# LTSPICE ASSIGNMENT

## ENEL372

TIM DOUGLAS 93298208
CADE HINMAN 27567040
CONNER HORN 36757479

This is the LTSpice schematic, that was designed against the specifications. This is a buck converter, that steps down a 15V input voltage to a ~7V output voltage. The RB050LAM-40 diode has been used instead of the SB240S, as this is not currently available in the LTSpice library. In the physical circuit, a SB240S diode will be used. Figure 1 shows the completed Buck Converter Circuit.

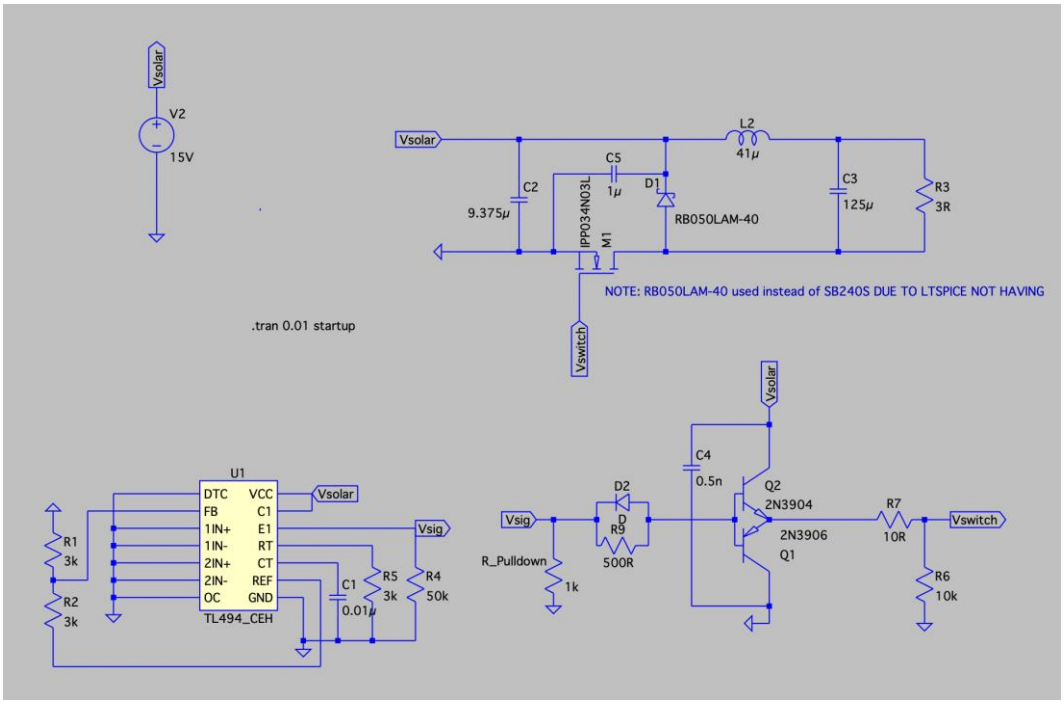


*Figure 1: Buck Converter Circuit*

There is a brief spike of noise in the MOSFET drain to source voltage, occurring between 0.3ms and 0.6ms, however this is at the beginning of the simulation. Once the simulation settles, the switching behaviour present smooths. This switching noise can be considered as negligible and ignored. Figure 2 shows the Drain to Source Voltage Output.

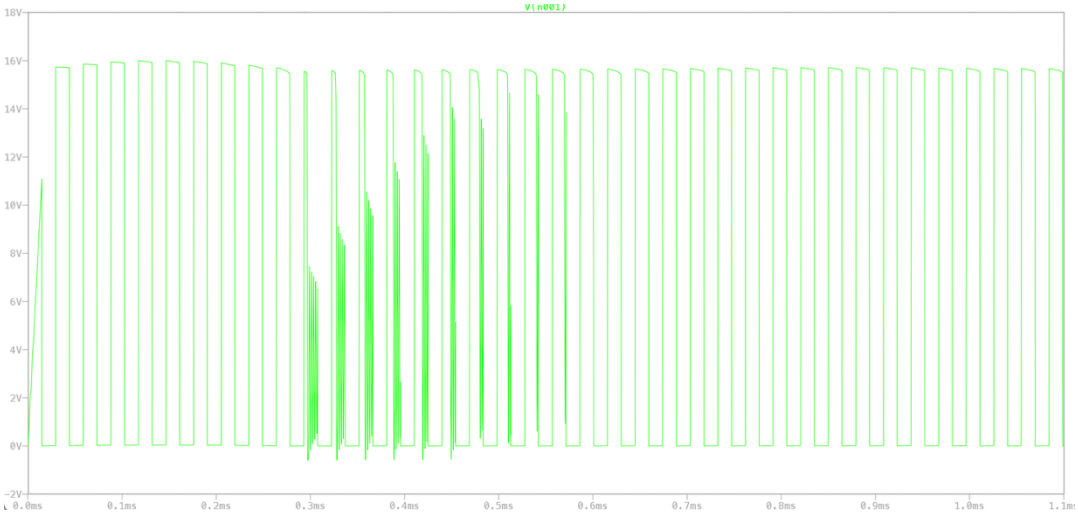


*Figure 2: Drain to Source Voltage Output*

There is a brief spike of noise in the MOSFET gate to source voltage, occurring between 0.3ms and 0.6ms, however this is at the beginning of the simulation. Once the simulation settles, the switching behaviour present smooths. This switching noise can be considered as negligible and ignored. Figure 3 shows the Gate to Source Voltage Output.

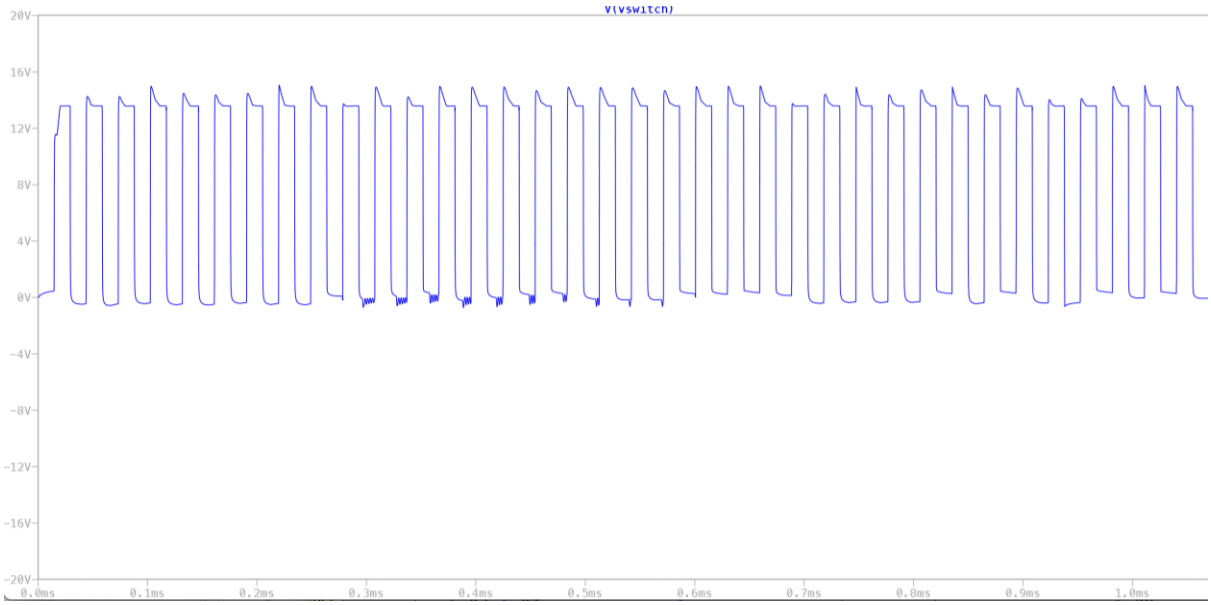


*Figure 3: Gate to Source Voltage Output*

The inductor current is depicted below in Figure 4. When looking at this graph, there is a clear instability in current from 0-0.5ms. This can be explained by the startup transient response. This occurs when the converter first turns on and the capacitor has no charge and acts as a short circuit as it draws large amounts of current to charge up to the desired voltage. Following this, the current begins to settle over time until reaching true steady state and outputting a stable triangular waveform, this is known as Continuous Conduction Mode (CCM). The average current can then be derived using this waveform.

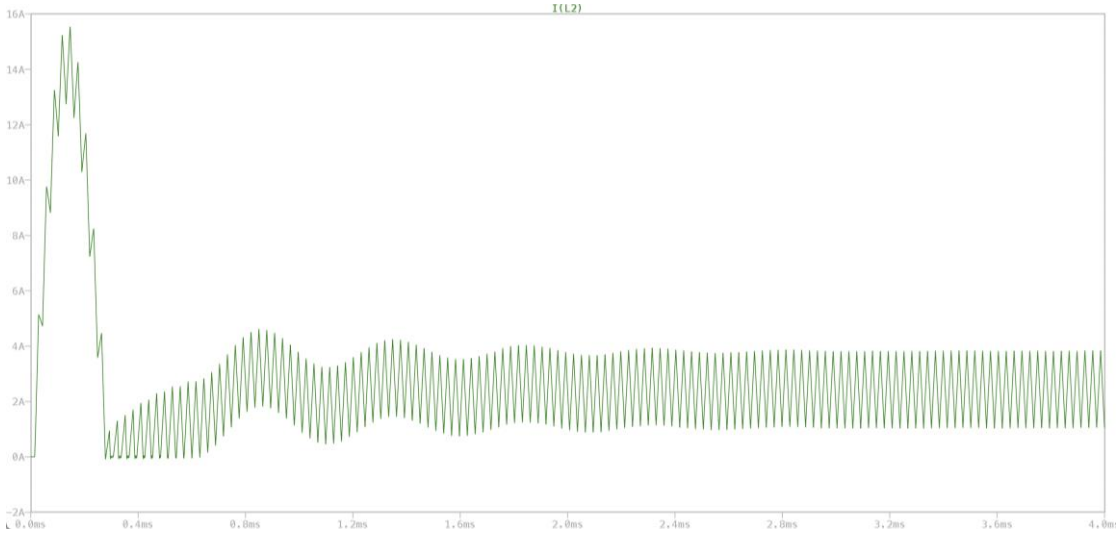


*Figure 4: Inductor Current*

As with the previous measurements and plots, there is an expected startup transient response with the output voltage (Vout). The overshoot seen here (Vmax in the startup) is important to consider and monitor to protect sensitive components withing the circuit that are not rated for this voltage. As seen in Figure 5, this overshoot is ~12.5V and thus should not cause any damage to the circuit due to the overall circuit design and use of necessary resistors. When looking at this graph, this initial overshoot and oscillation begin to settle in an appropriate amount of time (~2.5ms) at an output voltage of ~7.3V, thus meeting the specifications.

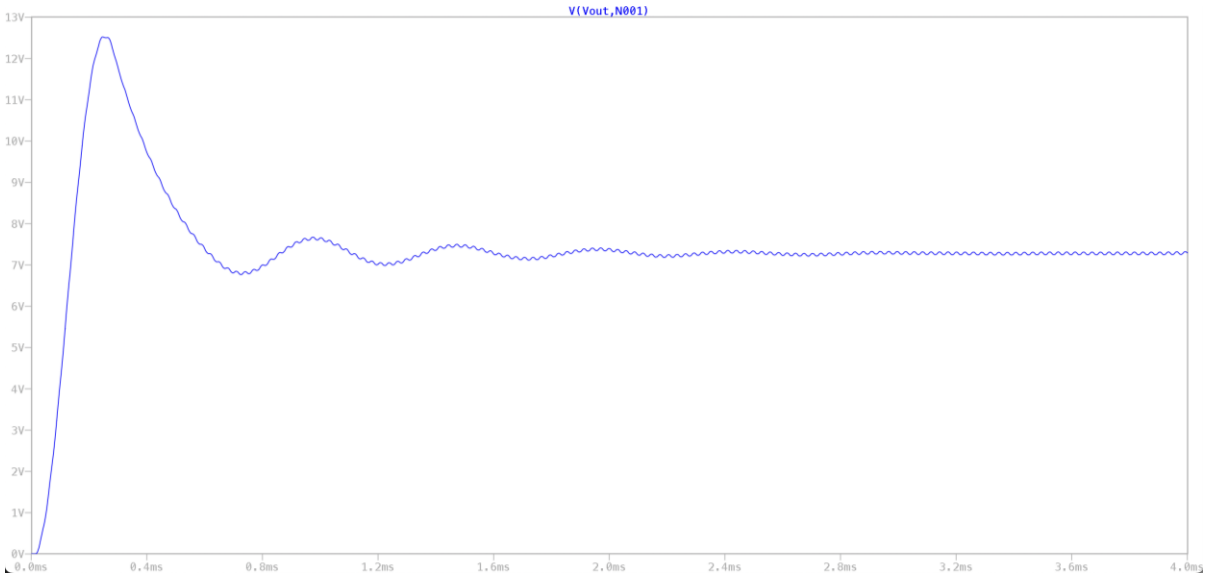


*Figure 5: Output Voltage*

The LTSpice setup requires calculating the inductance for the inductor, and the corresponding air gap to get the calculated inductance. Calculations for the input and output capacitor values were also determined with the maximum capacitance limited to 350µF. These calculated values provide a basis for translating the simulation results to into physical circuit values in the future. These are derived below.

$$V_s = 15\,V,\ \ I_o = 1.5\,A,\ \ f_s = 40\,kHz$$

$$L > \frac{V_s}{8 f_s I_o} = 31.25\,\mu H$$

$$i_{max} = \Delta i + \frac{\Delta i}{2} = 2.25\,A, \quad B_{max} = 300\,mT$$

$$NA_{main} = \frac{L i_{max}}{B_{max}} = 2.34 \times 10^{-4}\ Turns \cdot m^2$$

$$N_{min} = \frac{NA_{min}}{A} = 37\ Turns \implies 40\ Turns\ (Rounded)$$

$$R = \frac{N^2}{L} = 51.2 \times 10^6\ \frac{Amp \cdot Turns}{Weber}$$

$$l = R\,\mu_0\,\mu_r\,A = 0.6485\,m = 648\,mm$$

$$C_{out} = \frac{V_O(1-D)}{\Delta V_O(8Lf^2)} = 125\mu F$$

$$C_{in} > \frac{I_O}{4 f_S \Delta V_S} = 9.4\mu F$$

The main goal of this assignment is to step down a 15V input to ~7V stable output using a buck converter. The simulated buck converter in this report was designed using the values derived from the earlier mentioned calculations. These values were then used to design and simulate the buck converter on LTspice. Using this buck converter and simulation, this goal was clearly achieved when looking at the plots, and thus, further development in the real world can be undertaken.

The biggest challenges that occurred during this stage of development was around circuit design and getting used to the LTspice software. Familiarization with the software came naturally and was soon no longer an issue. However, the circuit design was a lingering issue that resulted in large ripple and wrong output voltages. Through discussion and further research, these issues were resolved.

As for future development, further improvements and optimization can take place alongside real world considerations. As for real world applications, the proposed design is currently built and implemented using a bread board, however, it would be far better to design and manufacture a PCB once real world testing has been concluded.