Cody Gabriel
9/30/2014
Dr. M. Jeske
Dr. R. Campbell
ECE321L

Lab I: Further Adventures in LTSpice

Abstract
This laboratory report consists of five experiments conducted by the TA for an introductory course on semiconductor devices. The experiments in this lab assignment are solely based on simulations using LTSpice. The lab experiments are a combination of new and review material. The first two experiments should be review for any student taking this course while the next two experiments introduce new devices that the student may not have encountered before. This gives the student an opportunity to familiarize themselves with some of the more basic concepts of these new components (diodes and transistors). The purpose of the TA conducting the experiments is to ensure that they are ready to help the students succeed and learn. The assignment consists of five parts: Op Amps, circuits that contain dependent sources, diodes circuits, transistor circuits, and the last sections covers adding new components to the library and how to parameterize some circuit values. The focus of the assignment is not on solving problems but becoming better acquainted with the simulator and components that will be used heavily in the coming weeks of this course.

Introduction/Prelab
The simulator has been shown to be one of the most useful tools in an engineer's toolbox when designing or verifying lab measurements. In this introductory laboratory assignment the student will not only review the fundamentals of using the simulator but also be introduced to some of the new electronic components that will be explored in this course. The simulator of choice for the Portland State Electrical and Computer Engineering Department is LTSpice because it is a free software that is relatively easy to use and learn while also being compatible with many industry Spice simulators. For any students new to this department and LTSpice, or for those who feel they need a refresher, there is an introductory tutorial on LTSpice available on the Lab's D2L page.

Part 1: Op Amps
<u>Experimental Procedure</u>
In this portion of the lab assignment the student will set up a schematic of a cascaded pair of inverting op amps in LTSpice and simulate them in various ways. The component values and configuration are supplied in the lab handout and are shown below in Figure 1. The student is first asked to calculate the voltage value of specified nodes in the circuit and then to run a DC operating point simulation to verify their calculations. Next, the student will sweep the input voltage to the circuit using the DC sweep simulation. The following step in the procedure is to simulate the frequency response of the system by using an AC sweep simulation. Lastly, the load of the circuit is varied and the effects are measured with more DC operating point simulations for each specified load value.

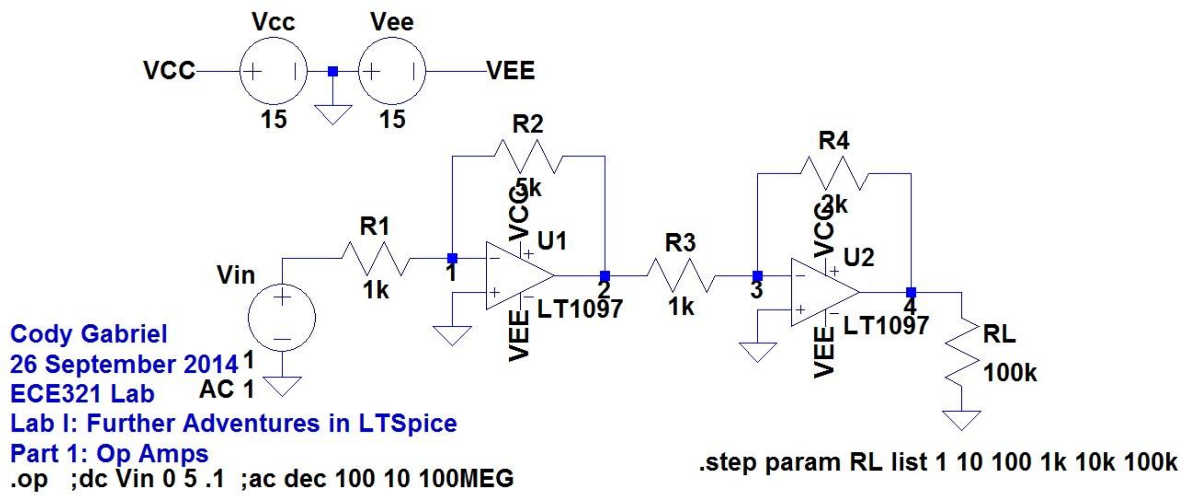

Figure 1: LTSpice schematic of op amp circuit used for Part 1.

Results and Discussion

The DC node voltage can be determined using the ideal op amp rules by inspection. The circuit consists of two stages, both of which are inverting amplifiers. The first stage has a DC gain of 5 while the second stage has a gain of 2. The gain of the two stages multiply because they are cascaded and inverted twice, so the total DC gain of the circuit is +10. By the ideal op amp rules nodes 1 and 3 are grounded and are therefore at zero volts. Node 2 is the output of the first stage which has a gain of -5, so for an input of 1 V has a voltage of -5 V. Node 4 is the output of the whole circuit which has been determined to be 10 times that of the input so for a 1 V input the output is +10 V. The DC operating simulation results as are presented below in Table 1 along with the expected values from the calculations.

Node Voltages	Calculated	Simulated
1	0 V	2.46 μV
2	-5 V	-4.99 V
3	0 V	-4.63 μV
4	+10 V	9.99 V

Table 1: Calculated and simulated results of DC operating point simulation.

As shown by the simulated results in Table 1, these op amps behave quite close to ideal in the simulator.

The input voltage was swept from 0 to 5 V in 0.1 V increments. The output versus input is plotted below in Figure 2.

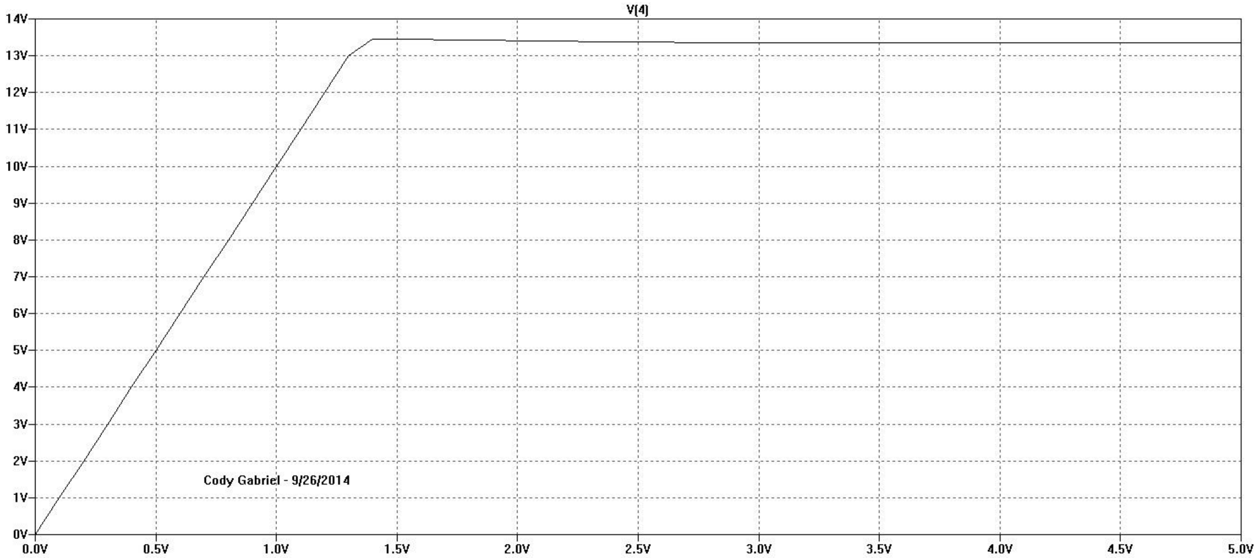

Figure 2: Output vs. Input voltage that resulted from the DC sweep simulation.

As shown in Figure 2, the output increases linearly until the input reaches approximately 1.4 V, where it then flattens out and remains constant for any further increase in input. The second stage op amp has saturated, the output by the ideal op amp approximation would exceed the DC power supplied to the op amps.

The circuit was then simulated using an AC sweep from 10 Hz to 100 MHz on a decade scale using 100 points per decade. The result of this simulation is shown below in Figure 3.

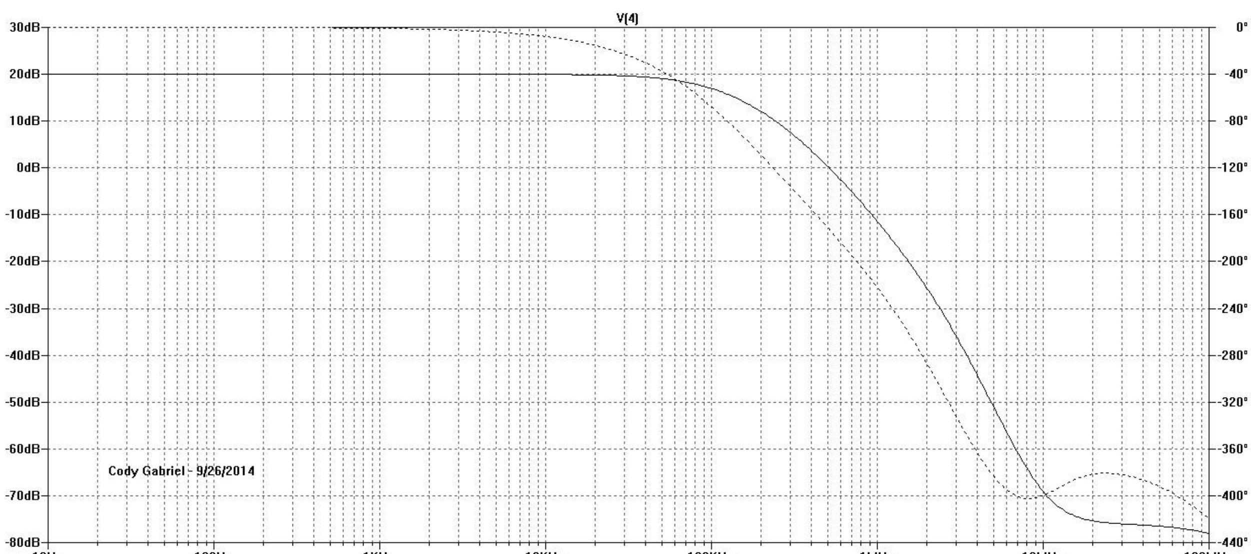

Figure 3: Frequency response of the op amp circuit as a result of an AC sweep simulation.

The circuit exhibits a low-pass response with a 3-dB bandwidth of approximately 100 kHz.

The final simulations for this circuit involve varying the size of the load resistance from 1 to 100 kΩ in decade sized steps using an input of 1 V. The output voltage was recorded in an Excel spreadsheet and plotted versus the load resistance. The table from the spreadsheet is shown below in Table 2 and the plot is shown below in Figure 4.

RL in Ohms	1	10	100	1000	10000	100000
Vout in V	0.0143	0.1431	1.3835	10	10	10

Table 2: Output voltage for decade spaced load resistance.

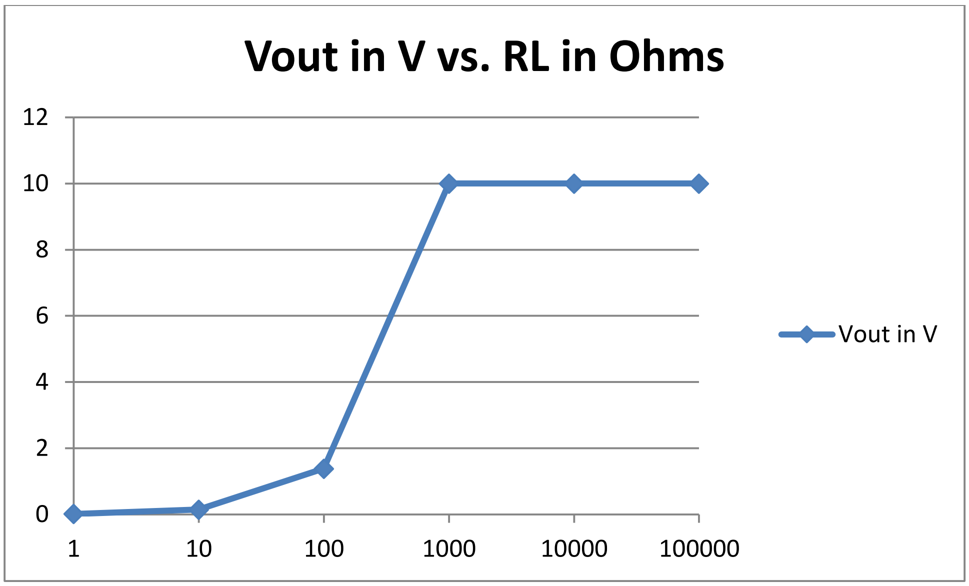

Figure 4: Output voltage versus load resistance on a log scale.

Not unlike in the previous case where the output voltage was limited by the DC supply voltage of the op amps, in this case the op amps are only able to so supply so much current to the output resulting in lower than desired output voltages for loads smaller than 1 kΩ.

Part 2: DC Analysis of Dependent Sources

Experimental Procedure

In this portion of the lab assignment the student will replicate the results of an example in the course textbook. The examples show the method of determining the Norton/Thevenin equivalent of a circuit that contains a dependent source. The schematic for this circuit is shown below in Figure 5.

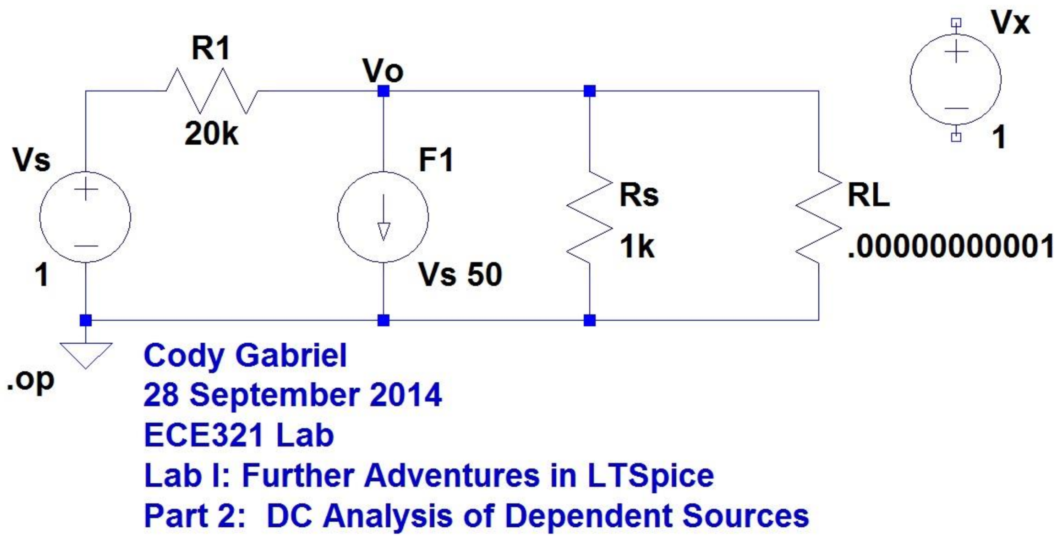

Figure 5: Schematic for Example 1.1 and 1.2 in the course textbook.

Results and Discussion

The library component that is used for a current dependent current source is labeled as simply "F" and for some reason has the opposite polarity as expected. To simulate the Thevenin equivalent voltage simply run a DC operating point simulation and probe the output node using a 1 V input. To determine the Thevenin equivalent resistance the input source must shorted, or set to zero, and add a 1 V test source on the output. Simulate this modified circuit and probe the current going through the test source; 1/(measured current) is the Thevenin resistance. To find the Norton equivalent current one must 'short' the output by adding a very small resistor (≪ 1); the Norton equivalent is the current through this resistor. As a check, divide the Thevenin voltage by this Norton current to verify the Thevenin resistance. The results are outlined below in the Table 3.

Example	Parameter	Calculated	Simulated
1.1	V_{th}	$0.718v_s$	0.71831 V
	R_{th}	282 Ω	281.69 Ω
1.2	I_n	$(2.55\ mS)v_s$	2.55 mA
	R_{th}	282 Ω	281.69 Ω

Table 3: Norton and Thevenin equivalents of the circuit in Figure 5.

Part 3: Diodes

Experimental Procedure

The PN junction diode is typically the first device that a student will encounter in an introductory solid-state electronics course. The purpose of this exercise is to help bridge the gap between theory and practice for the student; simulating diodes will help them understand what they read from the text and prepare them experimenting with actual devices in the lab later in the course.

This portion of the lab assignment will explore several different circuits that utilize diodes. The first of which, and simplest is the half-wave rectifier. The second circuit is the full-wave bridge rectifier. The student will perform a transient analysis on both of these rectifier circuits and observe the effect of diodes with a sinusoidal input. The third circuit is a simple Zener diode regulator circuit. This circuit will be simulated by sweeping the input DC voltage source and observing the output. The fourth and fifth circuits are simple ones that are used to observe the I-V characteristic curve of two different types of diodes: and ideal silicon diode and a germanium diode. The schematics for all of the circuits are shown below in Figure 6.

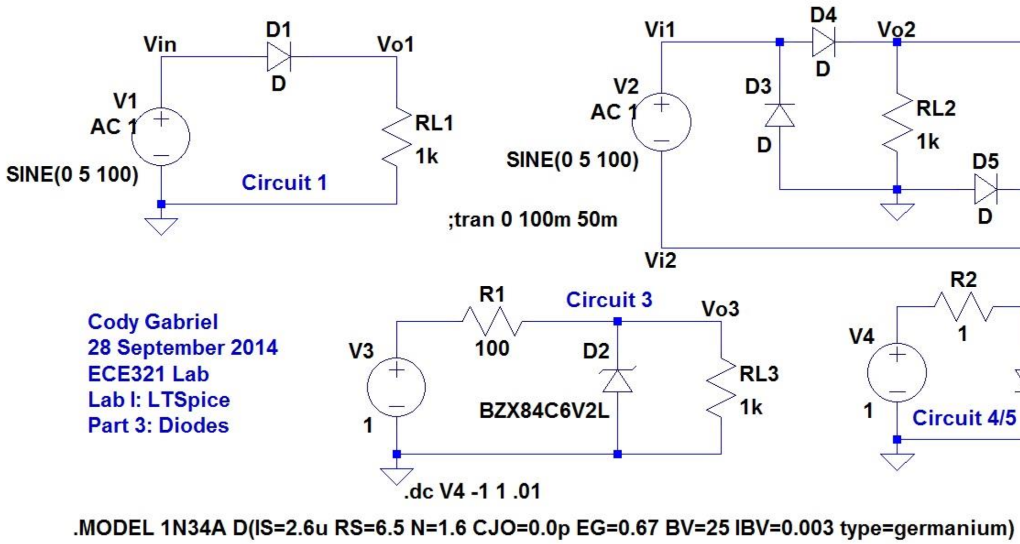

Figure 6: Diode circuit schematics.

<u>Results and Discussion</u>

Both circuits 1 and 2 were stimulated with a 5 V 100 Hz sine wave input and then simulated over 50 ms. The output was plotted over time as well as the input for comparison. The plots for circuits 1 and 2 are shown below in Figures 7 and 8, respectively.

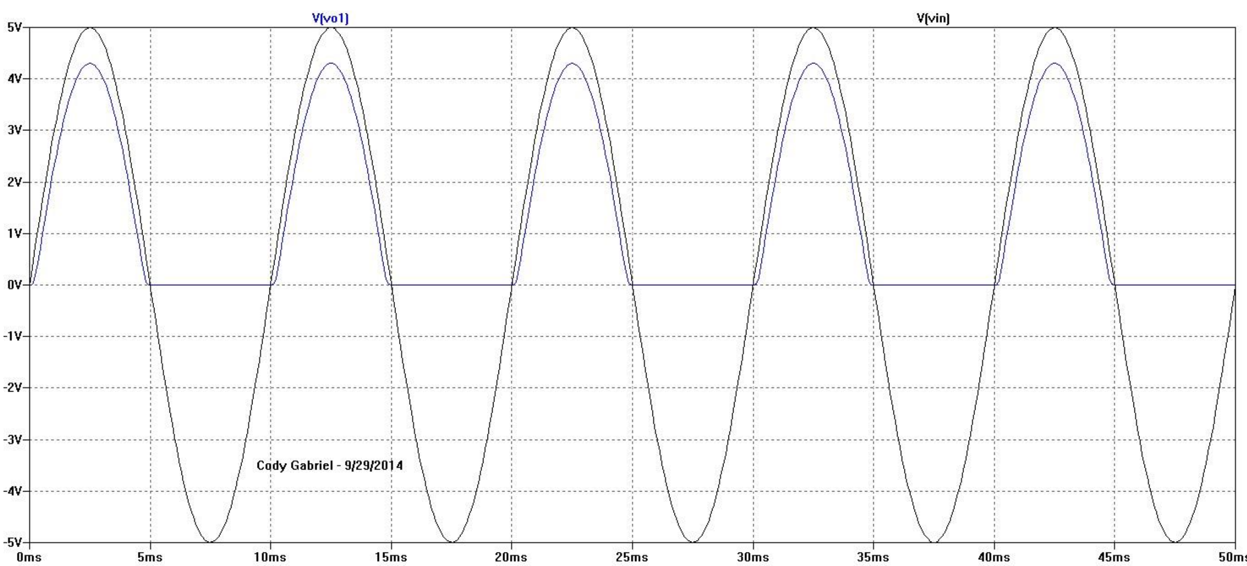

Figure 7: Output and input plots of half-wave rectifier circuit over 50 ms.

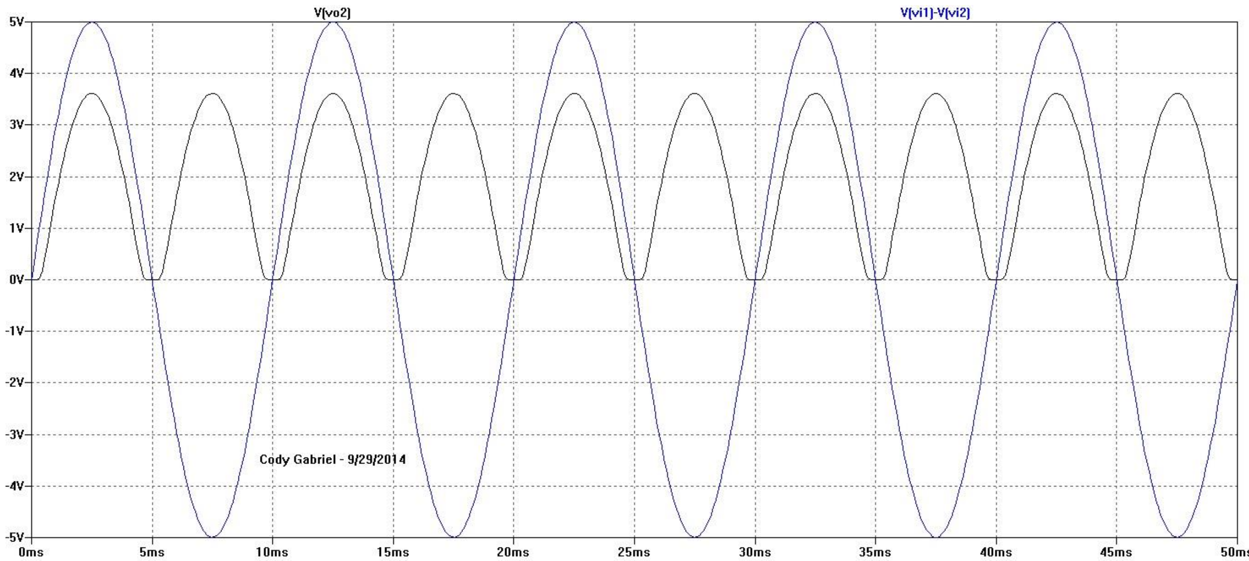

Figure 8: Output and input plots of full-wave bridge rectifier.

Some observations of the two different rectifier circuits would be that each output shows some loss in amplitude when compared to the inputs. The half-wave rectifier has a loss of one diode drop whereas the full-wave bridge rectifier has an amplitude loss of two diode drops. The output frequency of the full-wave bridge rectifier is double that of the half-wave rectifier and the input signal.

For circuit 3 in Figure 6, the DC input was swept from 10 V to 20 V in steps of 0.1 V. The output and input is shown below in Figure 9. A closer look at only the output is also shown in Figure 10.

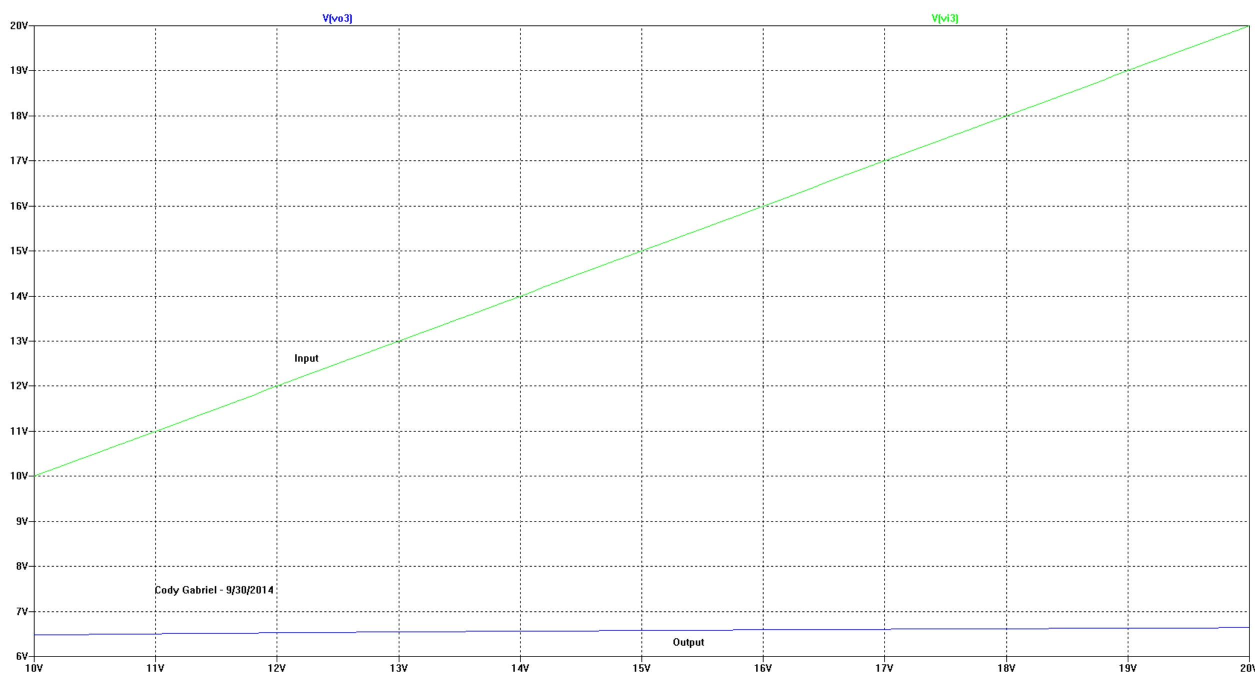

Figure 9: Input and output of Zener diode regulator circuit.

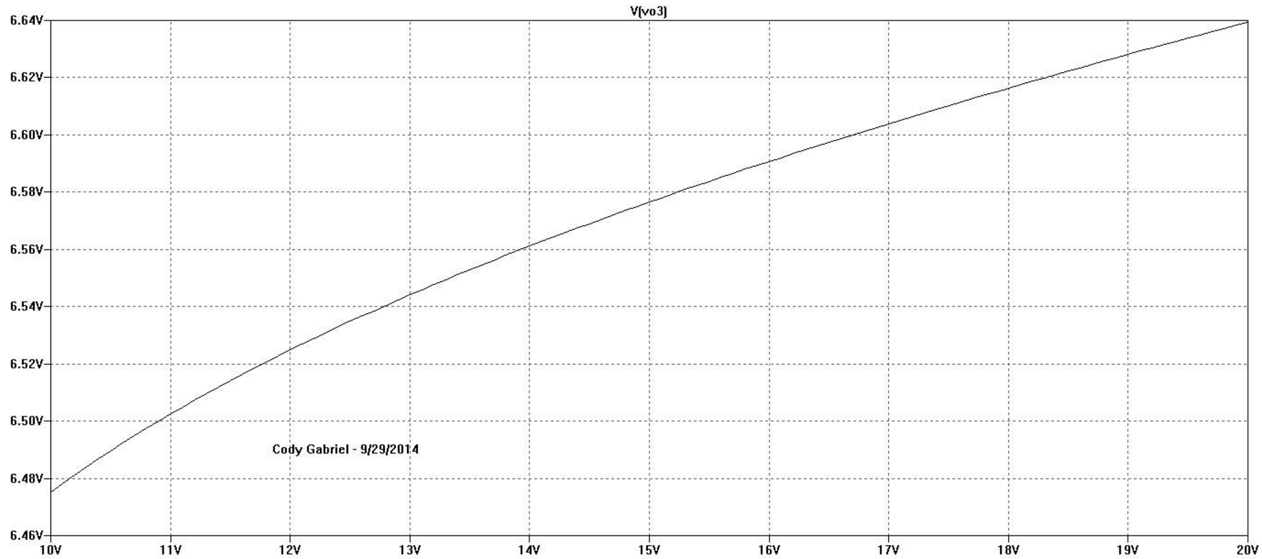

Figure 10: Output only of the Zener diode regulator circuit.

By observing the output versus input in Figure 9 it is obvious what the purpose of this circuit is. But when looking at Figure 10 one can understand that the output does not remain perfectly flat, suggesting that the Zener breakdown is occurring. This can be observed by plotting both the current through the output load resistor and the current through the Zener diode shown below in Figure 11.

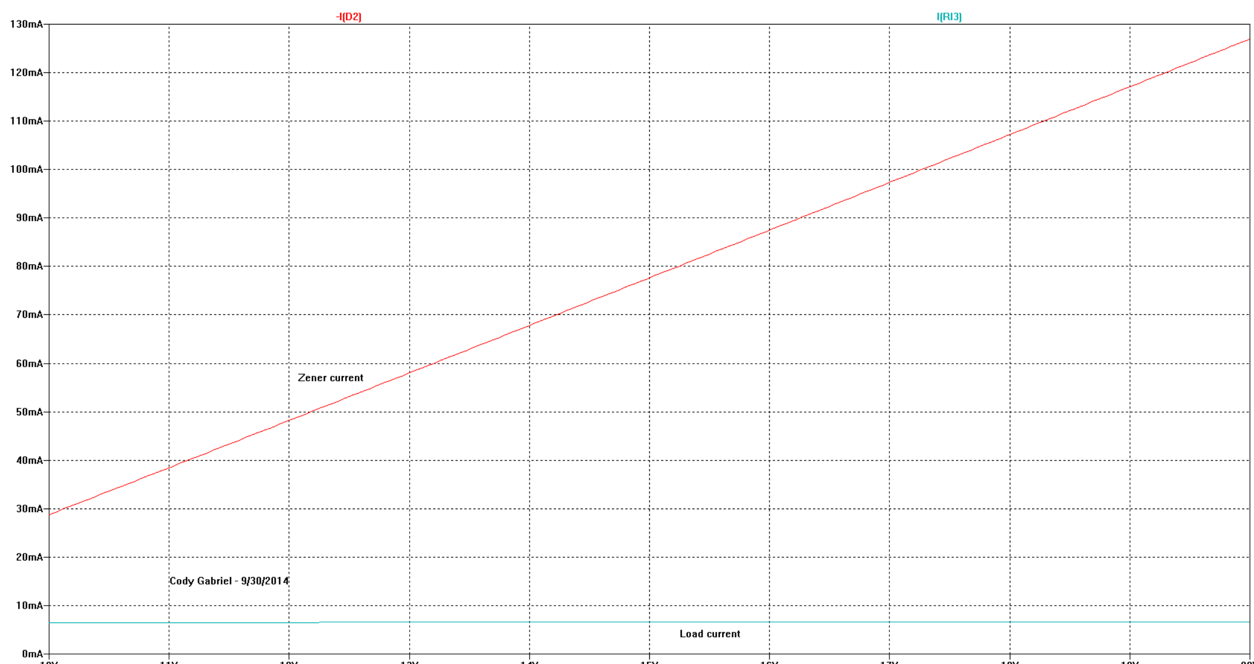

Figure 11: Zener current and load current plotted versus input voltage.

The final two diode circuits are identical with the exception of what type of diode is being used. The lab instructions stated to use a voltage source and diode only but that gave results of nonrealistic current values. I added a 1 Ω resistor in series to limit the current to more realistic values. The default library for LTSpice does not contain a germanium diode. The lab instructions provided a model that must be inserted or the germanium diode will not simulate properly. The voltage source was swept from -1 V to +1 V in steps of 0.01 V and the current through the diode was plotted versus the input voltage and is shown below in Figure 12 for the ideal diode and in Figure 13 for the germanium diode.

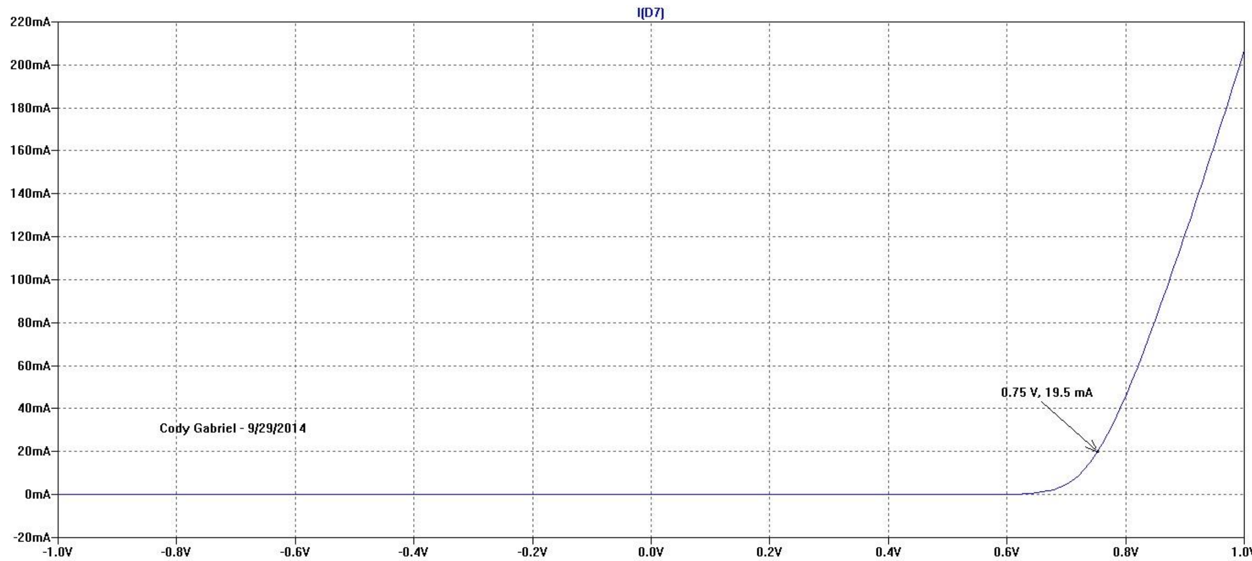

Figure 12: I-V characteristic curve for ideal diode.

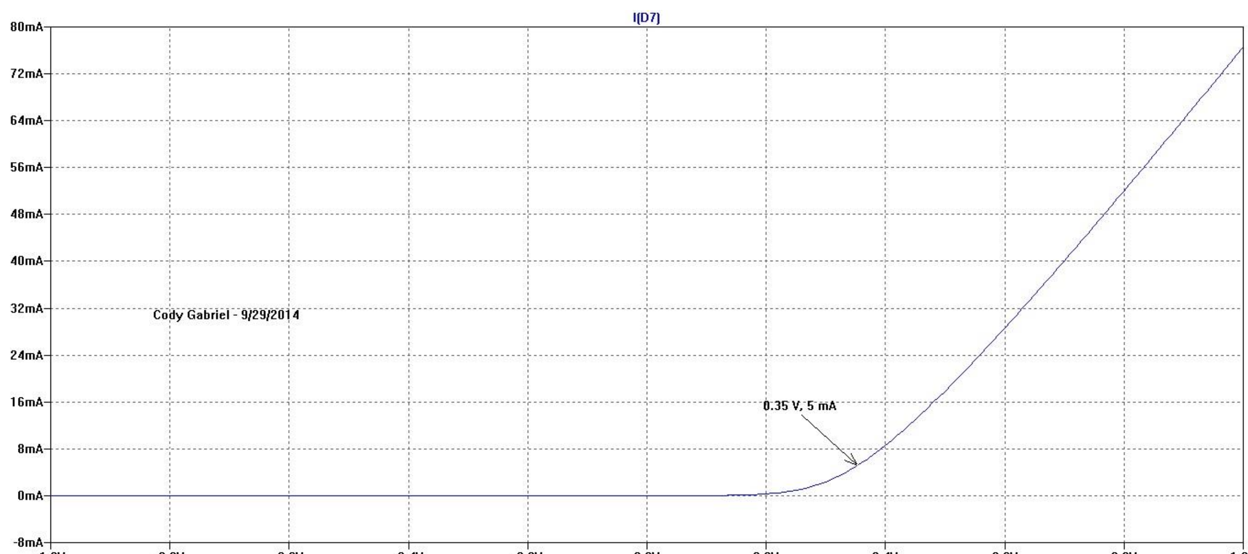

Figure 13: I-V characteristic curve for germanium diode.

The arrows on each I-V curve indicate approximately where the diode turns on and the output current shows a linear relationship with the applied voltage which is about 0.75 V for the ideal diode and 0.35 V for the germanium diode.

Part 4: Transistors

Experimental Procedure

Up until this point in the student's academic pursuit they have mostly only encountered and studied simple two-terminal devices. Transistors are the first three-terminal devices that the student will study intently later on in this course. Two transistors will be examined in this section of the lab assignment: the bi-polar junction transistor (BJT) and the metal-oxide semiconductor field-effect transistors (MOSFET). The circuits for the portion are shown below in Figure 14. The first circuit explored will be a simple BJT common-emitter amplifier using a NPN BJT. This circuit is biased using a 12 V DC voltage source and with an offset applied to the base with the sinusoidal input. The student will simulate the amplifier using a transient simulation and observe the input and output waveforms. The MOSFET circuits shown below implement two unknown digital logic circuits. The student will set the DC inputs through each combination in a truth table and measure the output to determine what logic gate are here.

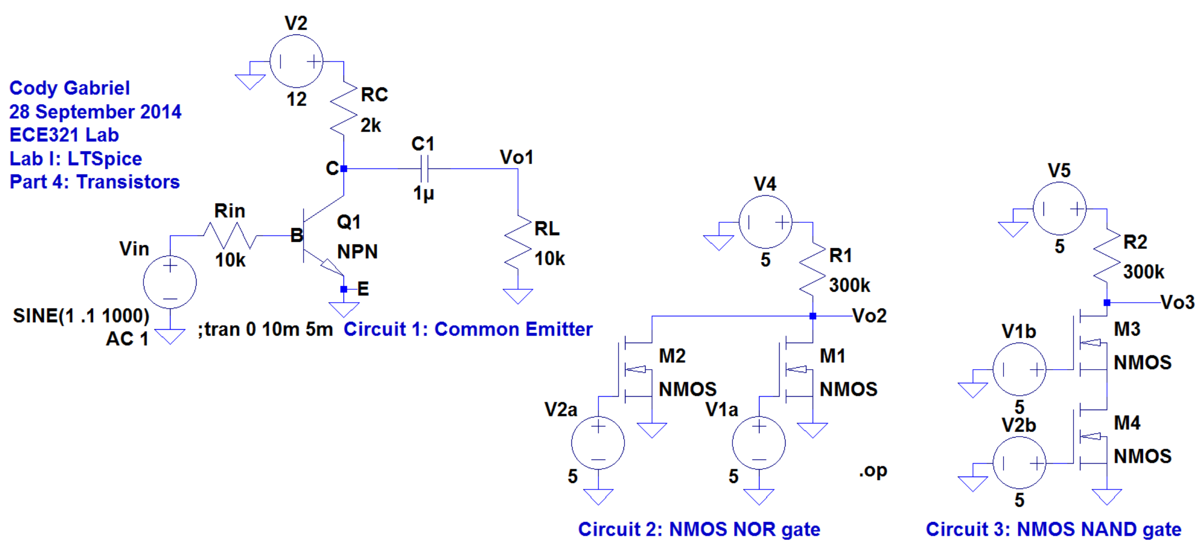

Figure 14: Transistor circuits, common-emitter BJT amplifier and two MOSFET logic gates.

Results and Discussion

The output of the common-emitter amplifier is plotted with the input below in Figure 15. The input voltage was instructed to be set to $v(t) = 1 + 0.1\sin(2\pi 1000)$, so the offset is visible and is what is forward biasing the base-emitter junction of the BJT. The output has an amplitude of 1.5 V, so for an AC input amplitude of 0.1 V the voltage gain is 15. The purpose of the capacitor, C1, the amplifier circuit is to block the DC bias voltage that is present at the collector of the BJT. This makes it so the output is ground referenced and symmetric about V = 0.

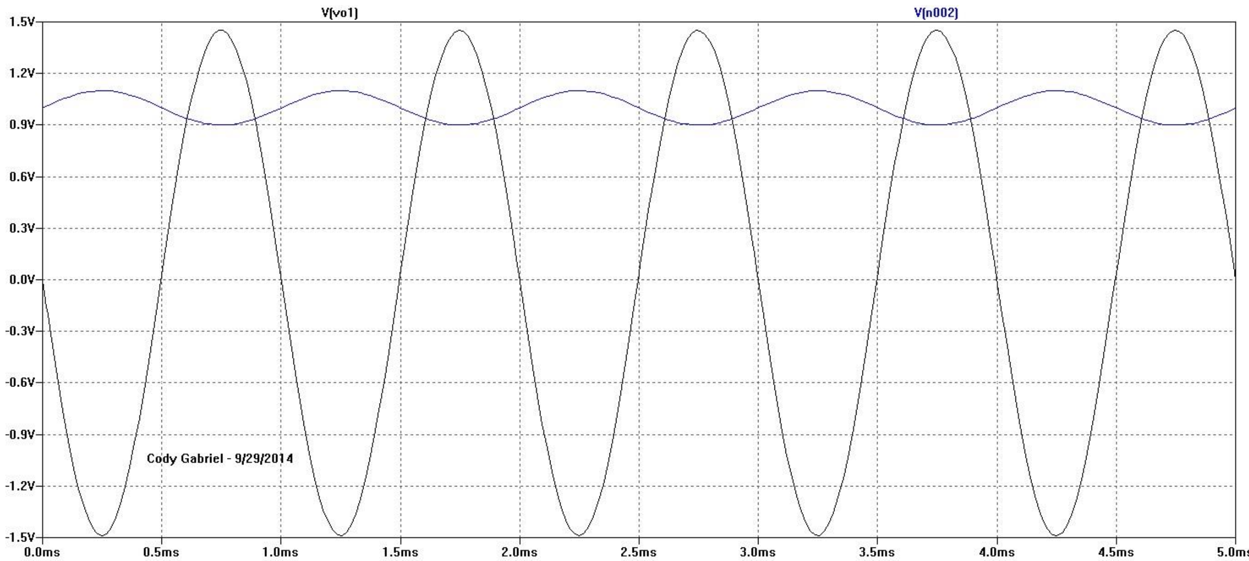

Figure 15: The output and input to the common-emitter BJT amplifier.

The two MOSFET logic circuits had a bias voltage of 5 V so I chose the same value for the input voltage in order to assure proper operation in the saturation region. The results of the DC operating point simulation of the MOSFET circuits are shown below in Tables 4 and 5 for circuits 2 and 3 respectively. Circuit 2 was determined to be a NOR gate and circuit 3 is a NAND gate.

V1a	V2a	Vo2
Low	Low	High
Low	High	Low
High	Low	Low
High	High	Low

Table 4: Truth table for circuit 2, a NOR gate.

V1b	V2b	Vo3
Low	Low	High
Low	High	High
High	Low	High
High	High	Low

Table 5: Truth table for circuit 3, a NAND gate.

Part 5: Adding New Components and Variable Parameters

Experimental Procedure

The purpose of the fifth and final experiment in this lab assignment is to learn how to add a new component to the LTSpice library and review how to use variable parameters. The instructions state to go the Yahoo! LTSpice group and download a .zip file that contains everything you need to add a new component to the library without having to create it. I added myself to this group and spent over half an hour looking for the right component. I contemplated creating my own symbol and subcircuit but decided that it was more effort than necessary. So I instead used some parameters and cleverness to create my own function potentiometer that works just as well. The schematic for the potentiometer is shown below in Figure 16. The potentiometer was made using two resistors in series with the output between them. The values of the resistors are parameterized such that an increase in one will decrease the other linearly. The circuit was simulated using a DC operating point simulation sweeping over the possible values in 1 kΩ increments.

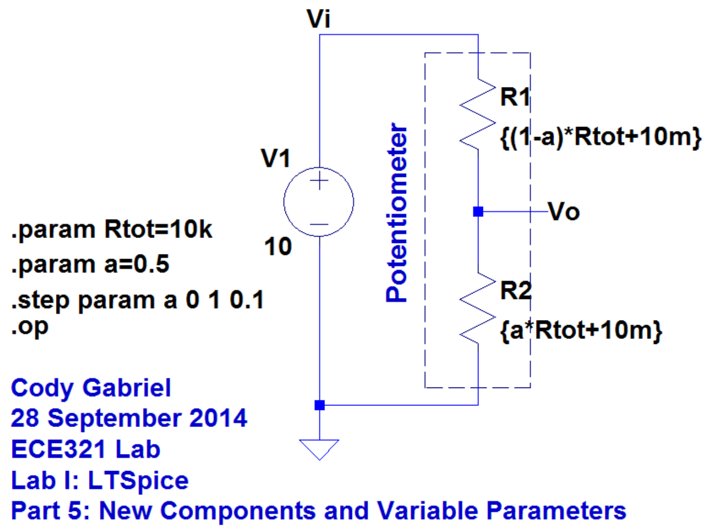

Figure 16: Potentiometer made with two resistors and parameterized dependent values.

Results and Discussion

The output of the potentiometer is plotted below in Figure 17. The linear relationship is clearly shown in the figure. The uniqueness of this circuit allows the user to specify the output as a percentage of the input which would aid in tuning a design. This potentiometer could easily be implemented as a new library component.

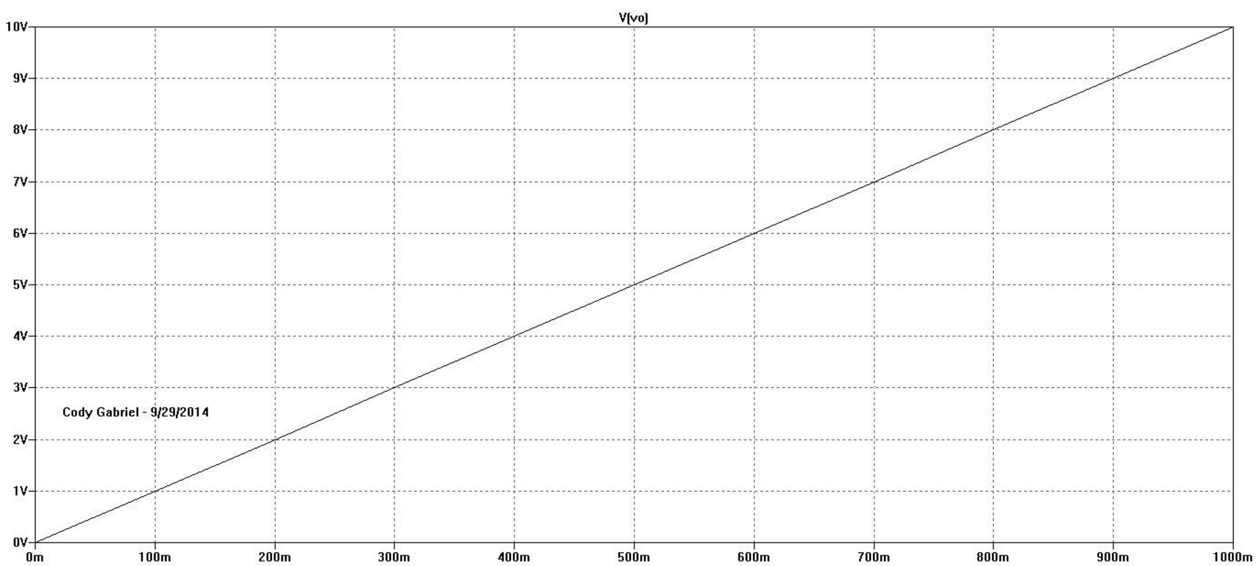

Figure 17: Output from the potentiometer sweeping over the values from 0 to 10 kΩ in 1 kΩ increments.

Conclusion

This laboratory assignment covers the most common functionality of LTSpice using some familiar examples as well as an introduction to some new devices that will be encountered this term in ECE321. The simulation types that this assignment covers are the DC operating point, DC sweep, transient analysis, and AC analysis. Also covered are parameterization techniques, multiple types of voltage/current sources, and defining model parameters.

I liked that for the examples that were review material, they were simple examples so that the student could focus on the LTSpice instead of trying to understand the circuit theory. Similarly, for the circuits that contain new elements, no prior knowledge of diode or transistor operation was required to complete the assignment successfully.

There are a few suggestions I would make to improve this lab in the future. Firstly, I would suggest re-writing the directions for Part 2, particularly where it states that Part 2 is directly related to Examples 1.1 – 1.4. That is not the case; Part 2 is only related to Example 1.1 and 1.2. This was confusing to me when I was doing this lab.

For Part 3 I will provide page numbers from the textbook to the students so that they can read up on the devices and circuits that we are simulating to better understand the results of the experiment. It would be nice to have these references on the lab assignment in the future so that the student can familiarize themselves with the behavior of diodes and transistors before running simulations with them.

The last thing on the assignment that could use some attention is Part 5. I had a hard time finding a potentiometer that was general enough for my taste. I think that it would be nice to have an additional document that goes through the process of creating my own component and how to add it to the library. I have done this before for the labs in ECE322 and remember it being more hassle than it’s worth because the instructions weren’t very straightforward. The help section in the software isn’t the most helpful either. For Part 5 I simply used two resistors and parameterized their values to get the desired functionality.

Appendix

N/A