ECE112 - Lab 3

Purpose

- Learn how to map a complex schematic into a physical form
- Clarify what a node is
- Further improve building skills
- Make a multi-voltage power supply to power future experiments

Parts/tools needed:

- Power supply printed circuit board and parts in lab
- Wall wart and protoboard
- Wire
- Soldering iron, tip and solder
- Diagonal Cutters and Pliers
- Digital Multimeter (DMM)

Build and Test the Power Supply

1. Using the schematic diagram in figure 2 and the layout of the printed circuit board (PCB) in figure 3, determine where all the parts go on the PCB. Do this before you come to lab. This will take some time.

Referring to figure 3, for each component to be mounted, there is a light blue rectangle or square. For each solder joint there is a small greenish oval marking the approximate position for soldering the component down. Starting with the power connector at the top left, work your way around the board and mark what component each blue rectangle represents and its value. You may want to put a value or schematic symbol for each component. You could even tape the components to the page one-by-one and draw a line to where they are to go.

After you have finished figuring out where all the parts go and are ready to solder parts on, your TAs will begin to hand out a few parts. Carefully make sure you have the right parts. Transistors, diodes, MOSFET's all have part numbers clearly stamped on them. Make sure the resistors are actually the correct value; measure them.

There is also one jumper wire that must be installed. Its marked as a red wire on the board. Use a short piece of insulated hookup wire to make this connection just like the other components. All the other interconnections on the board are made by the tin plated copper traces.

This may take some time. One hint is to realize the ground trace is at the bottom of the board as shown in figure 3. Another hint is to realize the board is patterned similarly to the way the schematic diagram is drawn with only small changes. When you have all the parts mapped out,

have your TA check it if you have any doubts.

When adding components in, observe the orientation and/or polarity on the transistors, diodes, and electrolytic capacitors. Use the inset drawing for the transistors and zener diode to make sure these are in correctly. The negative side of the electrolytic capacitor is marked with a band on its body that has minus signs on it. Solder the parts directly to the top traces on the board keeping the leads short enough so that nothing shorts out, but not so short that they are difficult to handle. If you leave the leads longer than about half-inch, they will fold over and short out to other components or the board itself whenever you transport the board.

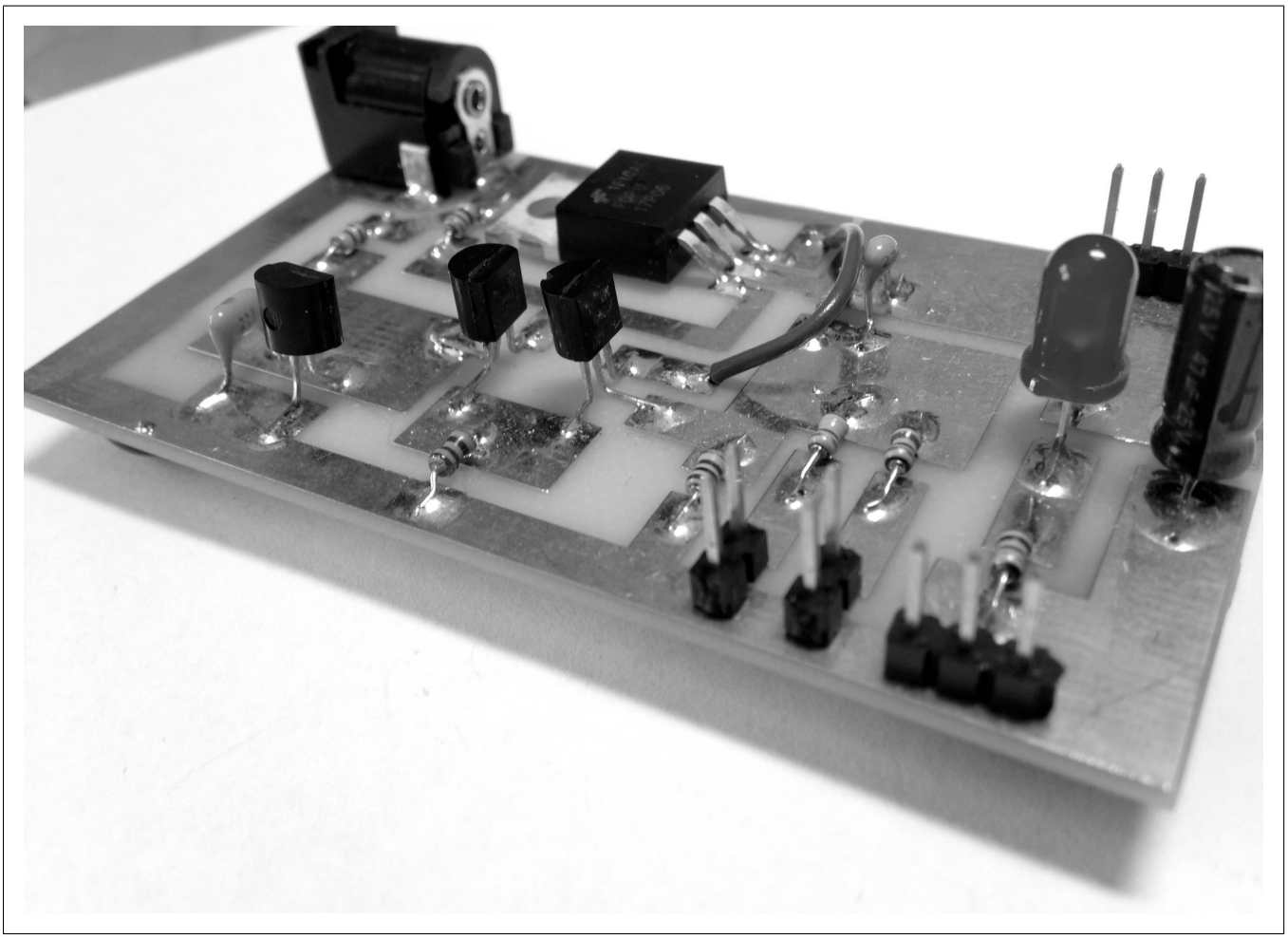

Figure 1: Power Supply Component Mounting Guide

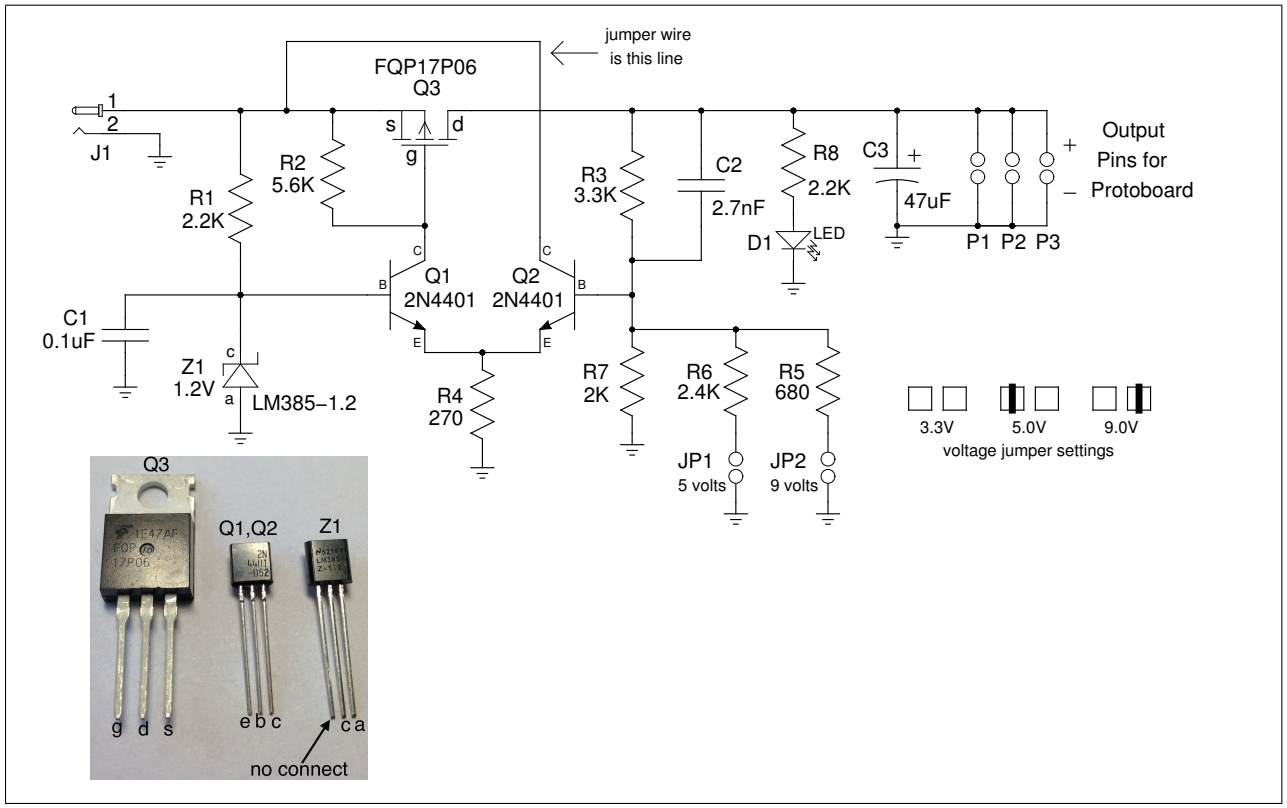

Figure 2: Power Supply Schematic

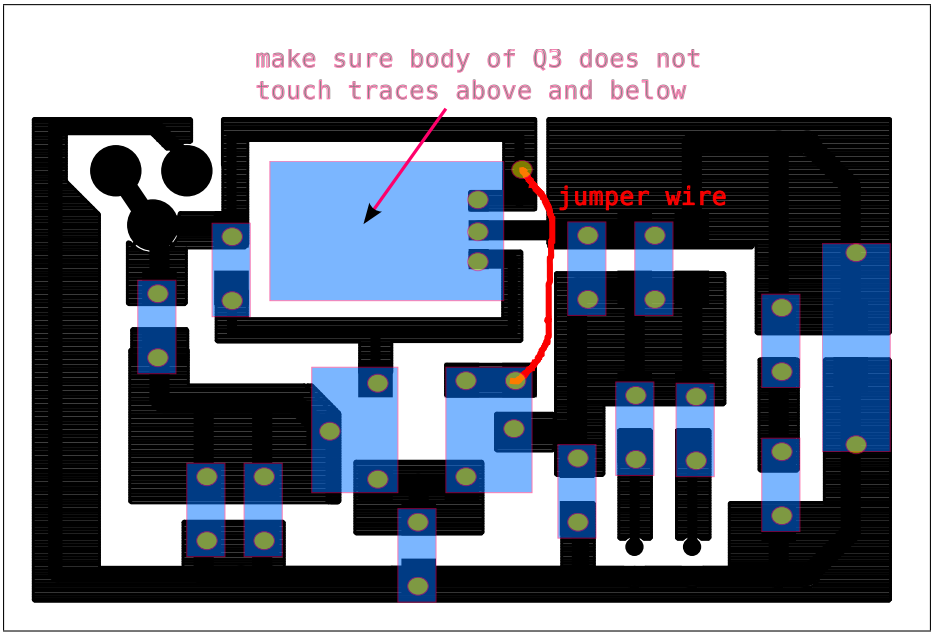

Figure 3: PCB Top Copper Layout

Once you have the parts soldered in, recheck your wiring. Be sure no leads are shorting together. See that the correct values have been placed correctly. Finally, make sure your solder joints are solid. Have your TA check your circuit if you are not sure. Leave all the jumper pins out for now. Then, plug the wall wart into the board and measure the voltage across C3. The positive lead of the DMM goes to the positive end of C3. You should measure about 3.3 volts. If you don't, immediately unplug the wall wart and check your circuit over for improperly placed components or bad solder connections.

If things are working well to this point, add the jumpers JP1 and JP2. These are located just below R5 and R6. The long side of the pins is on the board's top side. The shorter side is soldered from the bottom side of the board. Solder one pin, check the vertical alignment of the two pins, then once straight, solder the other pin.

In the same fashion, put the pins for P1, P2 and P3 into the board, long side of the pins on the top side of the board. These are the power and ground connections as shown on the right hand side of the schematic.

Insert one jumper into the voltage selection area to select a 5 volt output. Plug the wall wart into the board and measure the voltage at the supply output. It should be close to 5 volts. If not, determine where you went wrong. Likewise, move the jumper to the 9 volt position and you should measure 9 volts at the output.

1. What two, two terminal devices had a polarity that had to be observed?

_____

_____

2. What configuration of resistors does R3 form with either R5, R6, or R7?

_____

3. In preparation for later work, add a 5 pin strip of header pins on the side of your blue protoboard as shown 4. Put the center pin between adjacent pads and solder the outer two pins as shown. Then add two lengths of bus wire along the top and bottom of your board to serve as convenient power and ground connections.

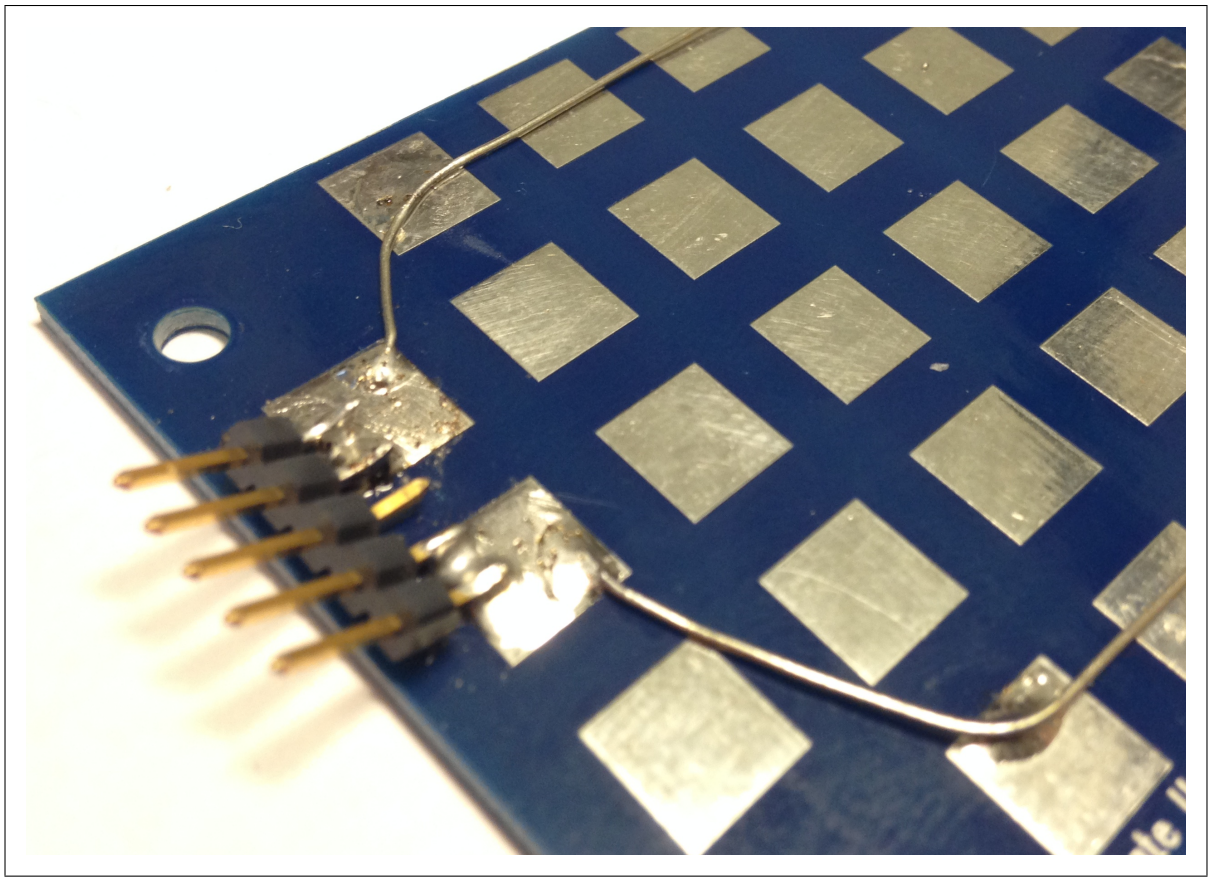

Figure 4: Power Supply Pins and Bus on Protoboard

3. Have your TA check off your work.
