d) Select the "Standard Service" option, then press "Next".

If no problems are discovered, the ExpressPCB program will display their normal price for having the board made. If there are problems, they will be reported. Any problems identified MUST be corrected or the board can not be submitted for manufacturing.

4) When the design is complete, email the .pcb file to Prof. Long (long@ece.ucsb.edu).