

ELEC 1908 Assignment 4: SPICE - An Introduction to Circuit Simulation.

Submit by email to **David**
~~Colin Fernandes~~

Due: ~~March 19th at 11:35AM~~

March 22 at 1:30

Assignment Outline

For this assignment you will be using LTspice to model 2 circuits and answer a few questions about them, the usual! Report must be submitted using L^AT_EX. Use references where appropriate.

RC Filter:

Here you will use that capacitor from your previous assignment! For the capacitor seen in figure 1 use the capacitance of your designed capacitor from assignment 3. For the resistor you will use a value based off the last 2 (example: 101xxxxYZ) digits of your student number in the equation as follows:

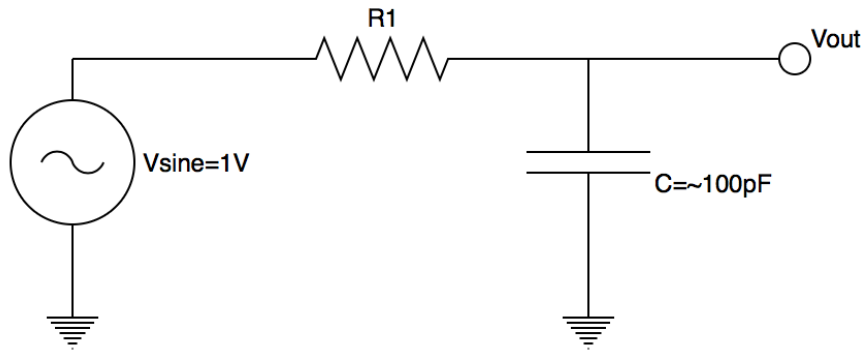


Figure 1: RC filter circuit diagram for question 1.

$$R = (Y + 1)(Z + 1) * 1 \times 10^5 \Omega$$

- With this circuit setup a frequency sweep from 1 mHz to 10 GHz. (This requires you to set the Vsine amplitude and frequency to "AC 1").
- graph the frequency vs voltage. Include a well formatted graph in your write-up. Include units, axis titles, properly formatted axis numbers.
- What type of filter is this circuit? Give a practical application of the circuit.
- Find and label the corner frequency for this circuit on the plot. Calculate the theoretical corner frequency for this circuit.

Active Circuits

Construct the non-inverting operational amplifier circuit seen in figure 2. The OP amp you will use will be the LT1351, this is available in the LTspice library under "opamp".

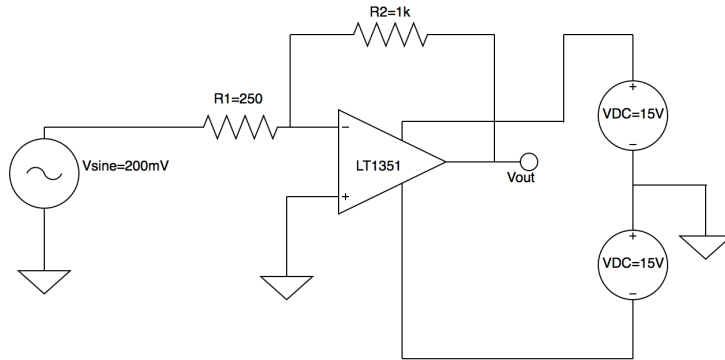


Figure 2: RC filter circuit diagram for question 1.

PART 1

- Set up a transient simulation with $f = 10 \text{ kHz}$ (only need a millisecond at most)
- plot voltage vs time for input (across the AC terminals) and the output (The point of the triangle). Include well formatted graphs with all relevant labels.
- Calculate the gain of the circuit via theory. Compare to simulation and discuss in a short paragraph in your report.

PART 2: Play with it until it breaks! Since we're using a model of a real OP-amp it isn't perfect and has a finite operational range.

- Set $R1 = 50 \Omega$ and $R2 = 50 \text{ k}\Omega$, Look at the transient analysis, how much gain are you expecting? what are you seeing?
- Reset $R1 = 250 \Omega$ and $R2 = 1 \text{ k}\Omega$, conduct a frequency sweep from 1 mHz to 10 THz. Look at the gain of the system again, what do you notice?

20 % Bonus Circuit Chao's Chaos – next lab you build it!

Run a transient, export the data, read into matlab and plot V1 and V2 versus time

Then plot V1 versus V2

