Professor: Mike Wilson

EE 346 Section 04

3:10 - 6:00 PM

Experiment # 1

Introduction to PSpice

Group/Bench Number: 7

Written By:
Joey Wilson
Partners/Reviewers:
Aiku Shintani & Aodhan Arias

Introduction:

Objectives of the Experiment:

The objective of this experiment was to familiarize ourselves with PSpice and LTSpice by creating circuits and simulating input/output relations.

List of Equipment:

1. PSpice
2. LtSpice

Procedure:

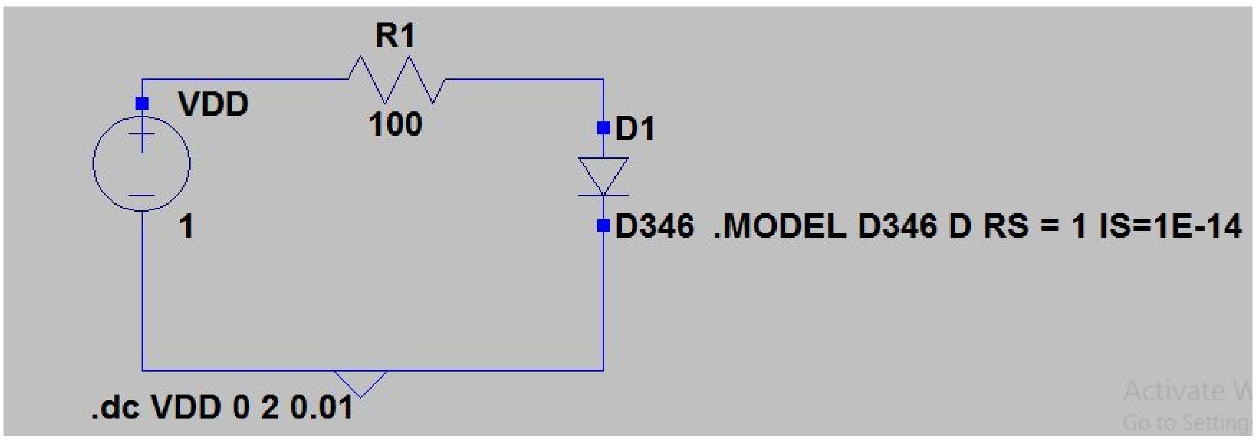

Figure 1: LTSpice Circuit Diagram for Part 1a

Part 1a:

The circuit in figure 1 above, consisting of a resistor and diode in series, was created using PSpice. Each component was modeled in a node net list entered as a text file. The resistor had a value of 100Ω and the diode had an internal resistance of 1Ω with a saturation current of 1E-14 A. The input voltage, VDD, was simulated as a sweep from 0 to 2V in increments of 0.01V. Probes were placed at nodes to record the current through D1 and R1, and the voltage across the diode. A simulation was then run and used to create graphs of current through the diode (Figure 2), voltage across the diode (Figure 4), and logarithmic current through the diode (Figure 3) with respect to the input voltage.

Part 1b:

A circuit comprised of an input and three diodes in parallel was then modeled using PSpice (Figure **X**). The input voltage, VDD, was again a sweep from 0 to 2V in increments of 0.01V. The diodes had internal resistances of 1Ω, but had saturation currents of 1E-14 A, 1E-13 A and 1E-12 A respectively. A simulation was then run from this node net list, and used to plot the current through each of the three diodes against the input voltage VDD.

Part 1c:

Both circuits in parts 1a and 1b including their plots were then replicated in LTSpice and compared to PSpice.

Data:

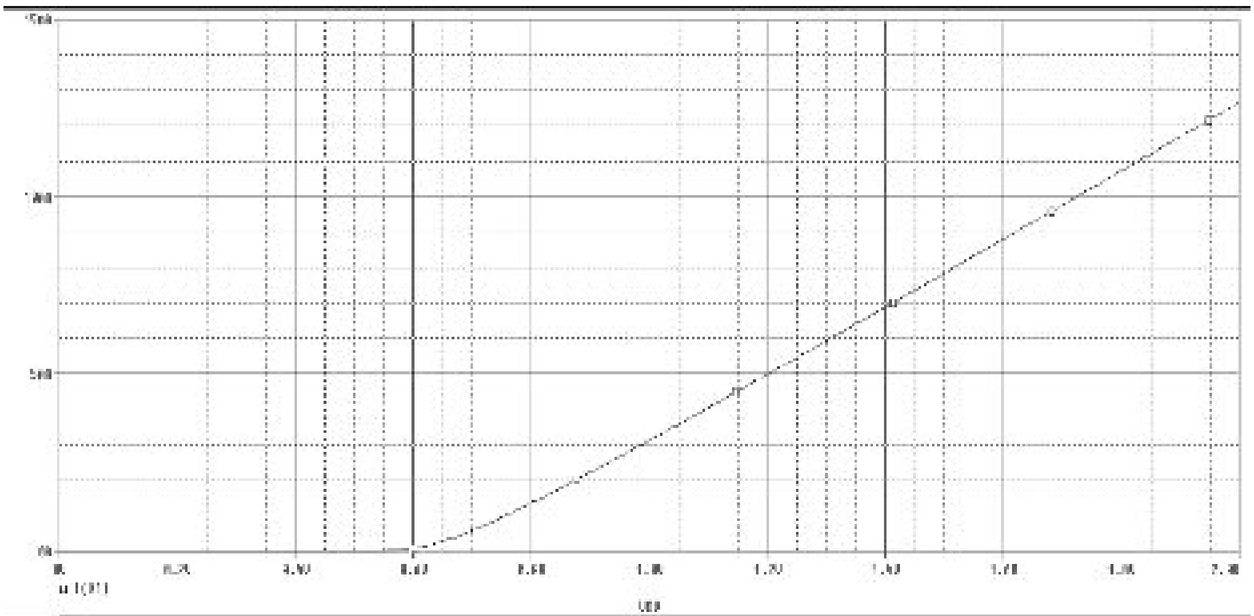

Figure 2: I vs. Vdd

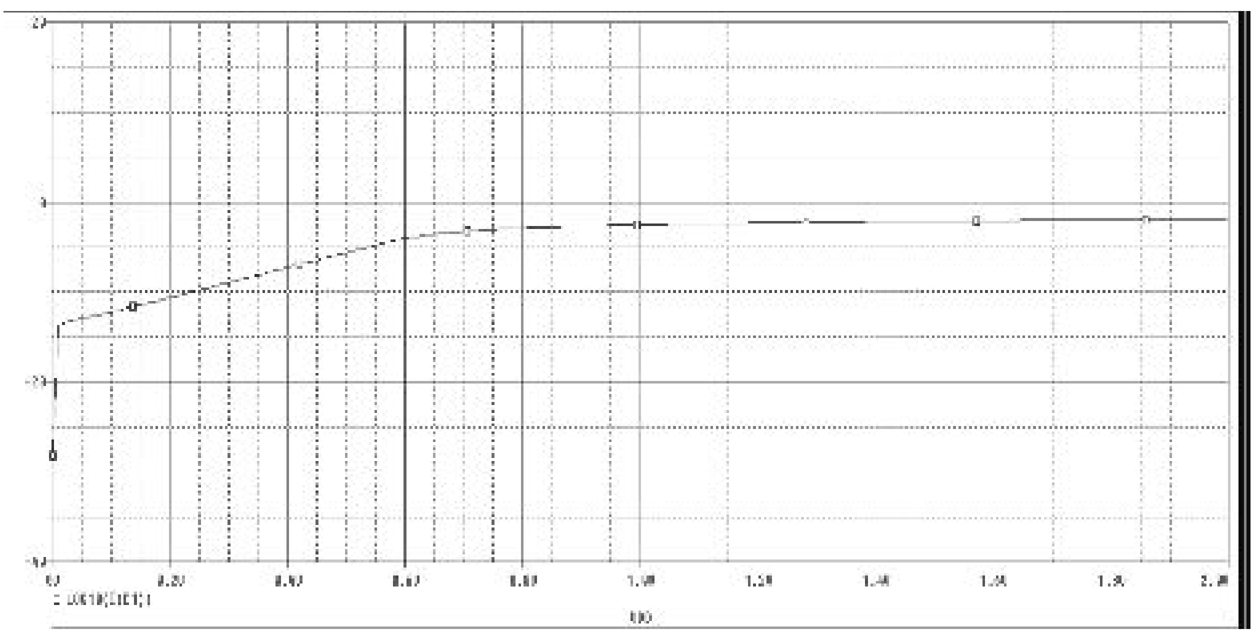

Figure 3: log10(I) vs. Vdd

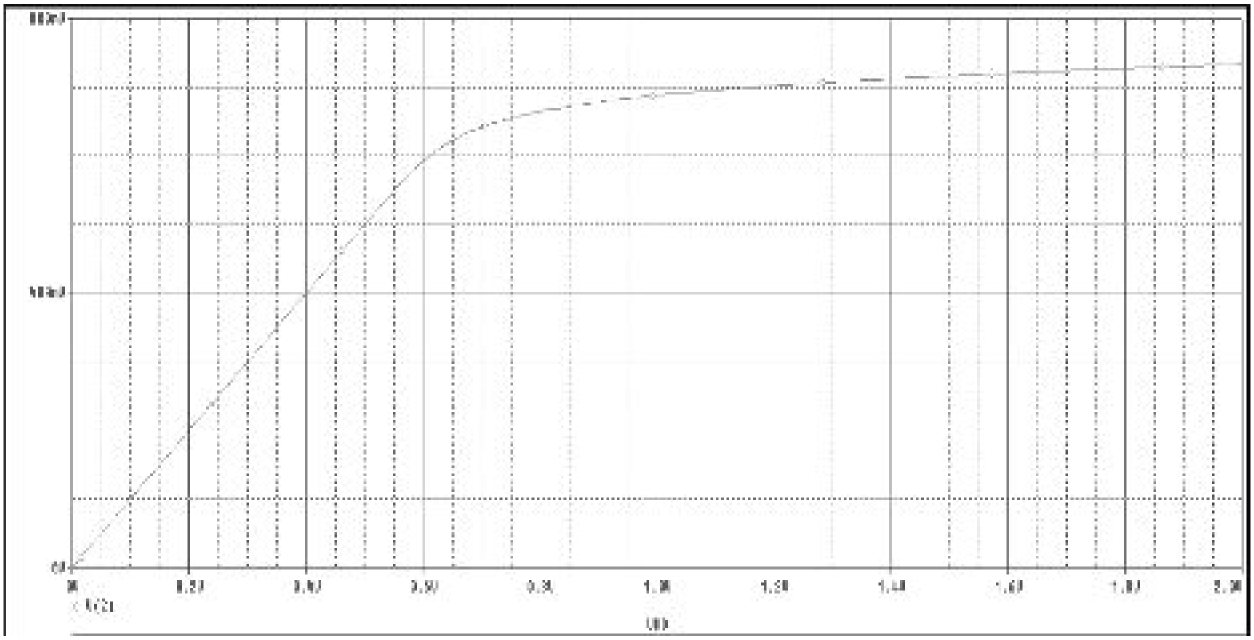

Figure 4: V2 vs. Vdd

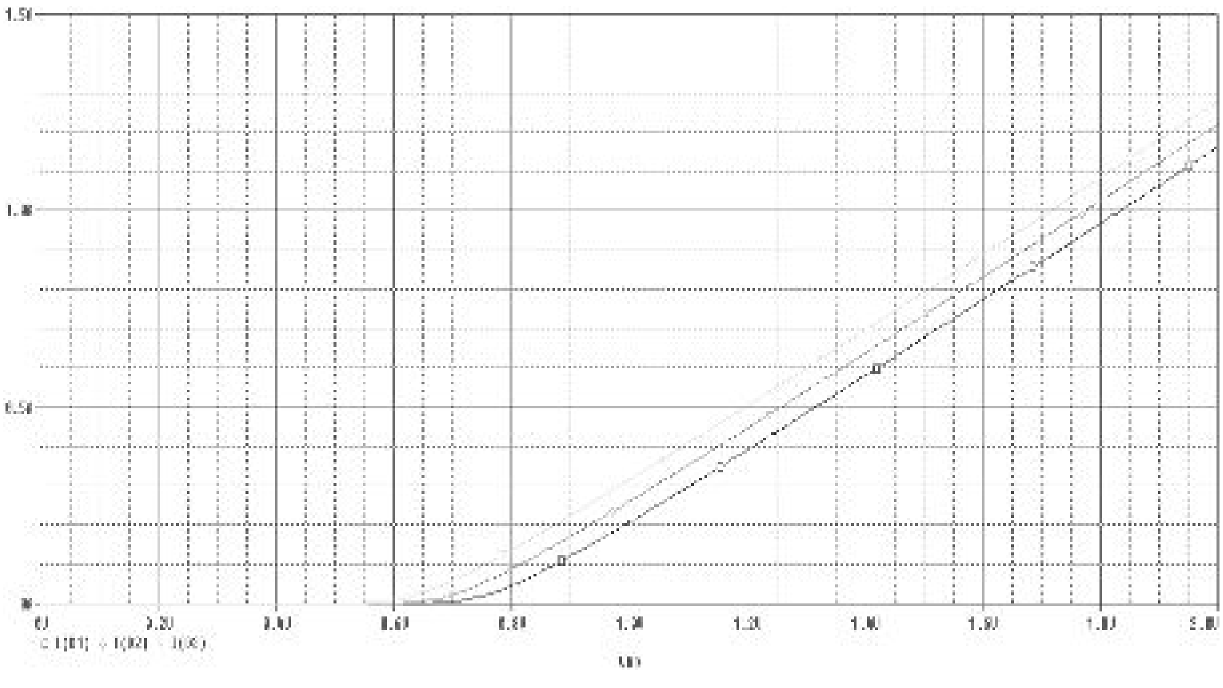

Figure 5: I(D1), I(D2), I(D3) vs. Vdd

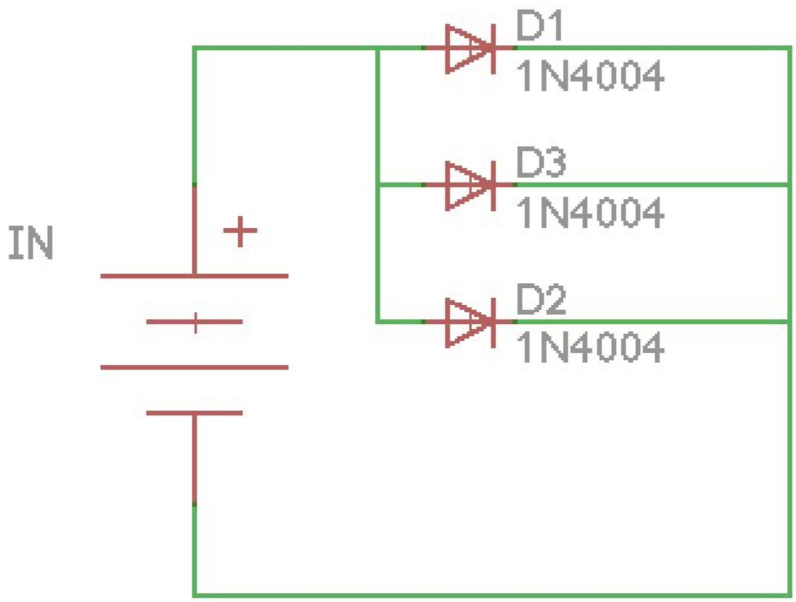

Figure 6: Input Deck 2 Circuit

Discussion Questions:

In section 1 of the lab, the current through the diode and resistor were found to be equal, which matches their theoretical relationship due to the components being connected in series. The current is exponential at first due to the characteristics of the diode transitioning from an open to a short-like circuit component, then becomes linear as the resistance of the circuit becomes approximately 100Ω. This relationship is shown in figure 3, the log scale of the current. As shown in figure 4, the diode voltage and input voltage have a linear relationship at first, with VDD equaling V2. This is because the diode starts with an extremely high resistance, causing most of the voltage to be dissipated by the diode. The curve smooths out at around 0.6V, at which point the dynamic resistance of the diode was similar to that of the resistor, causing more of the voltage to be dissipated by the resistor. As the voltage increases past 0.6V, the diode voltage increases more slowly since it has a much lower resistance, causing the circuit to have a resistance of about 100Ω.

In the second section of the lab, three diodes with different saturation currents were used. As expected, the diodes had different currents than each other as shown in figure 5. The currents appear to be offsets of each other, and seem like parallel lines at higher voltage values. The diodes with the lower saturation currents became saturated quicker, causing their voltage to increase at lower voltages. The larger current is the one with the saturation current of 1E-14 A, while the smallest current is the one with the saturation current of 1E-12 A. This circuit is shown in figure 6. The voltages across the diodes behave similarly to the voltage shown in input deck 1, which had only one diode. Without the resistor, it is easier to see the behavior of the diode as it isn't masked by the resistor.

Error Analysis:
This lab was done through simulation, so there were no components to contribute to error. Any error in this lab was extremely minimal, and either done by mis-entering a value, or rounding in LTSpice. No error was detected however.

Conclusion:

The purpose of this lab was to examine the voltage and current behaviors of diodes, and to gain more practice with modern engineering softwares LTSpice and PSpice. This lab was conducted entirely through simulation, so there weren't any errors to throw off our data. The diodes were shown to behave exponentially with current compared to voltage initially, then become linear at higher input voltages as they become closer to a short circuit. The voltage of a diode was shown to have a linear relationship at first, with VDD equaling V2. This occurred because the diode has an extremely high resistance, causing most of the voltage to be dissipated by the diode. At higher voltages, the diode voltage increases more slowly since it has a much lower resistance, which was evident in input deck 1. This lab was very beneficial in demonstrating the behavior of diodes and gaining more practice with simulating circuits.