ECE343 Lab #2: Diodes

1   Introduction

In this lab we'll be working with diodes and exploring some of their practical applications. The objectives for this lab are:

- Simulation of diode I-V curves

- Using diodes as reference voltages

- Using diodes as a rectifier

2   Diodes-Review

In previous courses, you should have been introduced to diodes and simple models of their I-V behavior. We can create an ideal model of the diode that satisfies the following equations:

$$I_D = 0 \text{ for } V \leq V_D \tag{1}$$

$$V = V_D \text{ for } I_D \geq 0 \tag{2}$$

This forms a "right-angle" curve on the current-versus-voltage $v - i$ graph. While this is a nice approximation, real life models are not quite as simple. A key difference between the "ideal" diode you have seen earlier and a practical diode is that for a practical diode $V_D \neq 0$ when $I_D \neq 0$. The full semiconductor model of a diode is beyond the scope of this class. The $v - i$ characteristic for a practical diode can be approximated using the following equation:

$$I_D = I_S \left(e^{V_D/V_T} - 1\right) \approx I_S e^{V_D/V_T}, \text{ if } V_D >> V_T \tag{3}$$

where I_S is the reverse saturation current, I_D is the diode current, V_D is voltage across the diode terminals, V_T is a constant called thermal voltage. Figure 1 Below, we plot an ideal model compared to a real one to show the difference.
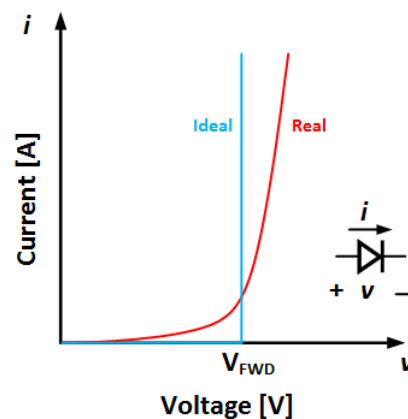

Figure 1: Ideal compared to real diode I-V curves.

3   Simulations

In this section we will perform *Spice* simulations to first obtain diode $I - V$ characteristics and then explore how this diode can be used as a simple voltage reference.
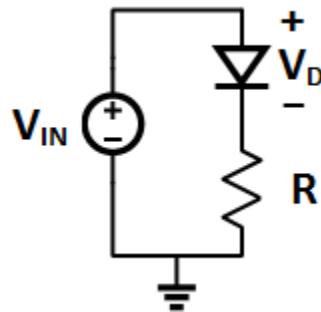
3.1   Diode Characterization

Figure 2: Diode characterization test bench

Component Values:

$$\text{Diode}: 1N914/1N4148, V_{IN} = 4V, R = 1k\Omega$$

1. Draw the circuit shown in Fig. 2 in LTspice.

2. Perform a DC sweep simulation on the circuit. Use the following parameters:

   - Source to sweep : V_{in}
   - Type of Sweep : Linear
   - Start Value : 0.2 V
   - Stop Value : 3 V
   - Increment : 0.01

3. Plot and take a screenshot of the I-V characteristics of the diode:

   - The current can be selected by clicking directly on the diode.
   - The voltage across the diode can be plotted by right-clicking the x-axis and typing in the correct formula in the "Quantity Plotted" box.

4. What is V_{FWD} of the diode? (An approximate value of V_{FWD} can be found by extrapolating a line from the steepest part of the curve. To draw a line, right-click the plot and select draw.)

3.2 Diode as a Reference

When designing circuits, we often only have one voltage as an input: V_{DD}. With this in mind, we may also need different reference voltages and biases elsewhere in the circuit. One simple way to generate a reference voltage is by using a diode. In this section, we will explore how a diode can be used as a simple voltage reference.
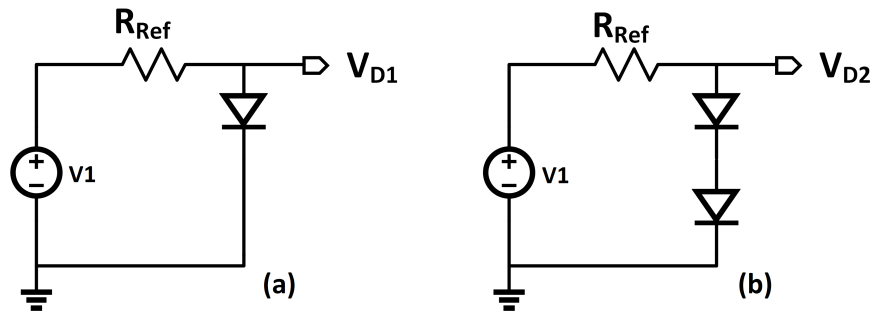

Figure 3: Left, one diode as a reference. Right, doubling our reference.

1. Assuming we are using the same diodes as in the previous section, compute the expected values of V_{D1}, V_{D2}.

2. Now assume that the source voltage has a sinusoidal noise component
   (i.e $\mathbf{V_1} = \mathbf{V_{DC}} + \mathbf{v_n}\sin(\mathbf{2\pi 100t})$). Derive an expression for the incremental output voltage $(\mathbf{\Delta V_0})$ in terms of the diode incremental resistance $\mathbf{r_d}$ for **each** circuit.

3. Draw the circuits shown in Fig. 3 in LTspice. Let $\mathbf{V_1} = \mathbf{4 + 0.5}\sin(\mathbf{2\pi 100t})\mathbf{V}$ and $\mathbf{R_{ref}} = \mathbf{1k\Omega}$. Make sure to use the 1N914 diodes as before. Perform a **transient** simulation on the circuits with the following parameters:

   - Stop time : 50m
   - Time to start saving data : 5m
   - Maximum timestep :1u

4. Note down the peak-to-peak variation in output voltage $(\mathbf{\Delta V_0})$ for both circuits. Take a screenshot of one of the plots.

3.3   Diode reference test circuit

In an ideal reference, the output voltage should not change under any load conditions. We will now see how well the diode based reference circuit discussed in section 3.2 acts under different load conditions.
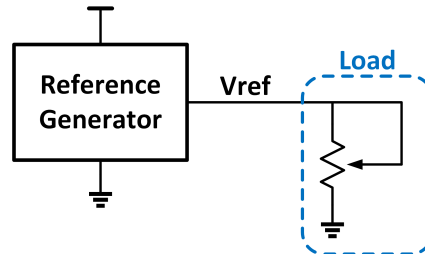

Figure 4: Simple reference generator measurement circuit.

1. Draw the voltage reference circuit shown in 3(b) with a resistive load ($\mathbf{R_L}$) as shown in Fig. 4. Set $V_1 = 4 + 0.5\sin(2\pi 100t)$ with $R_{ref} = 100\Omega$. Perform a DC operating point simulation while sweeping the resistive load from 10Ω to $2k\Omega$ with an increment of 10Ω. Please refer to the LTspice tutorial on the website for details on how to set the resistor as the sweep variable.

2. In LTspice, plot V_{ref} vs R_L and record the R_L at which the curve is 95% from the maximum voltage (use your best estimation in getting R_{Lmin}).

3. Repeat steps 1 and 2 for $R_{ref} = 470\Omega$ and $R_{ref} = 1k\Omega$.

4. Use $R_{ref} = 100\Omega$ and replace the load resistor with its corresponding R_{Lmin}.

5. Run a transient simulation and record the values of ΔV_o, $P_{supplied}$ and P_{load} in Table 1 (**HINT:** Instantaneous $P_{supplied}$ and P_{load} can be obtained by holding the **"alt"** key and clicking on the component to be measured. Average power can be obtained by holding **"ctrl"** button and clicking on the name of the measurement in the waveform window.)

6. Repeat steps 4 and 5 for $R_{ref} = 470\Omega$ and $R_{ref} = 1k\Omega$.

Table 1: Performance of a Diode based Voltage Regulator

$\mathbf{R_{ref}}$	Fig. 3 (b)			
	ΔV_o	$P_{supplied}$	P_{load}	R_{Lmin}
100 Ω				
470 Ω				
1 kΩ				

7. From the table above, briefly explain what happens as a larger $\mathbf{R_{Ref}}$ is used.

3.4   Practical Diode Usage

3.4.1   Rectifier

In this last section, we will look at some more practical uses of diodes. One of the most common uses of a diode is a rectifier: a device that removes the negative voltages at the output. The circuit for a half-wave rectifier is shown below.
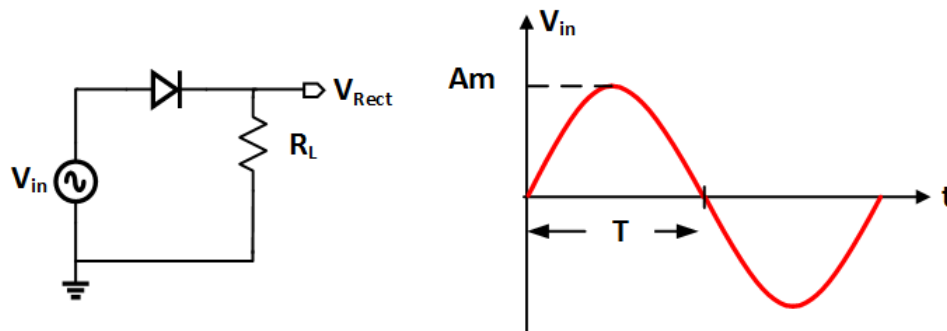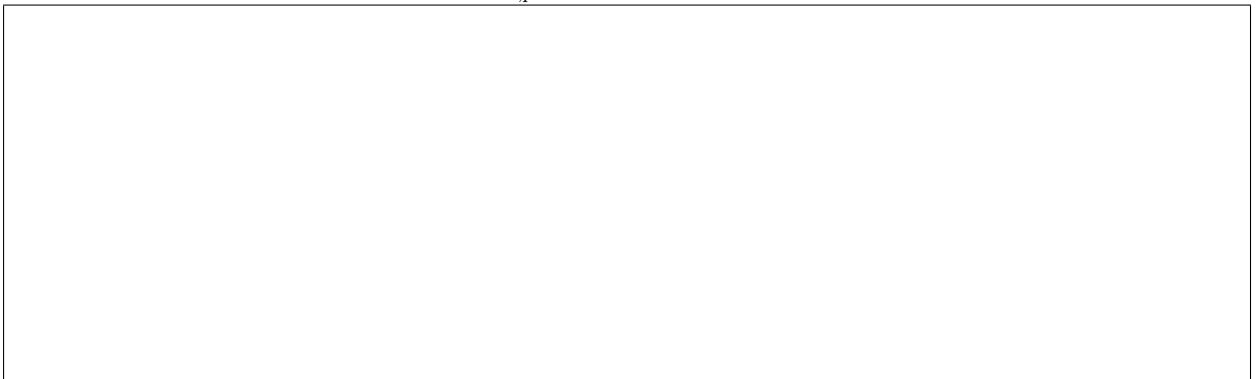

Figure 5: Input sine wave V_{in}. Draw in the rectified output V_{Rect} above.

1. Analyze the circuit shown in Fig. 5 and draw its output below for the sinusoidal input shown in Fig. 5. Assume $V_{FWD} < V_{in,peak}$.

2. Perform a **transient** simulation on the circuit shown in Fig. 5. Use an input voltage of $\mathbf{V_{in} = 2.5\sin(2\pi 60t)}$ and $R_L = 470\,\Omega$. Plot V_{Rect} and V_{in} vs time on the same graph. Save a screenshot of your results.

In many cases, rectifiers are used to convert AC to DC. We were able to get closer to DC by using a half-wave rectifier to remove the negative voltages as shown in the previous section. However, the output of this circuit still has large ripples and harmonics. To remove these ripples, we know from the previous lab that we can add a capacitor in parallel to the output to act as a low-pass filter (removing higher frequency content).

You may also think about the output waveform as charging and discharging the capacitor. The charging of the capacitor follows the output voltage waveform up to the peak voltage, but the discharging will be slower and occur at a rate of the RC-time constant. This half-wave rectifier with a capacitor is shown in Fig. 6 below.
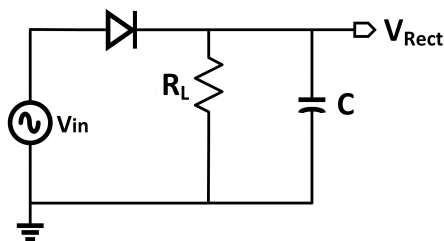

Figure 6: Half wave rectifier with a capacitor

3. Perform a **transient** simulation on the circuits shown in Fig. 6 for the different values of resistors and capacitors in Table 2.

4. Note down the average DC value of the output voltage (V_{DC}) and peak-to-peak output ripple voltage (V_{ripple}). Also calculate the RC-time constant τ. Take a screen shot of **one** simulation result.

Table 2: Performance of a Half-wave rectifier

C	$R_L = 470\Omega$			$R_L = 4.7k\Omega$		
	V_{DC}	V_{ripple}	τ	V_{DC}	V_{ripple}	τ
10μF						
22μF						
47μF						

5. How does the choice of load R_L and capacitor C affect average DC value and peak-to-peak ripple?

4   Measurement Data

List of Components Required:

- Diode: 1N914/1N4148 [x2]
- Capacitors : 10μF, 22μF, 47μF
- Resistors: 100Ω, 470Ω, 1kΩ, 4.7kΩ
- 5 kΩ Potentiometer

In this section, you will build the circuits from Section 2 on a breadboard and repeat the measurements done earlier.
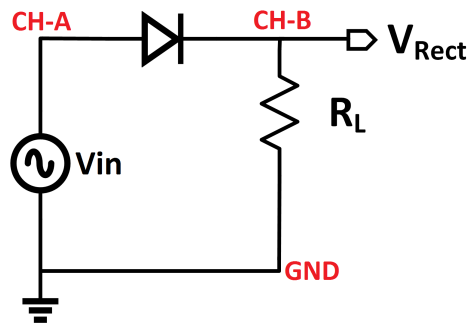
4.1   Diode Characterization

Figure 7: Half-wave rectified circuit

Component Values:

$$\text{Diode : 1N914/1N4148, } R_L = 1k\Omega$$

1. Assemble the circuit shown in Fig. 7. The labels in red are the connections to the ADALM1000.

2. In **ALICE Desktop**, configure **AWG CHA** in **SVMI** mode with a 1kHz **Sawtooth** shape with **min** value set to 0 and **max** value set to 5.

3. In **Curves**, choose **CA-V** and **CB-V**. Press **Run**

4. Export the data into MATLAB. The suggested method is shown below.

    - Save the data into a .csv file by pressing **File → Save to CSV**.
    - Read in the data into MATLAB by using **csvread(filename,1,0)**.

5. Plot I_D vs. V_{Diode} using the "scatter" function in MATLAB (rather than "plot"). Note: You will have to use the imported data to calculate I_D and V_{Diode}.

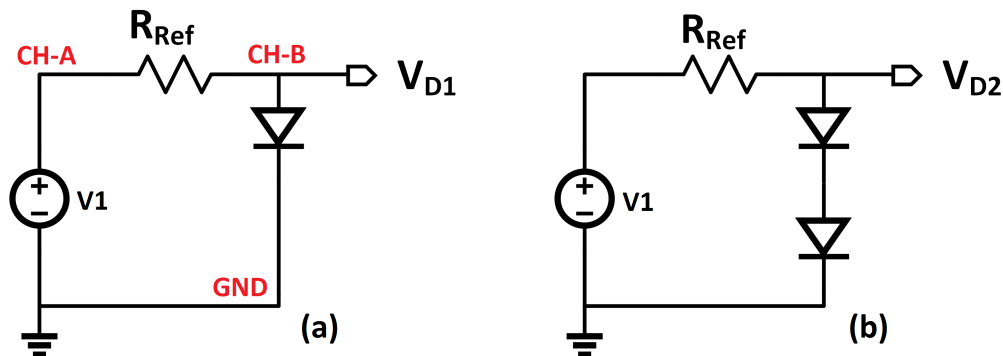6. What is V_{FWD} of the diode?

4.2 Diode as a Reference

Figure 8: Diode as reference

1. Assemble the circuit shown in Fig. 8 (a). Use $R_{ref} = 1\,k\Omega$.

2. In **ALICE Desktop**, configure **AWG CHA** in **SVMI** mode with a 100Hz **Sine** shape with **min** value set to 3.5 and **max** value set to 4.5. By using this setting, ADALM1000 will produce a signal with $\mathbf{V_{in} = 4 + 0.5sin(2\pi100t)}$.

3. Set **Time mS/Div** to **5**, **CB V/Div** to **0.01** and **CB V Pos** to **0.6**.
   (**NOTE:** This setting only serves as a guideline so you may need to modify it accordingly)

4. In **Curves**, choose **CA-V** and **CB-V**. Press **Run**.

5. Use **Meas** function in **ALICE Desktop** to obtain the variation in output voltage ($\mathbf{\Delta V_0}$). Take a screenshot of one of the plots.

	Figure 8 (a)	Figure 8 (b)
$\mathbf{\Delta V_0}$		

6. Repeat the steps above for circuit in Fig. 8 (b).

7. How do these results compare with your simulation results from Section 3.2?

4.3 Diode reference test circuit

Use the same ALICE Desktop setup as in the previous section:

1. Measure and record the average output voltage (V_{D1} and V_{D2}) without a load using the **Meas → Avg** function in **ALICE Desktop**.

$\mathbf{R_{ref}}$	Fig. 8 (a) $V_{D1,avg}$ (V)	Fig. 8 (b) $V_{D2,avg}$ (V)
100 Ω		
1k Ω		

2. Attach a **5kΩ** potentiometer as a load to the output voltage as illustrated in Fig. 4.

3. Turn the knob of the potentiometer until the average output voltage starts to drop significantly. (You may use 5% drop of the average value measured in step 1 as reference.)

4. Measure and record the value of the potentiometer as R_{Lmin}.

5. Complete the measurement of the parameters stated in Table 3 using the value of R_{Lmin} found in previous step.

   - P_{load} can be calculated by using R_{Lmin} and $V_{out,avg}$ (**-CB-V→Avg**) in **Meas**.
   - $P_{supplied}$ can be calculated using $V_{in,avg}$ (**-CA-V→Avg**) and $I_{in,avg}$ (**-CA-I→Avg**) in **Meas**.

Table 3: Performance of a Diode based Voltage Regulator

$\mathbf{R_{ref}}$	Fig. 8 (b) ΔV_o	$P_{supplied}$	P_{load}	R_{Lmin}
100 Ω				
1 kΩ				

6. Compare and comment on the simulation and measurement results.

4.4 Practical Diode Usage

4.4.1 Rectifier

In this final section, we will be building and testing our rectifier circuit on the breadboard. Unfortunately, the ADALM1000 is not able to work with negative voltages so we will have to improvise by adding a DC voltage rather than working with GND.
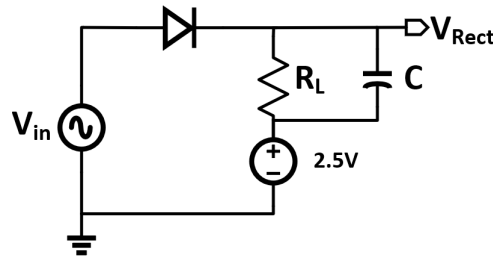

Figure 9: Half wave rectifier.

1. Assemble the circuit shown in Fig. 9 using values of R_L and C_L given in Table 4. Apply $\mathbf{V_{in}} = \mathbf{2.5 + 2.5sin(2\pi 60t)}$ from CH-A. The 2.5V voltage source at the output can be directly taken from the 2.5V DC pin of the ADALM1000.

2. Set the offset of **CA-V** and **CB-V** to 2.5 (on the bottom rightcorner of the window). This simply shifts the screen up so that 2.5V actually appears as the zero-point.

3. Set **CA V Pos** and **CB V Pos** to **0.0**, **Hold off** to **10.0**. In **Curves**, choose **CA-V** and **CB-V**. Adjust **CA V/Div**, **CB V/Div** and **Time mS/Div** so that the waveform fits nicely on the oscilloscope.

4. Use **"Meas"** to show the average DC value of the output voltage ($\mathbf{V_{DC}}$) as well as the peak-to-peak output ripple voltage ($\mathbf{V_{ripple}}$). **Run** and take a screen shot of **one** simulation result.

5. Repeat the steps above using different values of resistors and capacitors mentioned in Table 4.**Keep this circuit for demo in lab**

Table 4: Performance of a Half-wave rectifier

C	$\mathbf{R_L = 470\Omega}$		$\mathbf{R_L = 4.7k\Omega}$	
	V_{DC}	V_{ripple}	V_{DC}	V_{ripple}
$\mathbf{10\mu F}$				
$\mathbf{22\mu F}$				
$\mathbf{47\mu F}$				

6. Compare and comment on the simulation and measurement results.