

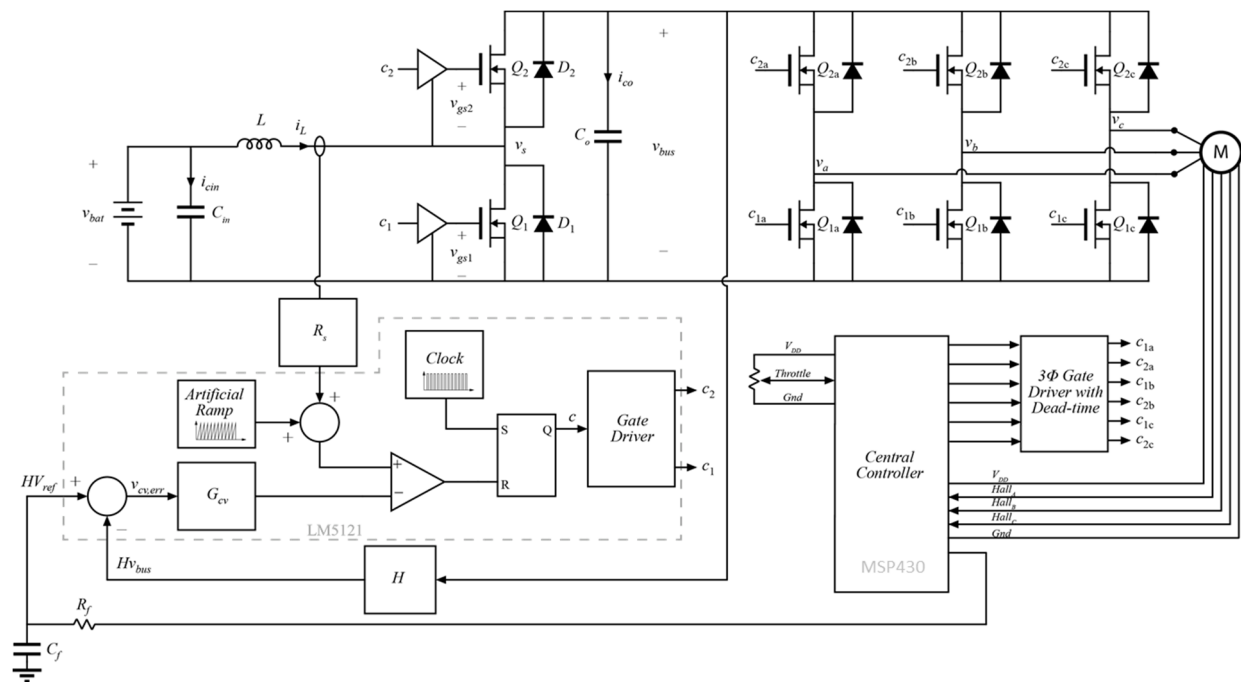
# Experiment 5

## System PCB Layout

ECE 482

The objectives of this experiment are:

- To gain experience with good practices of PCB layout including digital, analog, and power circuitry
- To use knowledge develop in Experiments 1-4 to formulate and implement system improvements



**Figure 1:** Example system diagram for drivetrain power conversion including DC-DC boost, motor driver, and central controller

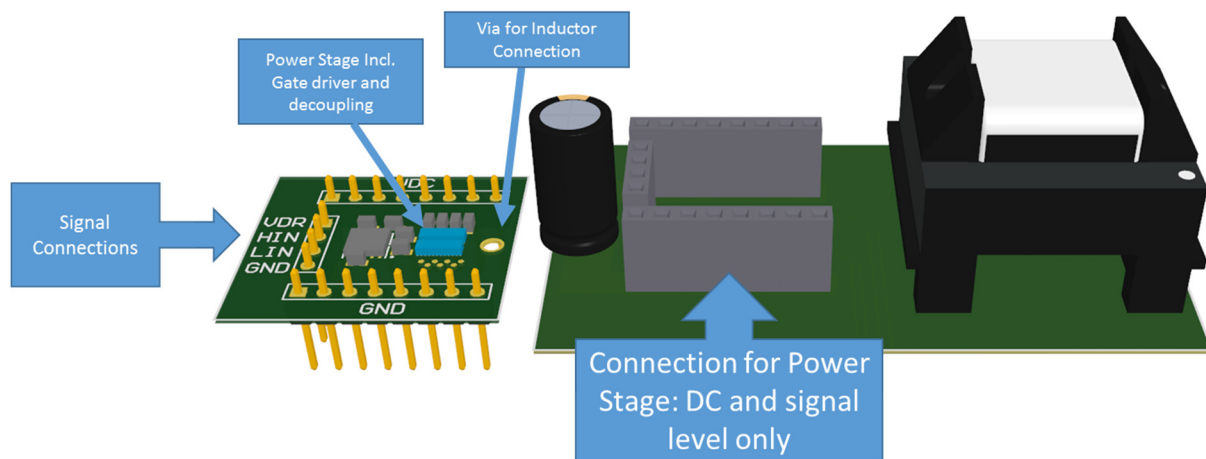
In this lab, you will use Altium Designer to create a two- or four-layer printed circuit board which implements your electric vehicle drive train. This will include both the DC-DC boost converter which you have previously constructed in experiments 3 and 4, as well as the motor driver.

### I. Design Finalization

Working as a group, finalize design decisions which will define your system. This includes the use of GaN devices, redesigning the boost power stage, selecting motor drive topology, and designing any additional circuitry per your prelab assignments.

Because we have time for only one PCB fabrication this semester, make sure to consider “backup plans” throughout the experiment as you perform PCB layout. As an example, if you have concluded that a PI compensator is sufficient for your boost converter, you may want to leave space for a more complex PID or similar compensator, which you do not intend to populate, but will have available in the event of an original design performing below your expectations.

If using GaN devices, consider implementing a two-board setup, where the power devices, gate driver, and relevant decoupling capacitors are on a separate PCB which is connected via board-to-board plugin connectors (as in the MSP430 breakout). This way multiple power stages can be swapped out in the event of a device failure. These will be fabricated as a single PCB, but can be cut apart in the mechanical shop. If this method is employed, make sure to consider connector parasitics.



**Figure 2:** Conceptual example of power stage daughter board. Note that, because the connectors will have  $L-C-R$  parasitics, only DC and signal-level connections are made from board-to-board

## II. PCB layout

Read through the materials on Altium Designer on the course website linked from the experiment 5 page. The PCBs ordered will be “no-touch”, standard specification boards from Sierra Circuits. A design rule file to include these specifications in Altium Designer has been posted on this page.

Additionally, PCB footprint and schematic libraries and a starter project for the experiment are provided. These files contain information on many of the parts included in the nominal system, and start with a nominal schematic. You will need to alter these and make additional connections; use these as a starting point for your PCB design. As you select new components for this lab, include the schematic symbols and PCB footprints that you create in these libraries

for better organization. **Do not** rely on the connections, footprints, or design in these starter files without reviewing to ensure they meet your needs.

### III. Motor Driver Implementation

The motor driver must consist of a three phase inverter with each leg comprised by a transistor half-bridge. Additionally, relevant high- and low-side transistor drivers should be selected so that the devices may be switched on and off. Your driver circuit implementation must be capable of switching each half bridge into one of three states. The states are given in Table I; the circuit must be able to hold each half bridge independently in any one of the three states for an arbitrary time (i.e. an adjustable dead time alone is not sufficient to implement State 0).

TABLE I: HALF BRIDGE STATES IN DRIVE TRAIN INVERTER

State	High Side Device	Low side Device
0	Off	Off
1	Off	On
2	On	Off

In the starter files, three example motor drive implementations are given. You may select to start your design from one of these three, or change it entirely.

### IV. Circuit Connections and Protection

Figs. 3 and 4 detail the electric bicycle platform to which you will attach your completed power converter drivetrain PCB, as well as the included circuit protection devices.

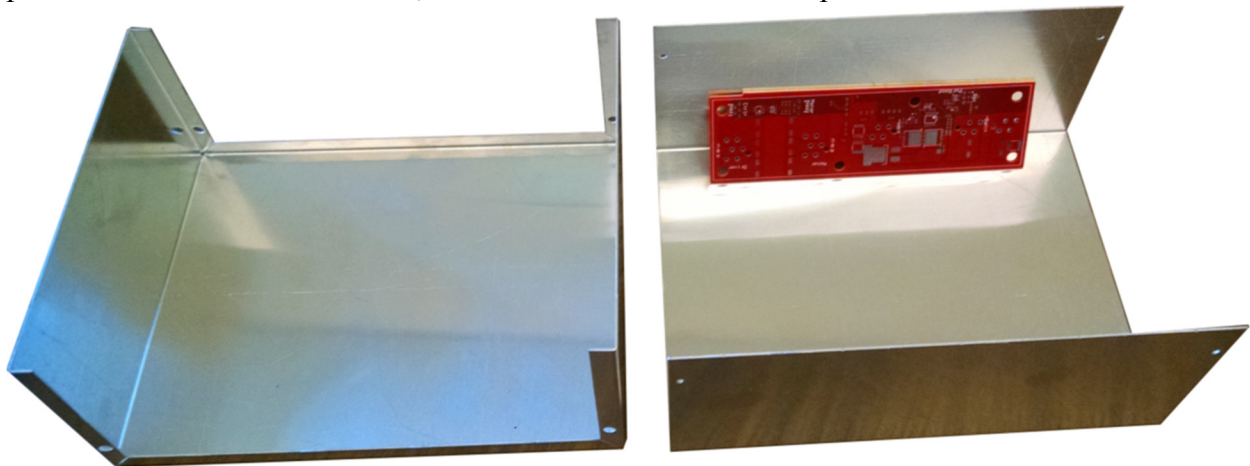
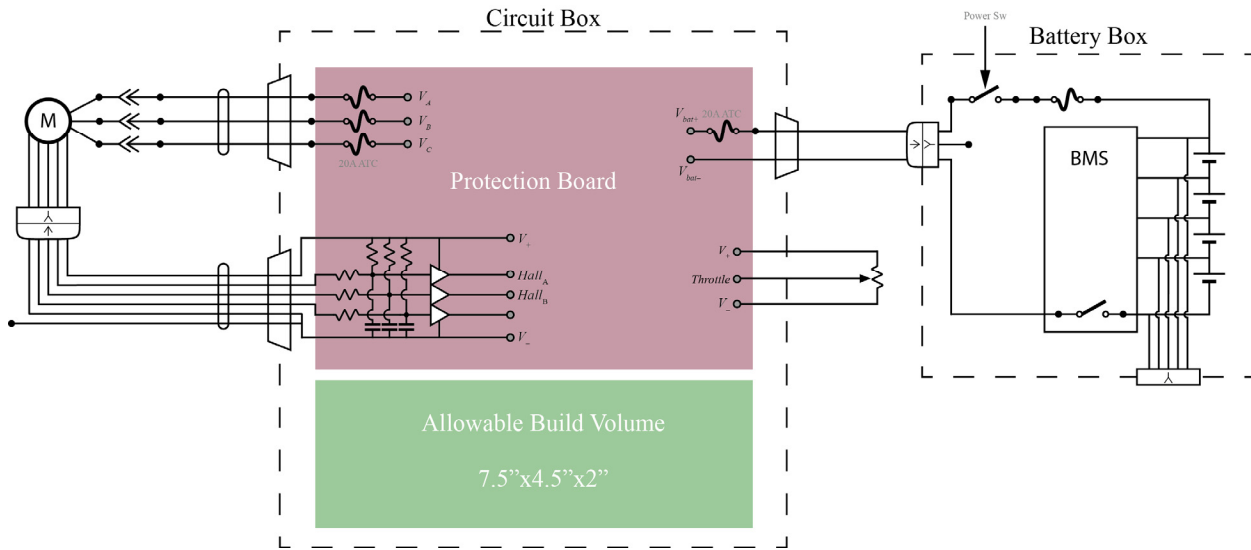


Figure 3: Circuit enclosure and protection board



**Figure 4:** Included circuit protection and connections diagram

Note that protections are included to prevent damage to the motors. Your circuit will not connect directly to the motor, but will instead connect to the protection board. All relevant screw-type connectors are included in the Altium starter files.

Your PCB should stick to the board size defined by the starter files in order to fit within the build space in the circuit box. Six holes are included for mounting to the included standoffs. You will not have access to the Battery Box.

### Auxiliary Supplies

To allow the electric bicycle to be a fully contained system, all components must obtain power from the battery pack itself. A pair of RSO-24xxS isolated power supplies are included in the design. These supplies are available with outputs of 3.3, 5, 9, 12, and 15 V with a common footprint. Depending on your design, you may be able to operate without these, or may require additional supplies.

## V. Deliverables

You will not complete a lab report for this experiment. Instead, you will turn in, by e-mail, the following:

1. Final .zip file of all gerber and NC drill files which passed DFM, along with your board dimensions (in e-mail text).
2. A .zip file of your altium project. Include only:
  - a. Schematics
  - b. Layout
  - c. Project file
  - d. Schematic and Footprint libraries
3. Spreadsheet of all components used which are not currently in your possession or available in the circuits store (See template on experiment 5 webpage).

Be aware that excessive component cost will not be permissible; new purchases must be justified with analysis showing why the benefits outweigh the additional costs. Your grade for this lab will be based on the timely completion of an errorless PCB layout which exhibits good layout practices, as discussed in lecture/notes, and proper implementation of system modification as developed in the prelab assignment.