Department of Electrical Engineering

Faculty Member: Qurat al Ain   **Dated:** February 15, 2024

Semester: 4th   **Section:** D

EE-215: Electronic Devices and Circuits

Lab 02: Introduction to PSpice and simulation using advanced features of PSpice.

		PLO4/CLO4		PLO5/CLO5	PLO8/CLO6	PLO9/CLO7
Name	Reg. No	Viva /Quiz / Lab Performance 5 marks	Analysis of data in Lab Report 5 marks	Modern Tool Usage 5 marks	Ethics and Safety 5 marks	Individual and Teamwork 5 marks
Haseeb Umer	427442					
Hanzla Sajjad	403214					
Irfa Farooq	412564					

Lab 02: Introduction to PSpice and simulation using advanced features of PSpice.

Objective

1. To gain working knowledge and to explore the features of PSpice the Simulation software.

EQUIPMENT REQUIRED

2. The following will be required in this lab experiment:

   - Lab PC
   - PSpice Software

Introduction to PSpice

3. PSpice is an acronym for **Personal Simulation Program with Integrated Circuit Emphasis**. Generally, an electronic simulation program uses mathematical models to replicate the behavior of actual electronic devices or circuits. Simulation software allows for modeling of circuit operation and is an invaluable analysis tool. Simulating a circuit's behavior before actually building it can greatly improve design efficiency by making faulty designs known as such and providing insight into the behavior of electronics circuit designs. As the student go along in the course the use of PSpice will become an essential tool for analysis, design, simulation and testing prior to circuit implementation on the breadboard and subsequently building of a Prototype.

4. PSpice is a powerful tool that allows you quickly obtain the complete list of voltages and currents for any given circuit. Moreover it can be used to display simulation results graphically.The graphing tool is really powerful and has many variations which will be helpful n understanding component characteristics, behaviour and performance under various stimulations and circuit conditions. The student is encouraged to explore various features of PSPICEand master its use.

5. During this lab the student will laso be using a document that has been dveloped by University of Pennsylvania titled *PSpice: A Brief Primer* . This is available as as a Adobe PDF file on your PC; in case you are unable to find it please contact the Lab Engineer, the Lab assistant or your instructor. Please read this document throughly especially it section 1 before proceeding ahead. The Lab Engineer will be conducting a Viva Voce during the lab and grade you accodingly.

6. The students are required to fill in various simulation results and graph that were generated during the corse of this lab as attachments to the LAB REPORT which will be submitted before start of next Lab.

EXERCISE 1

7. The first part of the lab is briefly given below:

   a. **Creating and Simulating a Circuit in PSpice.** For this exercise refer to section 2.1.1 – 2.1.3 and 2.2.1 of the primer. Using the tutorial provided as a guide, simulate the following circuit (figure 1) in PSpice and list down the required values mentioned.
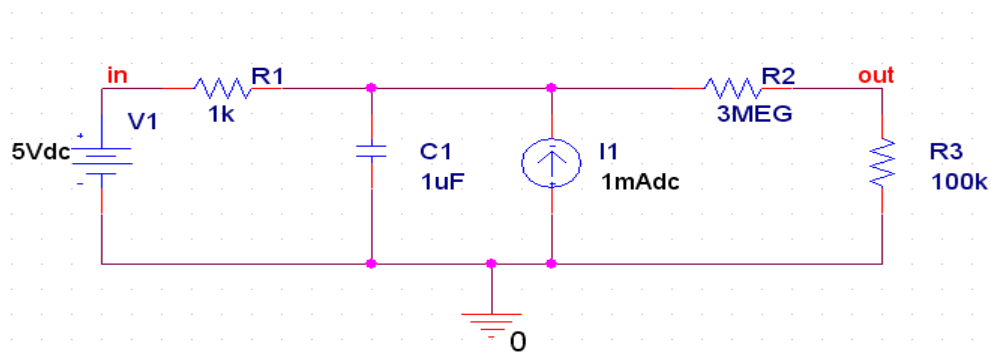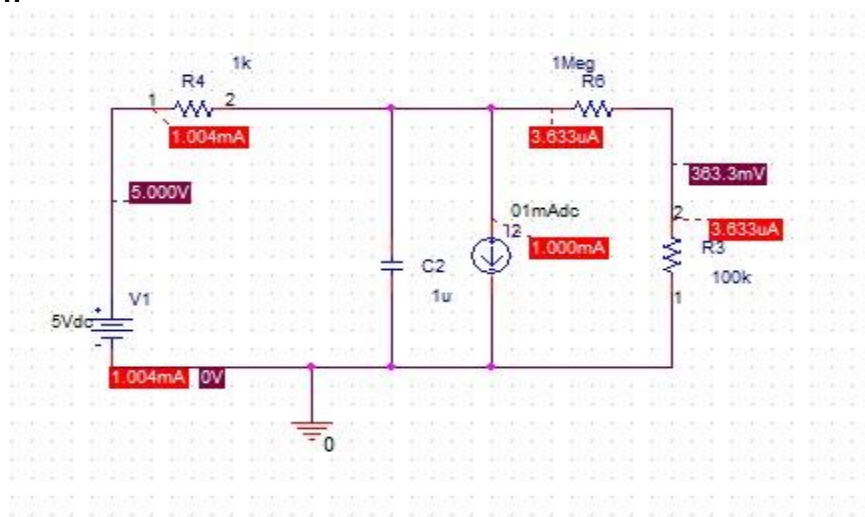

Figure 1

Circuit Simulation

   b. Determine the voltage and current across the resistor R1.

   $$V_{R1} = \underline{1.01 \ V}$$

   $$I_{R1} = \underline{1.001 \ mA}$$

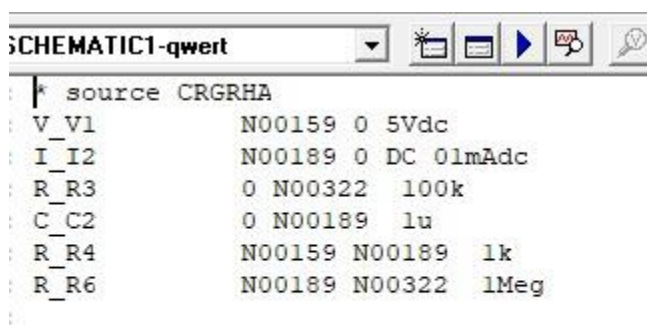   c. Determine voltage and current across the capacitor C1

   $$V_{C1} = \underline{3.99 \ V}$$

   $$I_{C1} = \underline{0 \ mA \ (Open \ Circuit)}$$

d. Determine the voltage and current across the resistor R2.

$$V_{R2}= \underline{\textbf{3.87 mV}}$$

$$I_{R2}= \underline{\textbf{1.29 μA}}$$

e. Determine the voltage and current across the resistor R3.

$$V_{R3}= \underline{\textbf{129 mV}}$$

$$I_{R3}= \underline{\textbf{1.29 μA}}$$

f. **Creating a Netlist.**   Netlists are a way of displaying all the elements of a circuit in the form of a list. Refer to section 2.1.4 of the primer on how to create a netlist. Create a netlist for the circuit in part a) and use print screen to save a screenshot.

Netlist of circuit 1:

```
SCHEMATIC1-qwert            ▾  🗎 🗆 ▶ 🔁 🔍
  * source CRGRHA
  V_V1              N00159 0 5Vdc
  I_I2              N00189 0 DC 01mAdc
  R_R3              0 N00322   100k
  C_C2              0 N00189   1u
  R_R4              N00159 N00189   1k
  R_R6              N00189 N00322   1Meg
```

EXERCISE 2

8. This part of the experiment is for dynamic simulations:

    a. DC Sweep Simulation. A DC sweep is used to check the response of the circuit across a range of DC voltages. For this exercise refer to section 2.2.2 of the primer guide and simulate the circuit given below (figure 2) for voltage 0-20V with a step size of 0.1V.
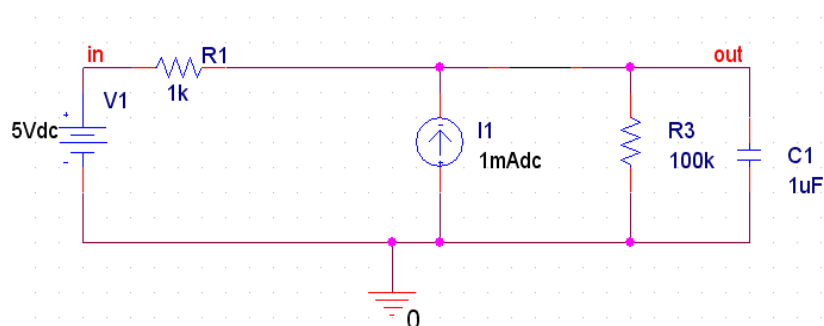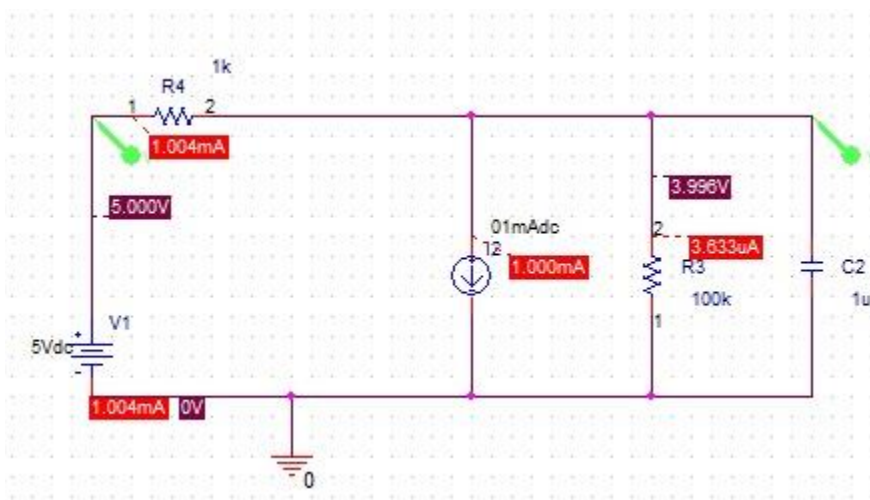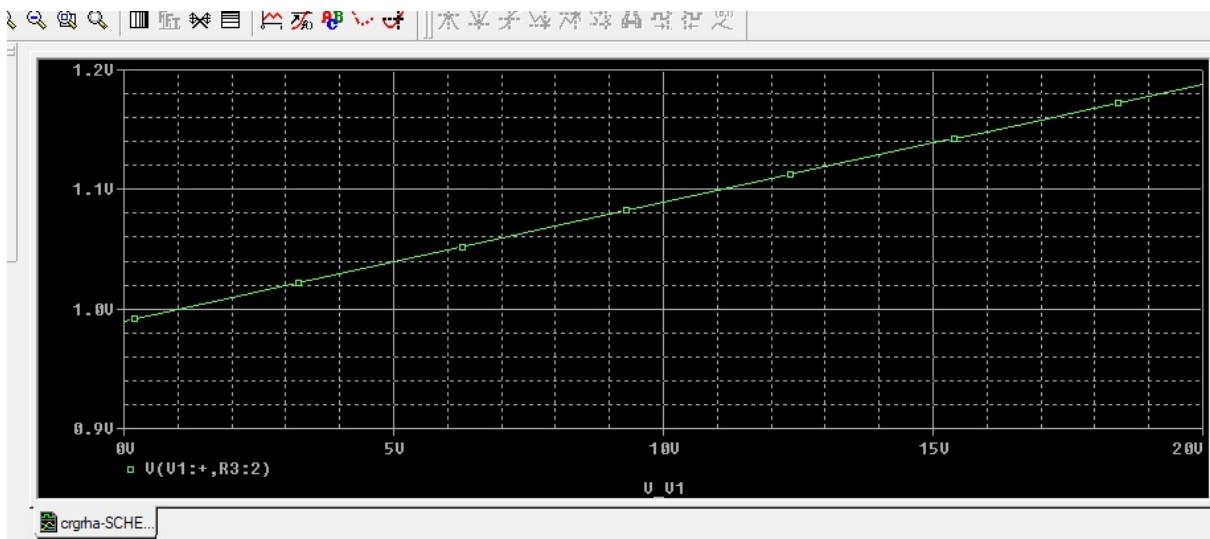

Figure 2

Circuit Simulation:

b. **Use DC Sweep simulation tool to create a graph of V_{in} vs. V_{out}. Print this graph or save it as a soft copy to be used later for printing purposes.**

Graph:

EXERCISE 3

9. This part will allow the students to use transient analysis for dynamic simulations.

   a. **Transient Analysis.** Transient Analysis involves simulating the circuit voltage or current w.r.t time. For this exercise refer to section 2.4.1 of the primer guide, and perform transient analysis on the following circuit
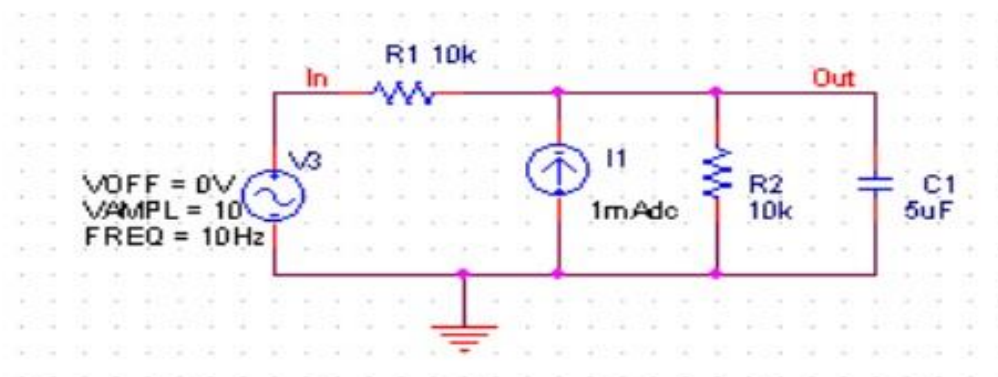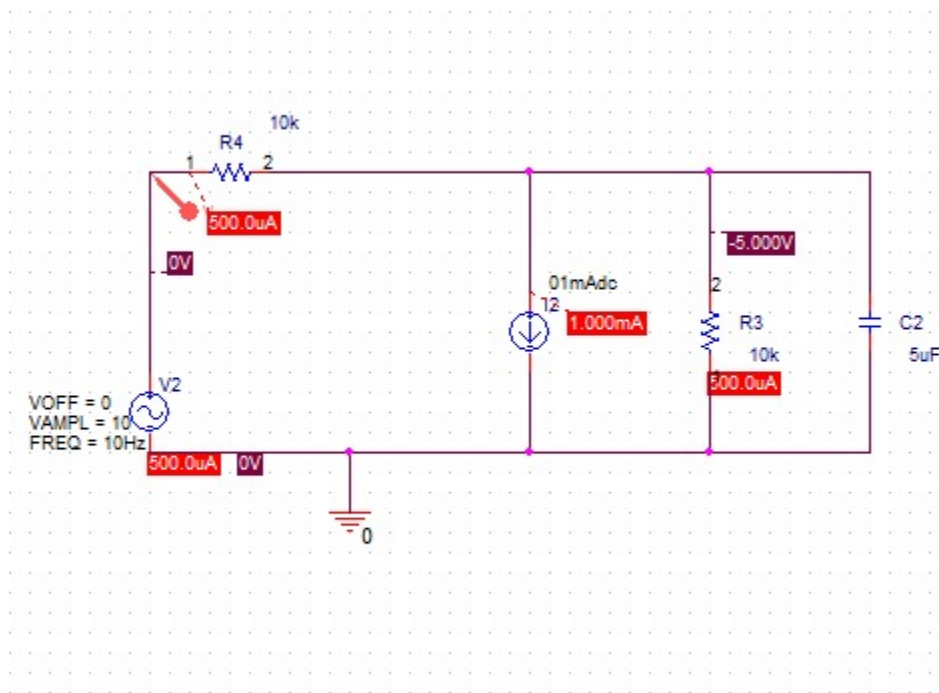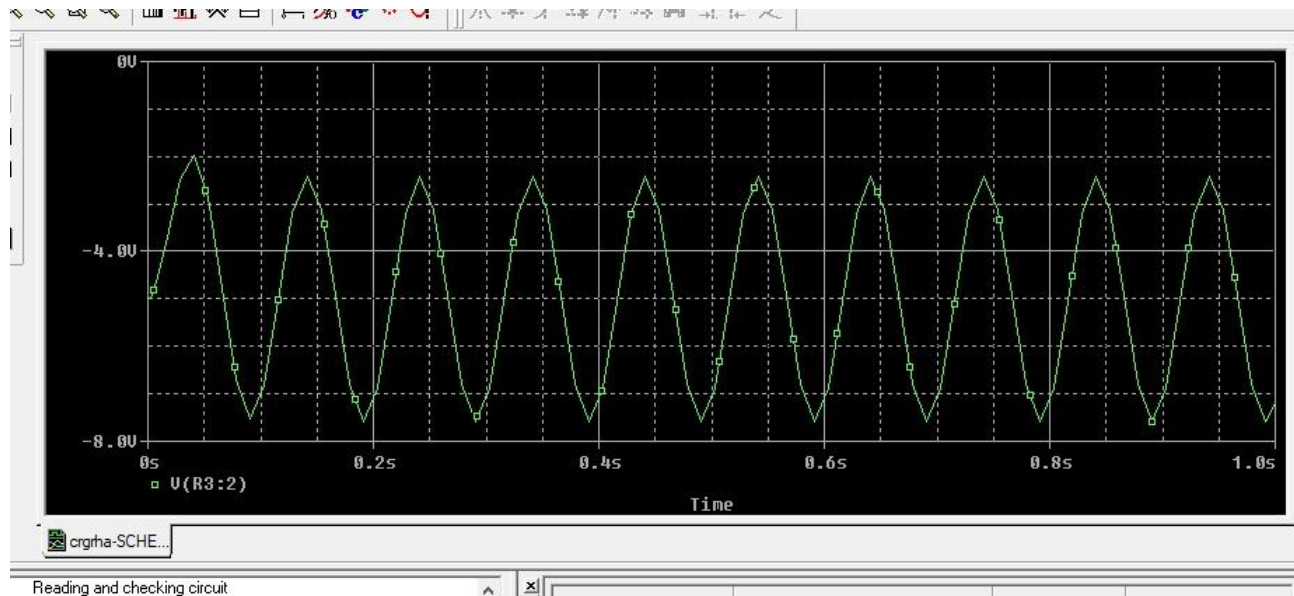

Figure 3

b. Enter the values of V1 and I1 as given in Figure 3. Use the transient analysis tool to create a graph of V_c and I_c versus time. Save the graph and get printout. Can you get a plot of V_{in} **versus** V_c?
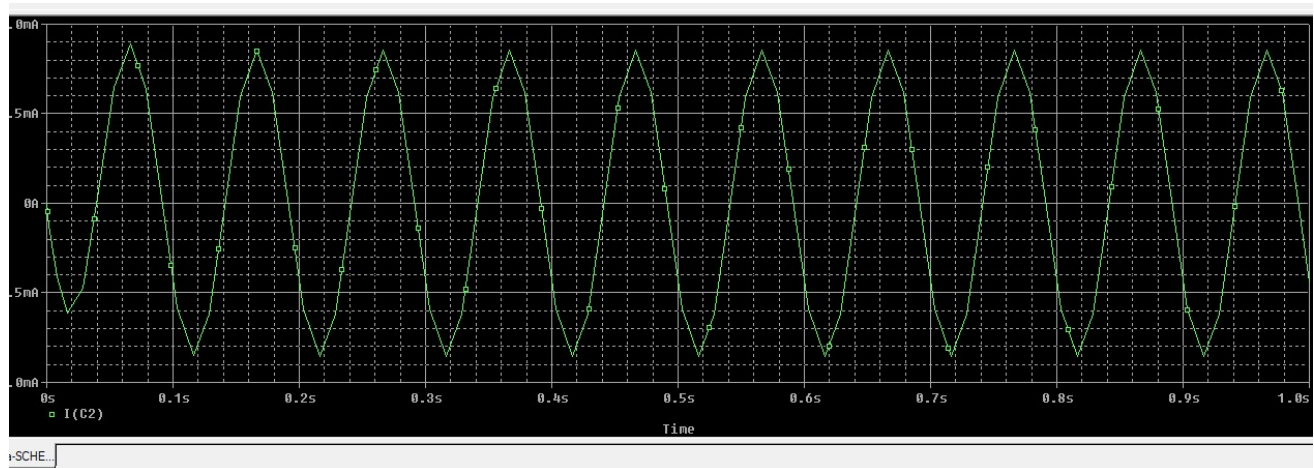
Circuit Simualttion:

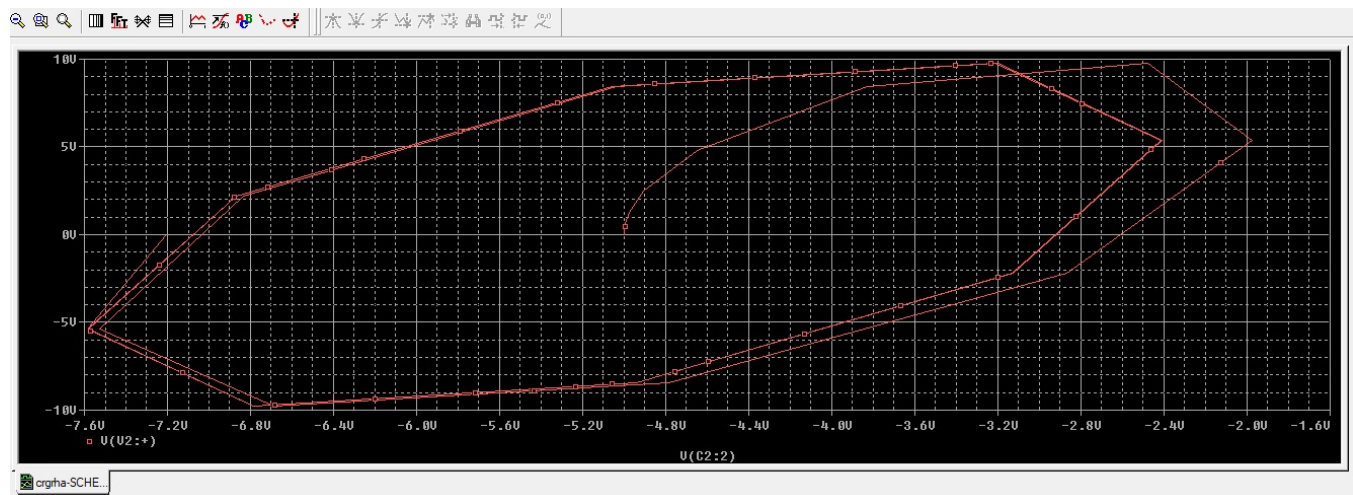**Voltage across Capacitor (V_c):**

Current through Capacitor (I_c):

V_in versus V_c:

Conclusion:

In conclusion, this lab provided an opportunity to acquire practical skills and delve into the functionalities of PSpice simulation software. Through hands-on experience, we gained a working knowledge of PSpice and explored its various features. The simulation exercises allowed us to apply theoretical concepts to real-world scenarios, enhancing our understanding of circuit behavior. Overall, this lab was instrumental in building our competence in using PSpice for circuit analysis and simulation.