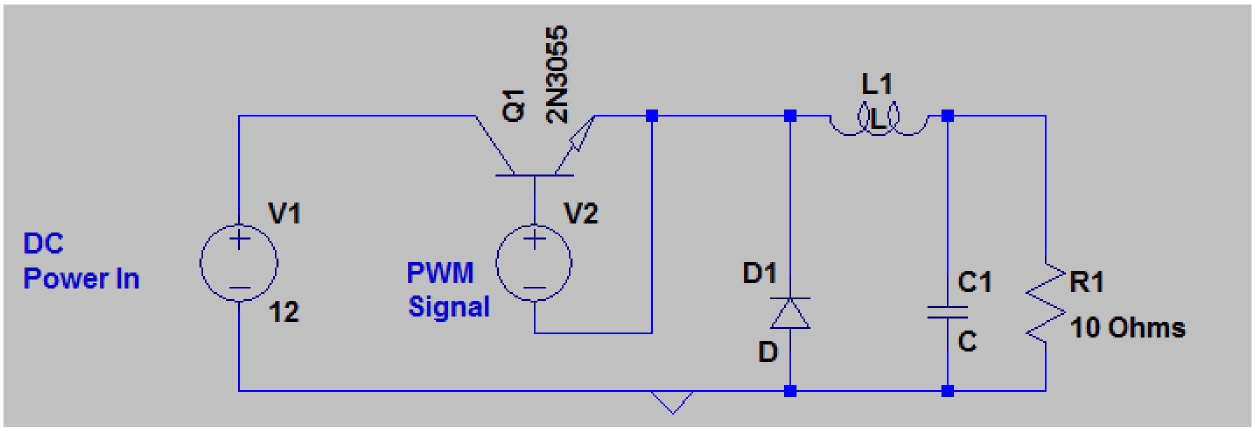
DC->DC Power Converters

Buck Converter Spice Simulation

Parameter Selection

We will be constructing a buck power converter using pulse width modulation.



To keep out of the audio range we will be using a 25kHz pulse frequency. We will down convert (the only conversion possible with a buck configuration) from 12VDC to 6VDC driving a 10Ω load. I have selected an NPN transistor capable of providing a significant current.

There are a number of circuit and signal parameters we still need to determine, (L, C, duty cycle ratio, and PWM on-voltage). Our first task is to determine the value of the duty cycle ratio, k:

$$V_{out} = kV_s \tag{1}$$

Is a good place to start. In truth it there is wiggle room as it is really the combination of L and k that sets the output voltage level.

Duty Cycle ratio for our circuit:______________________________.

We can choose the necessary inductance for the circuit, Eq. 5.64 from Rashid:

$$L = \frac{(1-k)R}{2f}$$

Inductance for our circuit:____________________________________.

Note that while we have used this to select a value of L based on k, we will keep this inductance fixed even later on when we change k, this is part of what makes this circuit nice, we'll be able to program its behavior after it is built.

Next we select capacitance, again Eq. 5.65 from Rashid gives:

$$C_{LB} = \frac{1-k}{16Lf^2}$$

This is an extreme lower limit, increasing the capacitance provides for vastly improved circuit behavior.

Capacitance lower bound:________________________________.

A more useful way to select the value of C is by selecting the acceptable amount of ripple in your output, from Rashid Eq. 5.53:

$$\frac{\Delta V_{out}}{V_s} = \frac{k(1-k)}{8LCf^2}$$

Or solving for this improved capacitance, $C=C_I$, and eliminating L

$$C_I = \frac{k}{4Rf\frac{\Delta V_{out}}{V_s}}$$

We will use $\Delta V_{out}/V_s=0.01$ to improve the behavior of our circuit.

Improved performance Capacitance:________________________________.

All that remains is the PWM voltage. This transistor is a BPJ transistor with a turn-on voltage of ~0.7V. We want to be well into the non-linear region of the transistor conductance. 2V, or anything higher, will do that for us.

Schematic Construction

Build your LTSpice circuit, run a transient analysis and show the first 10ms of the simulation using the parameters you determined above. Graph the PWM signal to be sure it make sense, then the voltage across and current the load.

Sketch (or snip) the current through the load for C_{LB} and C_I and briefly compare the two.

Data Collection

Controllability: We built this circuit to run a 10Ω load at 6V but we actually have a controllable power supply here, we simply need to change the width of our pulse.

Run simulations with $C=C_I$ to find the steady-state voltage V_{out} for k=0.1, 0.2, 0.4, 0.6, 0.8. Graph your data on a V_{out} vs k graph as well as a theoretical prediction from Eq. 1. Comment qualitatively on the consistency between the two.

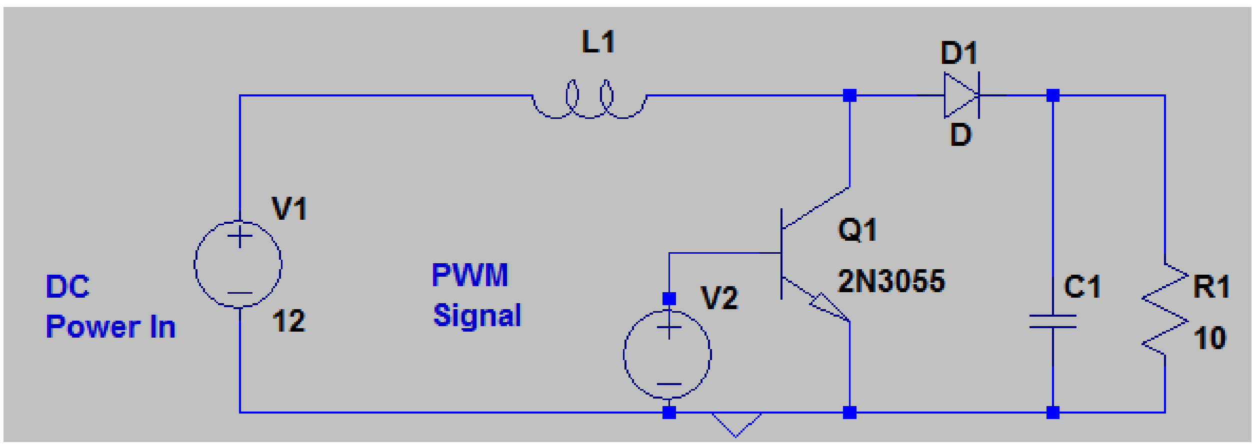
Efficiency: The point of using a buck circuit is to improve the efficiency of the down conversion compared to a voltage divider or LM317 regulator. We will now measure the efficiency as a function of V_{out}. Recall that $P_{DC}=\langle V\rangle \langle i\rangle$ and $P_{AC}=V_{RMS}\, i_{RMS}$ and that $\eta=P_{out}/P_{in}$. LTspice automatically calculates the averages and RMS values of each trace shown on the screen. To display the values for a trace, hold control and click (left click) the trace's name. You can change the time interval of the integration by dragging a box over the portion of the trace you are interested and then reshowing the integral values (control click).

Run simulations to find the efficiency on a DC basis (DC power out/DC power in) for k=0.1, 0.2, 0.4, 0.6, 0.8. Graph your η vs V_{out}. Comment qualitatively on how this compares to the result you found for the LM317 voltage regulator.

Boost Converter Spice Simulation

Parameter Selection and Schematic Construction

The circuit elements are the same, in the boost as the buck, only their placements change.



Rearrange the elements in your buck circuit into a boost circuit.

Data Collection

Controllability: We want this circuit to run a 10Ω load at 20V but we actually have a controllable power supply here, we simply need to change the width of our pulse.

Run simulations with C=50µF to find the steady-state calibration curve for voltage V_{out} for k=0.1, 0.2, 0.3, 0.4, 0.5, 0.6, 0.7. Graph your data on a V_{out} vs k and use it to infer the correct value of k for Vout=20V.

Efficiency: This is also an efficient circuit. We will now measure the efficiency as a function of V_{out}. Recall that $P_{DC}=\langle V\rangle \langle i\rangle$ and $P_{AC}=V_{RMS}\, i_{RMS}$ and that $\eta=P_{out}/P_{in}$. LTspice automatically calculates the averages and RMS values of each trace shown on the screen. To display the values for a trace, hold control and click (left click) the trace's name. You can change the time interval of the integration by dragging a box over the portion of the trace you are interested and then reshowing the integral values (control click).

Run simulations with C=50µF to find the efficiency on a DC basis (DC power out/DC power in) for k=0.1, 0.2, 0.3, 0.4, 0.5, 0.6, 0.7. Graph your η vs V_{out}.