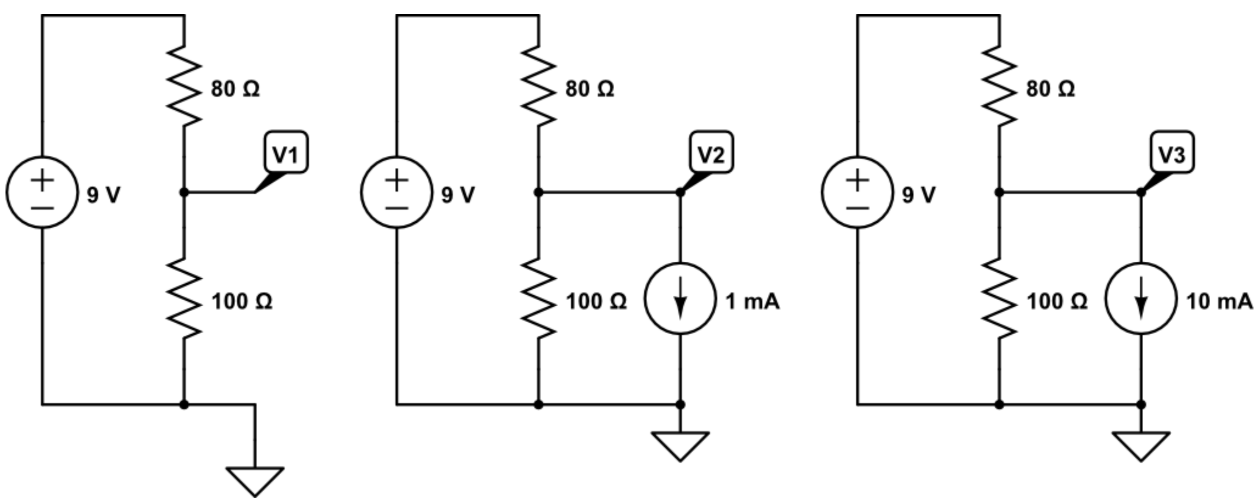
ECE 205 Lab – Regulators

Challenge: Voltage Regulation

The following is a recurring issue with electronics. You have a component or circuit which is designed to operate at "X" volts, but you only have a power source that supplies "Y" volts. We saw how to do this with a voltage divider in the lectures, but this approach is inefficient, as power is wasted in the resistor. In addition, there is another problem, as the voltage drops if the current load varies. Consider the following circuits:



The circuit attempts to generate a voltage of 5V from a 9V battery. However, as the load current (current source) increases, the voltage Vx will drop.

The LM317 (adjustable linear regulator) solves this problem by using feedback. It senses the output voltage and if the voltage is too low, turns on an internal transistor[1] to supply more current from the source. Alternatively if the voltage is too high, it turns off the transistor to lower the output voltage back to the target voltage.

Challenge: Using the LM317 linear regulator and other components, as well as 50 Ω resistor as the load, simulate an adjustable voltage regulator circuit that will provide a controllable output voltage, covering at least the range of 3V to 6V (demonstrate both), independent of the load. This circuit must also allow you to measure the current drawn by the regulator+load (the so-called "input current"). In LTSpice, the LT317A model (under /PowerProducts) may be used. This is the Linear Technology (original publisher of LTSpice) version of the LM317.

Prelab Deliverables:

- A schematic diagram of the circuit you are going to construct, with values for the resistors that you will use. (Hint: read the LM317 datasheet)
- Compute Vx in each of the above circuits (V1, V2, and V3) in the previous page.

---

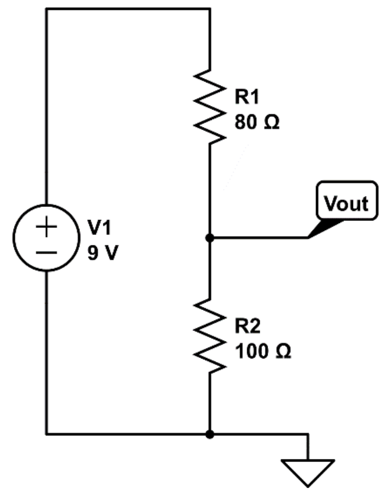
[1] We'll learn about this later in the class. For now, consider this an electronically controlled switch.

Required Deliverables:

- Provide two plots of the LM317 maintaining a constant voltage (use 3 and 6) for a varying load resistor. Your input voltage should be kept at 9V for these plots.

- Determine the efficiency of your LM317 circuit, at 5V (output), with respect to a 500 Ω load. For the circuit in this lab, the $Ef = \frac{output\ power\ (500\Omega\ load)}{input\ power\ (9\ V\ source)}$. Note that this requires measuring both the output current and the input current.

The voltage divider circuit to the right also gives you approximately 5V from a 9V input. Compare the efficiency of the voltage divider circuit to the LM317 circuit using a 500 Ω resistor. Note that the efficiency in this case is defined identically: $Ef = \frac{output\ power\ (in\ 500\Omega\ load)}{input\ power\ (from\ 9\ V\ source)}$. Note that you do not need to simulate this circuit in LTSpice, but you should compute the efficiency to use as a reference.



- Determine the so-called "drop-out" voltage for the LM317 in your circuit, with a 500 Ω load.
  - Drop-out voltage = the smallest voltage (difference), input-output, where the circuit continues to function properly. For example, if your battery is providing 8.9V, but your circuit will not regulate above 6.8V, then the drop-out is (8.9-6.8=) 2.1V.
  - Figure out how you would determine this parameter (under the test conditions) using LTSpice for the 500 Ω load condition. Corroborate this result with the relevant parameter in the datasheet.
- Determine the maximum current capability of the LM317 with LTSpice. What happens to the voltage output after this maximum current is exceeded? (You can simply point this out using an existing plot or make a new plot to explain).