Emre Şaş

22102764

EEE 313-1

25.09.2024

**EEE 313 Laboratory 1 Preliminary Work**

## Introduction

In this lab, we will characterize the 1N4148 diode and exploit its temperature dependency in the forward voltage under constant current to create a differential temperature sensor. This experiment aims to construct a temperature sensor that can detect even the smallest temperature variations, in addition to finding a technique for measuring the saturation current (Is) of a p-n junction diode. We will adapt the given circuit that detects temperature variation by examining the forward voltage behavior of the 1N4148 diode and operational amplifiers (LM358). In addition, the lab investigates using a comparator circuit to turn on an LED whenever the sensor registers a temperature differential of more than +5°C from ambient temperature.

A) Design a method to measure $I_s$ of a pn diode, 1N4148. The diode current is given by

$$Equation\ 1:\ I_D = I_S\left(e^{\frac{V_D}{nV_T}} - 1\right)$$

where n=1.752 and $V_T = \frac{KT}{q}$. For 1N4148, the maximum current should never exceed 50mA.

- $I_s: Reverse - biased\ saturation\ current$
- $V_T: Thermal\ Voltage =\ 0.026\ V$
- $n =\ 1.752$

By using the given diode current formula (Equation 1).  We can derive a formula for $I_s$.

$$Equation\ 2:\ I_s = \frac{I_D}{\left(e^{\frac{V_D}{nV_T}} - 1\right)}$$

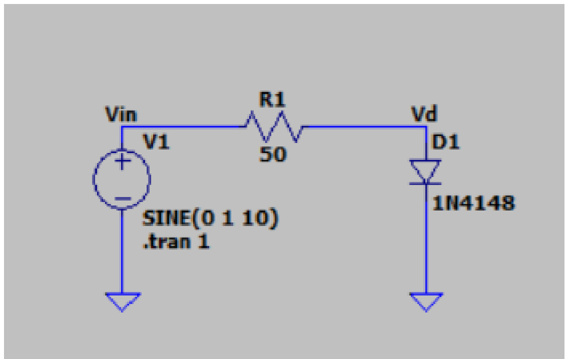


Figure 1: Test Design

System is designed to measure the $I_S$ of p-n diode (Figure 1).

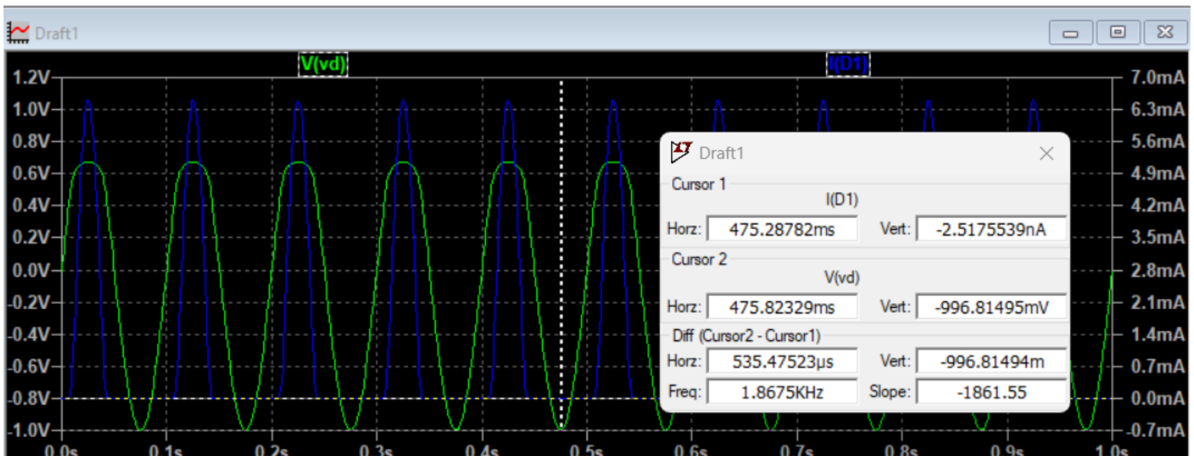


Figure 2: Measurements

As it is seen from Figure 2 , when the voltage at its lowest (-1V), the diode is reverse biased. The $I_s$ of the 1N4148 diode is shown by cursor 1 which is nearly 2.52 nA. When we calculate the parameters according to given formula:

- $V_T = 0.026\ V$
- $n = 1.752$
- $V_D = 672\ mV$
- $I_D = 6.42\ mA$

Therefore,

$$I_s = \frac{I_D}{\left(e^{\frac{V_D}{nV_T}} - 1\right)} \approx 2.52\ nA$$

B) In this part we are already given a generic circuit. The differential temperature sensor circuit described in the lab uses the temperature-dependent characteristic of diodes (in this case 1N4148) to detect temperature changes. The core idea is constructed on the fact that the forward voltage of a diode decreases as its temperature increases, and this principle is used to measure the temperature difference between two points. Reference diode is accepted to be 23 degree which is the room temperature. Since this is only a preliminary work, we only work on simulation apps which are DipTrace and LTspice.

After giving a short description about the circuit we can focus on the tasks which will ease our designing process. We will handle the tasks in the given order except we will deal with the 4$^{th}$ task before 3$^{rd}$ one since we will be finding a ratio which is related to some other equation in 3$^{rd}$ task. Tasks are listed below:

1. Using LTSpice, show that when both sensors are at 18°C and 28°C, the output voltage is nearly (Vcc−2)/4 ±0.3V.

2. Using LTSpice, show that the output voltage, vout, increases by 1±0.1 V/°C by sensor temperature when the room temperature is at 23°C

3. Using LTSpice, show that LED voltage goes high when the sensor's temperature increases to +5±0.5°C above the room temperature of 23°C. You can use the L128DRD1003500000 component in LTSpice diode library with a current limiting resistor, R8, in series. LED is assumed to be on when a current flows through it.

4. Using LTSpice, show that LED voltage goes low when the sensor's temperature is reduced to 0.1°C below its turn-on point.

Before digging into the tasks we should find the $V_{cc}$ to use. In order to find our personal $V_{cc}$ we should use the given formula: mod(BilkentID,5)+10V. [mod(xx,5) is the remainder of xx after division by 5.]

Therefore, the $V_{CC}$ = 14V for this experiment.

Since we now know the Vcc we can also find $R_9, R_{10}$ by using the equation. $I_D$ is 1mA

R9=R10= (Vcc−0.6)/(ID)

$$R_9 = R_{10} = \frac{(14 - 0.6)}{10^{-3}} = 13.4K\Omega$$

Let's begin with the first task. We should satisfy the given condition when both diodes' temperature are the same. Their voltage ($V_{D1}$ and $V_{D2}$) levels must be the same. When they are equal $V_{out}$ equation turns into this:

$$V_{out} = (R_2 / R_1)(V_{D1} - V_{D2}) + V_R = V_R$$

By calculating the $V_{out}$ according the given formula in task 1 we can find the value of $V_R$.

$$V_R = \frac{(14 - 2)}{4} \pm 0.3V \rightarrow 2.7V \leq V_R \leq 3.3V$$

Since we know the value of $V_R$ (ideally 3 V) , we now can find $R_3$ by using the given equation:

$$V_R = \frac{V_{CC} R_3}{(R_3 + R_4 + R_5)} \rightarrow 3 = \frac{(14 \times R_3)}{(R_3 + 10(K\Omega))}$$

We assumed R4+R5= 10K ohm since the potentiometer that is going to be used is 10K ohm. Therefore, $R_3$ = 30/11 KΩ≈2.7KΩ (Further adjustments may be made for fine tuning). When the left side of the circuit it can be foreseen that the current that will pass through $R_1$ and $R_2$ is so small ( micro-amperes) since all the current coming through the Vcc line will go through diode.

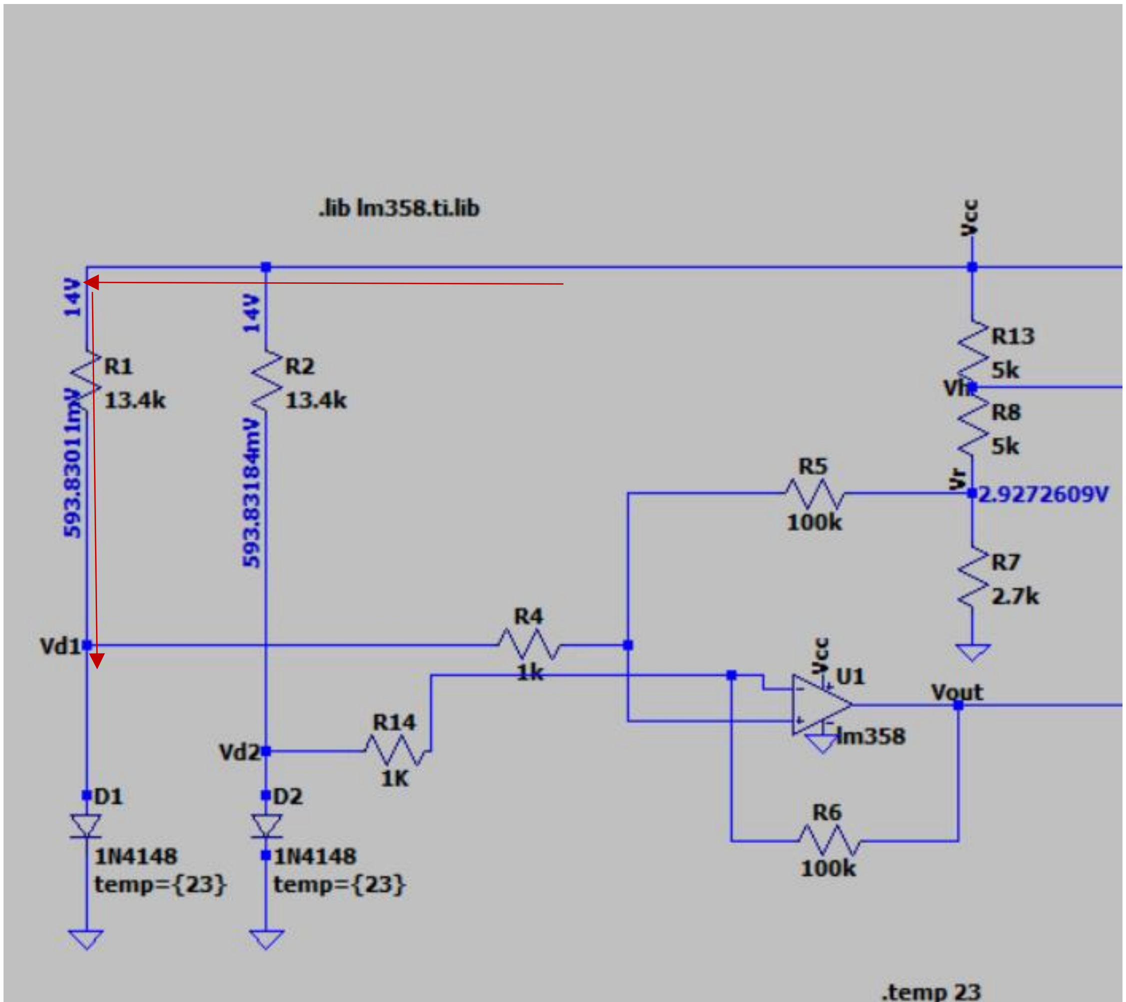


Figure3: Vcc line and Id

So the gain rate will be big. First, I gave 1K ohm to R1 just to ease the process. R1 and R2 values are not important , but their ratio is. My first guess about their ratio was 100 but it was not enough. After some trial and error ,I found out that when $\frac{R_2}{R_1} = 565$ $V_{out}$ is around 2.99 V which is enough, but since we don't have resistors valued 565KΩ I rounded it down to 0.56MΩ.

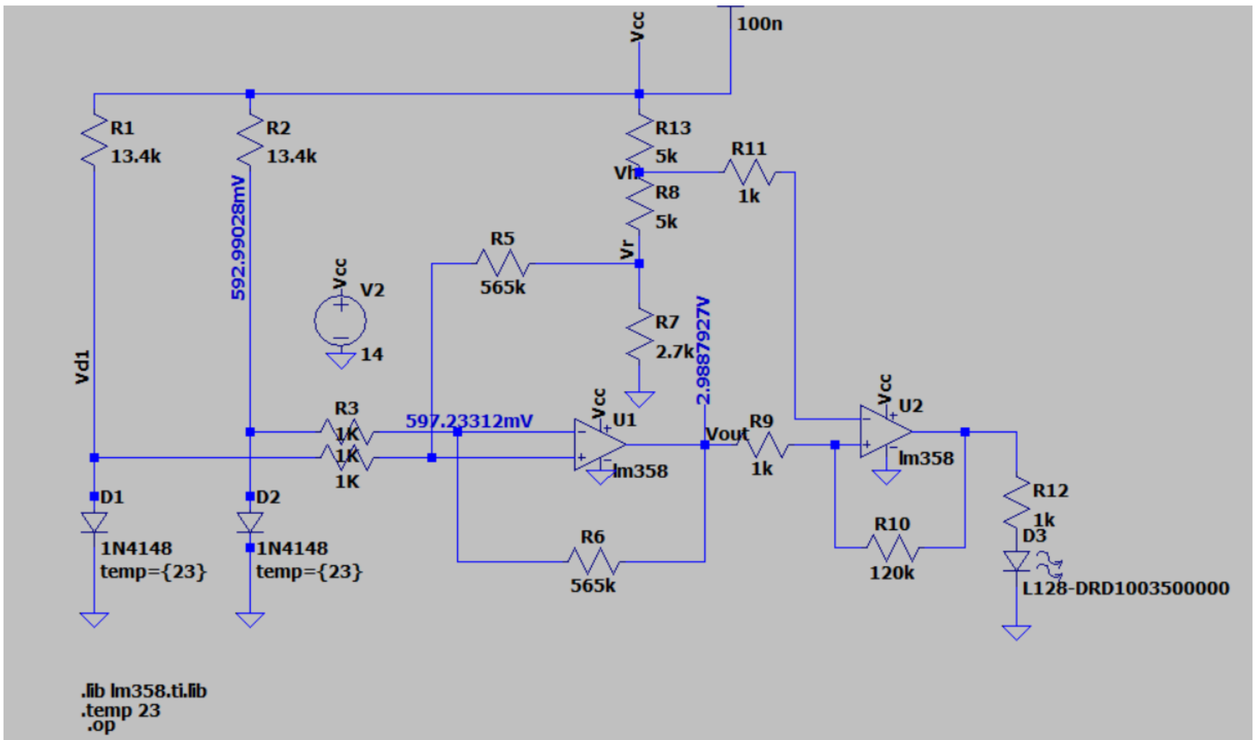


Figure 4: Whole circuit and Vout when diodes' temperatures are the same

After completing task 1 we can continue with the second task which is related to voltage temperature relation. Although we have found the value of $\frac{R_2}{R_1}$ by trial and error, we again try to find the best value for this ratio. While I am researching about the voltage-temperature characteristic of 1N4148 diode I found out that a silicon diode (such as the 1N4148) experiences a forward voltage decrease of about 2 mV/°C as the temperature rises while maintaining a constant current. However, this change ratio changes at extreme temperatures. For instance, our diode produced a sensitivity of 2.25 mV/K when the temperature is between 0 °C -65 °C [1].

By using the information above we can calculate the ratio of $\frac{R_2}{R_1}$. We want to get 1 V of amplification per degree. We now have 2mV (not exact so I experimented on the resistors to tune the circuit), but we need 1V so we can say that we need to amplify the voltage by 500.After arranging the gain rate, we should use the step parameter as recommended. The step parameter specifies the change in output voltage per degree Celsius of temperature variation. In this circuit, it is set to 1V/°C, indicating that the output increases by 1V for each 1°C difference. With a 10% tolerance, the expected range is 0.9V to 1.1V per °C. After adding the step directive parameter t is inserted into sens diode.

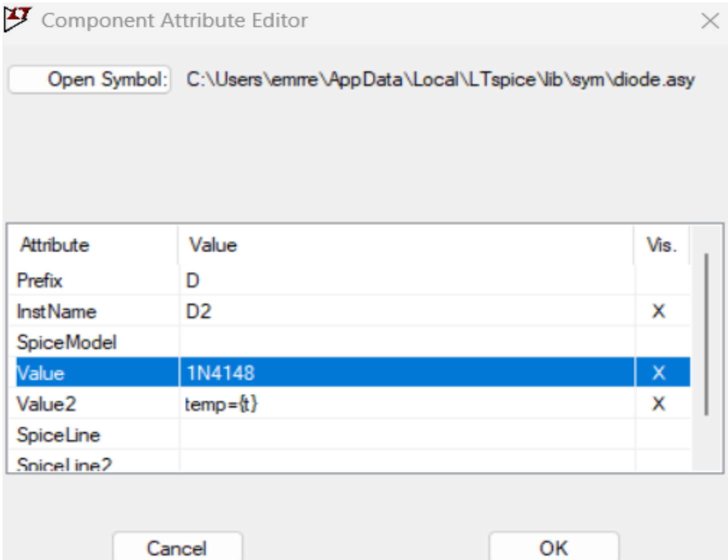


Figure 5: Adding parameter to diode

As it can be seen in Figure 6 slope is 1.07 which is acceptable

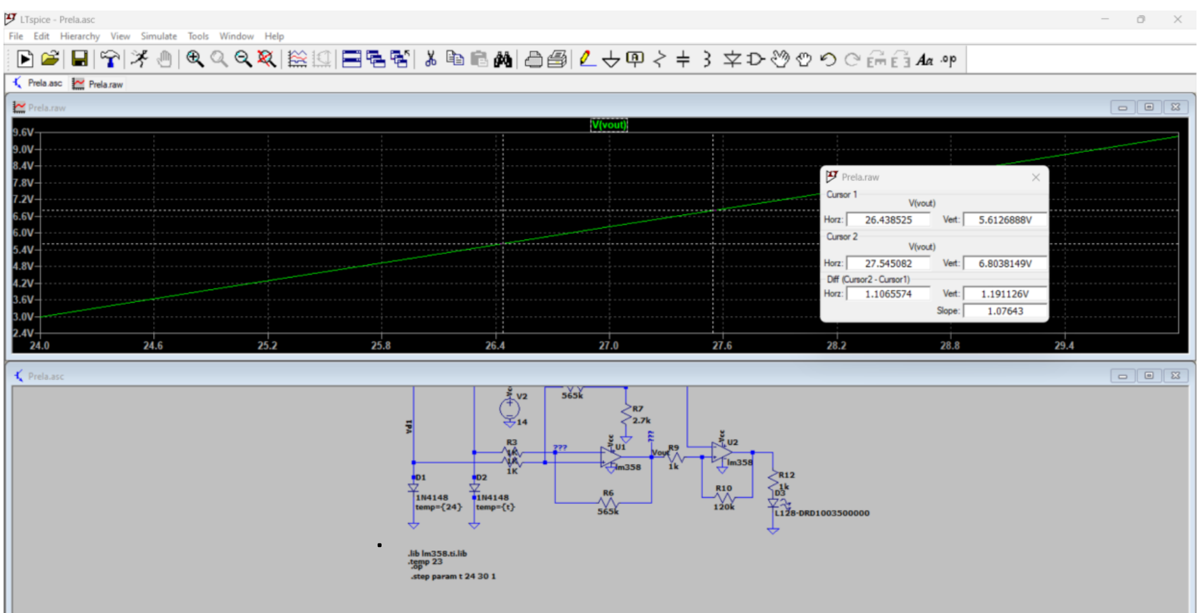


Figure 6: Voltage- Temperature Relation

Since we have completed the second task too we can continue with the third one. In the third task we are asked to turn on or turn of a LED depending on the temperature difference between the diodes. If the difference between them is more than 5C° LED should turn on. In our case 5C° difference creates 5V. The condition of LED is depending on two different equations (Figure 7) according to lab manual.

To turn on the LED, $v_{out} > V_H(R_7+R_6)/R_7$.

To turn off the LED, $v_{out} < V_H(R_7+R_6)/R_7 - (R_6/R_7)(V_{cc}-2)$

Figure 7: LED conditions depending on Vh

Before adjusting the resistor values I created a simple circuit to see what voltage level we need to open the LED. To find it I designed a simple circuit with piecewise linear voltage source (Figure 8 ).

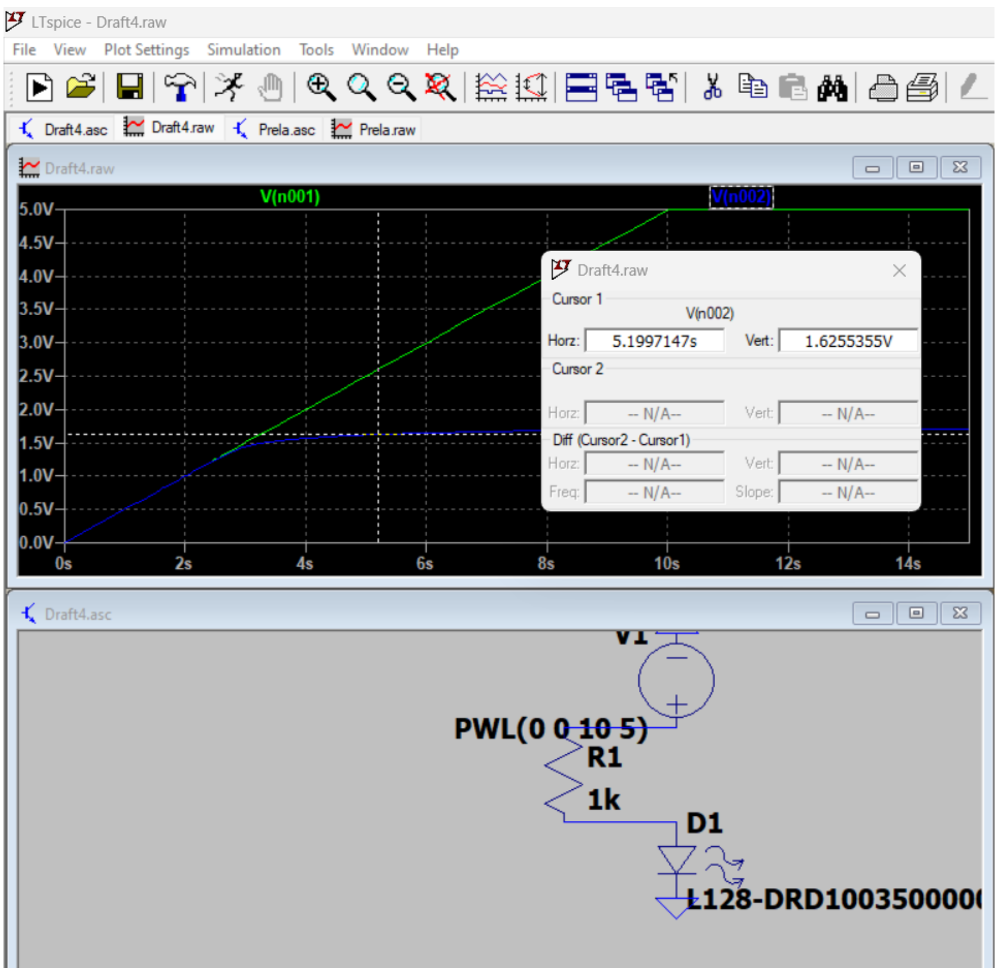


Figure 8: LED test circuit

We now have $V_{D\_ON} \approx 1.7V$, $V_H = \frac{V_{cc}(R_3+R_4)}{R_3+R_4+R_5}$ ( Equation 3)and the equations in Figure 7. Although these information are enough to make rough guess about $R_7$ and $R_6$ we should also consider hysteresis. In this context, hysteresis refers to the intentional delay or difference in the circuit's response to rising and falling temperature changes, specifically in the output voltage or the triggering point for the LED. As mentioned before dealing with the hysteresis value first will ease out job.

To find the hysteresis value simple equation were given:

$$\text{Hysteresis Value :}\frac{R_6}{R_7}(V_{cc}-2)$$

Since desired value is 0.1 V ratio of $R_6$ to $R_7$ is 1/120. Let's put this ratio in the equations in Figure 7:

$$To\ turn\ on\ the\ LED,\ V_{out} > V_H$$

$$To\ turn\ off\ the\ LED,\ V_{out} < V_H - 0.1$$

We can also see that hysteresis value is implied in these equations. In this case LED would not flicker when $V_{out} = V_H$. Although there are 4 parameters in equation 3 $R_3$ , $V_{cc}$ ,$R_4$ +$R_3$ are bounded by previous tasks so we can only play with $R_4$ and $R_3$ without changing their total resistance which is impossible since this is a potentiometer, and we are just turning a knob.

Let's find the last component's value ($R_8$) too and finalize the circuit then see the result of the last two tasks. The approximate value of $R_8$ is given as ($V_{cc}$-4)/0.01. Since our voltage source is 14V $R_8$ is 1KOhm. Complete circuit can be seen in Figure 9.

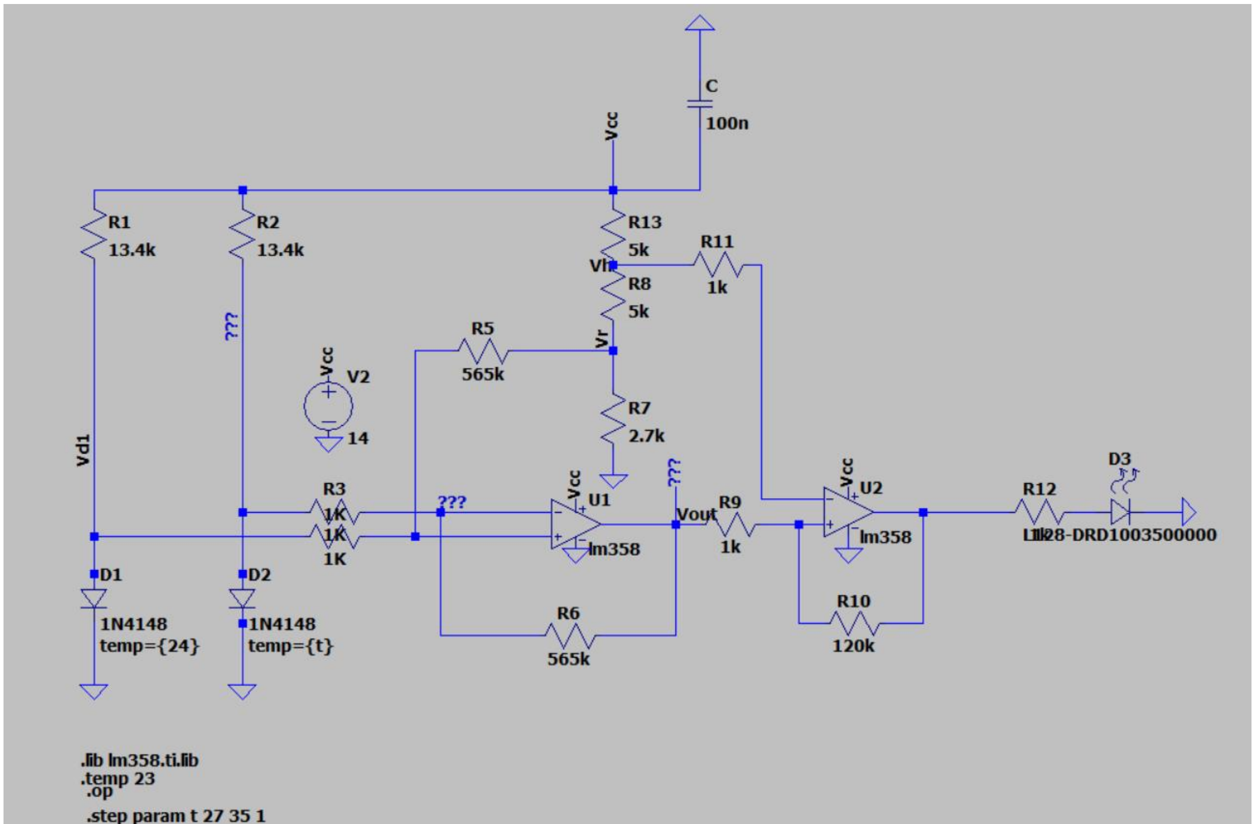


Figure 9: Finalized circuit with Spice directives

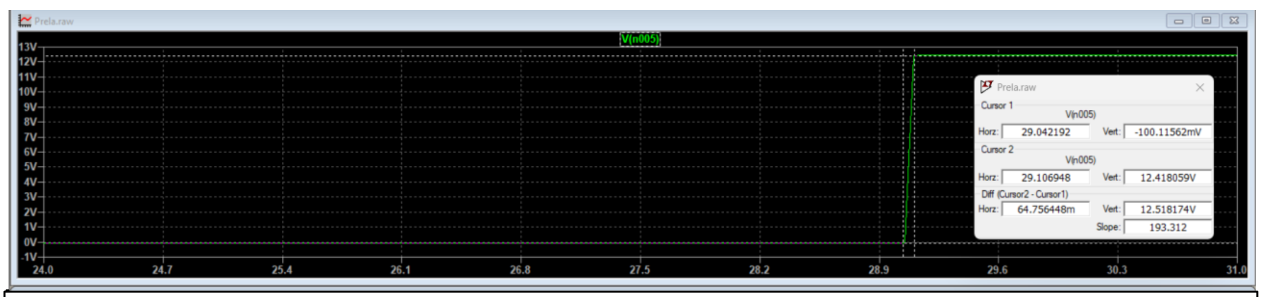

Figure 10: LED's reaction to temperature

As seen in Figure 10 and 9, when the reference diode is at 24C° diode does not react until there is a 5 C° temperature difference. To show the result of the final LTSpice task we need to manipulate the step directive a bit. (31-t) was written as parameter for the sens diode and the step directive updated as (.step param t 0 7 0.05). Therefore, when t=1.9 we want to see LED turning of and indeed we have the intended result (Figure 11)

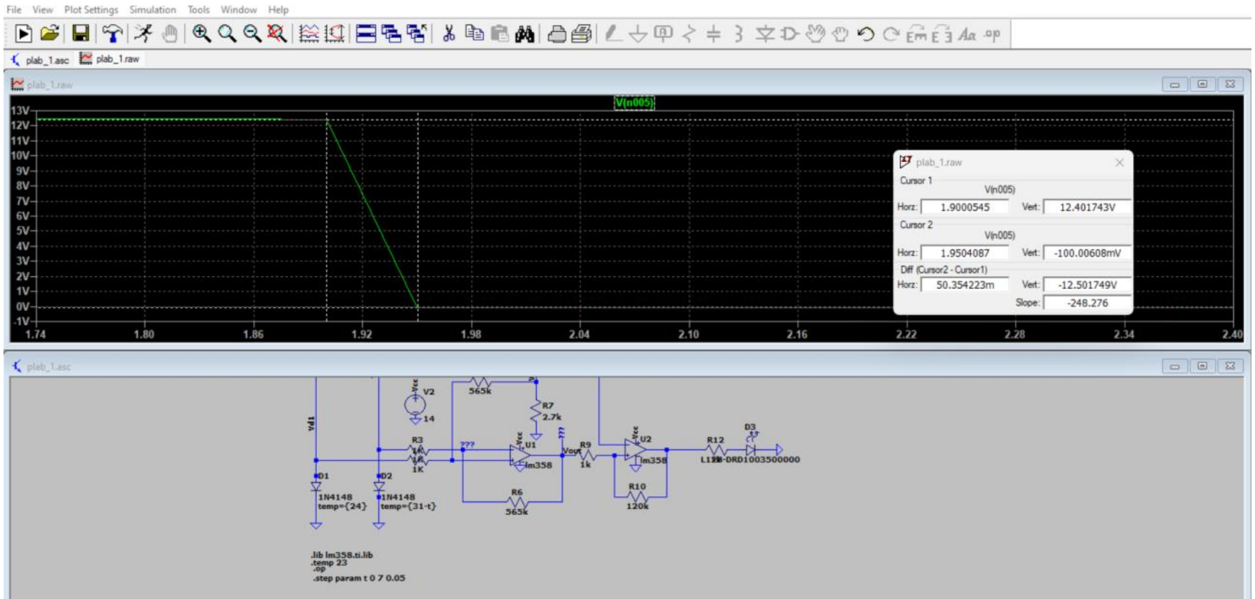

Figure 11: Hysterisis value

As the final step of this preliminary task, we should generate a schematic on DipTrace. DipTrace is PCB design software used for creating schematics, designing circuit boards, and generating files for manufacturing. Although finding the components and creating a schematic was fairly easy, there was only one thing that I had to tackle, and it is related to potentiometer. I could not find the schematic representation and footprint of potentiometer on DipTrace so I downloaded and imported the component from outer source [2].

By referring the DipTrace tutorials I created the schematic below:

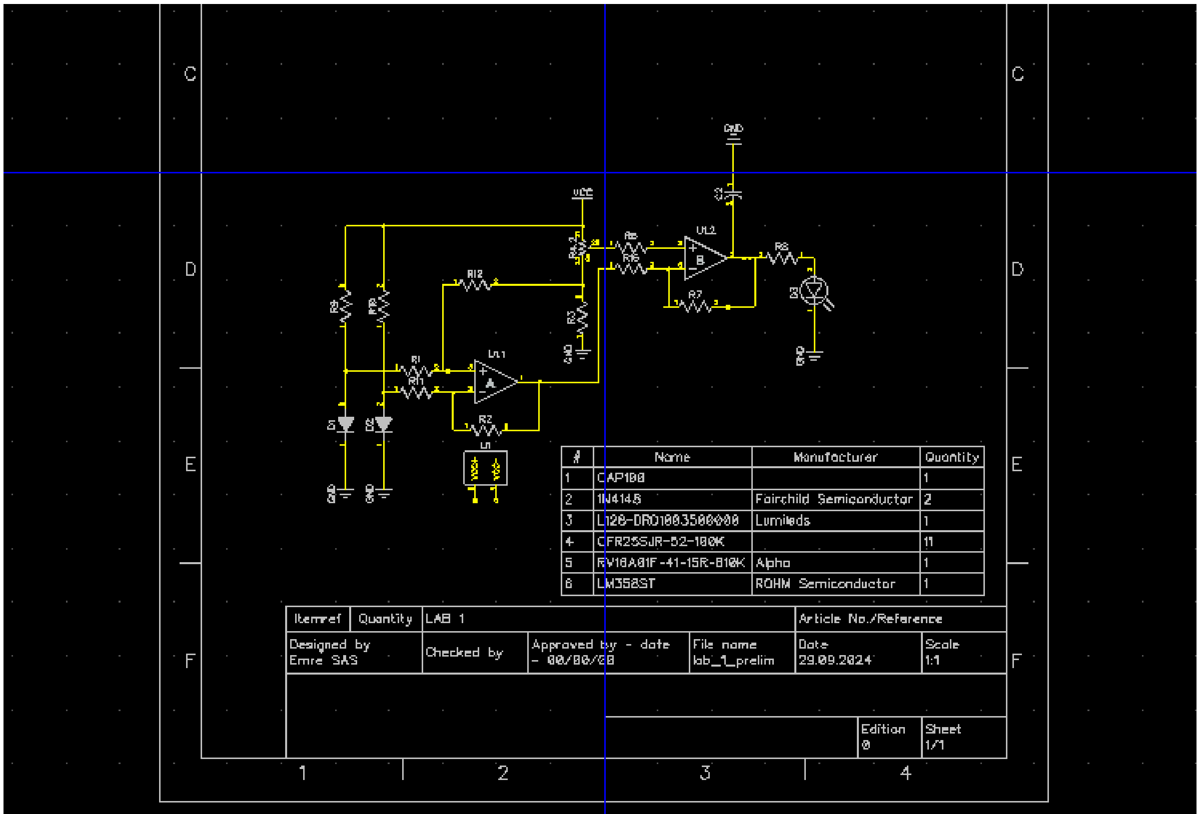


Figure 12: DipTrace Schematic and BOM

After creating the schematic, I also added the BOM list (Bill of Materials).

## References

[1]Kaitano Dzinavatonga, “Application of the 1N4148 to temperature measurements between -140 0C to 50C,” IEEE Sensors Journal, pp. 1–1, Jan. 2016, doi: https://doi.org/10.1109/jsen.2016.2614844.

[2] “HELP!! Looking for a part! - DipTrace Forum,” *Diptrace.com*, 2017. https://diptrace.com/forum/viewtopic.php?t=11673 (accessed Sep. 29, 2024).