ECE 343 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:

$$\mathrm{I}_D = 0 \text{ for } \mathrm{V} \leq \mathrm{V}_D$$
$$\mathrm{V} = \mathrm{V}_D \text{ for } \mathrm{I}_D \geq 0$$

This forms a "right-angle" curve on the current-versus-voltage $\mathrm{v} - \mathrm{i}$ 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 $\mathrm{V} \neq \mathrm{V}_D$ when $\mathrm{I}_D \neq 0$. The full semiconductor model of a diode is beyond the scope of this class. The $\mathrm{v} - \mathrm{i}$ characteristic for a practical diode can be approximated using the following equation:

$$\mathrm{I}_D = \mathrm{I}_S(e^{V_D/V_T} - 1) \approx \mathrm{I}_S e^{\mathrm{V}_D/V_T}, \text{ if } \mathrm{V}_D \gg \mathrm{V}_T$$

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. In figure 1 below, we plot an ideal model compared to a real one to show the difference.

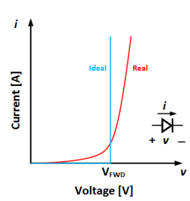

Figure 1: Ideal diode compared to real diode I-V curves.

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

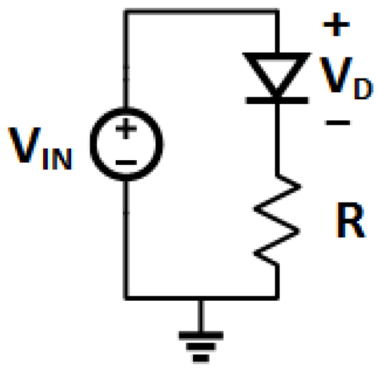

Figure 2: Diode characterization test bench

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

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

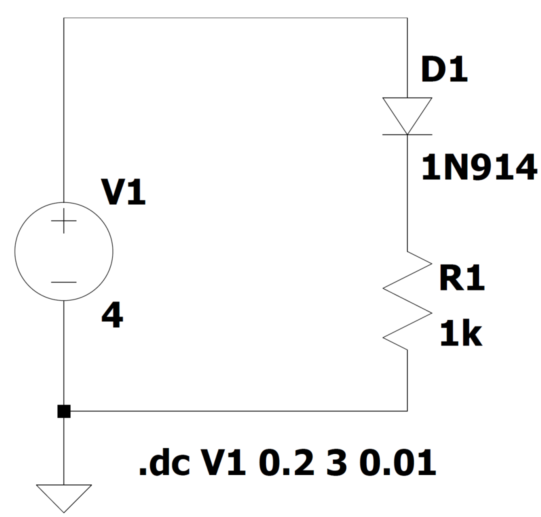

Figure Solution 1: Test bench circuit in LTSpice.

2. Perform a DC sweep simulation on the circuit. Use the following parameters:
    - Source to sweep: V_{in}
    - Type of sweep: Linear

- Start Value: 0.2V
- Stop Value: 3V
- 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 “Quality Plotted” box.

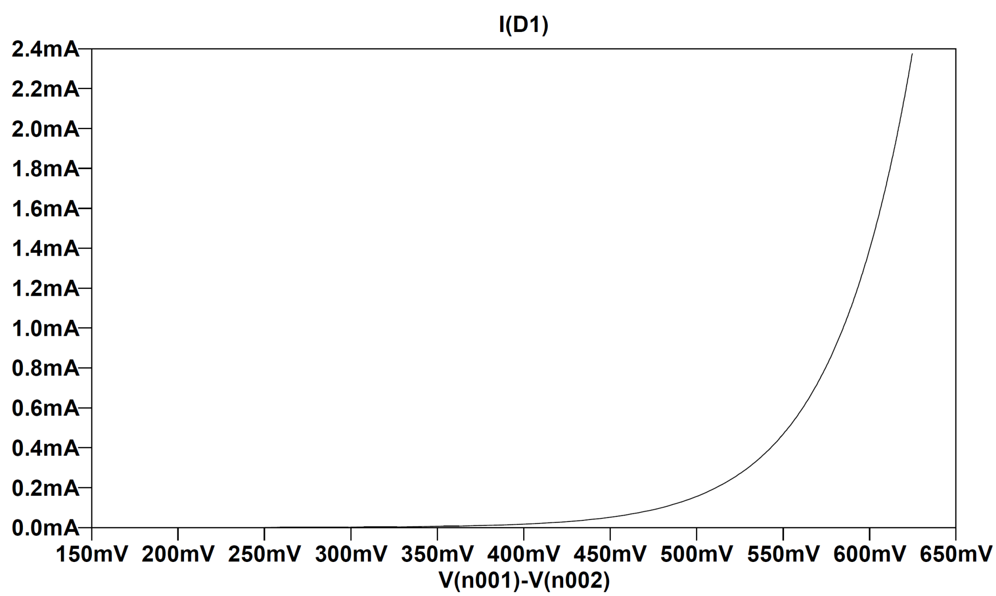

Figure Solution 2: Diode I-V characteristics

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.)

$V_{FWD} = 578.0\text{mV}$

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.

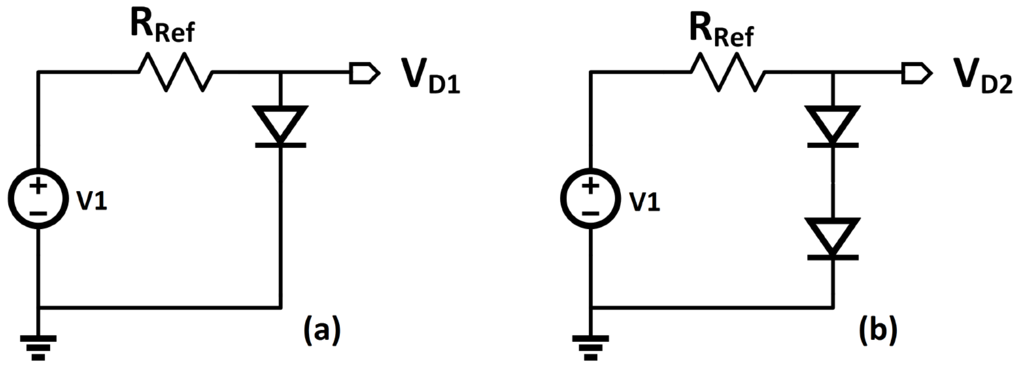

Figure 3: Left, one diode as a reference. Right two diodes as a reference.

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

$$V_{D1} = V_{FWD} = 578.0\text{mV}$$
$$V_{D2} = 2 \cdot V_{FWD} = 2 \cdot 0.578\text{mV} = 1.156\text{V}$$

2. Now assume that the source voltage has a sinusoidal noise component (i.e. $V_1 = V_{DC} + v_n \sin(2\pi \cdot 100t)$). Derive an expression for the incremental output voltage (ΔV_0) in terms of the diode incremental resistance $\mathbf{r_d}$ for **each** circuit.

The diode is essentially a resistor with resistance $\mathbf{r_d}$ in incremental analysis. So,

$$\Delta V_{01} = \frac{\mathbf{r_d}}{\mathbf{r_d} + R_{ref}} v_n;$$

$$\Delta V_{02} = \frac{2 \cdot \mathbf{r_d}}{2 \cdot \mathbf{r_d} + R_{ref}} v_n$$

3. Draw the circuits shown in Fig. 3 in LTspice. Let $V_1 = 4 + 0.5\sin(2\pi \cdot 100t)$V and $R_{ref} = 1\text{k}\Omega$. 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

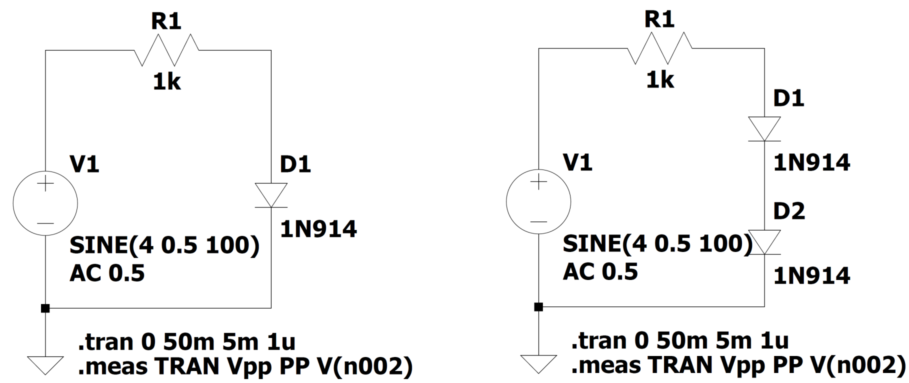

Figure Solution 3: Reference cirucit in LTSpice

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.

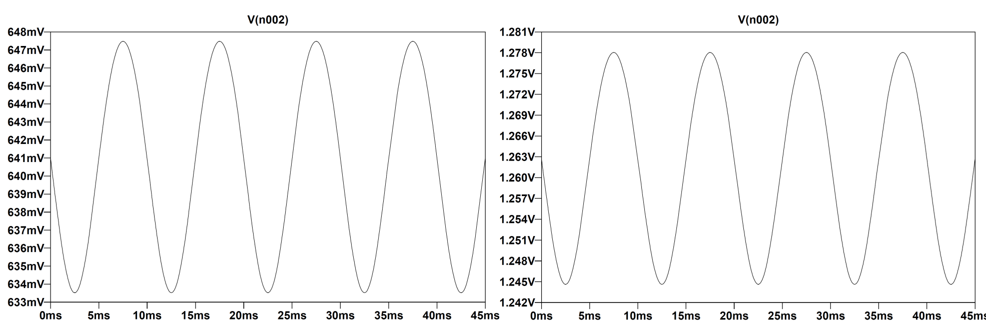

Figure Solution 4: Left is V_{D1}, Right is V_{D2}

$$\mathbf{\Delta V_{01}} = 13.96\text{mV}$$

$$\mathbf{\Delta V_{02}} = 33.42\text{mV}$$

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 3.1 acts under different load conditions.

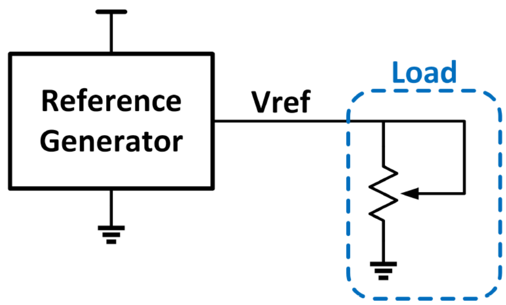

Figure 4: Simple reference generator measurement circuit

1. Draw the voltage reference circuit shown in 3(b) with a resistive load $\mathbf{R_L}$ as shown in Fig. 4. Set $V_1 = 4 + 0.5\sin(2\pi \cdot 100t)$ 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% of the maximum voltage (use your best estimation in getting $R_{L,\,min}$). Take a screenshot of your graph.

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

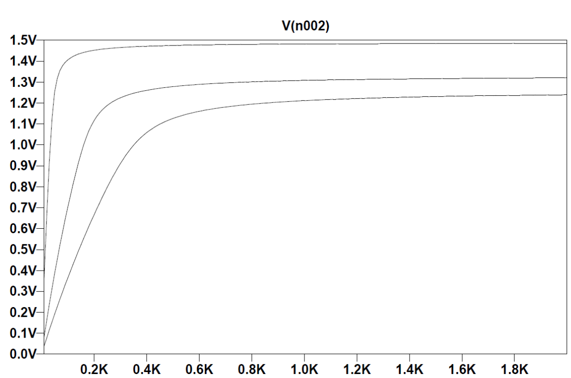

Figure Solution 5: The curve when `Rref = 100Ohms, 470Ohms, 1kOhms` (from bottom to top)

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

5. Run a transient simulation and record the value of $\mathbf{\Delta V_0}$, 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 $R_{\text{ref}} = 470\Omega$ and $R_{\text{ref}} = 1\text{k}\Omega$.

Table 1: Performance of a Diode based Voltage Regulator

$\mathbf{R}_{\text{ref}}$	ΔV_0	P_{supplied}	P_{load}	$R_{L,\min}$
100Ω	72.96mV	−103.38mW	18.21mW	110Ω
470Ω	68.66mV	−23.35mW	4.148mW	380Ω
$1\text{k}\Omega$	71.81mV	−11.29mW	2.039mW	680Ω

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

When a larger $\mathbf{R}_{\text{ref}}$ is used, the P_{supplied}, and P_{load} decreases (since current is limited, of course.), and the $R_{L,\,\min}$ increases. The ΔV_0 doesn't seems to change, it always around 70mV.