The City College of New York

EE 22100 EE Lab#2

09/27/2017

Experiment 2: Data Acquisition, Circuit Simulation, Data Analysis and Computer Report.

Fall 2017

Instructor: Obinna Igbe

Report: Shamim Babul & Hasibul Islam

<u>Data Acquisition in the Laboratory:</u>

- ❏ DC Measurements
- ❏ Time Dependant Waveforms

Purpose

The main purpose of the lab is to become familiarize with LabView, and Multisim, and Matlab. We will learn how to acquire data from LabView, and Circuit Simulation using Multisim, and then use Matlab for graphical representation of those data.

Equipment

- ❏ The DC Triple Power Supply
- ❏ The Function Generator
- ❏ The Digital Multimeter
- ❏ The Digital Oscilloscope
- ❏ The Breadboard
- ❏ 1 KΩ Resistor and 0.1 uF Capacitor.

Procedure

- ❏ DC Measurement

First we created the following circuit on the breadboard, and connected the DC power supply.

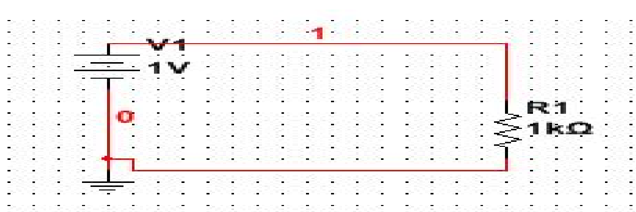


Figure1: Simple DC resistor circuit for sweep experiment.

Then we set the DC voltage power supply to 1V and measured the value of the voltage on the multimeter. Then we used the LabView V1”Power Supply Voltage Sweep” function in order to sweep the voltage from Vmin=1V to Vmax=10V and using 0.5V step.We saved the data on the computer with a file name dc_sweep_measured. Then we drew the same circuit on Multisim. On Multisim we only placed the resistor, power source and grounds. We connected all the components the way it supposed to be. The on the toolbar, we clicked on simulate, and then Analysis. Then we selected DC Sweep, and set the starting voltage to 1V and the final voltage to be 10V, and the voltage increment to be 0.5V. Then we clicked the run button to simulate. Finally, we saved the graph as a txt file on the computer with the name dc_sweep_simulation.

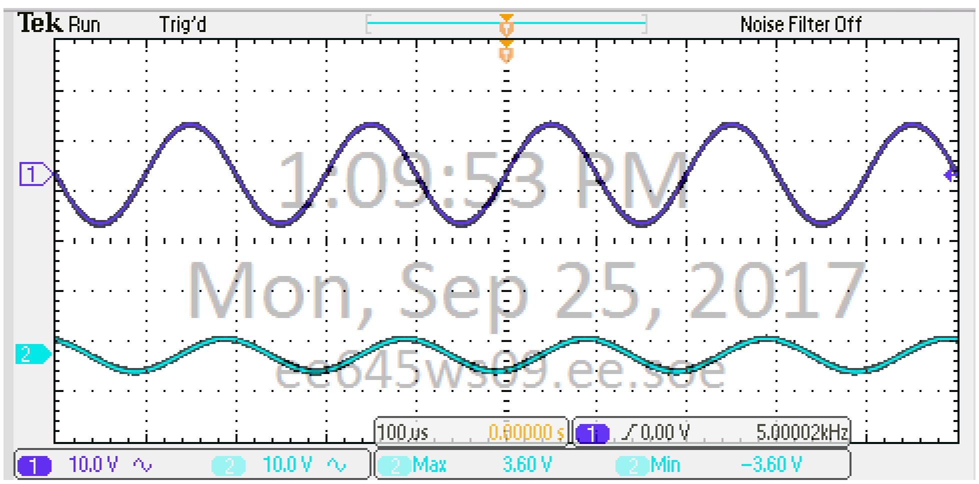


Figure2: Graph from Oscilloscope

<u>Time Dependant Waveforms:</u>

For the next part, we build the following circuit on the breadboard, and connected function generator and oscilloscope.

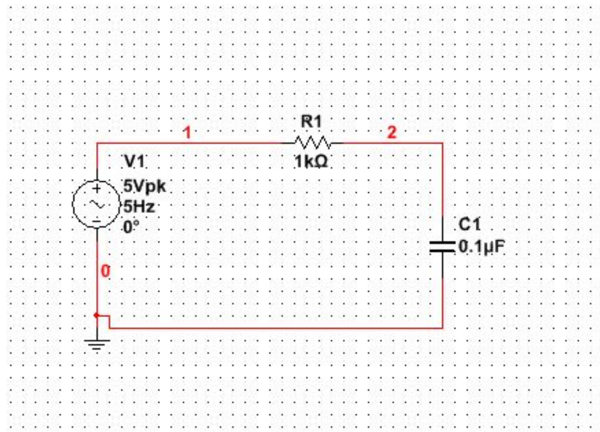


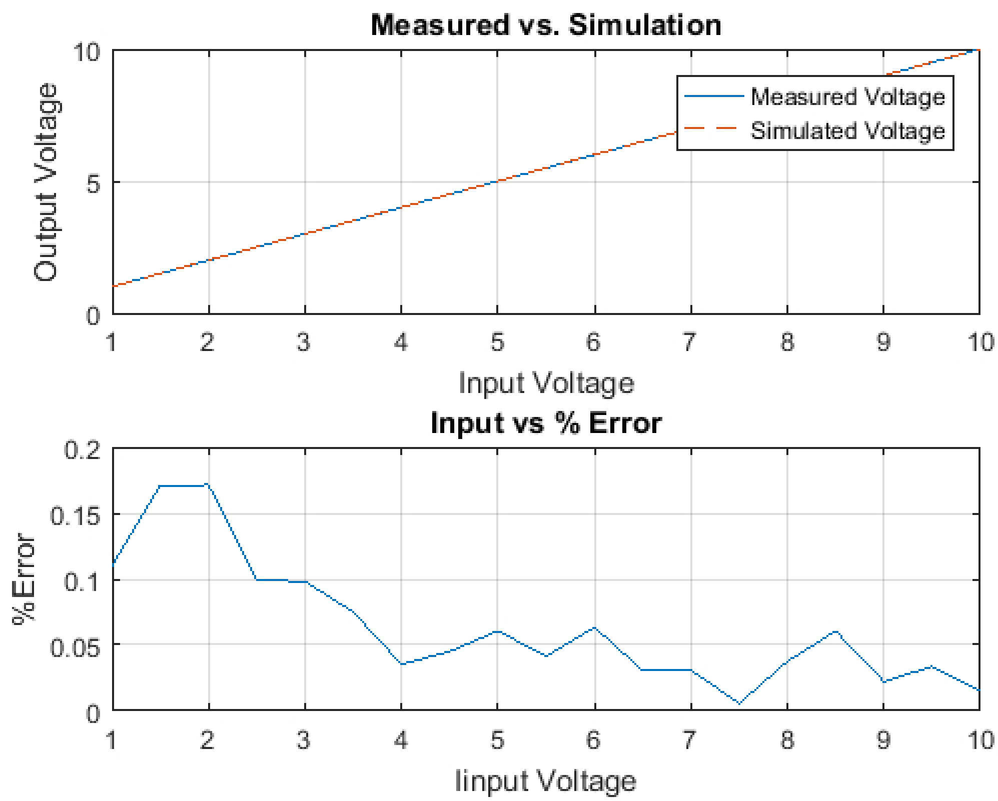
Figure 3: RC Circuit

We set the Vp-p=10V with 5 KHz of frequency. The we connect the oscilloscope channel 1 to monitor the function generator's output voltage and channel 2 to monitor voltage across the capacitor. We save the oscilloscope image using the LabView VI "Save Oscilloscope image" function. Then using LabView VI "Save Oscilloscope data:ch1 and ch2" function to save the numeric values of the graph and saved it on the computer with the filename "capacitor_measured.txt". Then using multisim, we drew the same circuit. We set the Vp to 5V, and frequency to 5 KHz on the AC power supply. Then we clicked on Analysis, and then transient analysis. We set the start to 0 second and the end to .001 second. We selected each output node at a time, and started the simulation. Then we saved the graph as .txt file on the computer with the names "capacitor_simulation.txt" and "input_simulation.txt." according to the node.

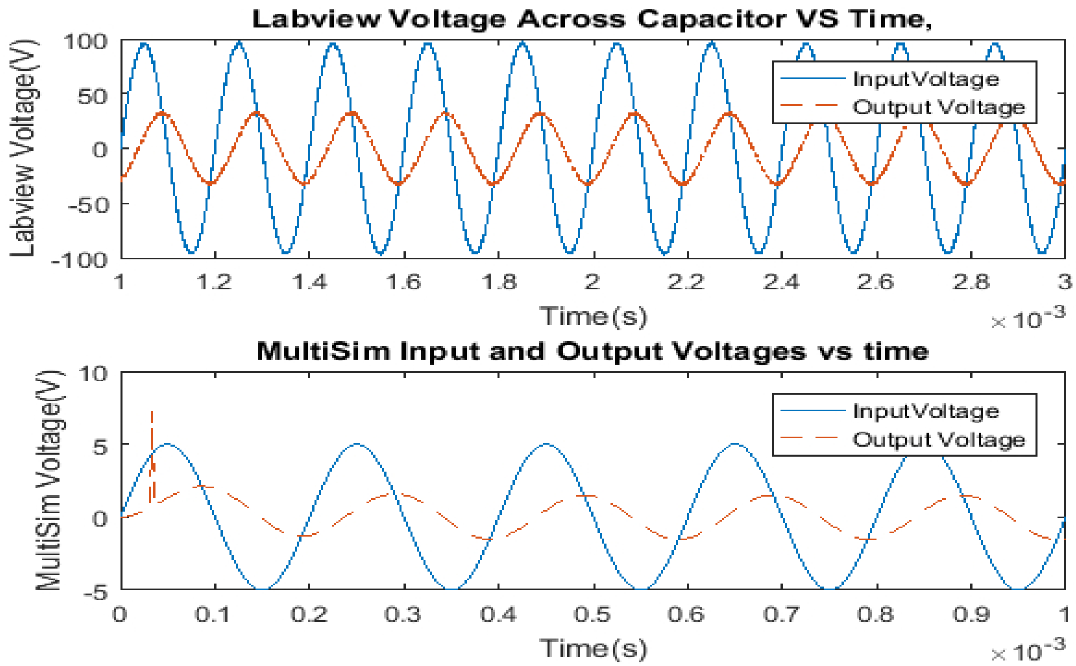
Data and Graphical Analysis:

After all the datas were collected, we used Matlab to represent all the datas into graphs. For the DC part, we first load the dc_sweep_measured, and dc_sweep_simulation to Matlab.Then separated each row of the data into two variable("Output Voltage", and "Input

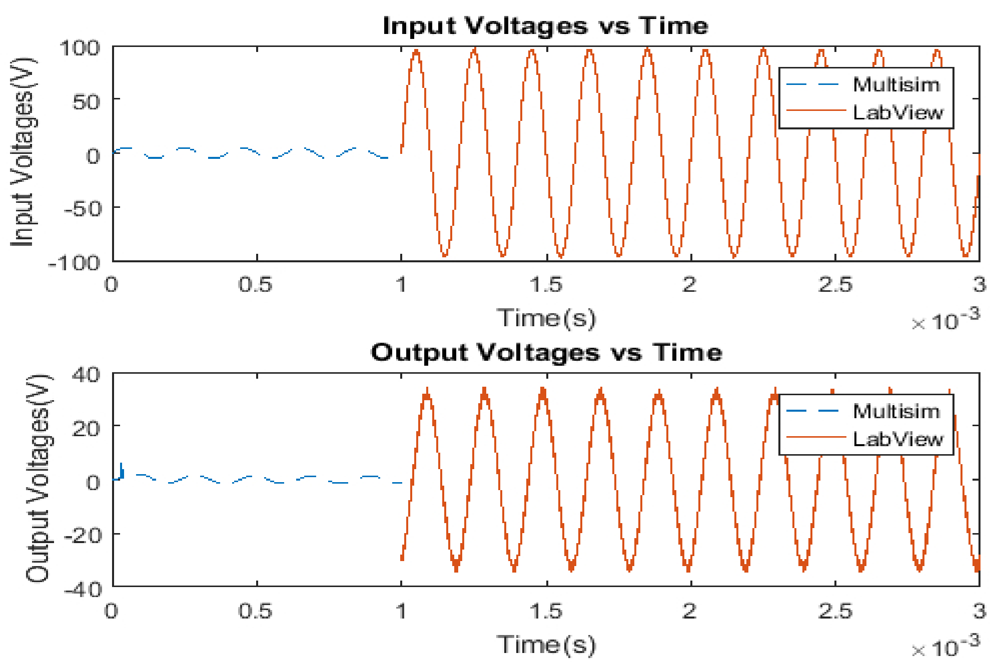
Voltage") and graphed them using plot. We also graphed the %error vs. input voltage using subplot. The graphs are shown below.



For the next part, we loaded capacitor_measured, capacitor_simulation, and input_simulation into Matlab. For the capacitor_measured, the first row gave us time, and the next two gave us input and output voltage. We graphed the input and output voltage over time using subplot and plot. We also graphed the input and output voltage from multisim vs time using subplot. The graphs are shown below.



- ❑ For the last part, we plotted the input voltage and output voltage as function of time using both Labview, and Multisim.



Post Lab: Question

1. **How accurate is the DMM in making voltage measurements? Does the Multimeter require any calibration?**

➔ It is really accurate and does not require any calibration. The error on the multimeter is really small. I would assume the conductors have some little resistance which resulted in some voltage loss.

2. **How do the RC circuit measurements obtained by LabVIEW differ with those obtained by Multisim? Explain any perceived differences.**

➔ The oscillation on the LabView is greater than on the Multisim. The amplitude is also different. The multisim will start from the moment the power is supplied but on the oscilloscope is different. The oscilloscope tries to input multiple sections of the graph in order to give us a clear steady graph.

3. **Why must we generate and save separate input waveforms in Multisim?**

➔ Multisim will save the data for each nodes vertically. In matlab we use matrices to plot graphs so we saved different files for different nodes.

4. **Explain how the input and output differs in an RC circuit.**

➔ The input voltage is the incoming voltage from the power source and does not undergo any changes. So it would be not in phase with the output voltage since as the electrons move through the entire circuit, changes will occur because of the resistor and the capacitor.

5. **If we define the phase delay as the angle difference between the output and input sinusoidal waveforms, calculate the phase delay.**

➔ We know that the f=5000 Hz and the C=.1 microFarads, R=1000 ohms.

So we can get the angular frequency ω=2*pi*5000=31415.9radians

The impedance of the capacitor will be = 1/(-j*31415.9*.1*10^-6F)=-j318.31ohms

The total impedance will be =1000-j318.31/_-90=1049.44/_-17.66°.

V(out)=(impedance of capacitor/total impedance) V(in). Now we can solve it.

V(out)=((318.31/_90°)/(1049.44/_-17.66°))V(in)=.3033/_-72.34° V(in)

From this we can conclude that V(out) and V(in) have a phase delay of -72.34°.

6. **If we define the gain as the amplitude of the output voltage sinusoid divided by the amplitude of the input voltage sinusoid, calculate the gain.**

➔ .3033V/W

7. **Is the gain and phase delay a function of the input peak voltage? A function of the input frequency?**

➔ They are dependent on input frequency but not on input peak voltage.

8. **How is the frequency of the output related to the frequency of the input?**

➔ They are equal.

Conclusion

This lab helped us gain a better understanding of the LabView, and Multisim software. We learned to acquire datas from bothe Multisim and LabView. Multisim helped us create a digital representation of the circuits, and did not require any physical tools to create and analyze the circuits. The LabView required us to create a physical circuit on the breadboard and analyze them. We learned to use those datas on Matlab to create a graphical representation. All of these graphs help us understand and compare the results.