UC San Diego, ECE65

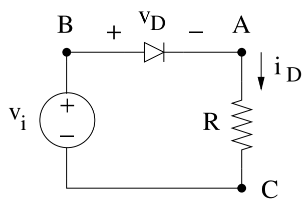
Lab 3: Diode iv Characteristics, Zener Diode, Clipper circuit

Experiment 1: General Purpose diode *iv* Characteristics

One way to measure the *iv* characteristics of an element is to apply a known voltage to the element and measure its current. By repeating these measurements at different voltages, we arrive at the *iv* characteristics. This experiment shows how we can measure/plot the *iv* characteristics of an element (a 1N4148 diode in this case) on the scope. It uses the (x vs. y) capability of the scope, which plots v_1 versus v_2, where v_1 and v_2 are voltages on scope channels 1 and 2, respectively.

The oscilloscope only measures voltages. To measure the diode current, we need to "convert" it into a voltage and measure the voltage using an oscilloscope.

In the following circuit, the currents through the diode and the resistor R, in series with the diode, are the same. Thus, measuring the voltage across the resistor and scaling it by the reciprocal of the resistor value will result in the diode current. ($i_D = i_R = v_R/R$). Furthermore, in order to "see" the plot on the scope, the plot should be refreshed continuously (*i.e.,* we need to use a periodic input voltage).



<u>Prelab:</u>

Simulation

Simulate circuit of Fig. 1 with v_i being a 1-kHz triangular wave with a peak to peak value of 10 V and a DC offset of zero (*i.e.,* input signal ranges from -5 to $+5$ V). Run the simulation for three periods. Use R=1 $k\Omega$.

1. Plot v_D and v_R as functions of time.

2. Plot $i_D = \frac{v_R}{R}$ as a function of v_D. On your plot, identify forward-bias and reverse-bias regions. (To accomplish this, you need to change the x-axis to v_D and then plot $\frac{v_R}{R}$. Do not change the simulation profile settings for this.)

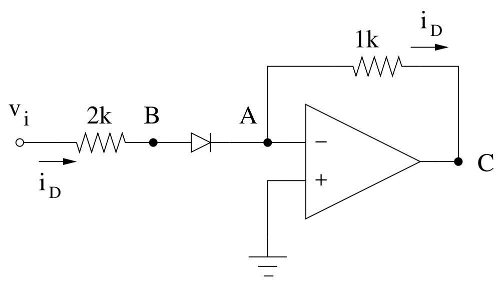
Lab Exercise:

It turns out that the circuit of Fig 1 will not work in the lab, and we need a more complicated circuit. To understand the reason, build the circuit of Fig. 1. v_i is supplied by the function generator and is a 1-kHz triangular wave with a peak to peak value of 10 V and a DC offset of zero (similar to simulation). The scope should be in its default mode of showing channel traces as functions of time.

1. Use scope probe for channel 2 to view v_R (i.e., attach the scope ground to point C and the probe to point A). Compare with your simulation.

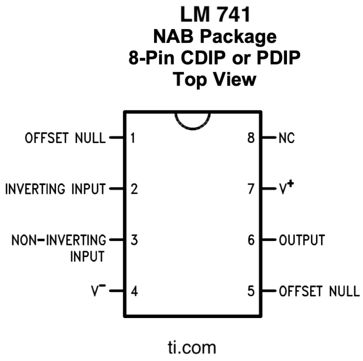
2. Try to simultaneously read v_D on channel 1 of the scope. What happens to the channel 2 trace? Explain why this setup does not work while the circuit simulator gives the correct answer.

3. Build the below circuit with a 741 Op-amp chip.



The chip should be powered with ±15 V supplies. This type of chip has two important properties which are relevant to this experiment. One, the current flowing into chip input terminals (marked by − and + signs) is very small and can be ignored. Second, if we operate this chip in the negative feedback mode (as is done in the above circuit), the voltages at the "inverting" terminal (marked by −) and the "non-inverting" terminal (marked by +) are almost equal.

The following picture shows the pinout of LM741 IC.



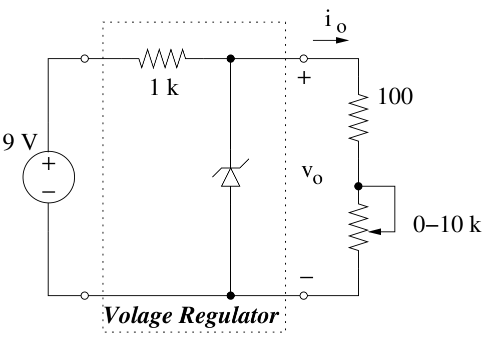
Set the function generator to produce a 1 kHz triangular wave with a peak to peak value of 10 V and a DC offset of zero. Attach the scope ground to the non-inverting terminal of the Op-amp, which is grounded (inverting and non-inverting terminals have the same voltage and, thus, point A is effectively grounded). Attach Channel 1 probe to point B (so Channel 1 reads v_D) and Channel 2 probe to point C (which will read $10^3 i_D$). Print out the traces and compare the results to your simulations. Explain why this circuit works.

4. Using the same function generator setting, set the scope to show (x vs y). Set both channels to 1 V/division. The scope should show one point. Move the point such that it is at the lowest, right-most voltage division marks on the scope. Slowly increase the amplitude of the input. The scope shows the $i - v$ characteristics of the diode. Increase the amplitude of the input wave until the diode iv curves "fills" the scope display. Print out the scope output and mark and label the axes.

5. What is the function of the 2 k resistor?

Experiment 2: Zener Diode Power Supply

Set up the circuit below with a 1N5232B Zener diode ($V_Z = 5.6$ V). In this circuit, the 9-V supply represents the "unregulated voltage." The elements in the box (1 kΩ resistor and the Zener diode) form the regulator circuit. The combination of the variable resistor (potentiometer) and 100 Ω resistor represents the "load" in this circuit (call their combination R_L). With varying the resistance of the potentiometer, we can draw a different amount of current from the regulator circuit.



<u>Prelab:</u>

Circuit Analysis

Using a "Constant voltage" model for the Zener region, calculate the output voltage of the regulator (v_o) as a function of its output current (i_o). Estimate the maximum load current for the circuit to act as a voltage regulator.

Simulation

Simulate the circuit with R_L (combination of the potentiometer and 100 Ω resistor) as a parameter with a range of 100 Ω to 10 kΩ (do NOT include the 100 Ω resistor in your simulation!). Plot v_o versus i_o and compare with your analytical results. (Change the x-axis to display i_o instead of resistance.)

<u>Lab Exercise:</u>

Assemble the circuit. Start with the potentiometer set at maximum resistance (*i.e.,* about 10 kΩ). Measure the load current and the load voltage. Then, vary the potentiometer resistance and measure the load voltage for a range of load currents. Plot v_o, versus i_o. Compare with your circuit analysis and simulation and explain the results (especially the observed slight drop in v_o when i_o is increased).

Experiment 3: Clipper Circuit

<u>Prelab:</u>

Circuit Design

Design a clipper circuit using 1N4148 general purpose diodes and a $R = 1$ kΩ to clip the input signal voltages that are above 5.7 V or less than -0.7 V. Make sure to use a DC source in your design, and do NOT use Zener diodes.

Simulation

Using Transient analysis, simulate the circuit you have designed with two input waveforms:

1. A sinusoidal wave with a frequency of 1 kHz, an amplitude of 8 V, and a DC offset of zero
2. A sinusoidal wave with an amplitude of 8 V and a DC offset of 4 V. In each case, plot v_o and v_i for two periods (both traces on the same graph).

Note: Make sure your step size is small enough. You should see smooth curves.

Lab Exercise:

1. Assemble the circuit that you have designed. Take a picture of the setup and include it in your report.
2. Apply a sinusoidal wave with a frequency of 1 kHz, an amplitude of 8 V, and a DC offset of zero to the circuit. Adjust the oscilloscope settings to display v_i and v_o properly. Take a picture of the oscilloscope display and include it in your report.
3. Increase the DC offset of the input. What do you see? Does it follow your simulation results? Explain.
4. Set the DC offset to +2V. Adjust the oscilloscope settings to display v_i and v_o properly. Take a picture of the oscilloscope display and include it in your report. Explain the results.

Helpful Tips for LTspice:

- To create a **Zener Diode** in Exp. 2, use the

  .model Z5p6 D(Is=1.5n Rs=.5 Cjo=185p nbv=3 bv=5.6 Ibv=1m)

  SPICE directive **in addition** to your simulation directives. After you add this directive, make sure to rename your Diode to **Z5P6** (Change 'D' to 'Z5P6'. Do not change the 'D1' label). Otherwise, LTspice will not know what to apply the .model directive to.

- Use a Parametric Sweep for Exp. 2. **Don't forget to add the .op directive in addition to the .step directive**