Dylan Falzone
9/4/2023

EE Design 1 Technical Report
PCB Design

Introduction

In every field with real world applications, from biology, to teaching, to electrical engineering, there is a separation between knowing and doing. Sometimes this separation is small, for example, knowing how to bandage a wound vs applying the bandage yourself. However, this separation can also be monumental, like knowing the anatomy of the human heart vs performing open-heart surgery on a patient. Similarly, electrical engineering has a colossal, yet well-hidden gap between theory and practice. I first encountered this gap many years ago, when I was first beginning to disassemble household electronics and recombine them into something new. I had the components, I had wires, I had a plan in mind, yet I struggled to make the physical connections between the parts, especially when moving parts were involved. Electrical tape, poorly soldered wire junctions and jumper wires were only capable of so much. The one thing standing between my idea and its implementation was a thin, rectangular piece of glass and copper: a Printed Circuit Board (PCB). A PCB is the essential delivery system for the electronic circuit. It allows for complexity and connectivity that is impossible by other means. Yet, its complicated manufacturing and design process creates a huge potential gap between those who know how to make one, and those who do not. In this Lab, we will finally cross the gap and learn how to design the backbone of the modern circuit.

Methods

Circuit

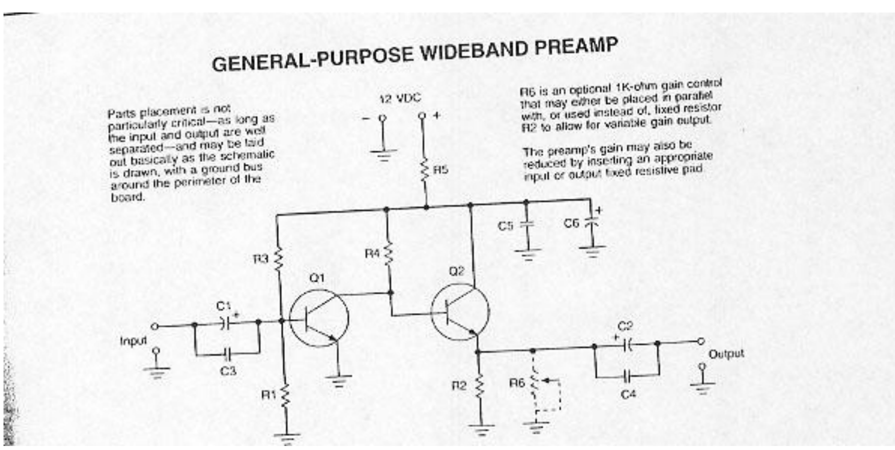

Figure 1: General-purpose wideband preamp circuit schematic provided.

For this lab, we were given a circuit schematic from which to create our first PCB. Mine was for a general-purpose wideband preamp (Figure 1), a circuit that aims to amplify electrical signals. First, I analyzed the circuit, noting which components were contained within, and in what quantities. Notably, this circuit used several bypass capacitors. A bypass capacitor aims to alleviate a noisy signal in a circuit, often an input or output signal. Since connections between circuits often vary in cord lengths and material compositions, one or more bypass capacitors can be used to account for the resultant noise [1]. Specifically, in this circuit, there are 3 parallel sets of two bypass capacitors used on the power, input, and output lines to ensure all signals remain as clean as possible. One of these capacitors is small, around 100nF to account for small variations in the signal, while the other is a much larger 100uF electrolytic capacitor intended to stabilize larger variations. For a DC signal, which is what is used in this circuit, capacitors act like an open, which will not change the circuit's functionality. But for an AC signal, or an AC component on top of a DC signal (A.K.A. noise), the capacitor will act like a short to ground, effectively removing the noisy part of the signal [2]. The other components are standard resistors and BJTs, though there was also an optional resistor, R6 that I chose to omit for my reconstruction of the circuit.

PCB

Next, I got to work on the schematic document in Altium. Using the provided ECE and Misc libraries, I drew the circuit to match the given schematic as closely as possible. I even chose to label the resistors, capacitors, and transistors the same as the given schematic in order to make checking and following the design more intuitive. I chose the values for the resistors arbitrarily, setting them all to 1kOhm. I also chose the specific type of BJT arbitrarily, just making sure that they were in the same direction as the ones pictured above.

Once the schematic was complete, I used Altium's 'update PCB document' function to transfer all the components onto the PCB. At first, this looked incredibly messy, so I followed the methods laid out by Professor Stapleton's videos [3] to make the connections as neat as possible, while still keeping key components relatively close together. The result of my optimization did not fit very well into a rectangular shape, so I drew the keep-out layer in a shape that better fit the components. The keep-out layer defines an area where no copper should be placed. This allows us to outline and define the edges of our board, since we don't want exposed copper around the outside edges. Using the keep-out layer as the outline for the board, I finalized the shape of the PCB.

The next step was the routing. Since I had already aligned the components in ways that would make their connections simple, the routing was as easy as tracing a handful of mostly straight lines. At this point, I also added holes to the board so that it was mountable inside a casing or on a larger device. For this, I had to revise the schematic and then update the PCB once more. Finally, I completed the board by adding a copper pour to the top and bottom layers. The top pour went to ground with a hatched 45degree fill while the bottom pour went to VCC with a solid fill. These pours helped to simplify the wiring.

Results

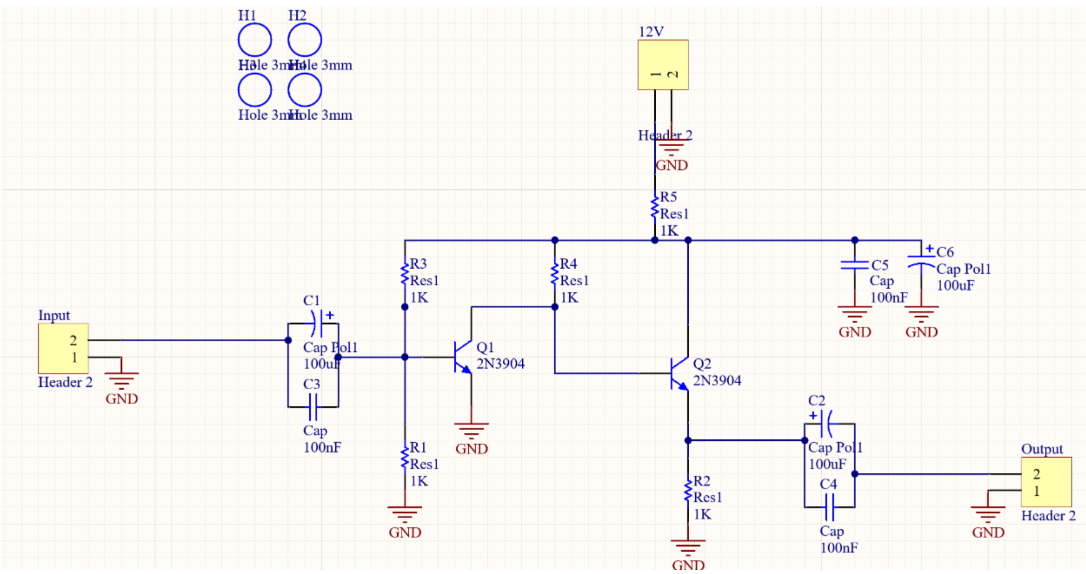

Figure 2: Altium Circuit Schematic.

The Altium circuit schematic I designed closely mirrored the provided schematic shown previously. Minor differences include the addition of the holes in the top left, which were needed for the PCB, as well as the input/output nodes. In the provided schematic, the ports were left open, however, for this schematic, I used the ‘Header 2’ component to represent the signal input, power input, and output ports.

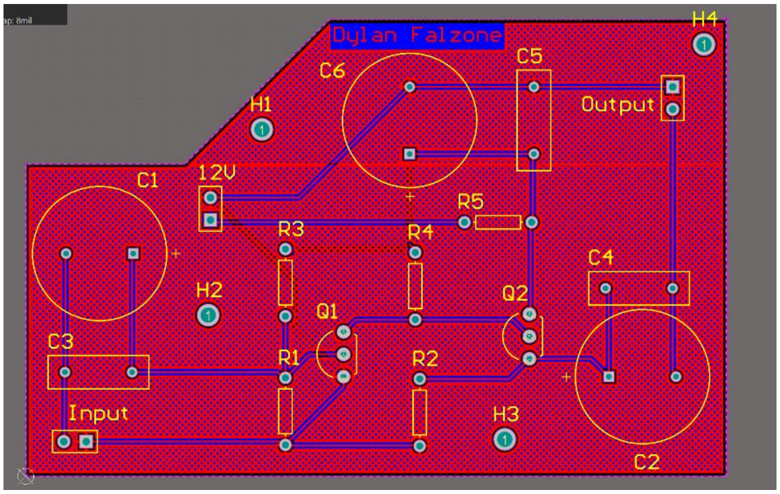

Figure 3: Top-layer of PCB.

The top-layer of the PCB includes most of the routing. This figure also displays the shape that I found most optimal for the board outline, as opposed to the default rectangle. Many components had to be rotated, flipped, and relocated to produce clean lines all around, but the only major change from the schematic is that the output has moved to the top right instead of the bottom right. Otherwise, most of the schematic lines up with the PCB layout.

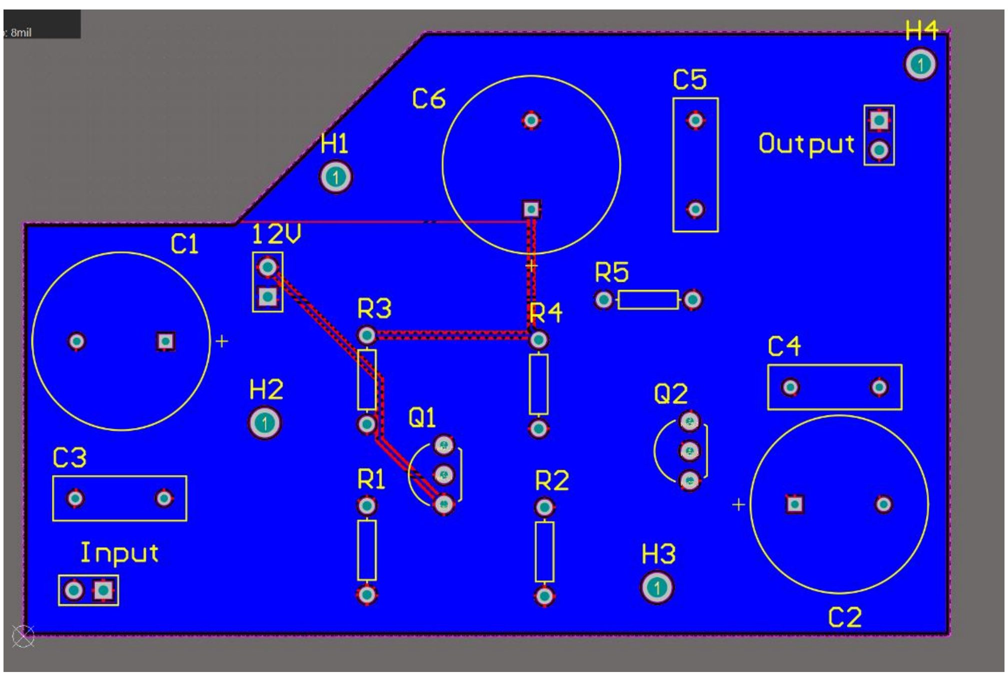

Figure 4: Bottom-layer of PCB.

The bottom-layer of the PCB includes just a few of the connections that would not fit onto the top layer. There is also a VCC-connected copper pour on this side which some components are connected to.

Conclusion

The divide between knowing and doing has been bridged. Leveraging Professor Stapleton's guidance, my independent research, and the Altium design platform, I crafted a PCB for a wideband preamp circuit. I constructed the schematic, selecting components and their values, and employing bypass capacitors at the inputs and outputs to effectively reduce noise. I then transferred the circuit onto the PCB and routed the connections in an efficient and logical manner and used the keep-out layer to outline the shape of the board, getting its profile just right. And finally, I chose the copper pours to further simplify the routing to ground and VCC, sealing the board and completing the project.

References

[1] T. S. Team, “What is the use of a decoupling capacitor?,” Sierra Circuits, https://www.protoexpress.com/blog/decoupling-capacitor-use/#:~:text=A%20bypass%20capacitor%20is%20used,spikes%20on%20the%20supply%20lines. (accessed Sep. 4, 2023).

[2] “What is a Bypass Capacitor?,” Learning About Electronics, http://www.learningaboutelectronics.com/Articles/What-is-a-bypass-capacitor.html (accessed Sep. 4, 2023).

[3] Altium Designer 23 Moving Parts to PCB, https://www.youtube.com/watch?v=RmO1pOV4288 (accessed Sep. 4, 2023).