Stepper Driver PCB Iterations

Skittle Sorter Project

Jaidon Lybbert, Motor Control Design Lead, EWU IEEE Student Chapter

08/23/20

This document serves as a history for the various changes that have been made to the motor driver PCB design over its many iterations, and why those changes were necessary.

Iteration 1

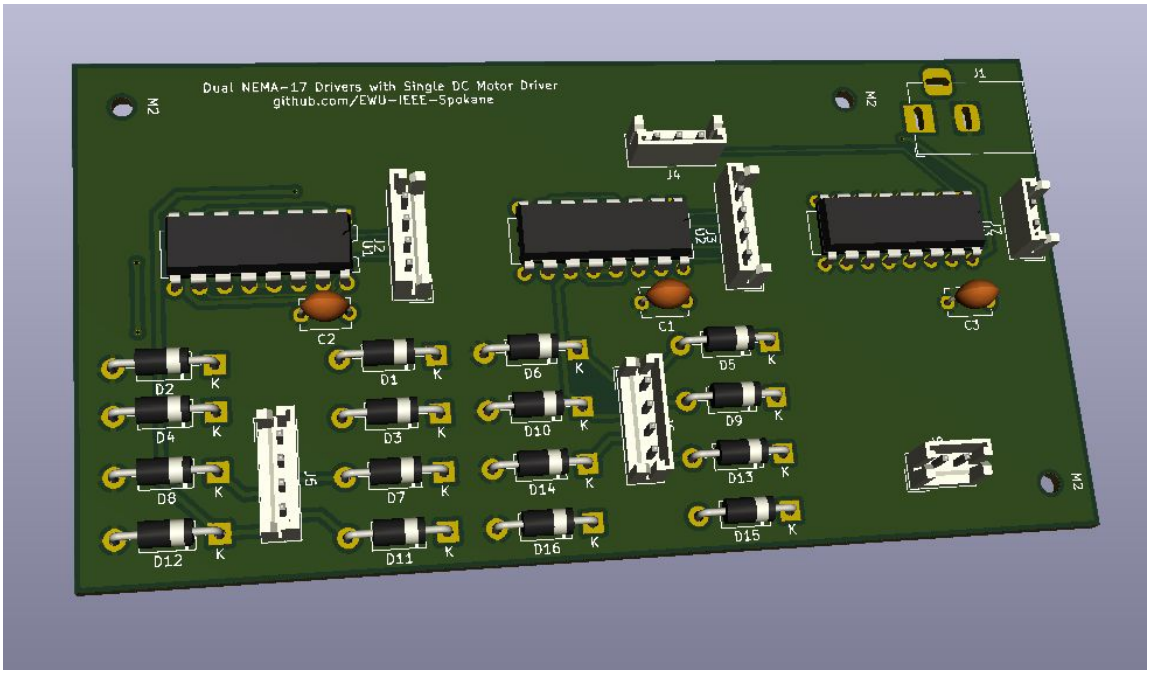

Fig. 1: 3D view of the first design of the motor driver PCB.

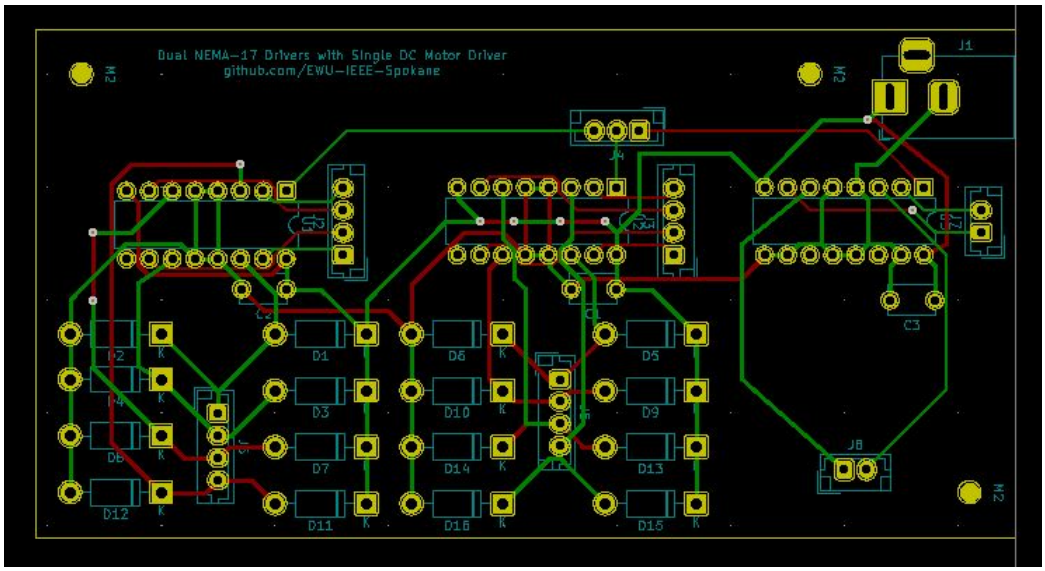

Fig. 2: Trace view of the first PCB design. Notice the poor layout of components, long traces, excessive vias necessary, and ridiculous daisy-chained power supply with slim traces.

Issues with iteration 1:

- The power supply is located at one end of the board, and the ICs are daisy-chained together. When one IC draws a lot of current, the ICs up the chain will likely experience a lot of noise at both their logic supply, and power supply inputs.
- The trace widths for the power supply pins are not wide enough. They would burn and short with less than 1 amp of current. Since the chips are daisy-chained, a lot more than 1 amp (closer to 4) could be drawn through these traces.
- There is a lot of unused space on the board. Traces are long, which will increase noise and thermal losses.
- The outputs are far from the ICs and placed in an awkward position, making running traces to them difficult (there are two output pins on either side of each IC) and long.

Iteration 2

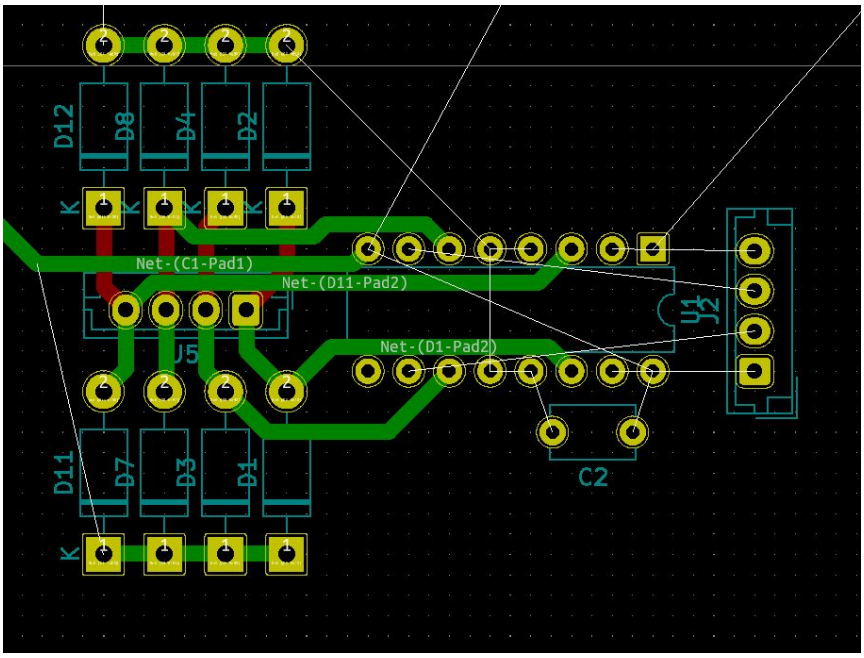

Figure 3: The second iteration of the PCB design. At this stage I focused only on the layout of one L293B, since it would be duplicated for the other. When I noticed issues, I stopped and restarted instead of carrying through and finishing the board.

Improvements in iteration 2:

- The trace widths are now wide enough to carry the desired current, so long as ICs are not daisy-chained, and a copper weight of 2 oz is used.
- The placement of the output pins at the bottom of the IC, instead of the side, allows for shorter, more direct connections between the IC and connector.
- Traces around the heatsink/ground pins are eliminated, allowing better thermal dissipation across the ground planes.
- Due to the better layout, no vias are necessary, and the footprint is reduced in size.

Issues with iteration 2:

- The flyback diodes from output to VCC are on the opposite side of the IC from the VCC input pin. This means a high current trace would have to be run around the board or underneath it. Rotating the flyback diode/output connector section 180° would allow a shorter direct connection.

Iteration 3

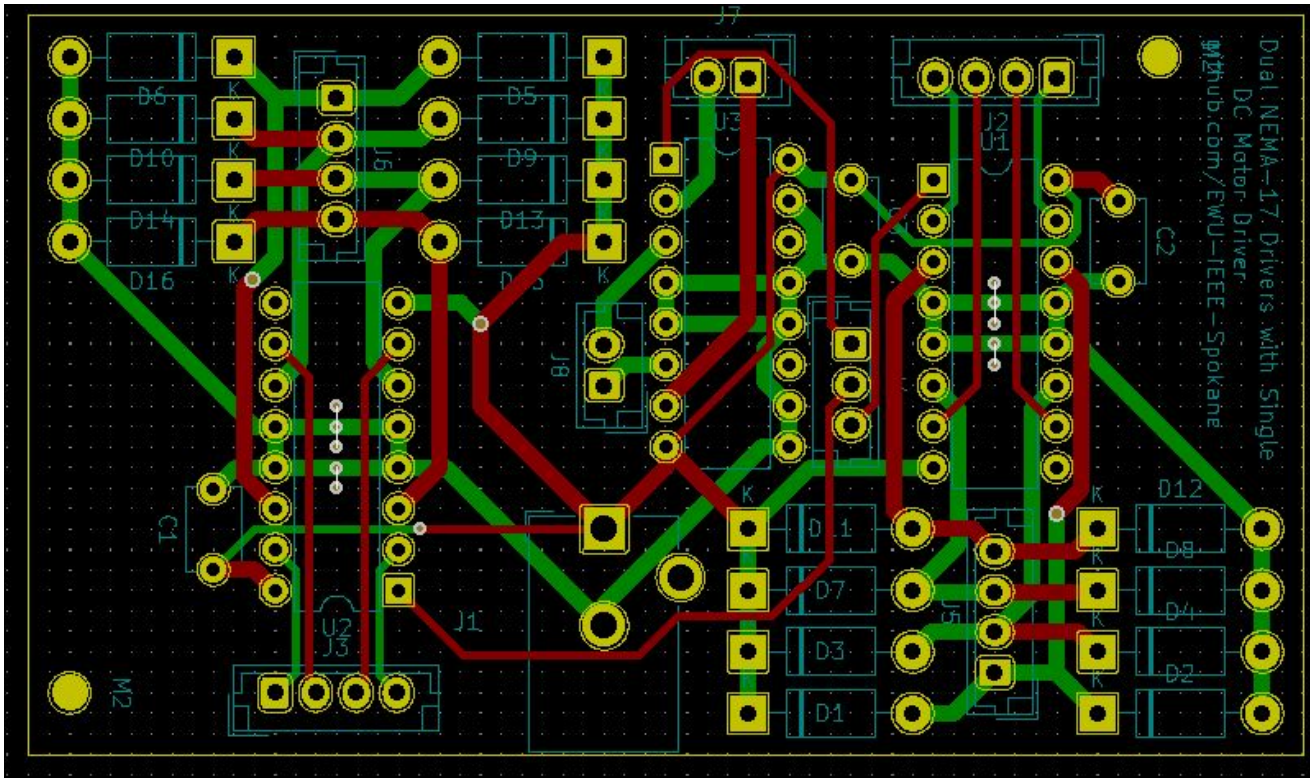

Figure 4: Iteration 3 of the driver PCB. A small layout means high current traces are kept minimized and direct, and noise is reduced.

Iteration 3 improvements:

- Total area is reduced from 64 cm^2 to 34 cm^2 when compared to iteration 1.
- Power connections are direct (the daisy-chaining seen between U1 and U3 does not occur in the design, that is the one modification not shown here).
- Thermal vias are included to conduct heat from the center of each l293B to the ground plane on the backside of the board.
- Total number of vias (excluding thermal vias) is reduced from 9 to 4.

Iteration 3 issues:

- This design will require external heatsinks for the l293B chips as there is simply not enough copper on the ground planes to dissipate the heat