

# EE133 PC Board README

Created by Thomas T. Lee, 27 JAN 2004  
leet@stanford.edu

## Contents

- 1) General features the EE133 PC Boards
- 2) How to use the EE133 PC Boards
- 3) Block-specific instructions
- 4) Schematic of transmit board
- 5) Component layout of transmit board
- 6) Component layout of transmit board with connections
- 7) Schematic of receiver board
- 8) Component layout of receiver board
- 9) Component layout of receiver board with connections

## General Features of the EE133 PC Boards

In an effort to reduce the amount of wiring work in lab, an effort was made last year to create pre-connected PC boards for EE133. The boards being provided to you by your TA's are the first fruits of that effort. The goal is to speed your initial learning of each circuit block by providing pre-connected areas on each PC board. Toward the end of the quarter, we expect that the time you save will be devoted to using your creativity to come up with new and improved designs or architectures. To this end, both boards have breadboard-style prototyping areas (called "headers") where you can place circuits of your own design.

You will be given two boards, one transmitter and one receiver. The receiver board is the smaller of the two, and has two separate header areas.

Please note that this is the first time pre-printed PC boards have been used in EE133. We apologize for any mistakes in the boards. They have been designed so that mistakes can be corrected easily. Shorts can be removed by scratching metal away with a razor blade, and misconnected lines can be reconnected with wire. (These boards do not have a solder mask, so you can solder directly to the metal lines.) We also ask that you please give us your feedback on these boards so that we can improve them in the future.

## How to use the EE133 PC Boards

This packet includes schematics and layouts of both the transmit board and receive board. In addition, your TA may provide you with transparencies of the board layouts so that you can lay them on top of the board to determine the proper placement of your parts.

Aligning the layout and PC board - To line up the printout with the holes on your board, align the printout (or transparency) with the board along the right-hand edge.

Standoff Holes – The holes in the corner of each board are for standoff posts. Use these posts to lift your board off the table surface.

Ground plane – the side with the ground plane (the large areas of metal) should face up.

Components – Components may be placed on either side of the board, but try to place as many components on the top of the board so that you have easy access to them for debugging.

BNC connectors – Each board has four footprints for BNC connectors. These BNC connectors are not actually connected to anything. There is a metal line that extends from the signal post of the BNC to a drill hole. After soldering your BNC into the connector, run a wire from the drill hole to the input or output you wish to test. The BNC should be placed on the same side as the ground plane so that the BNC casing touches ground.

Connecting board-level power supplies – Power the board through the two holes marked “Pow”. This is in the upper left-hand corner of the transmit board, and the upper right-hand corner of the receiver board. You will see that one hole is connected to the ground plane. Be sure to connect ground (and not power!) to his hole.

ON/OFF Switch – In series with “Pow” is two holes labeled “ONoff”. This is where you can put a power switch, if you so choose. If you do not wish to have a power switch, short these two holes together.

Connecting block-level power supplies – Local to each block is a set of two holes labeled “<block name>\_VCC”. One hole is connected to the board VCC, the other to the power pin for the block. When testing a single block, apply power to the block only. When you’re ready to test more of the system, add a (small!) wire to short the two holes together. This is also true for blocks requiring a 4.5V reference.

Connecting blocks to each other – In order to facilitate testing, the output of one block is separated from the input of the next by two drill holes. When testing, probe the output or drive the input using the appropriate hole. When you are ready to connect the blocks together, simply connect the holes with a short wire.

Potentiometers – There are two types of potentiometers in the lab, long rectangular (usually blue), and small square (usually yellow). To accommodate both types, the board has footprints for both placed in parallel.

Probe pads – at the inputs and outputs to each block, we have provided pads of metal for you to probe, if necessary. They appear on the schematic as little circles with labels for node names.

Headers – Note that some rows of the headers may be pre-connected to VCC or ground.

**Block-specific instructions**

4.5V Reference – The circuit here is slightly different from that given in your lab handouts. Specifically, there is a LPF on the output that helps keep the output more stable (see Horowitz & Hill pp. 944-946). If you do not want to use this option, replace R\_ref3 and C\_ref with wires and omit R\_ref4.

Colpitts Oscillator – There is a footprint given for a crystal, X1. If you do not want to use the crystal to stabilize your Colpitts (e.g. if you're using it as a VCO) simply replace this with a wire. For the variable trim capacitor (C\_VAR) the incorrect footprint was used. The footprint provided has three holes, while the trim capacitors in the lab have only two holes.

IF Filter – On the receiver board, the smaller header section is intended for the passive part of your IF filter. If there is insufficient space, you may use the larger header area.

LM566 – On the LM566 schematic, pay attention only to the pin NUMBERS, and ignore the PIN NAMES. This symbol was “borrowed” from a 555 timer for the footprint and as such, the pin names do not correspond to signal names from the LM566.

Power Amp (GALI-5) – The symbol in the schematic is for a bipolar transistor. The GALI-5 is not simply a bipolar transistor. It is a pre-packaged amplifier. Use the GALI-5 datasheet from the EE133 website to determine the proper pinout.

TRANSMITTER









