Voltage Regulator Using LTSpice

Objectives

1. Design a simple voltage regulator circuit using a zener diode.
2. Contrast the performance of this circuit with an IC voltage regulator.
3. Check the noise performance of the circuit.

Reading Exercise

Please read about filter circuits at:
https://www.electronics-tutorials.ws/diode/diode_7.html

Circuit Overview

A simple voltage regulator circuit is shown below in Fig. 1.

The input voltage is fed to a resistor connected to a shunt zener diode. A zener diode is a special type of diode that is operated in the reverse direction when it is used in a voltage regulator. When operated in reverse bias, the diode maintains a constant voltage across the diode irrespective of the diode current (provided the current is above the minimum breakdown current and below the maximum current rating).

The series resistor connected to the zener diode is used to limit the diode current. The load is connected in parallel to the zener diode so that the voltage across the diode remains constant irrespective of the value of the load.

One problem with zener diode stabilizer circuits is that the diode can sometimes generate electrical noise on top of the DC supply as it tries to stabilize the voltage. Hence, a large value decoupling capacitor should be connected across the zener's output for additional smoothing.

We will not build the regulator circuit. Instead, we will use a previously fabricated integrated circuit (IC). Specifically, we choose the IC7805 circuit.

Lab Exercise

1. Connect a DC input of 15V, a resistor of 1kΩ, and a zener diode rated at 5V. Connect a load resistor R_L across the diode. Draw this circuit in

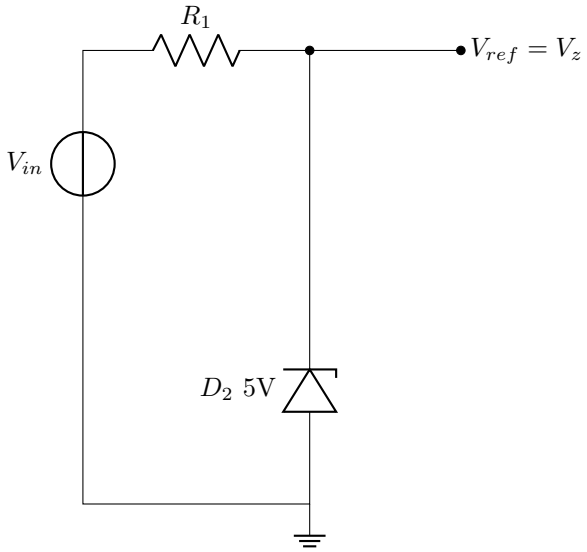

Figure 1: Voltage regulator circuit with flipped polarity zener diode.

LTSpice & run transient simulation.

2. Insert a DC input of 15V to the regulator circuit and measure the output voltage across R_L. Vary R_L from 100Ω to 5KΩ and note how the output voltage is changing (.op).

3. Now connect the voltage regulator circuit at the load of a rectifier circuit with a smoothing filter (C = 2200μF) and repeat the study of the previous step (transient).

4. Now instead of the zener circuit shown in Fig. 1, we will consider the Spice model of an IC7805 circuit and connect it to the input DC excitation and output load R_L.

5. Repeat the tests from steps 2 and 3.

6. Turn off source excitation and perform a noise analysis on the circuit at the output up to 10kHz.

7. Introduce a smoothing capacitor at the output of the IC7805 in parallel to the load. Change the capacitor from 5μF to 500μF and observe the noise performance of the resulting circuit.