



University of California Merced  
School of Engineering  
Department of Electrical Engineering

## **ENGR 065 Circuit Theory**

### **Lab #7: Introduction to SPICE**

#### **Authors**

Andre Martin

Luis Mora

#### **Instructor**

Ricardo Pinto de Castro

#### **TA**

Haoyu Li

#### **Section**

Thursday 9:00 am - 11:50

ENGR065-03L

#### **Date**

11/04/2021

Fall 2021

## Objectives

- Learn what SPICE, (Simulation Program and Integrated Circuit Emphasis) is and its basic functions
- Learn how to use SPICE to construct a simple circuit.
- Learn how to use SPICE to simulate and analyze the circuit

### 1. Introduction

The purpose of this lab is to learn how to use SPICE software to simulate and analyze the behavior of an RC circuit, which is a circuit composed of an AC voltage source, a resistor (R) and a Capacitor (C). By using SPICE, the behavior can be seen over a period of time to compare the voltages in an AC sinusoidal wave.

### 2. Procedures

#### Creating the Circuit

To begin this lab, access to the SPICE software is needed and a multisim account will have to be created. The instructions are included in the lab. After creating an account, a circuit can be made.

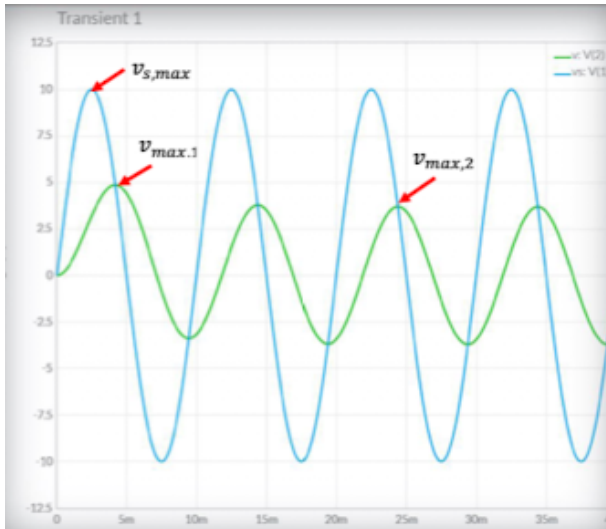
1. An AC voltage source and a ground reference is needed to make the circuit which can be found in the sources tab of the toolbar to left of the screen. Set the voltage source to 10V.
2. A resistor is then needed which can be found in the passive tab, set the resistance to 1k $\Omega$ .
3. A capacitor will need to be inserted into the circuit to complete this RC circuit, this can be found in the same passive tab. Set the capacitance to 1 microFarad.

#### Simulating the Circuit

To obtain the voltage values at the desired nodes over a period of time, we'll need to change the simulation mode to transient.

1. After changing the mode, we can run the simulation and view the values of the voltmeter over a period of time.
2. In order to get a better view of the graph we'll change the end time to 0.04s and enable Manual step to 1e-5s.

3. Since Voltage of a capacitor changes exponentially, we can see the peak of the voltage maximum change and note it down in our data tables.



### Sensitivity Analysis

To see how the resistance affects the voltage of the capacitor we will change the resistance and repeat the procedure.

1. Change the resistor's value to  $50\Omega$  and repeat the above "Simulating the Circuit" procedure. Note down the  $V_{s,max}$ ,  $V_{max,1}$ ,  $V_{max,2}$ .

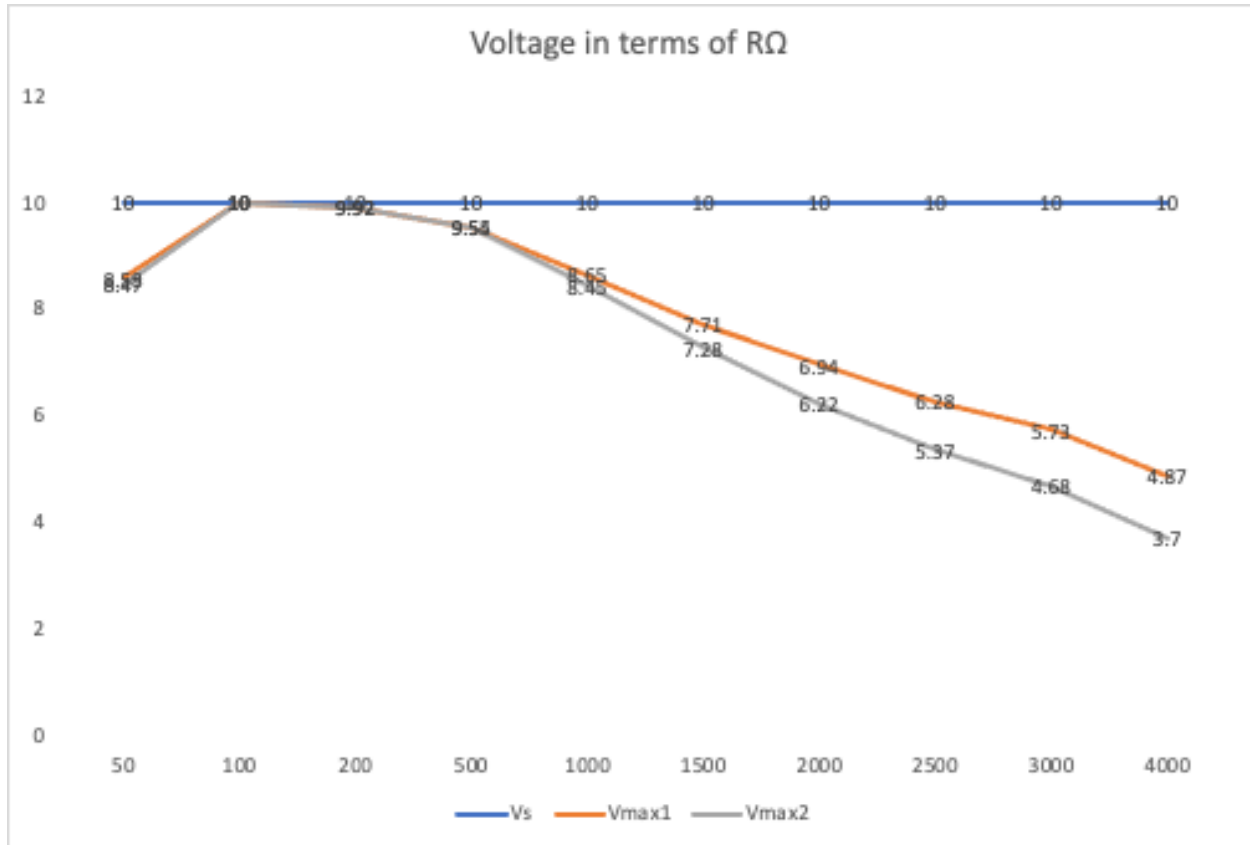
2. Repeat for resistance values  $100\Omega$ ,  $200\Omega$ ,  $500\Omega$ ,  $1000\Omega$ ,  $1500\Omega$ ,  $2000\Omega$ ,  $2500\Omega$ ,  $3000\Omega$ , and  $4000\Omega$ .

$R1 (\Omega)$	50	100	200	500	1000	1500	2000	2500	3000	4000
$v_{s,max} [V]$	10	10	10	10	10	10	10	10	10	10
$v_{max,1} [V]$	8.58	10	9.92	9.55	8.65	7.71	6.94	6.28	5.73	4.87
$v_{max,2} [V]$	8.47	10	9.92	9.54	8.45	7.28	6.22	5.37	4.68	3.70

### 3. Discussion and Analysis

We can see the change in voltage when the resistance is changed by plotting the data:

$R1\Omega$	50	100	200	500	1000	1500	2000	2500	3000	4000
$V_s$	10	10	10	10	10	10	10	10	10	10
$V_{max,1}$	8.58	10	9.92	9.55	8.65	7.71	6.94	6.28	5.73	4.87
$V_{max,2}$	8.47	10	9.92	9.54	8.45	7.28	6.22	5.37	4.68	3.7



The maximum voltage at the capacitor occurred when the resistance was  $100\Omega$  and any value greater than  $100\Omega$  the voltage began to decrease.

#### 4. Conclusions

The objectives of this lab were performed successfully. Students became familiar in using the Simulation Program and Integrated Circuit Emphasis (SPLICE). An RC circuit using an AC voltage source was effectively constructed and analyzed. By changing the resistance of the resistor in the circuit, students were able to see the behavior of the voltage at the capacitor in different cycles of the sinusoidal wave and compare them with a plotted graph.