Bilkent University

Electrical and Electronics Department

EE313-02 Lab 1 Preliminary Report:

“Diode Characterization and Differential Temperature Sensor”

28/09/2024

Fatih Mehmet Çetin - 22201689

Introduction:

This experiment consisted of two parts: A and B. In part A, we were asked to design a method to measure Is of a p-n diode, 1N4148. In part B, we were asked to design a differential temperature sensor using the temperature dependence of a diode forward voltage under constant current with certain specifications.

In the lab manual, we were given the following recommended circuit for part B **(Figure 1.1)**:

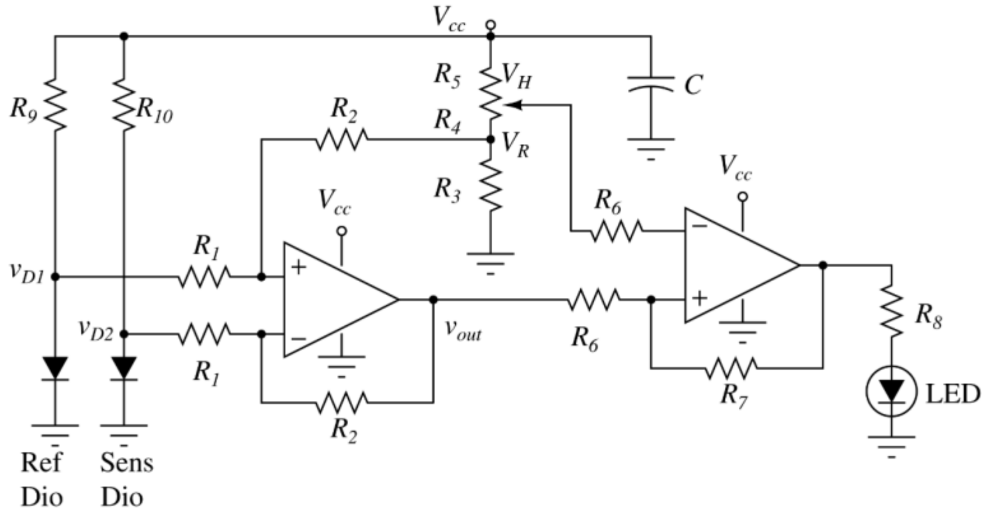

Figure 1.1: The Example Circuit in the Lab Manual for Part B

Simulation&Analysis:

Part A:

In this part of the experiment, we were asked to design a method to measure Is of a p-n diode, 1N4148. The mathematical relationship between I_s and I_d is given in (1&2) where n is 1.752.

$$I_D = I_S \cdot \left(e^{\left(\frac{V_d}{n \cdot V_T}\right)} - 1 \right) \qquad (1)$$

$$V_T = \frac{kT}{q} \qquad (2)$$

The LTSpice circuit and simulation for Part A can be seen here **(Figures 2.1&2.2)**:

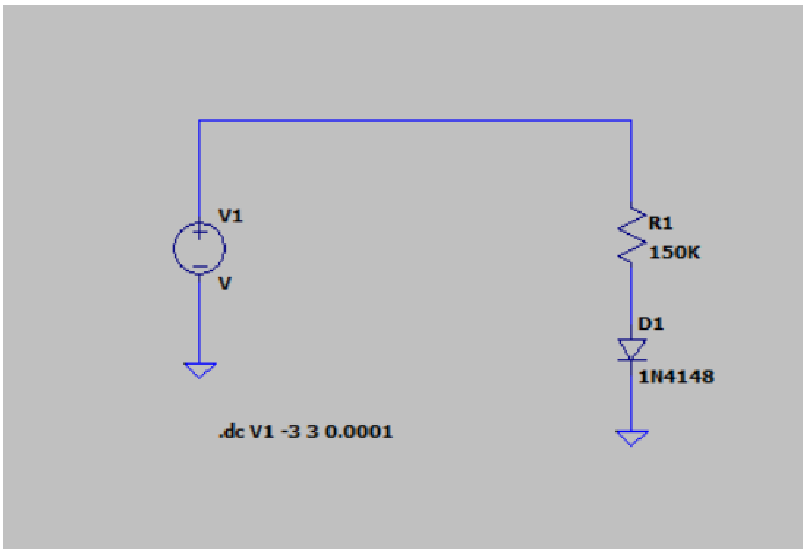

Figure 2.1: The LTSpice Circuit for I_s Analysis

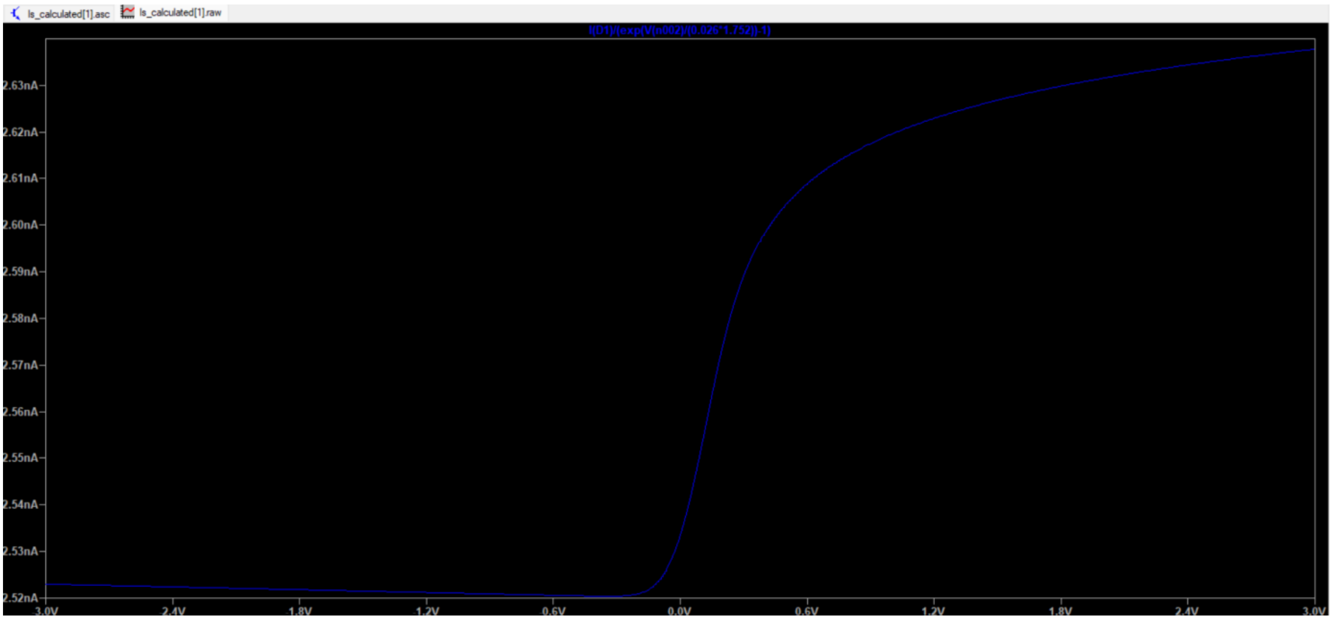

Figure 2.2: The LTSpice Simulation for I_s

It can be observed in the results that reverse bias region has 2.52nA I_s, as expected.

Part B:

In this part of the experiment, we were asked to design a differential temperature sensor using the temperature dependence of a diode forward voltage under constant current with certain specifications.

Since my Bilkent ID number ended with number 9, the DC supply voltage I used was 14V. Here is the overall circuit I used for the part B of the experiment **(Figure 2.3)**:

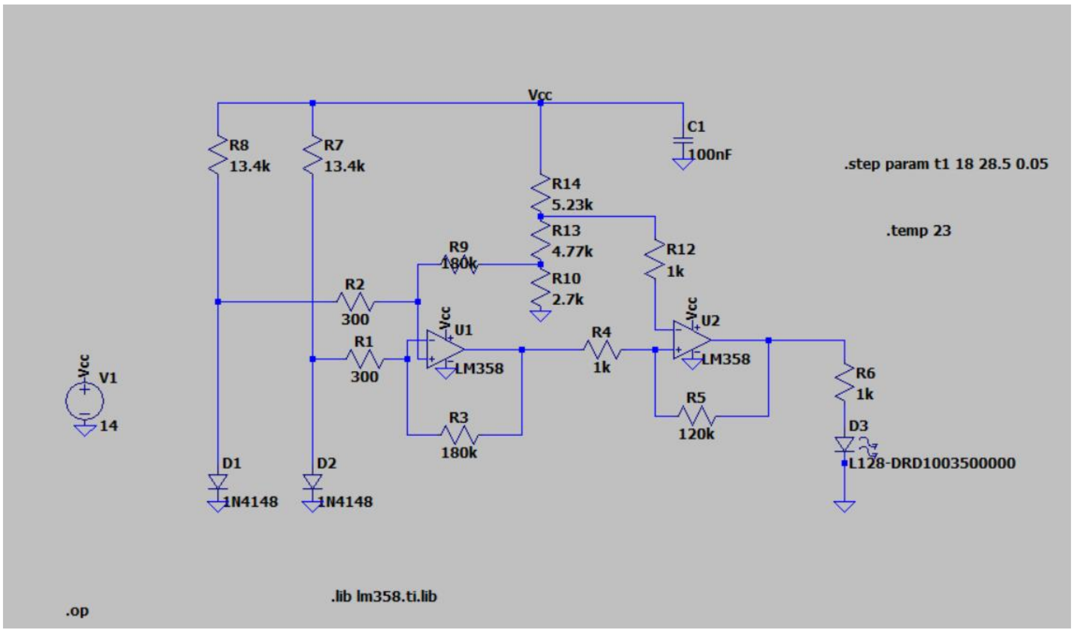

Figure 2.3: The overall circuit I used for the part B of the experiment

There are 4 requirements that the circuit should met. These were given in the lab manual **(Figure 2.4)**:

1. When both diodes are at room temperature (v_{D1}=v_{D2}), the output voltage, v_{out}, should be at (V_{cc}−2)/4
2. The output voltage should show the temperature difference between the room temperature and the temperature of the sensor diode in degrees with a 10% tolerance. For example, if the sensor diode is +1-degree warmer, v_{out} should change by +1V.
3. A red LED should turn on when the sensor's temperature exceeds +5±0.5°C the room temperature.
4. The LED should never flicker around the thresholds: It should have a 0.1°C (0.1V) hysteresis.

Figure 2.4: The Requirements the Circuit should Meet

Here you can see the simulation results for each criterion **(Figures 2.5 to 2.7)**:

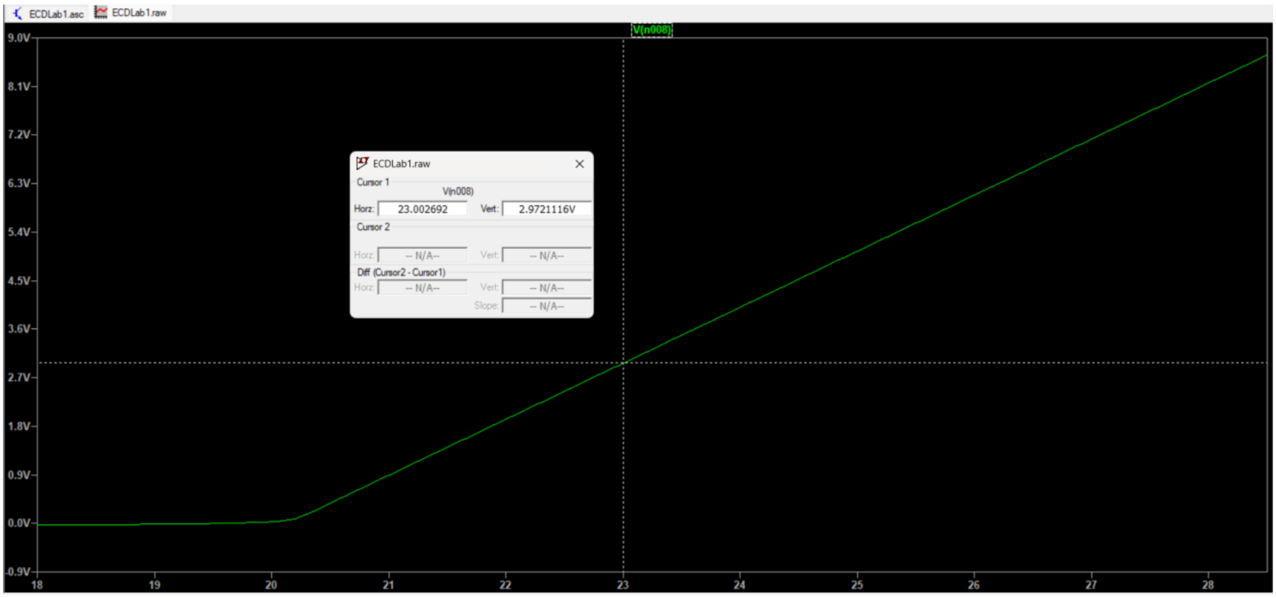

Figure 2.5 (Criterion 1): Output Voltage being 2.97 Volts as expected when both diodes are at 23°C

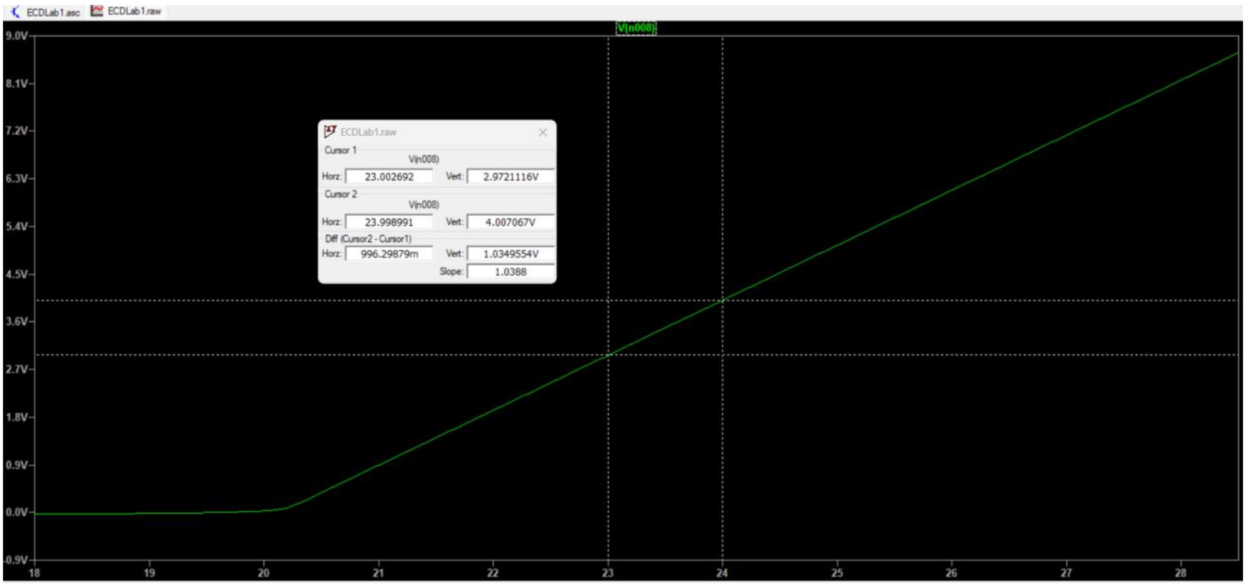

Figure 2.6 (Criterion 2): The Voltage Difference being 1 Volt when there is a 1°C Difference Between the Diodes as expected

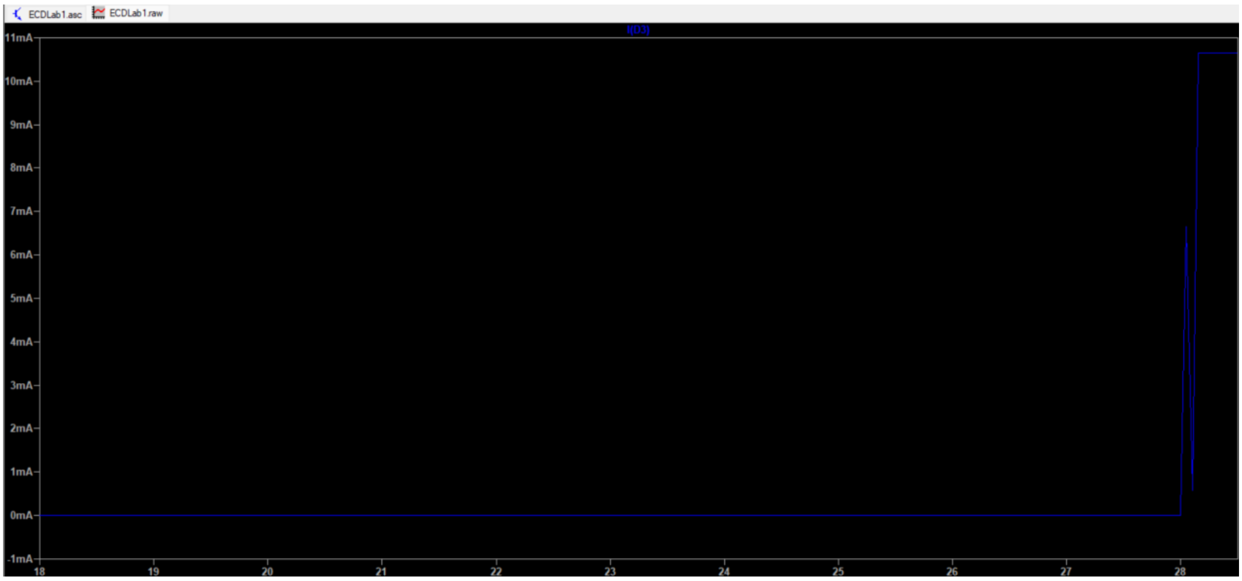

Figure 2.7 (Criterion 3): The current on the LED rising up when there is a 5°C difference between two diodes as expected – one is 23°C, the other is 28°C

For criterion 4, the equation for the hysteresis values given to us in the lab manual is given in (3):

$$(V_{cc} - 2) \cdot \left(\frac{R_7}{R_6}\right) = Hysteresis\, Value \qquad (3)$$

The values I selected for R7 and R6 are 120kΩ and 1kΩ respectively. Therefore, the hysteresis value is 0.1 Volts as required.

Here you can see the output voltage when two diodes have the same temperature value **(Figure 2.8)**:

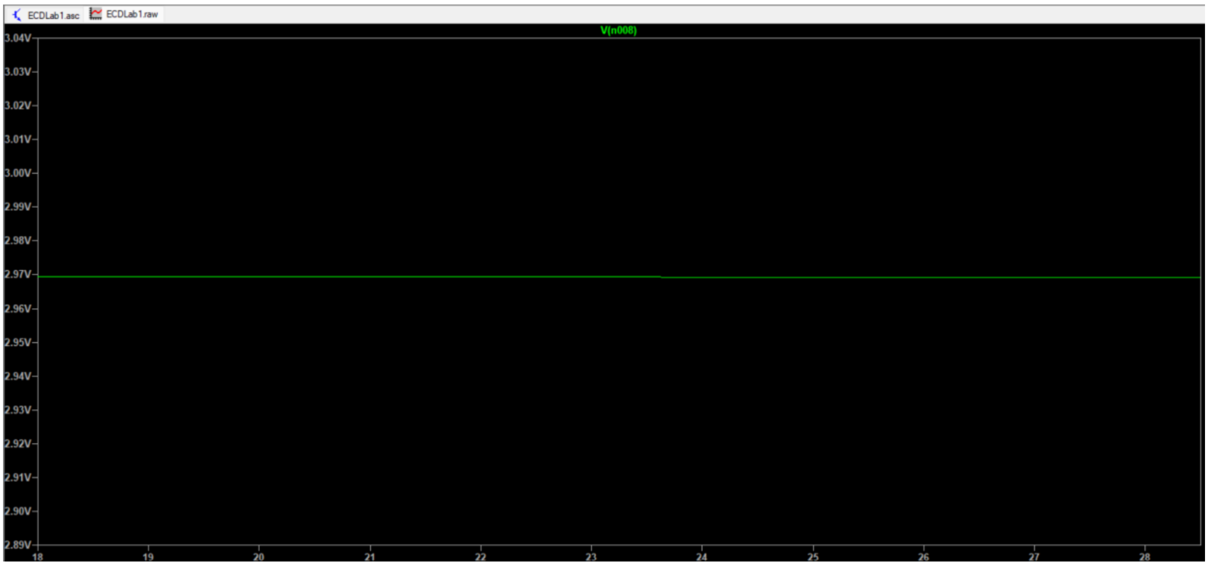

Figure 2.8: The Voltage Level is nearly 2.97 Volts for all temperatures between 18°C and 28°C.

Here you can see the Diptrace Schematic of my design as well as the component list **(Figure 2.9) (Appendix 1)**:

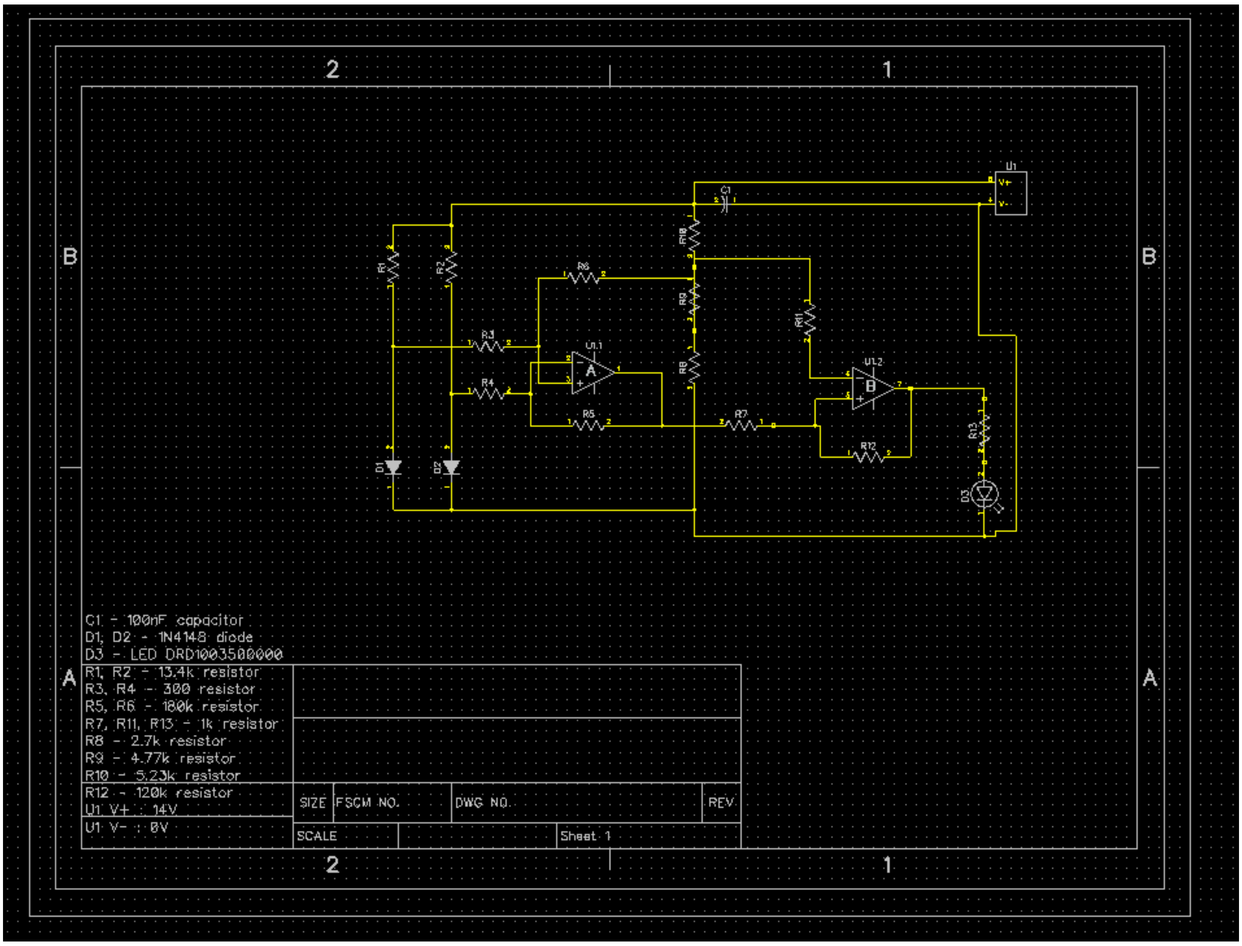

Figure 2.9: The Diptrace Schematic with the Component List of the Circuit

Conclusion & Thoughts on HW Part:

The preliminary work was successfully finished. The requirements were met both in part A and part B. The LTSpice & Diptrace softwares were used. Properties of diodes were learned and used practically. I believe this lab will help us a lot in our courses because we learned so much about diodes. Also, the temperature mechanics of LTSpice was totally unfamiliar to most of us before the lab, we learned how to use temperature as a variable in our simulations.

I believe the hardware implementation of this lab will be relatively easy but the precise measurements of the temperature of the diodes will be a hard challenge for both us and the TA's.

Appendices:

1- https://diptrace.com/books/tutorial.pdf