

ECE322 Laboratory Experiment 2 –SPICE Subcircuit Syntax and Evaluating Nonlinearities

I. Introduction/Objective

In the first lab, you became familiar with the SPICE syntax and techniques to perform DC, AC, and transient simulation tasks. In this lab, you will extend your knowledge of Spice syntax and use it to characterize basic amplifier circuits.

II. Prelab Discussion and Simulation – Dependent Source Models and Hierarchical Simulation Method

A. Dependent Source Models

Spice provides four basic types of dependent sources. The first two devices (VCCS and VCVS) are important in this course and we will focus more on these. These devices are outlined below with further syntax information available in your Spice manual. The second two types (CCCS and C CVS) are mentioned for completeness.

1. **Voltage-Controlled Current Source (VCCS) – G device**
The circuit of Figure 1 represents a typical usage of a VCCS in a Transconductance Amplifier.

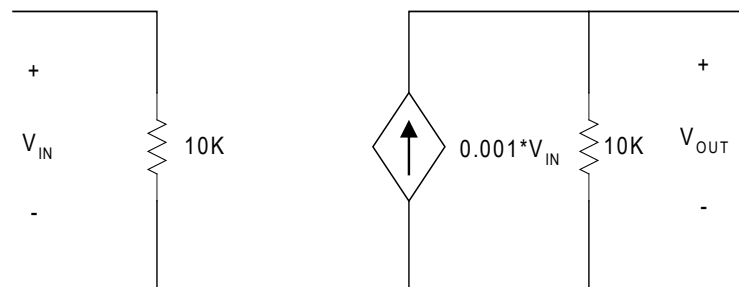


Figure 1. Transconductance Amplifier

VCCS Instantiation : `gxxxx from_node to_node vctrl+ vctrl- gain`

2. **Voltage-Controlled Voltage Source (VCVS) – E device**
Figure 2 represents a common usage of the VCVS in a voltage amplifier configuration.

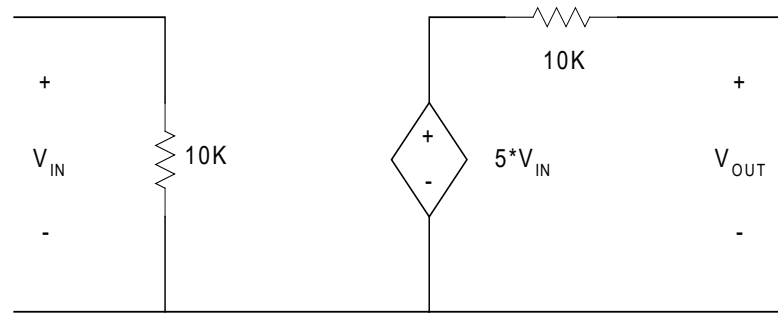


Figure 2. Voltage Amplifier

VCVS Instantiation : `exxxx +_node -_node vctrl+ vctrl- gain`

3. Current-Controlled Current Source (CCCS) – **F** device

Figure 3 demonstrates the usage of a CCCS in a Current Amplifier. An issue with current controlled devices is that we often need to add a 0V source from which to measure the controlling current.

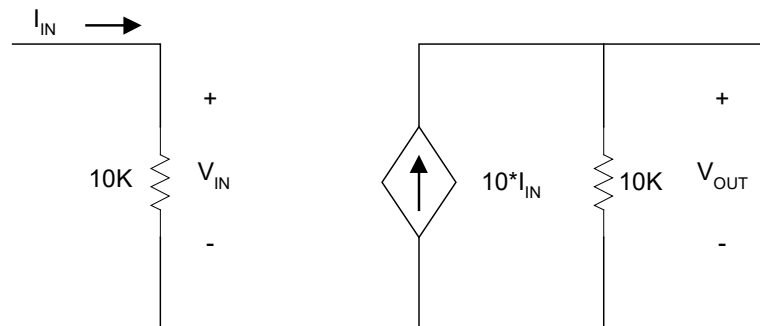


Figure 3. Current Amplifier

4. Current-Controlled Voltage Source (CCVS) – **H** device

The CCVS is unlikely to be used in this course, but shown for completeness in a Transresistance amplifier configuration in Figure 4.

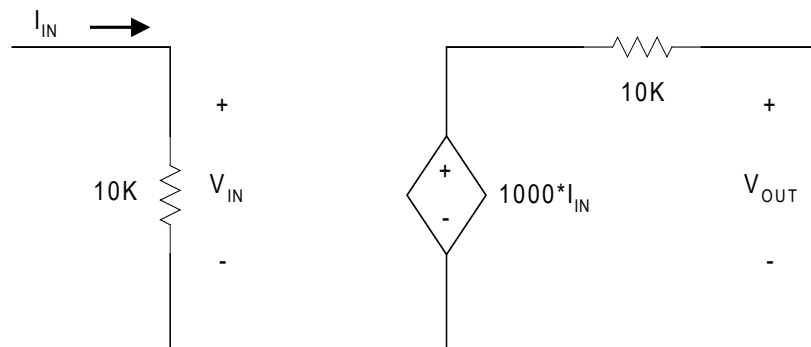


Figure 4. Transresistance Amplifier

B. Hierarchical Design and Simulation Approach

Often circuits are captured as reusable subcircuits and modeled in test fixtures or higher complexity circuits. You should be familiar with this concept from digital design and software design where lower-level routines are developed and tested and then called by higher level code. Devices such as opamps are modeled as subcircuits by the analog integrated circuits (IC) vendors to support their simulation in analog circuits. Consider the simulation test fixture in Figure 5 that evaluates a three terminal amplifier circuit that has interface pins V_{IN} , V_{OUT} , and V_{GND} .

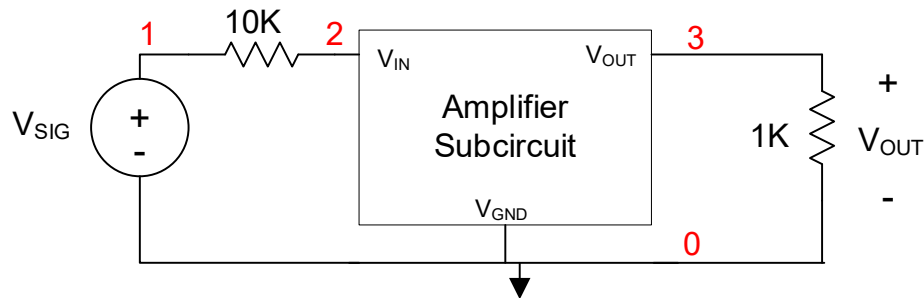


Figure 5. Test fixture for an Amplifier Subcircuit with input source with 10k Thevenin equivalent resistance and 1k load resistance. The circuit node names at the test fixture are named 0, 1, 2, and 3.

The Amplifier Subcircuit being tested is shown in Figure 6. It consists of a transconductance amplifier followed by a voltage amplifier. Notice it has a lowpass characteristic based on the argument of capacitor impedance at frequencies of zero and infinity. The node names used in the subcircuit definition are shown in red. Only three nodes feed out of the amplifier subcircuit block V_{IN} , V_{OUT} , and V_{GND} . The nodes V_X and V_Y are internal to the subcircuit and are required to define the subcircuit component connections.

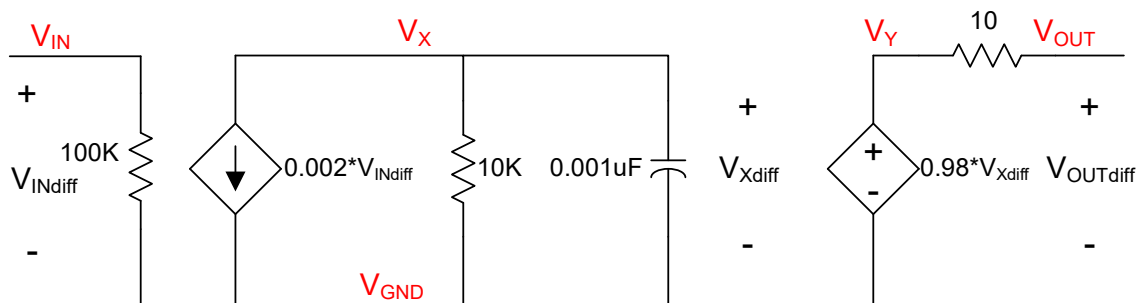


Figure 6. Amplifier Subcircuit Model and Node Definition

The minimum required cards in a SPICE subcircuit deck consist of (1) the subcircuit command line which defines the unique subcircuit name and the subcircuit pinorder for higher circuit level instantiation, (2) the cards to define the subcircuit components, and (3) the .ends command line.

SPICE Subcircuit File: Amplifier.sub**.Subckt Amplifier VIN VOUT VGND**

* .subckt is command to SPICE to start the subcircuit definition. Next is the unique subcircuit name. Next
 * is the subcircuit instantiation pinorder. Upper level subcircuit instantiations must have nodal connections
 * in the same order not same name as in subcircuit. This allows for one subcircuit such as an opamp to be
 * used many times in a single circuit.

* Circuit components instantiation

R1 VIN VGND 100K
G1 VX VGND Vin VGND 0.002
R2 VX VGND 10K
C1 VX VGND 0.001u
E1 VY VGND VX VGND 0.98
R3 VY VOUT 10

.Ends Amplifier

* .Ends command to tell simulator this subcircuit definition is completed

****Test Fixture Simulation file below calls separate amplifier.sub file with an include command!!**

Main SPICE Deck File: Amplifier_test.cir

* Example of Amplifier Subcircuit instantiation and test
 * Filename: Amplifier_test.cir, Subcircuit called: Amplifier, Subcircuit Definition file: Amplifier.sub

* Input Voltage source and resistance

Vsig 1 0 DC 0V
R1 1 2 10K

* Load Resistance

Rload 3 0 1K

* IMPORTANT DESCRIPTION OF HOW SUBCIRCUITS ARE INSTANTIATED

* Subcircuit instantiation. All subcircuits start with leading character X and then unique identifier.

* Note: The PINS must be in the correct order to match that of the subcircuit.

* In this case **2 = VIN** **3 = VOUT** and **0 = VGND** which matches order in sub file .subckt command.

Xamp 2 3 0 Amplifier

* Include card tells Spice to pull named file into this deck

.include Amplifier.sub

*Usual Spice Simulation Commands

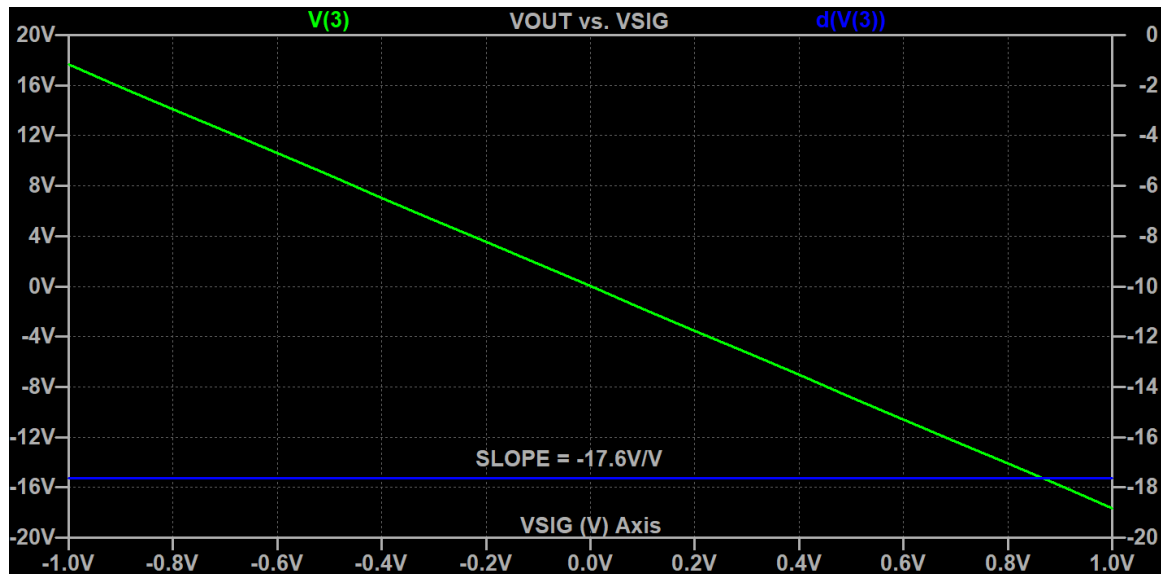
.dc Vsig -1 1 0.1
.probe
.end

IV. Pre-Lab Exercise

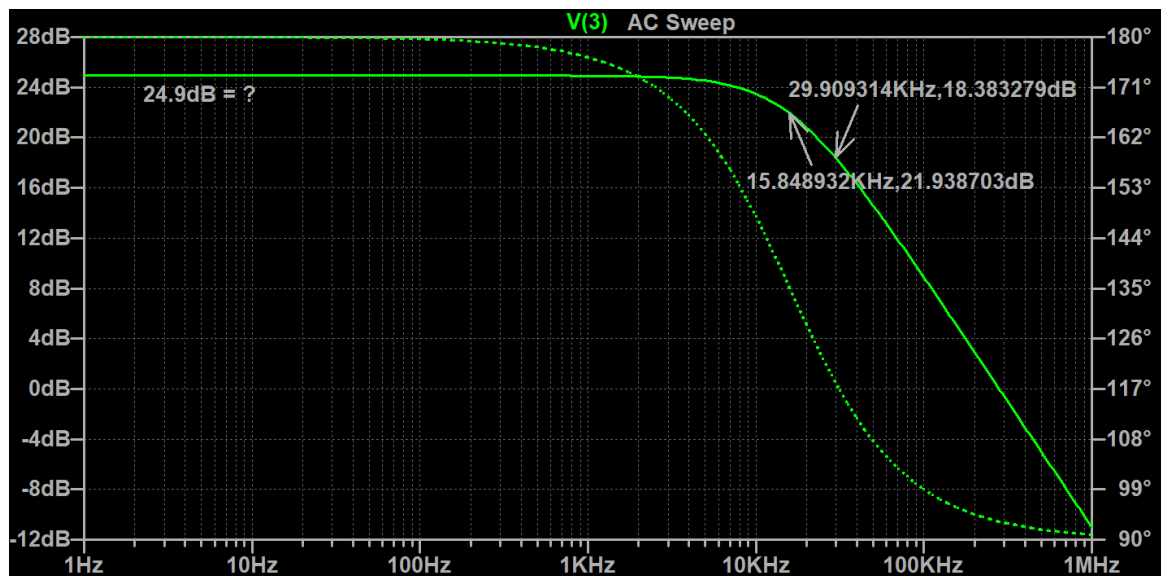
1. *DC Voltage Sweep*
Run Spice sweeping V_{SIG} from -1V to 1V in 0.01V increments. Use the post-processor to provide a plot of V_{OUT} vs V_{SIG} . Also provide a plot of the derivative of V_{OUT} vs V_{SIG} and source current vs V_{SIG} . Does this agree with an analysis of the circuit? See graph in appendix for verification of your simulation result.
2. *AC Simulation Sweep*
Run Spice and perform an AC sweep on the circuit from 1Hz to 1MHz with 20 points per decade. Let the DC value of the input source be 0V and the AC value be 1V . Use the post-processor to provide the circuit voltage gain in dB as a function of frequency. Does the low frequency gain and -3dB frequency agree with your own analysis (you need to calculate the circuit time constant to answer this)? See graph in appendix for verification of your simulation result.
3. *Transient Simulation*
Run Spice and perform a transient sweep over $300\mu\text{s}$ with the input voltage source defined as a sinusoid with an amplitude of 1 volt , zero offset, and frequency of 30kHz . Plot V_{OUT} and measure the output amplitude. Compare this result with the AC simulation result measured at 30kHz . See graph in appendix for verification of your simulation result.
4. From your results, comment on the AC sweep analysis relative to the transient simulations and the usefulness of the AC sweep tool.

APPENDIX

Prelab Simulation Result 1.



Prelab Simulation Result 2.



Prelab Simulation Result 3.

