

Lab Number 1
EEE 108L – Electronics I - Laboratory
Introduction to SPICE Analysis (One week)

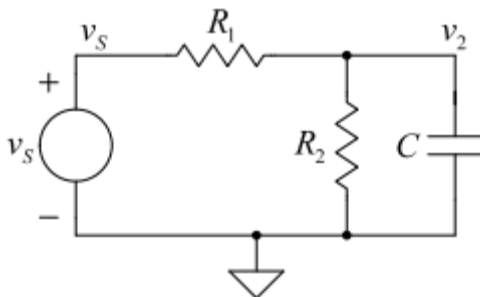
Background

In this experiment, a simple circuit will be investigated using SPICE simulations. The experiment has been designed so that the student will encounter situations that are common in the simulation of electronic circuits. The three major types of analyses will be used: DC, AC, and transient. The text of this experiment gives some guidance regarding the use of PSPICE.

Preliminary Calculations:

For the circuit of Figure 1:

1. Find the expression of transfer function.
2. At which frequency the capacitor can be considered an open circuit?
3. Calculate the magnitude of the transfer function at this frequency.
4. At which frequency the capacitor can be considered a short circuit?
5. Calculate the magnitude of the transfer function at this frequency.
6. Calculate v_2 and gain for $v_s = -1.5V, -1.0V, -0.5V, 0.0V, 0.5V, 1.0V, 1.5V$.
7. Find the RC time constant (τ).
8. Calculate the upper -3db, cutoff, corner or pole frequency.
9. Calculate the magnitude and phase of v_2 at frequencies 1kHz, 5kHz, 10kHz and -3dB, if $v_s = 1 \angle 0^\circ V$.
10. Calculate the 10% to 90% risetime of v_2 in response to a step input at $v_s = 1V$.
11. Find the bandwidth (B). $BW = \frac{0.35}{RT}$



$$R_1 = 4.7k\Omega, R_2 = 2.2k\Omega, \text{ and } C = 0.02\mu F$$

Figure 1. Drawn Schematic

SPICE Simulations:

DC Sweep: This type of analysis is used to find the DC transfer characteristic of a circuit. The simulation will sweep the DC value of a particular source specified by its part reference. Any of the PSPICE voltage (or current) sources may be swept in a DC sweep.

1. Enter the circuit of Figure 1 into PSPICE. Use source part “ v_{sin} ” for v_s . Use $V_{off} = 2V$, $V_{ampl} = 1V$, $F_{req} = 1k$, $AC = 1$
2. Create a DC sweep PSpice simulation profile. Specify a linear DC sweep of v_s from -2 to $+2$ volts, with an increment of $10mV$.
3. Run the DC Sweep simulation. Plot the simulated curves for v_s and v_2 .
4. Use the cursors to measure the voltage gain for the circuit, for $v_s = -1.5V, -1.0V, -0.5V, 0.0V, 0.5V, 1.0V, 1.5V$ and compare them to the values in preliminary calculation.
5. Display the simulation results for $\frac{v_2}{v_s}$ by entering this expression in the PSpice calculator. Notice the discontinuity in this curve when $v_s = 0$.

AC Sweep

AC analysis is used to determine the frequency response of a circuit. The analysis calculates the terminal characteristics of each element at each frequency of the sweep, and finds a solution for all of the voltages and currents in the circuit. For this type of analysis, the signal source must be assigned an AC magnitude and phase.

1. Assign an AC magnitude of 1 volt to the input voltage source v_s .
2. Create a new PSpice simulation profile and choose an AC Sweep simulation.
3. Choose a logarithmic sweep by selecting Decade. Set up the AC sweep to start at $100Hz$, end at $100kHz$, and use 30 points per decade.
4. Run the AC Sweep simulation. Plot the simulated curves for magnitude and phase of v_2 .
5. Use cursors to find the magnitude and phase of v_2 at frequencies $1k, 5k$ and $10k$ and cutoff frequencies. compare them with the values in preliminary calculations.
8. Add a blank plot to the window by going to the Plot menu.
9. Use the PSpice calculator to add the trace “ $P(V(v_2))$ ” in this plot. This is the phase of v_2 .
10. From this phase plot, determine the phase of the v_2 at the upper $-3dB$ frequency.

Transient Simulation

Transient simulations are used to obtain the circuit voltages and currents as functions of time. This type of simulation is different from those previously discussed. Here the sweep must start at time zero ($t = 0$), and the results at each time step depend on the results at the previous time step. In order to approximate a continuous-time system, the time steps must be small enough so that the circuit voltages and currents do not change too much between any two-time steps.

1. Create a new PSpice simulation profile and choose a Transient simulation.
2. Edit the voltage source v_s to give it the following values needed for a Transient analysis:
 - i. DC offset voltage = 0V
 - ii. Amplitude = 1V
 - iii. Frequency = the -3db frequency found from your previous results.
3. Set the simulation end time to 5 times the period of the sine wave.
4. Run the simulation, and plot the two traces " v_s " and " v_2 ".
5. Measure the peak-to-peak magnitudes of v_2 and v_s , and use these values to determine the voltage gain, $\frac{v_2}{v_s}$. (Note that in transient analysis it is always best to use peak-to-peak values to find voltage gain, since this avoids errors when the DC offset voltage isn't 0V.)
6. Compare this gain to the values you got in preliminary analysis.

10% -90% rise time:

7. Replace the v_{\sin} voltage source with a v_{pulse} source.
8. Assign this source the parameters shown in Table 1.
9. Run a transient simulation with an end time of 2 ms and a time step of 0.1 μs .
10. Plot both v_s and v_2 .

Property	Spice	Value
initial voltage	V1	0
pulsed voltage	V2	1
time delay	TD	0.25m
risetime	TR	1u
falltime	TF	1u
pulse width	PW	0.25m
period	PER	0.5m

Table 1.

11. Measure the "time" to which v_2 settles after v_s transitions from 0V to 1V, but before it goes to low again. Express this time in term of time constant.
12. From the simulation results, determine the 10% to 90% risetime and compare this value with that calculated in preliminary part.