Experiment #1 - SPICE Circuit Simulation and Equipment Usage

# Youssef Samwel

# [yo800238@ucf.edu](mailto:yo800238@ucf.edu)

# EEE3307 Electronics I

Section 0014

# Due Date 9/18/2023

# **Project Description**

# The students were tasked to construct a half-wave rectifier using a single diode on a breadboard. The students were also tasked to use the function generator (Tektronics AFG3022) to generate the sinusoidal voltage input source and to verify the circuit output voltage using the oscilloscope (Tektronics MSO 4034). The last circuit parameter to be measured was the current through the resistor.

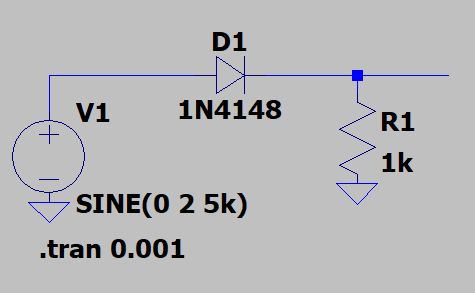
# **2.0 About Laboratory Day**

# The laboratory session took place on the Monday section between 6:00pm and 8:50pm on September 11, 2023. Regrettably, my lab partner was unable to attend the session, which meant I had to conduct the experiment independently.

# **3.0 Computer Simulation (SPICE)**

# We performed a simulation of circuit using LT-spice. The following was the input circuit given to the simulation software.

LT-Spice Circuit



We measured the voltage at the R1 resistor node; the following is the simulated waveforms.

Simulated Waveforms from LT-Spice

A graph on a black background

Description automatically generated

# **4.0 Experiment Procedure**

The initial step involved constructing the circuit using the components provided, which included a bag of diodes and a 1kΩ resistor. We connected the voltage source, diode, and resistor in series.

For the sinusoidal stage, as depicted in Figure 1, we utilized the function generator to produce a 4V peak-to-peak voltage source with a frequency of 5 kHz. Subsequently, we measured both the input voltage source and the output voltage using the oscilloscope.

Measurement Screenshot

A screen shot of a computer

Description automatically generated

In the DC voltage stage, as illustrated in Figure 2, we employed the power supply to generate two different input voltage sources: 2V and 1V. During this phase, our objective was to measure the current flowing through the resistor. Here are the measured and simulated results for this stage.

|  |  |  |
| --- | --- | --- |
|  | DC Voltage Source | Measured Current |
| Experiment | 2 V | A |
| Simulation | 2 V | A |
| Experiment | 1 V | µA |
| Simulation | 1 V | µA |

# **5.0 Observations and Simulation Comparison**

The simulation results for this lab have demonstrated a high level of accuracy, typically falling within a 2% margin of the measured values. It's important to note that any disparities between the measured data and simulated waveforms can be attributed to the tolerance levels of the electrical components employed in the lab setup.

One key factor contributing to these variations is the assumption made during simulation that the resistor had ideal characteristics, with no capacitance or inductance. In reality, resistors possess some level of capacitance and inductance. However, the primary source of deviation arises from the 5% tolerance in the specified resistance value. Although we selected a resistor rated at 1000 ohms, our measurements revealed an actual resistance of 983 ohms, resulting in a 1.7% deviation from the expected value.

Furthermore, it's worth noting that the SPICE model used in LT-Spice may not perfectly align with the real-world characteristics of the diode. Real diodes can have specifications that differ from those described by the SPICE model, leading to variations in the results.

Despite these differences between simulation parameters and physical parameters, the simulation has proven to be remarkably accurate in replicating real-world behavior and outcomes.

# We also noted that for enhanced measurement accuracy, it was beneficial to place the oscilloscope probes on parallel nodes when measuring voltage waveforms. This practice was employed to mitigate the impact of the pressure exerted by the oscilloscope probes, which could otherwise result in poor connections with the components on the breadboard. Implementing this technique significantly reduced noise levels observed on the oscilloscope, contributing to more precise measurements and a clearer signal representation.

# **6.0 Learned Objectives**

* Use of function generator
* Measurement using oscilloscope and practical probing.
* Simulation via LT-spice
* Diode performances and diode IV curve
* Half-wave rectifier
* DC current measurement

# **7.0 Conclusion**

In conclusion, our laboratory experiment focused on constructing and analyzing a half-wave rectifier circuit using LT-Spice simulation, diodes, and resistors. Through a series of steps involving sinusoidal and DC voltage stages, we gained valuable insights into the behavior of this circuit.

Our findings revealed that the simulation results closely matched the measured data, demonstrating a high degree of accuracy, typically within a 2% deviation. We attributed any disparities to the tolerance levels of the electrical components, especially the 5% tolerance in the resistance value, as well as the differences between real-world diodes and their SPICE models.

Furthermore, we learned the importance of careful probe placement on parallel nodes when measuring voltage waveforms with the oscilloscope. This technique helped alleviate the pressure applied by the oscilloscope probes, resulting in improved connections and reduced noise, thereby enhancing the accuracy of our measurements.

Overall, this laboratory experience provided valuable hands-on learning opportunities, deepening our understanding of electronic circuits, component tolerances, and the intricacies of simulation. It underscored the significance of meticulous measurement techniques in achieving reliable results and the practical application of theoretical knowledge in real-world scenarios.