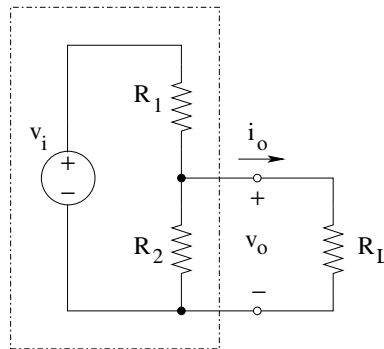


Lab 1: Circuit Simulations

Circuit Simulations

The PSpice_primer.pdf file is created to be used as a reference for PSpice simulations in ECE 65 labs. Please review the file to find more information on the simulation settings, etc. If you use LTspice for simulations, you can refer to the **Helpful Tips for LTspice** section of this document.

Problem 1: Voltage Divider (Bias Point, DC Sweep, Parametric sweep, plotting)



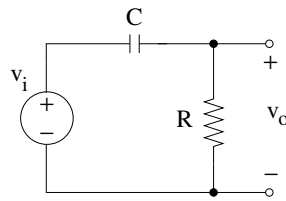
In many cases, we need to bias the circuit components with a voltage that is different than the power supply voltage. Other times we need to feed back a signal proportional to the output signal of one part of the circuit to another part. The voltage divider circuit above (circuit in the "box") is the simplest circuit that does these functions and, therefore, is used extensively in electronic circuits.

1. Calculate the Thevenin equivalent circuit of the voltage divider (the "box"). Use the Thevenin equivalent circuit to find v_o and i_o in terms of v_T , R_T and R_L . Prove that for $R_L \gg R_T$, $v_o = v_T$.
2. Use a circuit simulator to simulate the above circuit with $R_1 = 1 \text{ k}\Omega$, $R_2 = 1 \text{ k}\Omega$, $R_L = 50 \text{ k}\Omega$, and $v_i = 5 \text{ V}$. Use "Bias Point Details" option to find the value of v_o and i_o (attach the circuit with bias-point details).

Note: If you are using PSpice, please refer to the **PSpice_primer.pdf** file posted on the class website. If you are using LTspice, please refer to the **Helpful Tips for LTspice** section on this document.

3. Simulate the above circuit with $R_1 = 1 \text{ k}\Omega$, $R_2 = 1 \text{ k}\Omega$, and $R_L = 50 \text{ k}\Omega$. Use DC SWEEP to generate a plot of v_o as a function of v_i for v_i ranging from 0 to +10 V. Does it match the expression from part 1? Hint: Note that $R_L \gg R_T$.
4. Simulate the above circuit with $R_1 = 1 \text{ k}\Omega$ and $R_2 = 1 \text{ k}\Omega$, and $v_i = 5 \text{ V}$. Use parametric SWEEP to generate a plot of v_o/v_i as a function of R_L for R_L ranging from 0 to 50 k Ω (choose the increment in R_L such that you have a meaningful plot, *i.e.*, the curve looks nice and smooth). Does it match your expectations (consider cases of $R_L \rightarrow 0$, $R_L \rightarrow \infty$, and $R_L = R_T$)?
5. Without changing the simulation settings in the previous part, plot i_o (current in R_L) versus v_o . Does it match your expectations? (consider cases of $R_L \rightarrow 0$, $R_L \rightarrow \infty$, and $R_L = R_T$)?

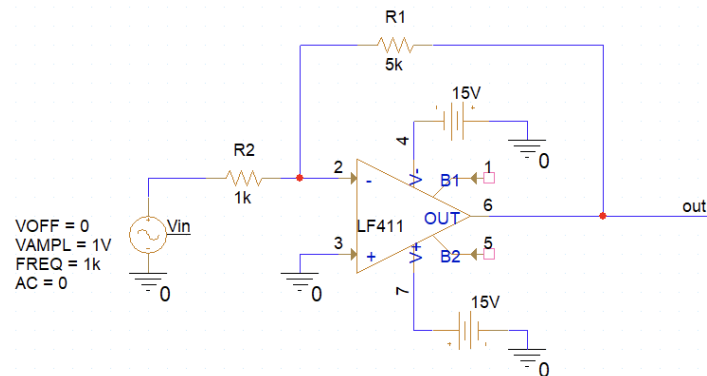
Problem 2: RC circuit (VPULSE function, time-domain analysis, plotting)



Simulate the circuit above with $R = 1 \text{ k}\Omega$ and v_i being a square wave function with a frequency of 500 Hz, a peak to peak amplitude of 5 V, and zero DC bias (*i.e.*, v_i switches between -2.5 and 2.5 V).

1. Plot v_o and v_i as a function of time for two periods for $C = 100 \text{ nF}$, $1 \text{ }\mu\text{F}$, and $10 \text{ }\mu\text{F}$.
 - Make sure both traces are on the same plot.
 - For each case, simulate the circuit for 100 ms and plot the waveforms.
 - For each case, simulate the circuit from 90 ms to 100 ms and plot the waveforms.
2. Repeat part 1 for v_i having a peak to peak amplitude of 10 V, and zero DC bias.
3. Compute the time constant, $\tau = RC$, of the three circuits (*i.e.*, 3 different values of capacitor) and compare them to the half-period of the input voltage. What do you conclude from the above two simulations?

Problem 3: Op-amp as a voltage amplifier (Sine function, time-domain analysis, plotting)



Set the input voltage as a sine wave with peak amplitude 1 V, zero DC offset, and frequency 1 kHz as indicated in the circuit diagram. Simulate the circuit, and

1. Plot v_o and v_{in} as a function of time for two periods (both traces on the same plot.) Is there a linear relationship between v_o and v_i ?
2. Plot the output current of the op-amp as a function of time for two periods.
3. Plot the node voltage at the input inverting terminal as a function of time for two periods. How does the amplitude of this node voltage compare to the amplitude of v_{in} and the node voltage at the input non-inverting terminal?

Helpful Tips for LTspice:

- For **Bias Point** details, use the **.op** SPICE directive
- To open a user friendly settings menu for simulation settings:
 1. Right click anywhere on the canvas
 2. "Draft"
 3. "SPICE directive"
 4. Right click on the text box
 5. "Help Me Edit"

6. "Analysis Cmd"
 7. Now choose the simulation you need
- To accomplish a **Parametric Sweep**:
 1. Repeat steps 1-5 above
 2. Choose ".step command" in step 6 above
 3. Set the parameter **name** (any name you'd like to choose)
 4. Rename the needed component to {**name**} (The name you gave it written in curly braces)
 5. Add **.op** directive to your canvas in addition to the **.step** directive.
 - To add LF411 op-amp to your schematic on LTspice:
 1. Download the LF411C PSpice Model (LF411C.text) file from the Canvas website. The file includes the spice model for the LF411C op-amp. Save the file in your desired folder.
 2. Create a spice directive to include the LF411 library file. For example, if you use a Windows computer and your username is *Max* and you saved the file in the *ECE65labs* folder on your desktop, you should use the below spice directive.

```
.lib c:\Users\Max\Desktop\ECE65labs\LF411C.txt
```

If you are a MAC user with username *Max* and saved the file on your desktop, you should use the below spice directive.

```
.lib /Users/Max/Desktop/LF411C.txt
```
 3. Place an "opamp2" component on your schematic. Once placed, right click on the component, verify it contains the prefix "X", and change its value to "LF411C". Use the following picture as a guide.

