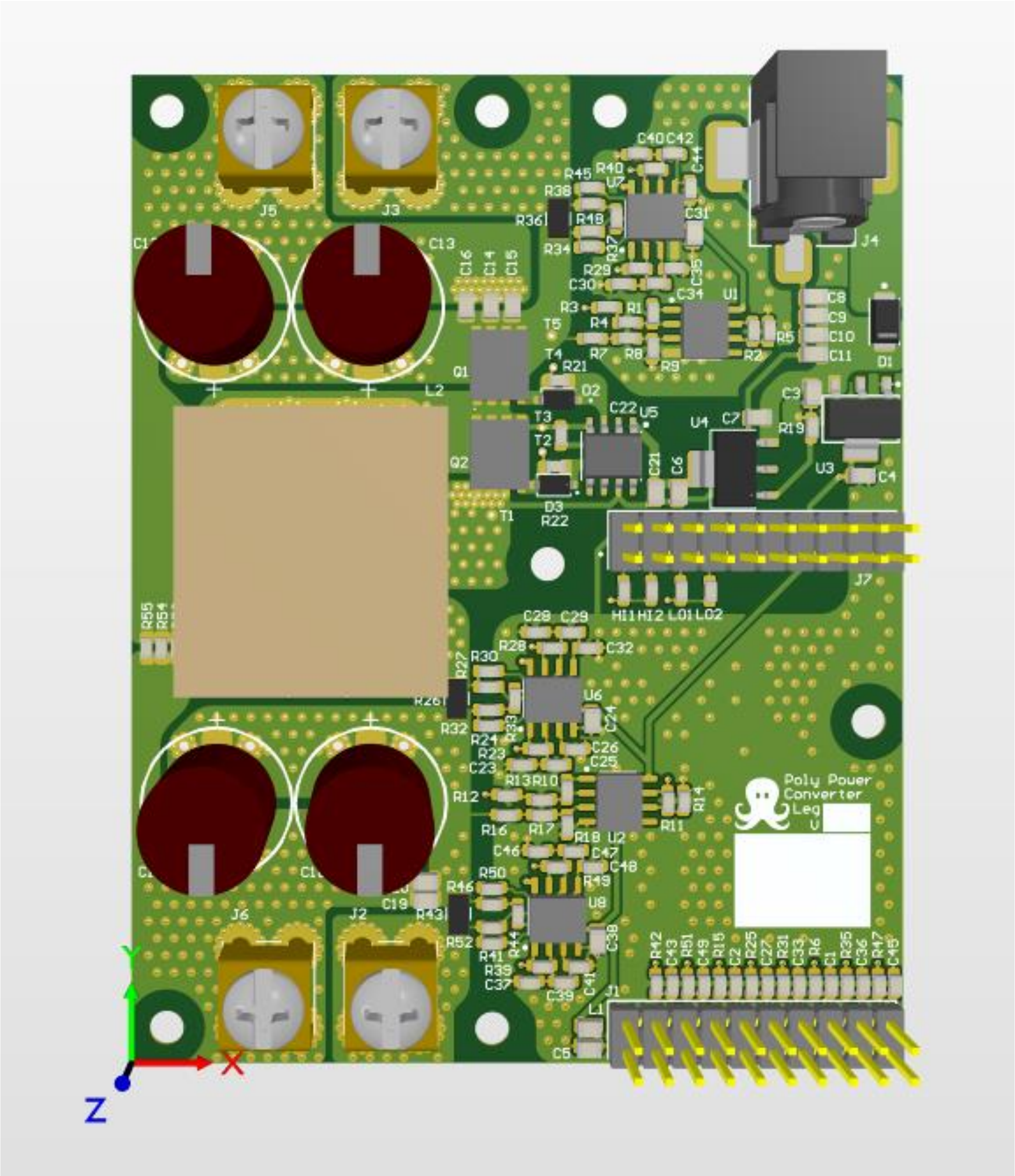
Modular Power Converter Design Notes

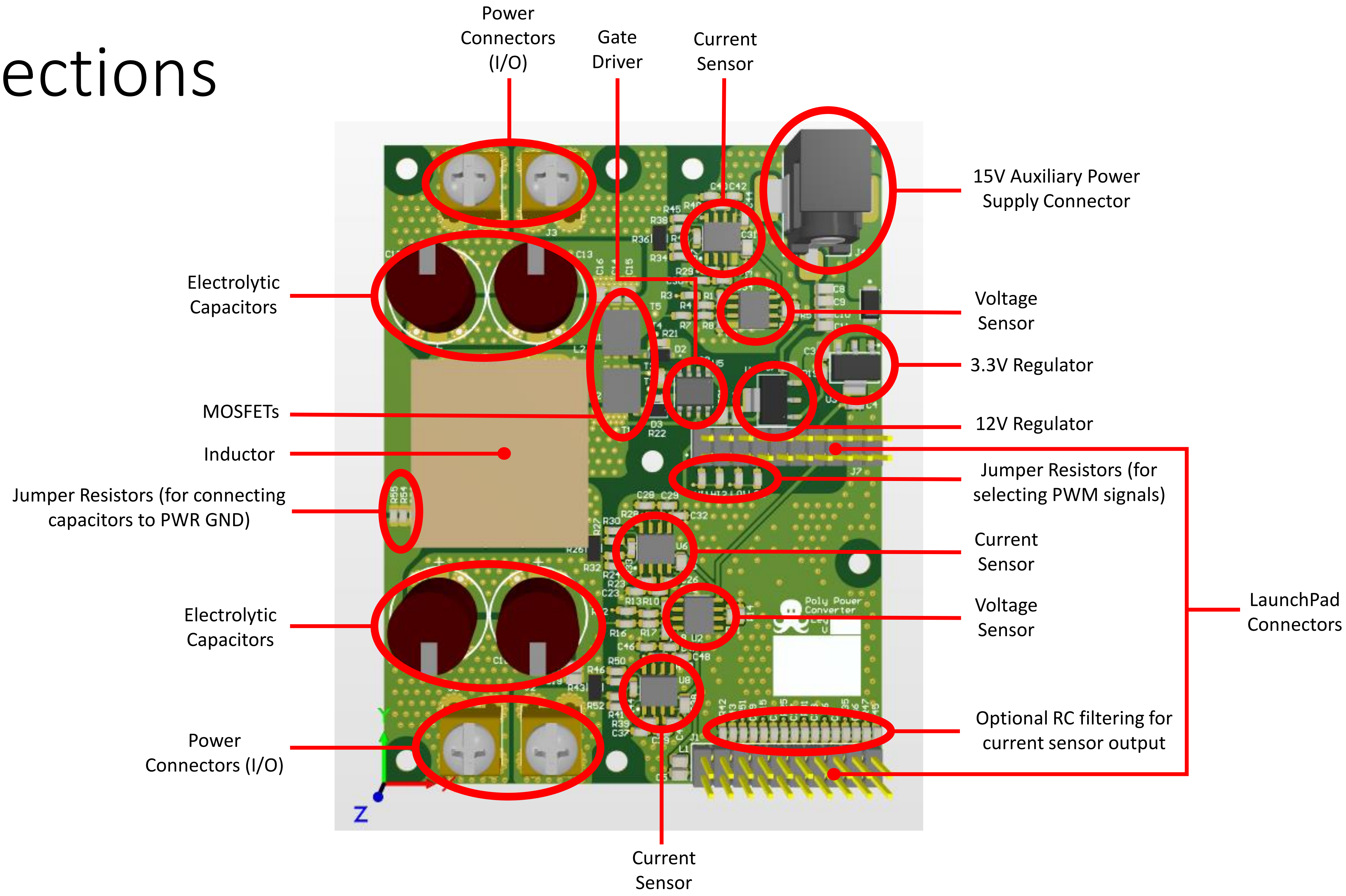
Compiled by Dr. Ignacio Galiano and Anastasiya Rybitska

Overview

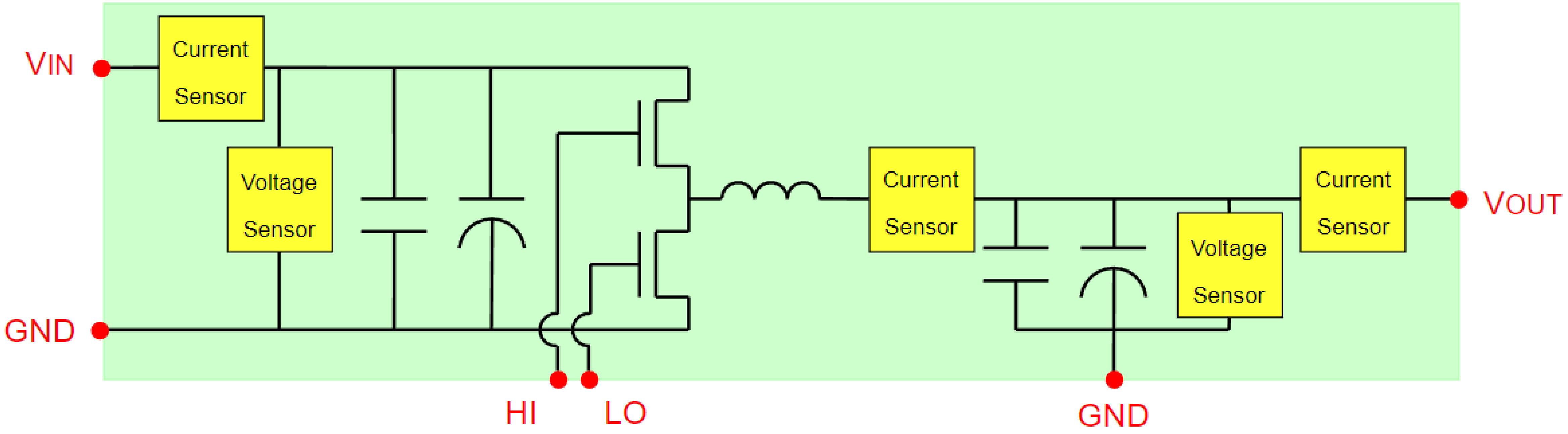
- The Poly Power Converter V1 is a module that can be used on its own, or duplicated and interconnected with itself to create multiple power converter topologies
- It can be used to make buck, boost, back-to-back, half LLC/resonant, full bridge, and dual active bridge converters
- The module on its own can be used a buck or boost converter
- This version is rated for 50V and 10A
- This version uses a single bus for input and output
- It is used along with the TI C2000 Piccolo MCU F280049C LaunchPad™ development kit
- It measures and outputs 5 signals to the LaunchPad, these signals are:
  - Voltage across input electrolytic capacitors
  - Voltage across output electrolytic capacitors
  - Input current
  - Output current
  - Current through the inductor



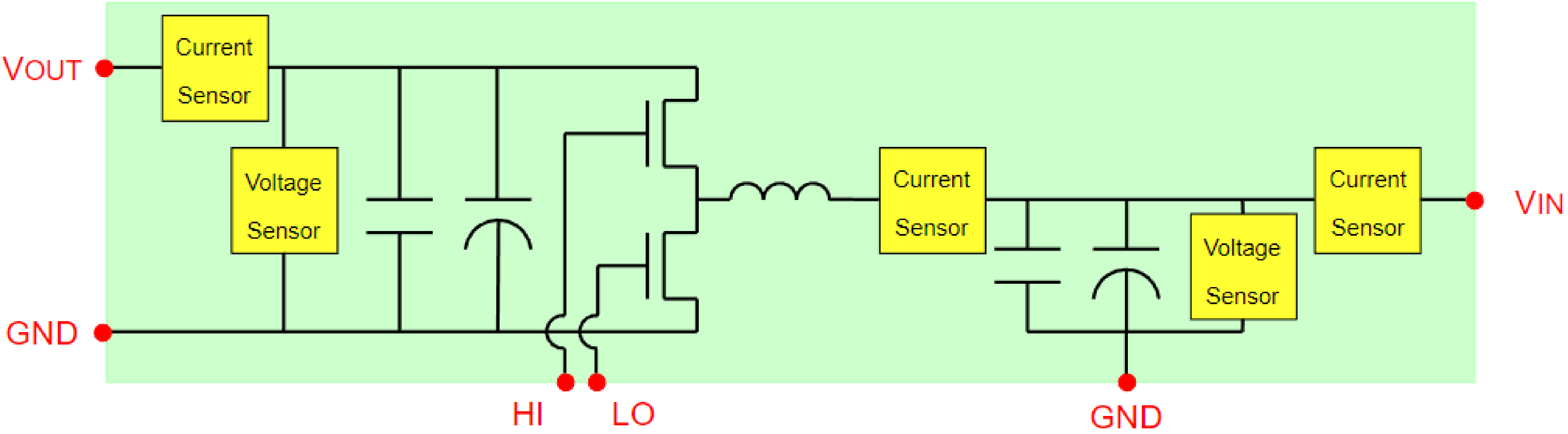
Key Sections



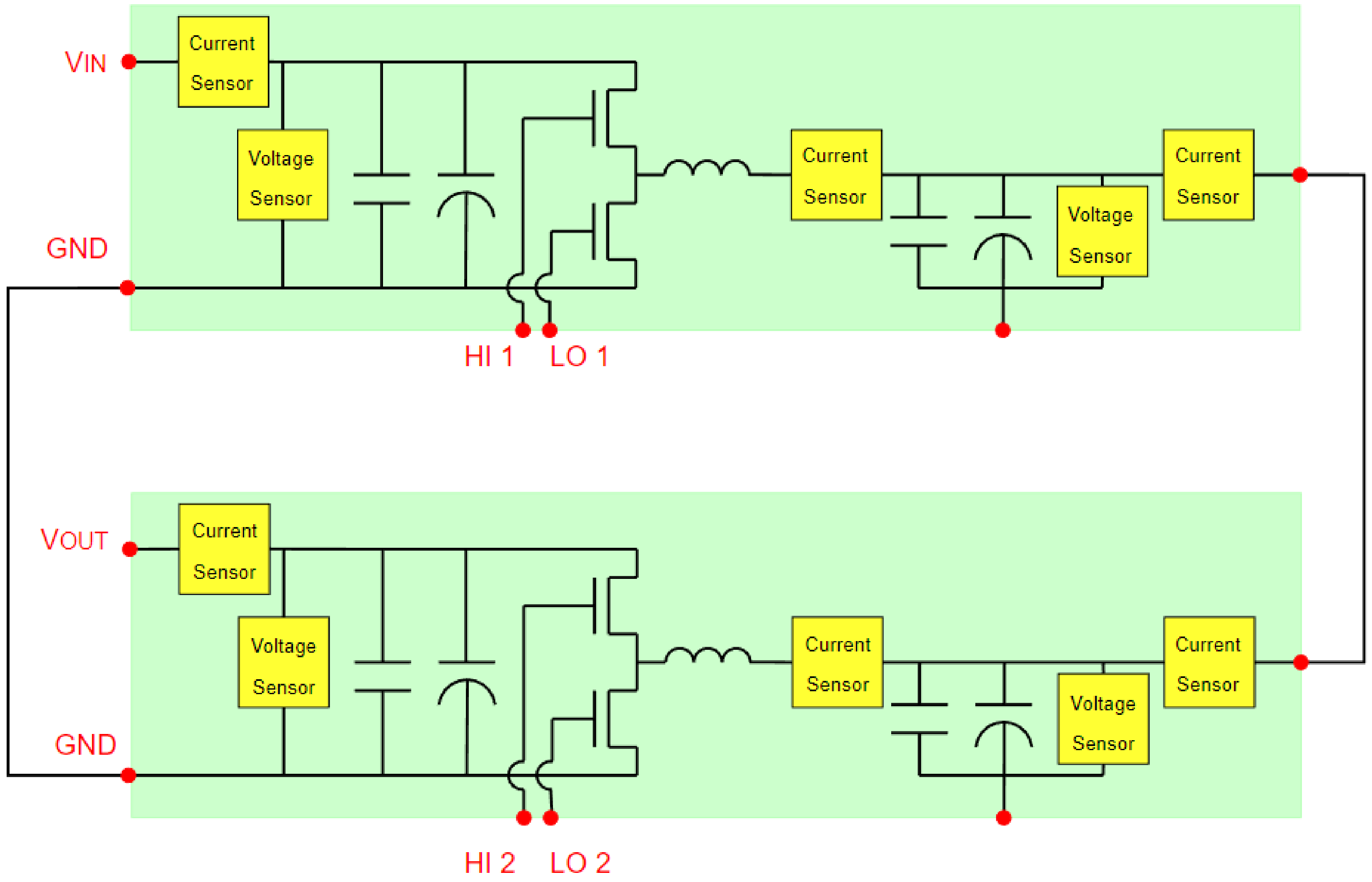
Configurations: Buck Converter



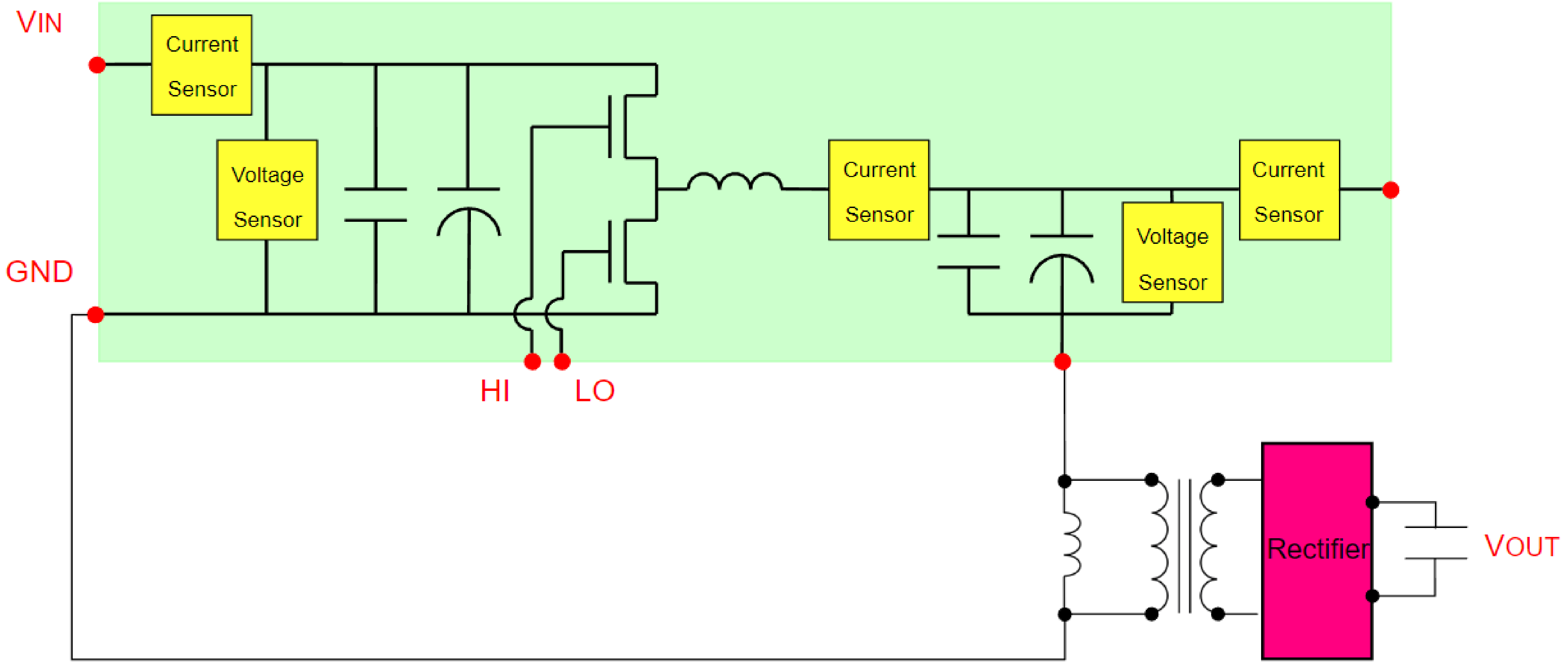
Configurations: Boost Converter



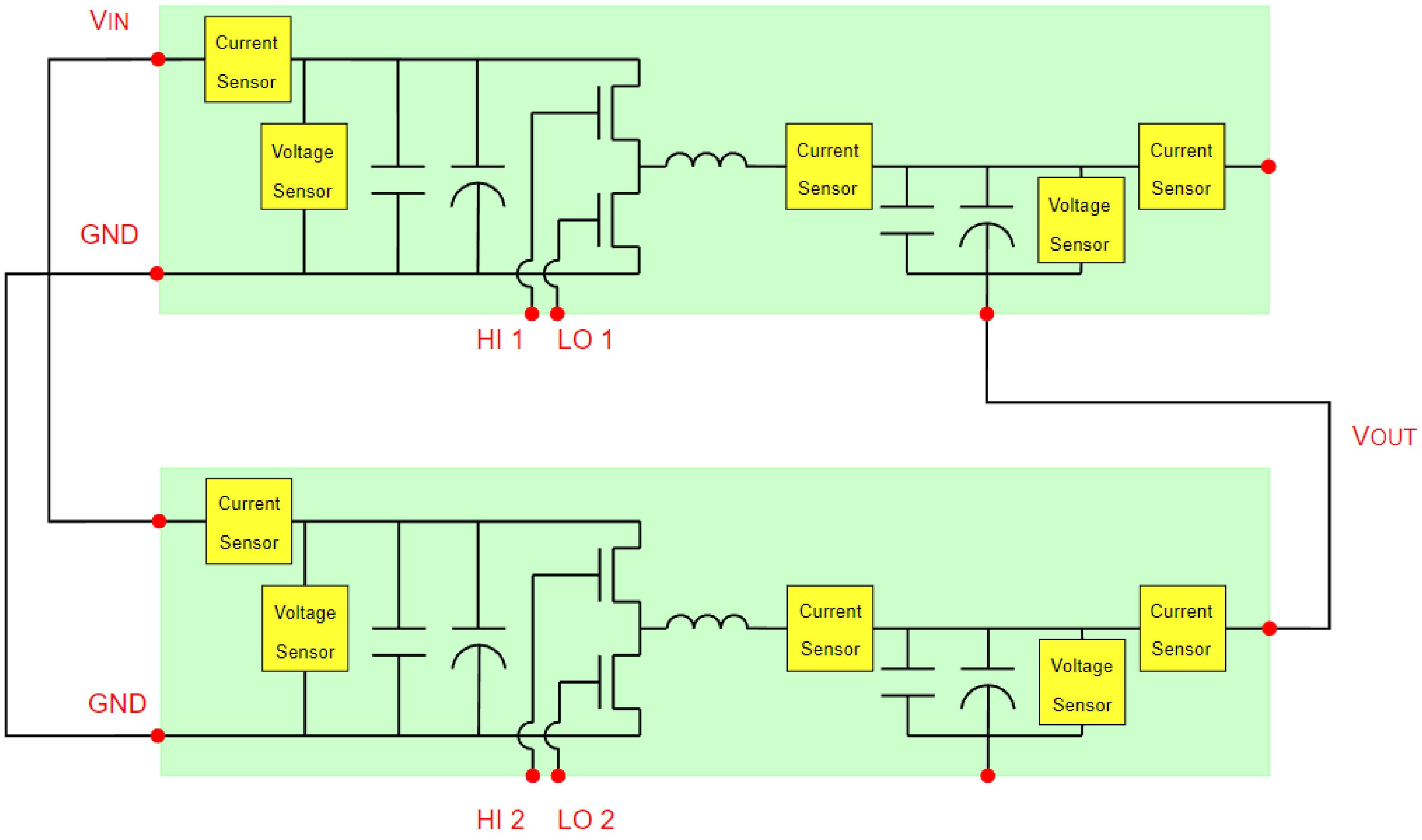
Configurations: Back To Back Converter



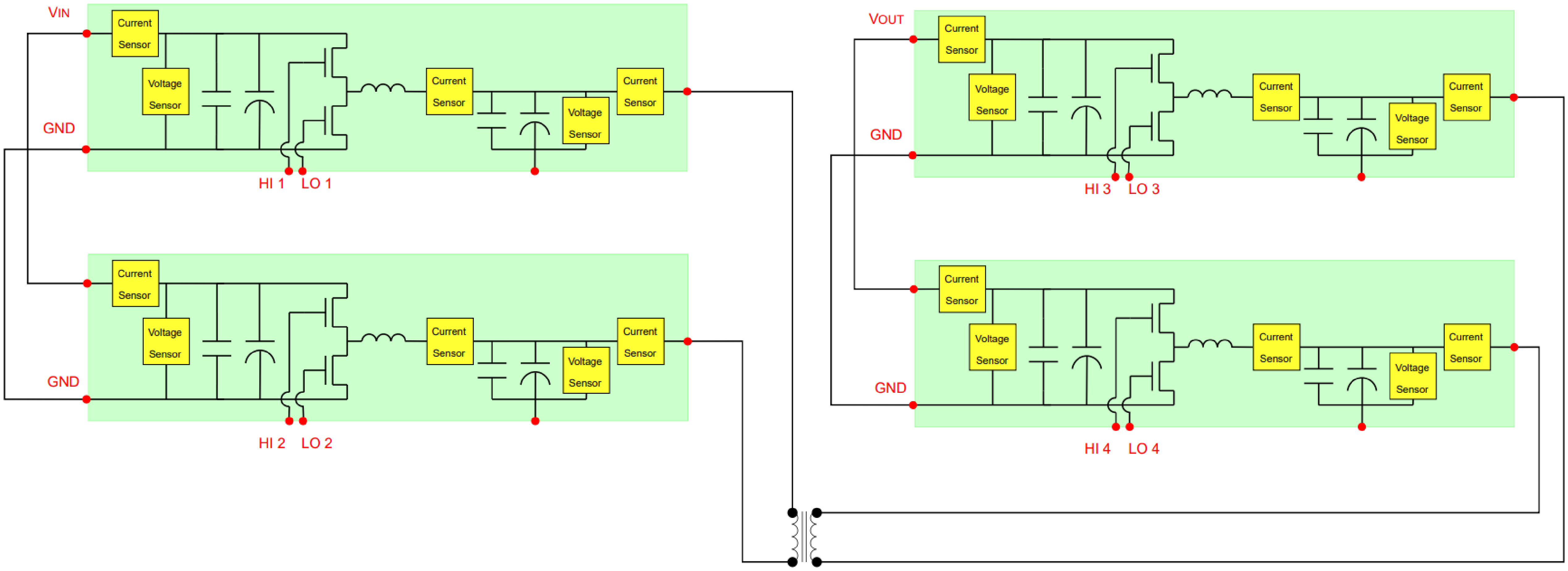
Configurations: LLC/Resonant Converter (Half)



Configurations: Full Bridge Inverter

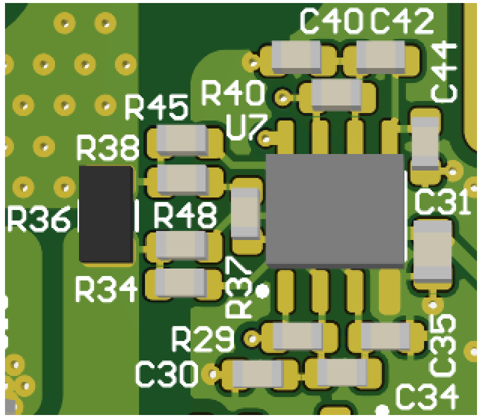


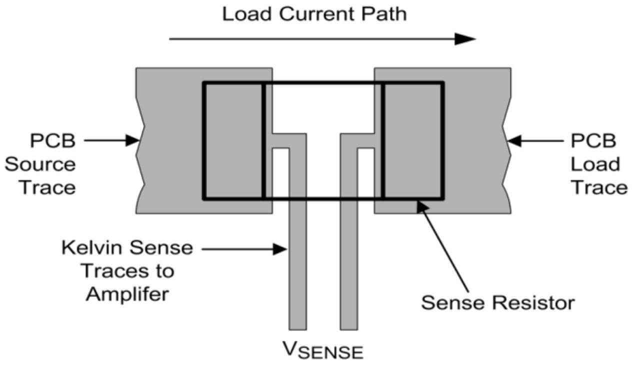
Configurations: Dual Active Bridge Converter

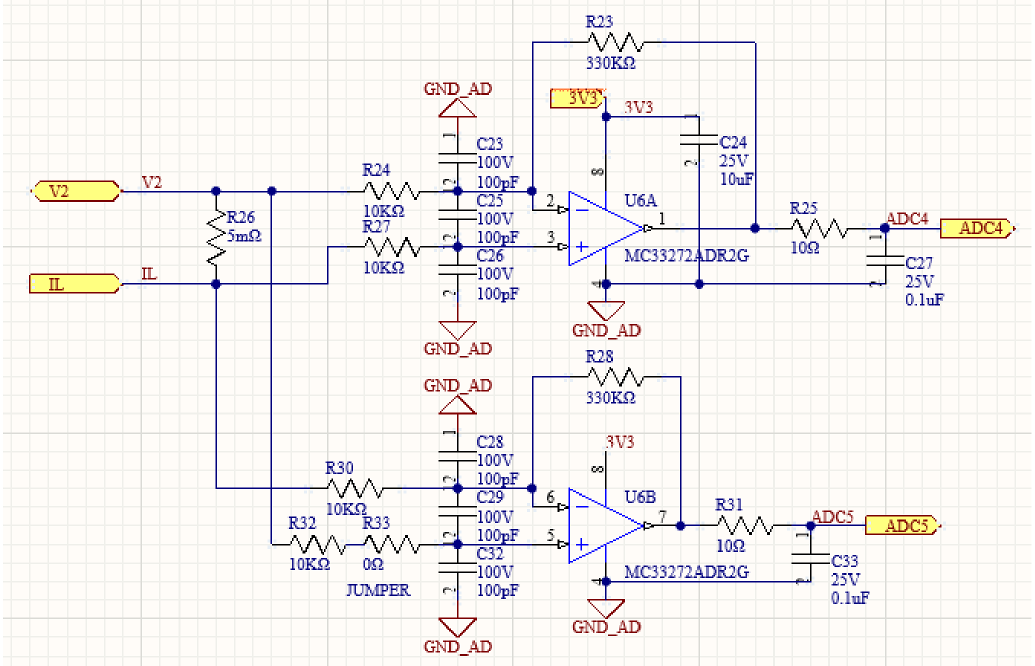


Design Considerations – Current Sensor

- Both hall effect and shunt resistor current sensing methods were considered:
- Hall effect sensor pros:
    - Small footprint, comes made as an IC
    - Isolated
    - Can measure AC and DC
    - Good for high current
    - Has PC protection
- Hall effect sensor cons:
    - Small bandwidth
    - High cost
    - Not in stock (chip shortage)
- Shunt resistor current sensor pros:
    - High accuracy
    - Large bandwidth
    - Cheap
    - More freedom for design
- Shunt resistor current sensor pros:
    - Susceptible to noise
    - Requires external circuitry (op-amps)
    - Not isolated
    - More difficult to measure AC
- Ultimately, the shunt resistor current sensor method was chosen due to the unavailability of IC hall effect sensors (check out the Allegro ones)
- When selecting the shunt resistor, pick based on the voltage expected to be across it and based on the power dissipated
- Use of Kelvin Connection on shunt resistor

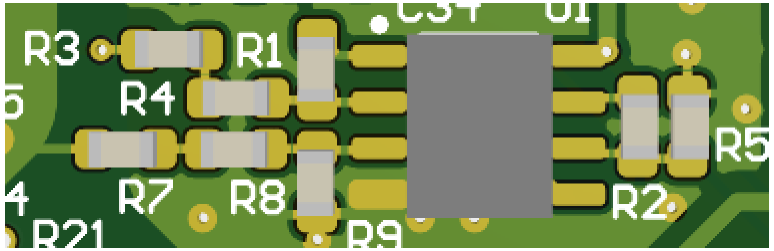


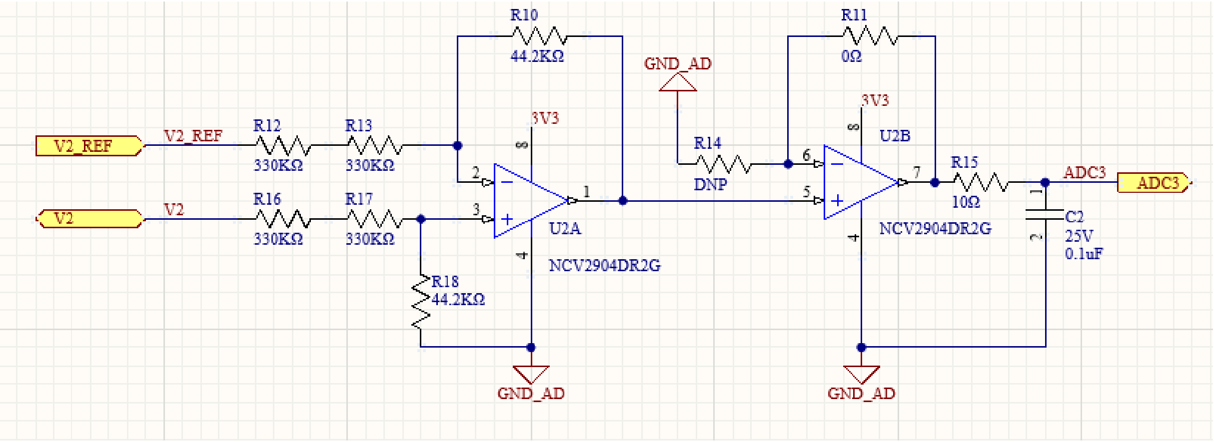




Design Considerations – Voltage Sensor

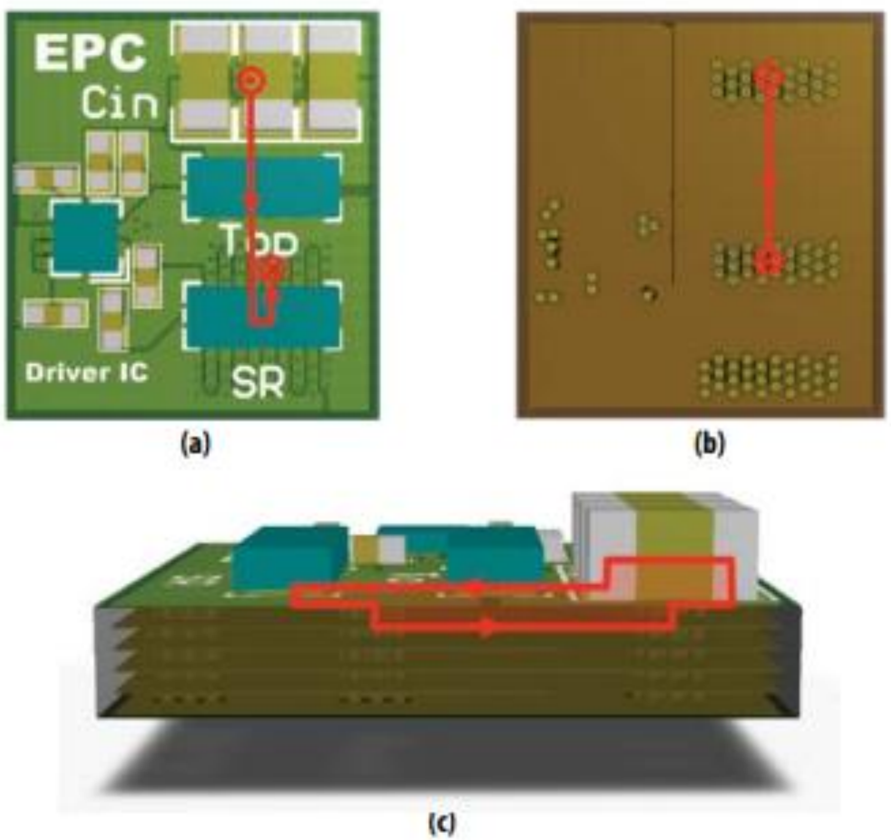
- Both voltage divider and differential op amp methods were considered:
- Voltage dividers are simple and cheap to implement, but their main downfall is that they are not easily implemented for differential voltage sensing
- This reason is why a differential op amp was chosen for voltage sensing
- When looking for differential op amp, look for:
  - High slew rate
  - High bandwidth
  - Low cost
  - High stock
- When picking resistor values, stay withing 250 kOhm and 1 MOhm range, too high and you'll get more noise
- Resistors can be split up into two or more to reduce required power rating across each one
- Use of Kelvin Connection on capacitors (see current sensor slide for diagram)
- Voltage sensing resistors are placed as close as possible to the capacitor

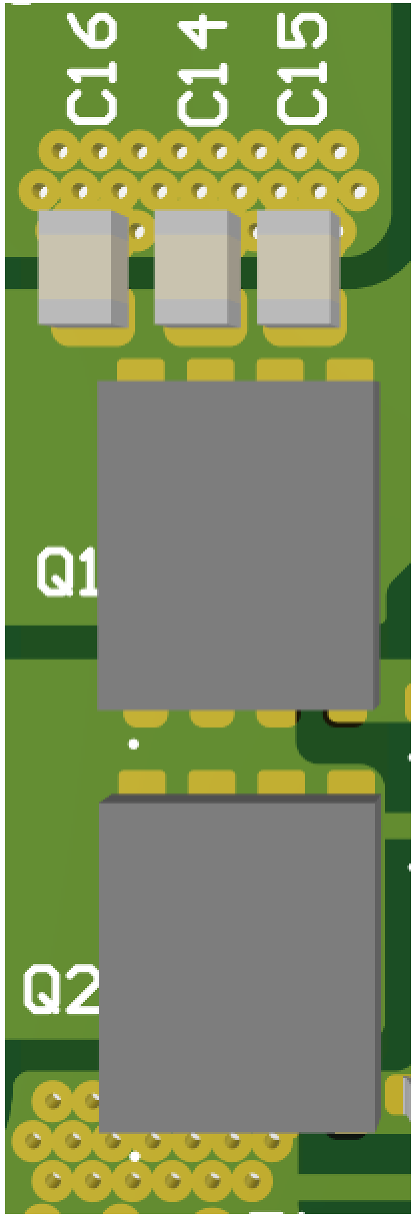




Design Considerations – Switches

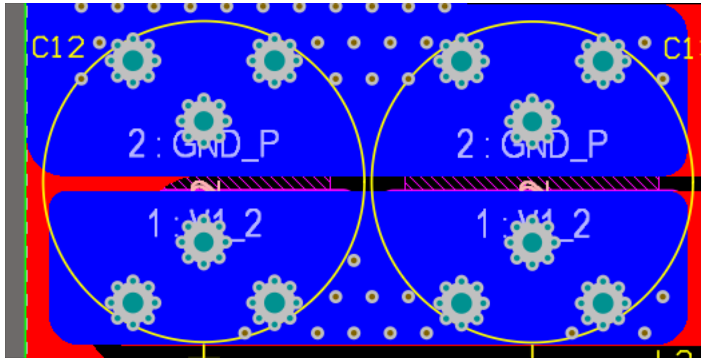
- Voltage rating of switches was made double of voltage rating of board to accommodate for voltage spikes (100V)
- Rds On: Less than 50 mOhm is acceptable, lower than 20 mOhm is best
- Cost: preferred under $10, acceptable under $20
- Junction to case parameter: don’t want it to exceed 110 Celsius (typically)
- Considered making a custom footprint that would accommodate 8-powerVDFN, 8-powerTDFN, 8-SOIC, DPak, and D2Pak packages
- Will operate at frequency range of 50 to 100 kHz, but 250 kHz is absolute maximum
- Layout of switches and ceramic capacitors of power loop was based on recommendations from the “Optimizing PCB Layout with eGaN FETs” white paper (https://epc-co.com/epc/Portals/0/epc/documents/papers/Optimizing%20PCB%20Layout%20with%20eGaN%20FETs.pdf)

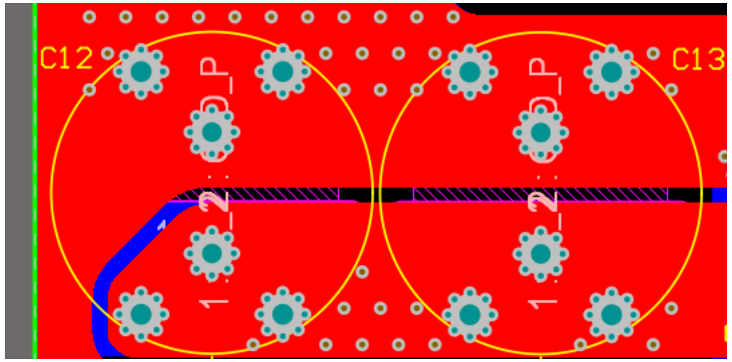




Design Considerations – Input/Output Capacitors

- Voltage rating of electrolytic capacitors was made double of voltage rating of board to accommodate for voltage spikes (100V)
- Customized footprint made to accommodate electrolytic, ceramic, and film capacitors
- Allow for multiple capacitors to be used in order to reduce current through each capacitor
- PCB had copper pour on top and bottom of board to increase heat dissipation
- Formulas used for calculations of capacitor value shown to the right. Top formula is for the buck configuration, and bottom is for the boost. The maximum value calculated is chosen as the reference value for the design





$$C = \frac{V_{in}(1-D)D}{8Lf_{sw}{}^2\Delta v_o},$$

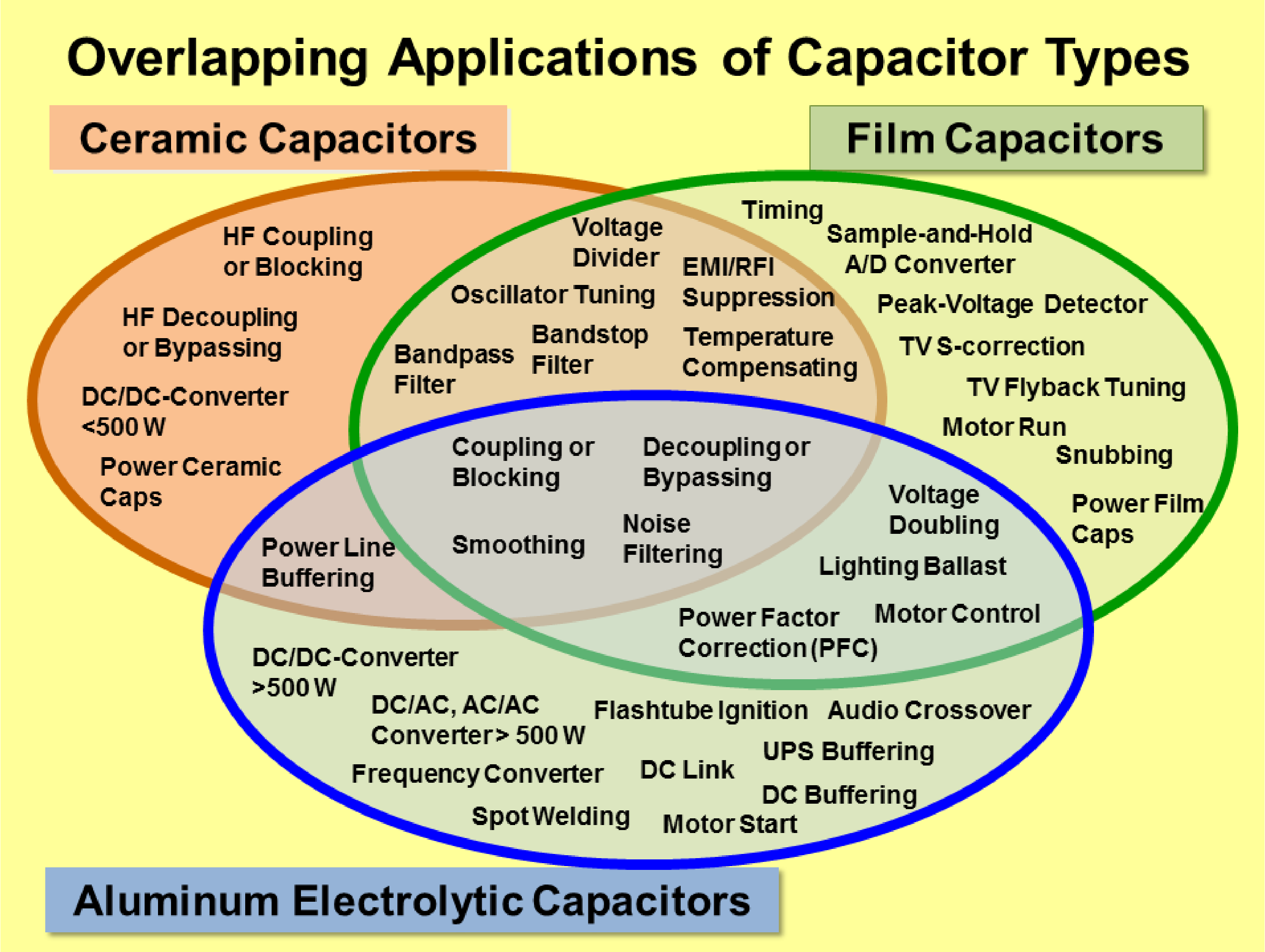
$$V_{in} = 0\ to\ 50V, D = 0.1\ to\ 0.9, \Delta v_o \ll 5\%, f_{sw} = 50\ to\ 100\ KHz$$

$$C = \frac{V_{in}D}{(1-D)R_o f_{sw}\Delta v_o},$$

$$V_{in} = 0\ to\ 50V, D = 0.1\ to\ 0.9, \Delta v_o \ll 5\%, f_{sw} = 50\ to\ 100\ KHz, R_o = 5\ Ohm$$

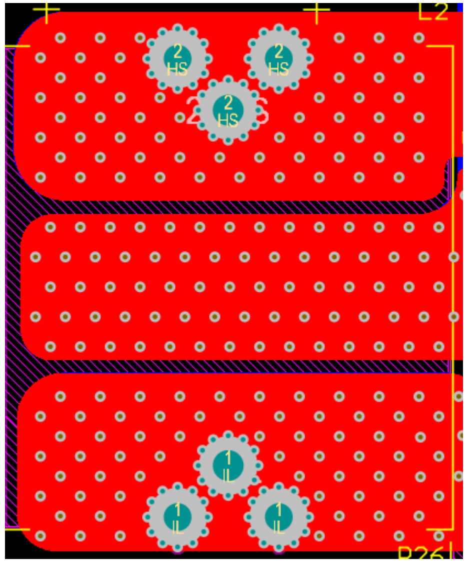
Design Considerations – Input/Output Capacitors

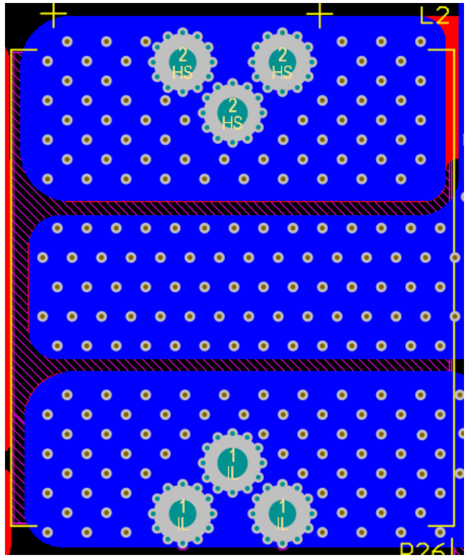
- Ceramic:
    - Low ESR
    - Non-polarized
    - Effective at high frequency
- Electrolytic:
    - High ESR
    - Polarized
    - Have the highest capacitance per volume compared to other capacitors
    - Effective at low frequency
- Film:
    - Low ESR
    - Non-polarized
    - Very high precision capacitance
    - Retain capacitance longer than other types of capacitors
    - Effective at high frequency



Design Considerations – Inductor

- Current rating was set to be 1.5x of max inductor current to accommodate for current spikes (15A)
- Max inductor current ripple: 20%
- Series resistance is important to consider when choosing a component (go for lower when possible)
- Customized footprint made to accommodate common through hole/surface mount inductors (for our requirements)
- PCB had copper pour on top and bottom of board to increase heat dissipation
- Formulas used for calculations of inductor value shown to the right. Top formula is for the buck configuration, and bottom is for the boost. The maximum value calculated is chosen as the reference value for the design





$$L = \frac{V_{in}(1-D)D}{\Delta i_l f_{sw}},$$

$$V_{in} = 0\ to\ 50V, D = 0.1\ to\ 0.9, \Delta i_l \ll 20\%, f_{sw} = 50\ to\ 100\ KHz$$

$$L = \frac{V_{in}D}{\Delta i_l f_{sw}},$$

$$V_{in} = 0\ to\ 50V, D = 0.1\ to\ 0.9, \Delta i_l \ll 20\%, f_{sw} = 50\ to\ 100\ KHz$$

Known Issues

Version 1:

- The 15V auxiliary power supply is connected wrong, the right and left pins (looking down on the PCB) should be ground, and the bottom one should be power
- All connectors need silkscreen labelling to indicate input voltage, output voltage, GND, and type of ADC signal
- Everything is challenging to hand-solder because it is all so close together, and so small
- Footprint for U3 needs to be bigger (make the pads wider and/or longer)
- Fix the pin 1 dot designation on D1 (the dot on the diode actually indicates pin 2)
- Make the 0805 footprint pads longer (need more space for soldering)
- Components near the header connectors need to be moved a little further away
- Consider adding LEDs to indicate power ON/OFF (for 15V, 3.3V, 12V, and voltage input/output)
- HPWM and LPWM pins are incorrectly assigned on the LaunchPad connector J7. Currently it goes pin 1: HPWM1, pin 3: HPWM2, pin 5: LPWM1, pin 7: LPWM 2. It should be instead, pin 1: HPWM1, pin 3: LPWM1, pin 5: HPWM2, pin 7: LPWM 2
- Really large voltage drop from 15V to 3.3V for the linear regulator U3. It heats off, and triggers thermal shut off. Consider cascading another regulator or using a switching regulator
- All current sensing circuits are missing two resistors in the schematic that are needed for differential mode sensing
- Current orientation of LaunchPad connectors forces user to connect the LaunchPad board with the long side pointing to the left (if looking down on the PCB). Initially, the LaunchPad was meant to be placed with the long side pointing to the right. This issue is not a mandatory fix
- Test holes should be made larger if possible
- Current sensing amplifiers do not output at a steady gain. Gain should be 33, but gain decreases as current across shunt resistor increases (for ex. At 5A, gain is 30)
- Footprints for MOSFETs need to be made bigger for easier soldering and troubleshooting
- Gate driver on BOM is incorrect (wrong VIL/VIH), the correct part number is UCC27211DR