

Lab 10: Diodes and Transistors using LTSpice

Getting started with LTSpice

Hello again!! I hope you are healthy and safe! :) If you have already installed LTSpice and learned how to work with it, ignore this section and move on to the next.

LTSpice is a very cool and simple software to use for simulating circuits. Please download the software from:

https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html?gclid=EAIaIQobChMIIsqGC6r-h4AIVSbjACh0y5QOmEAAAYASAAEgLz4_D_BwE

LTSpice could be slightly different for Mac and Windows users. Therefore, we provide two video links, the first one for Mac users and the second one for Windows users. They are very similar though.

1. <https://www.youtube.com/watch?v=6AA4YBtqhwE> (Mac users)
2. <https://www.youtube.com/watch?v=JRcyHuyb1V0&t=19s> (Windows users)

And a small written tutorial could be found here:

<https://eecs.oregonstate.edu/education/docs/tutorials/LTSpiceIntro.pdf>

Please create a report and include all the steps explained in each experiment.

Note: the values used in the experiments below, might be different from what have been used in lab10 handout in the website.

Lab 10: Diodes and Transistors

Experiment 1, Rectification: (15pts)

Halfway rectifier:

Please read this section in lab 10 handout. We are going to mimic the circuit behavior in LTSpice.

1. Open LTSpice, load the lab10_Ex1_Rectifier.asc file.
2. Make a plot of V_{in} (the signal from the function generator) and V_R (the voltage across the resistor). Attach the plots to your report. (5pts)

Explanation: when you run the simulator file, you may not see the desired waveforms at the first place. After the empty plot window pops up, you need to click on desired spots on the simulator's circuit so that the waveforms show up on the plotting window.

3. Place a 47 micro Farad capacitor in parallel with R_L in the simulator.
4. Make a plot of V_{in} (the signal from the function generator) and V_R (the voltage across the resistor) after placing the capacitor. Attach the plots to your report. (5pts)
5. Add a second capacitor to the circuit. What happens to the ripples of waveforms and their amplitudes? (2 pts)
6. Attach a picture of your simulated circuit to your report. (3pts)

Experiment 2, types of diodes: (12pts)

Zener diodes

Please read this section in lab 10 handout. We are going to mimic the circuit behavior in LTspice.

1. Please draw the circuit in Fig. 1 in LTspice simulator. You may want to use 1N750 Zener diode in LTspice. (3pts)
2. Make a plot of V_{in} (the signal from the function generator) and V_R (the voltage across the resistor). Attach the plots to your report. (3pts)
3. Please draw the circuit in Fig. 2 in LTspice simulator. You may want to use 1N750 Zener diode in LTspice. (3pts)
4. Make a plot of V_{in} (the signal from the function generator) and V_o (the voltage across the Zener diode). Attach the plots to your report. (3pts)

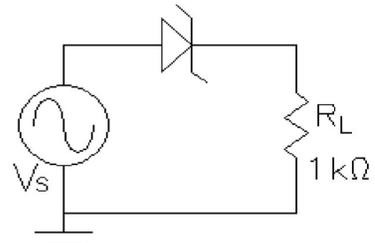


Fig. 1. Circuit using a Zener diode. Measure the voltage across R_L in LTspice.

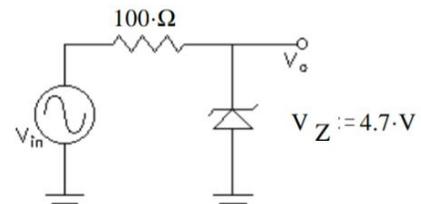


Fig. 2. Circuit using a Zener diode. Measure the voltage across the Zener diode in LTspice.

Light Emitting Diodes (8 pts)

Please read this section in lab 10 handout. We are going to mimic the circuit behavior in LTspice.

1. Please draw the circuit in Fig. 3 in LTspice simulator. You may want to use NSPW500BS. (3pts)
2. Calculate the LED current, assuming its voltage is about 3V. (2pts)
3. Measure the LED voltage and calculate its current based on the measured LED voltage. (2pts)
4. Are the calculated and measured currents similar? (1pts)

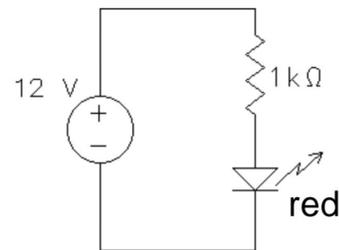


Fig. 3. Circuit using a light emitting diode. Measure the voltage across the LED and resistor in LTspice.

Experiment 3, transistors:(15pts)

Transistor amplifier

Please read this section in lab 10 handout. We are going to mimic the circuit behavior in LTspice.

1. Watch the YouTube video below and make the circuit explained there in LTspice. (5pts)
https://www.youtube.com/watch?v=9_mOp01TCKc
2. Measure the peak to peak voltage of the signal generator and the voltage across the collector (2pts)
3. Calculate the gain (3pts)
4. Change the resistor connected to the emitter to 100, 1k. Record the gains for each case. (5pts)

Conclusion(5pts)

Total score: 50 points