

ECE231: Introduction to Electronics

Lab #1

Introduction to Circuit Simulation using SPICE

Version: 1.4

INTRODUCTION

The objective of this lab is to familiarize students with SPICE (Simulation Program with Integrated Circuit Emphasis) based Computer Aided Design (CAD) tools for the simulation and analysis of circuits. The core of SPICE was originally developed at the University of California Berkeley and has since formed the basis for numerous circuit analysis tools, including Multisim from National Instruments.

SPICE based simulation tools use a netlist (a list of “nets” or nodes) to represent circuits and simulator settings. Netlists are used to precisely define the electrical connections between circuit elements using a simple text file. Modern circuit simulators have evolved to allow graphical entry of the design schematic (also known as schematic entry), such that a netlist is automatically generated and only visible to the user upon request. Using the netlist, SPICE produces a set of differential equations based upon KCL/KVL, which it solves using discrete time steps to determine the node voltages and branch currents in the circuit at each point in time.

SPICE requires models to represent all circuit elements (including semiconductor devices, amplifiers, etc.) within the netlist. Device models vary significantly in complexity and accuracy based on the needs of the circuit designer.

SPICE based tools allow the designer to simulate circuit behavior, refine design choices and investigate the role of component variation on design performance.

Goals of this experiment

1. Create models of circuits using NI Multisim.
2. Understand representations of circuits suitable for simulation tools.
3. Use SPICE to simulate transient and AC characteristics of circuits.
4. Understand and analyze data produced by SPICE simulations.

Lab Preparations

1. Install Multisim on your personal laptop:
www.ni.com/gate/gb/GB_ACADEMICEVALMULTISIM/US
2. Review the NI Multisim tutorial available on the national instrument's website <http://www.ni.com/white-paper/10710/en/>. In addition, there are many Multisim tutorial videos available on YouTube.
3. Prepare a bounded lab book and use it for the whole semester.
4. Preparation exercises is an **INDIVIDUAL** work.
5. Read detailed marking scheme in "Lab Policies and Marking Scheme.pdf".

Preparation Exercises (3 pts)

- PE1. Compute the time constant, τ , for the RC circuit shown in Fig. 1.
- PE2. Sketch the bode plot for the STC circuit shown in Fig. 1. The transfer function is defined by: $T(j\omega) = V_{out}/V_{in}$. Include both magnitude and phase and clearly indicate the -3 dB point.
- PE3. Consider the circuit shown in Fig. 2. Sketch the magnitude of the transfer function, T , and clearly indicate the -3 dB point. Is this a high-pass, low-pass, band-pass, or notch filter?

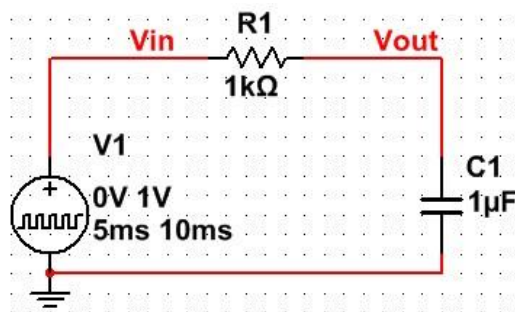


Fig. 1: RC Circuit.

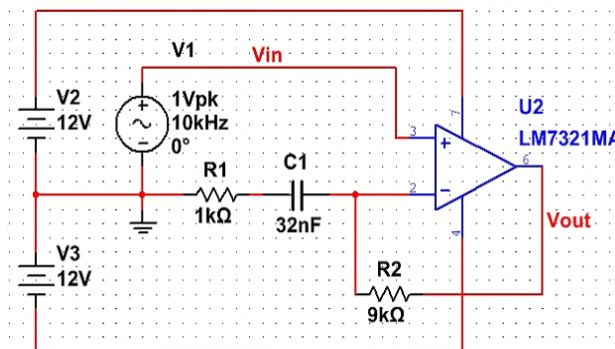


Fig. 2: Op-amp Circuit.

Lab Experiments

Part 1 RC Circuit

1.1 Transient Simulations

- E1. Start NI Multisim with a blank schematic.
- E2. Using components from the CAPACITOR, RESISTOR, POWER_SOURCES and SIGNAL_VOLTAGE_SOURCES families draw the circuit shown in Figure 1. These component families can be found under “Sources” and “Basic” groups. *(Note: There is a search function that allows you to type in the component name. Please avoid it for this lab, as it does not return the right resistor model, we need to perform parameter sweep in Part 2.2. For the RESISTOR and CAPACITOR models, Go to BASIC → RESISTOR or CAPACITOR, and pick the appropriate value.)*
- E3. Draw wires to create circuit connections. Ensure that a ground, located in the POWER_SOURCES family is attached to the circuit. A ground is required in all SPICE simulations to provide the reference for all measured values.
- E4. Use a PULSE_VOLTAGE source from SIGNAL_VOLTAGE_SOURCES database to excite the RC circuit.
- E5. Right click and select properties for each net in the circuit and rename as them shown in Fig. 1 by entering alternate net names in the ‘Preferred net name’ field.
- E6. Save your work before proceeding to simulations.
- E7. A SPICE transient simulation is used to plots circuit quantities as a function of time. The duration of the simulation, step size and relevant variables to be measured must be selected prior to the simulation.
- E8. Open the transient analysis dialog box from Simulate → Analysis and Simulation → Transient.
- E9. Choose a stop time to display 0.1 s.
- E10. In the output tab, select the voltages at nodes V_{in} and V_{out} .
- E11. Run the simulation.
- E12. Using the output graph, measure the circuit time constant by using cursers, τ . Save a clear screenshot with cursers and measurements.
- E13. How does it compare to the theoretical value? What are some sources of error? Write your explanations on the lab book.

1.2 Netlists

- E1. In NI Multisim, export the netlist to .cir file using the Export SPICE netlist functions from the Transfer menu.
- E1. Open the resulting .cir file in notepad or another text editor.
- E2. How are passive components (resistors/capacitors) represented? Explain the relationship between net names in the schematic and numbers in the netlist. What units for parameter values are used in SPICE netlists? Write explanations on your lab book.

1.3 AC Simulations and Bode Plots

- E1. Frequency domain analysis of circuits described using netlists or schematic capture can be performed in SPICE using an “AC analysis”. An AC analysis calculates node voltages and branch currents in the frequency domain, based on the concept of sinusoidal steady-state that was discussed in ECE212. This is contrast to the time-domain outputs from transient simulations. The frequency domain voltages and current are complex numbers, which contain both magnitude and phase information.
- E2. Open the AC analysis dialog from Simulate → Analyses and Simulation → AC Sweep.
- E3. Select an appropriate frequency range to include the expected -3 dB point of the filter and take note of the key features of filter magnitude and phase responses.
- E4. Select “Vertical Scale” to “Decibels”.
- E5. In the output tab add a custom expression representing V_{out}/V_{in} . (i.e. $V(V_{out})/V(V_{in})$)
Note: This is equivalent to plotting V_{out} , since V_{in} from Figure 2 is represented by the phasor $1\angle 0$.
- E6. Run the simulation. Save a clear screenshot with -3 dB measurements.
- E7. Repeat the simulation with ‘Number of points per decade’ in the range 1-10. Save a clear screenshot with -3 dB measurements.
- E8. What is the -3 dB frequency computed from the graph? How does it differ from the analytically computed point? What is the effect of the “Number of points per decade” parameter? Write explanations on your lab book.

Part 2 Op-amp Circuit

2.1 Circuit Characterization

- E1. Op-amp based circuits are frequently used to filter and amplify analog signals. In this exercise we will use the SPICE techniques from the previous examples to analyze a proposed op-amp filter circuit.

Create a new blank schematic in NI Multisim and construct the circuit shown in Figure 2. (Note: In BASIC \rightarrow CAPACITOR, and there is no 32nF. Pick one and you can change the values under properties once it is placed in the circuit.)

(Another Note: Use AC_VOLTAGE source from SIGNAL_VOLTAGE_SOURCES)

- E2. Change the net names as described in the examples above.
- E3. Conduct transient simulations showing approximately 10 cycles of input signal for input amplitudes 1 Vpk and 2 Vpk. Is there any non-ideality illustrated in this model? Describe how changing the power supply voltages affects the output. Save a clear screenshot showing V_{in} and V_{out} with 10 cycles.
- E4. Conduct an AC simulation and produce a bode plot for the op-amp circuit in figure 2. Choose frequency ranges based upon the calculated value in the lab preparation.
- E5. Determine the -3 dB point from the out graph. How well do the simulated -3 dB point values relate to the calculated values? Save a clear screenshot with -3 dB measurements.
- E6. Repeat the transient simulation with an input frequency of 100 Hz and again display only 10 cycles. Explain how the changes in circuit behaviour relate to the circuit transfer function as determined in the preparation and above in the AC analysis. Save a clear screenshot showing V_{in} and V_{out} with 10 cycles.

2.2 Component and Parameter Variation

- E1. The component values specified in the model and resulting analysis do not yet account for natural variation in real-world electronic parts. Electronic components are generally sold with a specified guaranteed tolerance, which has a major impact on cost as shown in Table 1.

Table 1. Tolerance of Typical Resistors with their respective costs (Source: Digikey.ca)

Nominal resistance	Tolerance	Min Resistance	Max Resistance	Unit Cost
1k Ω	$\pm 0.01\%$	999.9 Ω	1000.1 Ω	\$8.62
1k Ω	$\pm 0.1\%$	999 Ω	1001 Ω	\$0.05585
1k Ω	$\pm 1.0\%$	990 Ω	1010 Ω	\$0.00296

- E2. Parameter sweep simulations can be used to understand the subtle effects of component variation on circuit functionality. This information can further be used to design circuits that are robust to parameter variations. We will use the SPICE parameter sweep tool to analyze the effect of variation in passive component values on circuit functionality.
- E3. Open the Parameter Sweep dialog in the analysis menu.
- E4. Select “Device parameter” as the sweep parameter.

- E5. Choose “Device Type” as “Resistor”
- E6. Select Resistor R1 (Note: If you don’t see R1, and see something like r1:xl1, you are probably using the wrong resistor model. See note in E2)
- E7. Sweep the resistance from 0.5 k Ω to 4 k Ω with a total of 8 points.
- E8. Choose “Analysis to Sweep” to be “AC Sweep”
- E9. Click “Edit Analysis” and for Vertical Scale select “Linear” (This will display the y-axis with the magnitude on a linear scale, so it is easier to determine $\pm 20\%$ without converting the magnitude from magnitude expressed in dB)
- E10. Select an appropriate output variable in the “output” tab
- E11. Run the simulation.
- E12. What is the effect of variation of R1 on the circuit behavior?
- E13. Assume that the design is only acceptable if $|T(j\omega)|$ at 10 kHz lies within $\pm 20\%$ of the ideal value (Results obtained by using all components with the original value shown in Fig. 2). What is an acceptable range for R1 in this circuit? You will need to edit the sweep conditions in E7 and run a few more simulations to get a more accurate range for R1. Save a clear screenshot with all measurements that help you get the conclusions. Write calculations on your lab book.
- E14. Repeat the above analysis for C1 to meet the **SAME** $|T(j\omega)|$ requirements ($\pm 20\%$ tolerance). Assume that C1 is available with tolerances of $\pm 30\%$ (This determines the sweeping range for C1. You may need to sweep with more points). Save a clear screenshot with all measurements that help you get the conclusions. Write calculations on your lab book.