

Introduction and Top-SPICE Demo

Name: _____ Lab Day: _____

Objective

The goals of this lab are to meet your T.A. and other students in your lab section, and perform several simulations of a circuit in Top-SPICE.

Pre-lab Assignment

A. Before lab, please fill out page 2 of this lab and hand it in to the lab TA.

Exercise I. (10 points)

The main objective of this exercise is to communicate orally with your peers. In addition, the Career Fair is just around the corner. We would like you to practice talking about yourself in front of others.

Name: _____

Preferred E-mail: _____

Hometown: _____

Expected Degree from NMSU: _____

Expected Graduation Date: _____

Why did you choose to major in EE? _____

Most Recent Job Title¹: _____

Company & Location: _____

Responsibilities: _____

Next Most Recent Job Title¹: _____

Company & Location: _____

Responsibilities: _____

What are your personal areas of strength? Why might someone choose to hire you?

List one unique talent or accomplishment, something that will help us remember you by:

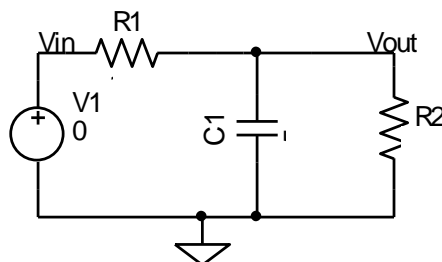
¹ Work experience does not necessarily have to be in engineering. There is a lot to learn about the work environment from non-engineering jobs, e.g., responsibility, management, determination, working with people...

From the Help files of TopSPICE “Simulating Your Own Circuit.” *Italics are added.*

- *In Windows, from the taskbar, Start/Programs TopSPICE/Launch TopSPICE.*
- Click on the New Circuit button. Enter your project file name.
- From the Schematic Editor menu select Edit/Title block to add a title for your circuit and other descriptive information. *Not necessary.*
- Place your circuit parts as described in the Schematic Capture Front-end section. *Under Draw...Part, choose the appropriate part to put in your schematic. Leave a little space between each part. Make sure to include a ground connection on your schematic. If you are not familiar with schematic capture programs, it is highly recommended that you hand draw your circuit on paper first. This will minimize the need to edit and move components around the schematic, which is the most time consuming part of drawing a schematic on screen. Moving parts is cumbersome using the Select commands under the Edit menu.*
- Wire all the parts on the schematic. *When using Draw...Wire, be careful to wire the tips or ends of each part. If you wire across the middle, or even near the ends, you'll miss the part. All part pins must be either connected to a wire or another part pin (the ground, power rails and I/O pin symbols are also considered parts), or given a node label. You can wire the parts as you place them, or whenever is most convenient or efficient. Unfortunately, wires are dark blue and parts are black and it's hard to tell them apart.*
- To indicate an electrical connection between two crossing wires, or between a wire or a part pin contacting another wire at right angles, place a junction. *Under Draw...Junction, put in junctions. The rule of thumb is to place a junction whenever you are joining more than two parts or wires.*
- All circuit nodes are automatically assigned node numbers. *It's best to turn on View...Node Numbers. However, these numbers are not fixed. The node number may change as you edit the schematic. If you need to reference a circuit node elsewhere on the schematic or in a simulation command or plot, you should label the node. Under Draw...Label node, write names for the input and output nodes by putting the node label near the appropriate wire.*
- You can also add any text on the drawing for documentation or descriptive purposes. *When you use Draw...Text, it is considered a command by the simulator, unless you begin the text with a *.*
- After the circuit drawing is complete, select Analysis/Setup to specify the desired analyses to be simulated and the results to be plotted.
- Select Analysis/Run Simulation to simulate your circuit and plot the results.
- If the simulation does not proceed because of an error, you can browse the output file by selecting Analysis/Browse Output File for error messages. Most error messages are *not* self-explanatory. After making the necessary changes run the simulation again.
- *If you want help on certain SPICE commands, go to Help...Simulator Reference and then choose the appropriate sub-menu.*
- *It is good to do the simulations on drive C:, since it reads and write much faster than drive A:. However, make sure to save your work on a floppy drive before you leave or reboot the computer.*

Exercise II. Circuit Analysis and Simulation (50 points)

The objective of this exercise is to demonstrate how to DC, AC, and transient circuit simulation using Top-SPICE **must be done in future homework assignments**. Your report will include these pages plus printouts you produce from Top-SPICE with added handwritten calculations.

**Figure 1 – Lowpass Filter**

- A. (2 points) Input the circuit of Fig. 1 in Top-SPICE. Call the circuit “lowpass”. Let $R_1 = 12\text{ k}\Omega$, $R_2 = 15\text{ k}\Omega$, and $C_1 = 0.015\text{ }\mu\text{F}$. Be sure to label the input and output nodes. **Print your schematic.**
- B. (2 points) Calculate the expected DC voltage gain and the corner frequency of the circuit using the formulas:

$$A_{V0} = \frac{R_2}{R_1 + R_2} =$$

$$f_o = \frac{1}{2\pi C_1 (R_1 \parallel R_2)} =$$

- C. (8 points) Now perform a DC analysis. To do so:
- Rename the voltage source VS, setting the DC voltage to 0V.
 - Under simulation, set up the DC (transfer) analysis to sweep the source VS from 0V to 2V in increments of 5 mV.
 - Now run the simulation.
 - **Plot the output voltage.** Since V_{in} is along the x -axis when you do a DC simulation, do not plot V_{in} , because it would just be a line with a slope of 1 V/V.
 - **Measure the DC voltage gain (slope) using cursors.** Print out the waveform with the measurement.

- **Compare to the calculated value in B using %error analysis on the graph.** When you calculate %error, do not take the absolute value in this class. The formula is below:

$$\%error = 100\% \times (\text{actual} - \text{expected}) / \text{expected}$$

D. (12 points) Now perform an AC analysis. To do so:

- For the voltage source VS, set AC specifications to 1V amplitude and 0° phase (defaults).
- Under simulation, set up an AC sweep with 100 points per decade (so that the curves are smooth). Let the start frequency be 100 times smaller than the corner frequency f_o and the end frequency 100 times larger than the corner frequency.
- **Plot the output voltage magnitude (in dB) and phase.**
- In this exercise, **measure the low frequency (DC) gain and the corner frequency (3-dB down) using cursors.** To measure the 3-dB frequency, first you measure the maximum gain. Then move the cursors until the gain has dropped by 3dB, which corresponds to $1/\sqrt{2}$ V/V. The frequency where the gain has dropped by 3dB is the corner frequency. Print the output waveforms with the measurement.
- **Compare the gain to the calculated gain using %error analysis on the graph.** When you compare gains, you must first convert the value in dB to V/V. Then do %error analysis. Never do %error analysis on dB values.
- **Compare the corner frequency to the calculated corner frequency using %error analysis on the graph.**

E. (2 points) Why did we set the voltage source VS to have an AC input signal is 1V amplitude and 0° phase? Because then the plotting the output voltage is the same as plotting the transfer function. Prove it below!

$$T(jf) =$$

F. (12 points) Now, perform a transient sinusoidal analysis. In general, you will be given the amplitude and frequency of the sinusoidal input signal whenever you do a transient sinusoidal analysis. **For this exercise, use an amplitude of 1.25V and a frequency equal to the corner frequency, f_o , you calculated earlier.** To perform a transient analysis:

- For the voltage source VS, set the sinusoidal specifications as follows:
 - offset = 0V (default, unless a value is given in the problem)
 - amplitude is given in the problem
 - frequency is given in the problem
 - delay = 1 period = 1 / frequency.
- Under simulation, set up the transient analysis to have:
 - start time of 0 seconds
 - end time of 4 periods
 - time increment (step time) of 1/100th of a period
 - step ceiling equal to step time

By setting the step time and step ceiling equal to 1/100th of a period, you are specifying 100 points in each cycle of the sine wave, which is enough to make it smooth, but not too many that the simulation takes long.

- **Plot the input and the output voltages on different axes.** Always plot the input voltage when doing a transient analysis, since the axis is now time.
- **Measure the peak-to-peak voltages of the input and output waveforms using cursors.** Write the values on the graph by hand if you don't have enough cursors. Print the waveforms with measurements.
- **Calculate the gain by dividing the peak-to-peak output over peak-to-peak input on the graph.**
- **Compare the measured gain to the calculated gain using % error analysis on the graph.** Both gains must be in V/V (not dB). See note below before completing this step.

Note: In this exercise, we chose a sinusoid with a frequency equal to f_0 . In this case, the expected gain is given by the formula below (calculate the actual value):

$$A_v(f_0) = \frac{A_{v_o}}{\sqrt{2}} =$$

- G. (12 points) Now, perform a transient pulse (square wave) analysis. In general, you will be given the frequency and voltage levels of the pulse whenever you do a transient square wave analysis. **For this exercise, let the pulse go from 0V to 5V and let the frequency be equal**

to **1/2 of the corner frequency, f_o** , you calculated earlier. Thus, the period (in this problem) will be $T = 2 / f_o$. To perform a square wave transient analysis:

- For the voltage source VS, set the pulse specifications as follows:
 - initial value is given in the problem
 - final value is given in the problem
 - delay time = T
 - rise time = 1ns (approximate rise time of function generator)
 - fall time = 1ns (approximate rise time of function generator)
 - pulse width = $T/2$
 - period = T
- Under simulation, set up the transient analysis to have:
 - start time of 0 seconds
 - end time of 4 periods
 - time increment (step time) of $1/100^{\text{th}}$ of a period
 - step ceiling equal to step time
- **Plot the input and the output voltages on different axes.**
- **Measure the delay time of the input waveform going up** (crossing through 50% of its final value, in this case 2.5V) **to the output waveform going up** (crossing through 50% of its final value) **using cursors**. Write the values on the graph by hand if you don't have enough cursors. Print the waveforms with measurements.
- **Compare the measured delay time to the calculated gain using % error analysis on the graph.** See note below before completing this step.

Note: The delay of an RC circuit is related to the time-constant, $\tau = RC$. One time-constant is the time needed to move to 63.2% of its final value. In this exercise, we want the delay for getting to 50% of its final value, which is given by the formula below (calculate the actual value):

$$t_d = 0.7RC$$

Exercise III. Circuit Design and Simulation (40 points)

The objective of this exercise is design a high-pass filter and to review DC, AC, and transient circuit simulation using Top-SPICE.

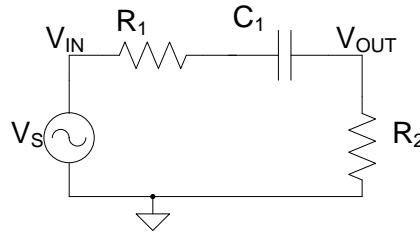


Figure 2 – High Pass Filter

- A. (8 points) Design the circuit (determine **appropriate** values of C_1 , R_1 , and R_2 , **showing the work below**) of figure 2 to satisfy the following constraints:

$$1k\Omega \leq R \leq 1M\Omega, 100pF \leq C \leq 1\mu F$$

$$A_{Vh} = \frac{R_2}{R_1 + R_2} = 0.75 \text{ V/V}$$

$$f_o = \frac{1}{2\pi C_1 (R_1 + R_2)} = 300 \text{ Hz}$$

- B. (2 points) Input the circuit of Fig. 2 in Top-SPICE. Call the circuit “highpass”. Be sure to label the input and output nodes. **Print your schematic.**
- C. (6 points) Perform a DC analysis similar to what was done for the lowpass filter. **Plot the output voltage. Measure the DC voltage gain (slope).** Do not compare to the calculated value since the DC gain is 0. Why is the DC gain 0? How does a capacitor behave at 0Hz?

- D. (12 points) Perform an AC analysis. **Plot the output voltage magnitude in dB and phase. Measure the high frequency gain (A_{vh}) and the corner frequency and compare using %error analysis on the graph.** Print the waveforms with measurements.
- E. (12 points) Perform a sinusoidal transient analysis. For the voltage source VS, set the sinusoidal specifications to 300mV amplitude and a frequency equal to the designed corner frequency. **Plot the input and the output voltages. Measure the gain and compare to the theoretical gain using % error analysis on the graph.** Print the waveforms with measurements. For this exercise, the theoretical gain of the high-pass filter at f_o is:

$$A_v(f_o) = \frac{A_{vh}}{\sqrt{2}} =$$

What to include in your Lab Report

- 1) (10 points) Provide a 0.75-page typed (12-point FONT, 1.5 line spacing) summary of your results in the form of a **table** (see the sample below). In addition, write one to two paragraphs that describe any major challenges or learning experiences you went through. Put the lab partner names on this page.
- 2) Attach, Exercises II and III from the lab report (filled in neatly), along with circuits and simulation results with measurements and % error analysis from Top-SPICE. Again, do the %-error analysis on the Top-Spice graphs.

Table 1 - Sample Summary of Major Results

	Low Pass		High Pass	
	Value	% error	Value	% error
Expected A_v	793 mV/V		0.350 V/V	
Expected f_o	2.84 kHz		1.20 kHz	
DC Analysis Measured A_v	793 mV/V	0.0%	0 V/V	
AC Analysis Measured A_v	793 mV/V	0.0 %	0.350 V/V	0.0 %
Transient Expected $A_v(f_o)$	0.560 V/V		0.248 V/V	
Transient Measured $A_v(f_o)$	0.5791 V/V	+1.9%	0.247 V/V	-0.4 %