THE GEORGE WASHINGTON UNIVERSITY

WASHINGTON, DC

SCHOOL OF ENGINEERING AND APPLIED SCIENCE
DEPARTMENT OF ELECTRICAL AND COMPUTER ENGINEERING
ECE 2110: CIRCUIT THEORY LABORATORY

Experiment #9:
AC Thévenin Circuits, RCL Meter, AC Multisim

EQUIPMENT

Lab Equipment	*Equipment Description*
(1) Function Generator	Supplied by the AD2
(1) Digital Multimeter (DMM)	Harbor Freight Model 63759 Handheld Digital Multimeter
(1) Digital Oscilloscope	Supplied by the AD2
(1) RCL Meter	Supplied by the AD2
(1) Breadboard	Prototype Breadboard
(1) Test Leads	Banana to Alligator Lead Set

Table 1 – Equipment List

COMPONENTS

Type	*Value*	*Symbol Name*	*Multisim Part*	*Description*
Resistor	3.3kΩ	R_P	Basic/Resistor	---
Resistor	6.8kΩ	R_P	Basic/Resistor	---
Resistor	15kΩ	R_P	Basic/Resistor	---
Resistor	22kΩ	R_P	Basic/Resistor	---
Resistor	33kΩ	R_P	Basic/Resistor	---
Resistor	47kΩ	R_P	Basic/Resistor	---
Resistor	68kΩ	R_P	Basic/Resistor	---
Resistor	110kΩ	R_P	Basic/Resistor	---
Resistor	220kΩ	R_P	Basic/Resistor	---
Resistor	470kΩ	R_P	Basic/Resistor	---
Resistor	(2) 270Ω	R_1, R_2	Basic/Resistor	---
Capacitor	820pF	C	Basic/Capacitor	Ceramic Disk, 821J
Capacitor	0.01μF	C_1	Basic/Capacitor	Ceramic Disk, 103K
Capacitor	1000pF	C_2	Basic/Capacitor	Ceramic Disk, 102M
Inductor	1mH	L_1	Basic/Inductor	---
Inductor	10mH	L_2	Basic/Inductor	---

Table 2 – Component List

OBJECTIVES

- To use the Oscilloscope to measure the amplitude, frequency and phase of an AC signal
- To use the RCL Meter to measure the impedance of resistors, capacitors, and inductors
- To create and explain a simple AC impedance model
- To determine the Thévenin equivalent of an AC circuit by hand calculation
- To find the Thévenin equivalent of an AC circuit by Multisim Simulation
- To measure the AC Thévenin Voltage, Current, and Impedance

INTRODUCTION

To this point of the semester, we have focused primarily on DC circuits and their applications. In this lab and for the remainder of the semester, we will be looking more closely at **AC circuits**. Although we are working with a different type of circuit, the laws and theorems discussed in earlier labs that applied to DC circuits can also be applied to AC circuits. **Ohm's Law**, **Kirchoff's Voltage** and **Current Laws**, and **Thévenin's** and **Norton's Theorems** are all applicable with AC circuits as well. In this lab, we will examine general AC circuit concepts as well as the application of Thévenin's Theorem in AC circuits.

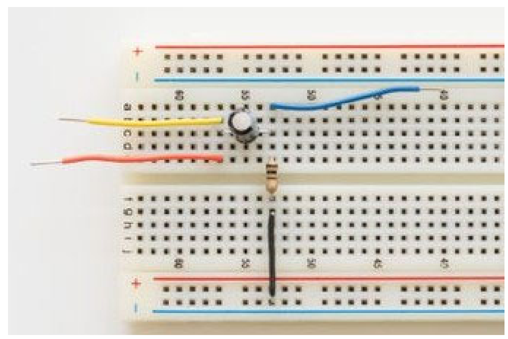
Phasors (Phase Vectors)

In physics and engineering, a phase vector, or phasor, is a complex number representing a sinusoidal function whose amplitude (A), frequency (ω), and phase (θ) are time-invariant. Phasors separate the dependencies on A, ω, and θ into three independent factors. This can be particularly useful because the frequency factor (which includes the time-dependence of the sinusoid) is often common to all the components of a linear combination of sinusoids. In those situations, phasors allow this common feature to be factored out, leaving just the A and θ features. The result is that trigonometry reduces to algebra, and linear differential equations become algebraic ones. The term phasor therefore often refers to just those two factors [1].

AC Thévenin Circuits

The general concepts of AC Thévenin's and Norton's Theorems are the same as DC Thévenin's and Norton's Theorems. The important difference here is that the signals V_T, I_N, V and I are phasors, Z_{TH} and Z_L are complex numbers representing the source and load impedances. Finding the Thévenin equivalent of a phasor circuit involves the same process as for DC resistance circuits, except that we must manipulate complex numbers.

Introduction to the "Impedance" tab in WaveForms



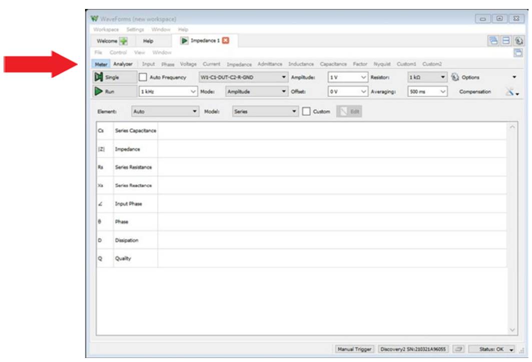


Figure 1 - WaveForms Impedance Screen

Capacitance	Reference Resistor	Inductance
100 pF	1 MΩ	
1 nF	100 kΩ	
10 nF	10 kΩ	1 µH
1 µF	10 kΩ	1 µH
10 µF	100 Ω	1 µH
100 µF	10 Ω	1 mH

Figure 2 - Reference Resistor Table

As we have seen throughout this class, the AD2 has a multitude of tools and functionalities that we are able to utilize to simulate a real lab experience. In the lab, there is a tool that is similar to the digital multimeter, but it only measures Resistance, Capacitance, Inductance, and Impedance. This device is known as an RCL meter, and the AD2 also has one as well. You can find this RCL meter under the "Impedance" tab. THIS DEVICE IS NOT AS STRAIGHTFORWARD AS THE OTHER TOOLS WITHIN THE AD2 SO PLEASE READ ALL INSTRUCTIONS CAREFULLY IN ORDER TO COMPLETE THIS LAB AND YOUR FINAL PROJECT. In order to measure with this tool, you will need four wires, the WaveGen wire (W1/yellow), both Channel 1 and Channel 2 of the oscilloscope (1+/orange, 2+/blue respectively), and Ground (Down Arrow/black).

First, you must set up your wires. In order to explain how to use this device, an example scenario will be used. In this scenario, we want to measure the exact capacitance of a 4.7uF electrolytic capacitor. It is important to know the exact value of your components in any laboratory setting, but for this class it will be especially important during the design phase of your final project. To measure this 4.7uF capacitor, you will need a few things: The AD2, a breadboard, the capacitor, and a specific "Reference Resistor". To pick a Reference Resistor, you will have to look at the chart in Figure 2. Because the capacitor is greater than 1uF, but less that 10uF, we must choose a 100 Ω resistor. Now that we have found what reference resistor we need; we can wire this testing circuit. To do this, you must put the WaveGen wire, the Channel 1 wire, and the positive end of the capacitor into the same node on the breadboard. Because this is an electrolytic capacitor, polarity does matter. Next, we wire Channel 2, the negative end of the capacitor, and one end of the reference resistor to another node on the breadboard. Lastly, the other end of the reference resistor will be connected to the Ground wire.

Once the circuit has been set up, we can go into "WaveForms" and open an "Impedance" tab. This will take us to the screen shown in Figure 1. We do not have the proper adapter to use the "Analyzer" tab, but we can click on the "Meter" tab. Now we are at the Meter tab, we must change the resistor value to the value of our Reference resistor. IT IS EXTREMELY IMPORTANT THESE VALUES ARE THE SAME OR THE METER'S VALUES WILL BE INCORRECT. Once the Reference Resistor value is chosen, we can hit play, and then stop, and we will see the values that we need to record. The series capacitance is the value we needed, and it is the very top value. We may also need the Impedance as well, which is the very next value. If we were to measure an inductor, the capacitance value will automatically change to the inductance value. The Reference Resistors still apply to these values as well. So if you have a 4.7uH inductor for example, you will still need a 100 Ω resistor.

Using Multisim to Find the AC Voltage and Phase Difference

This section will explain how to perform an AC Analysis on a simple AC circuit. These general steps can be used with any AC circuit. The following circuit shown in **Figure 4** will be used as the example here.

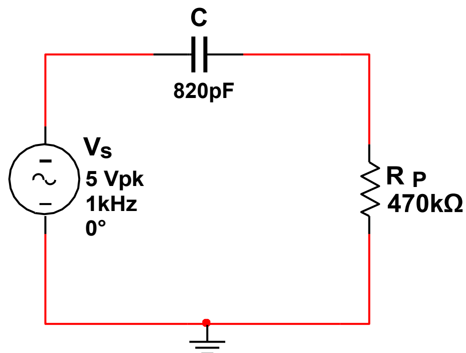


Figure 4 – AC Circuit

1. **Build the circuit in Multisim:**

   Start with your desired circuit, like the one in **Figure 4** above. The voltage source's part name in Multisim is **AC_Voltage**. $\mathbf{V_{pk}}$ is the amplitude of the signal. When defining the values of your AC voltage source, it is **extremely important** that you not only set V_{pk} but also the **AC analysis magnitude**, as shown below in **Figure 5**.

   Note: *Multisim treats the AC analysis magnitude and simulation voltage separately, so they must both be changed each time you are setting up a circuit.*

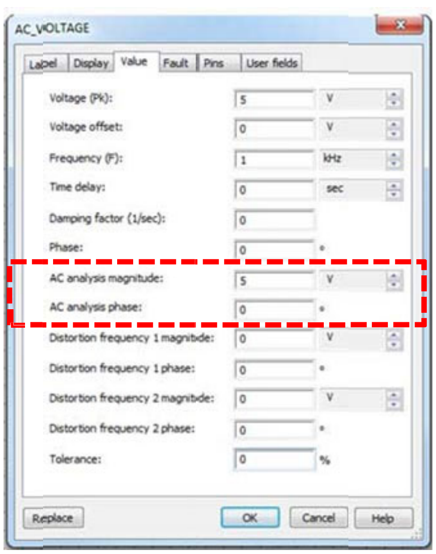


Fig re 5 – AC_Voltage Source Paramaters

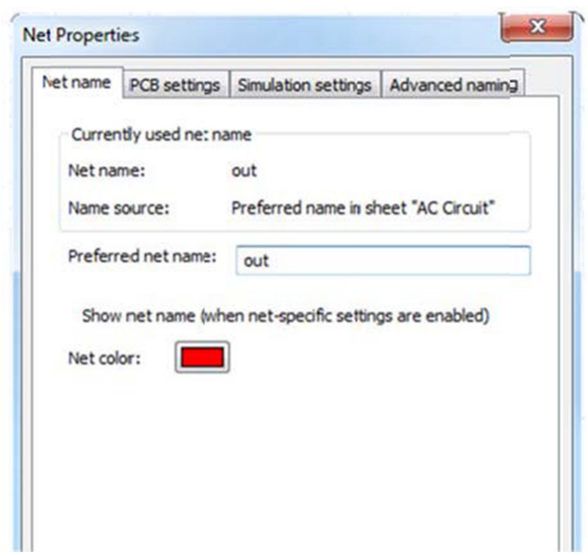


Figure 6 – Net Properties

2. **Name voltage nodes (nets):**

   Once your circuit is constructed, it is also good practice to name the **nets** or nodes in your circuit, especially those that you will be measuring later. For instance, you will often be looking at the output voltage, and naming this net will make it easier to choose the signal we want later. To name the net, you simply **double click on the wire** in Multisim, and the window in **Figure 6** will appear. Type in the desired net name and apply your changes. Now, when you simulate the circuit, the voltage signa will have a specific name such as V(out) instead of something arbitrary like V(1).

3. **Set up the AC Analysis:**

The next step is to prepare the simulation settings for the AC Analysis. To do this, navigate to the AC Analysis, **Simulate » Analyses » AC Analysis**, and the window in **Figure 7** will appear. The AC Analysis will perform a **frequency sweep** to show the behavior of your circuit over a range of frequencies

Note: *Remember that the impedance of capacitors and inductors is dependent on frequency. Changing the frequency of your voltage source will obviously have a significant impact on any AC circuit containing these components.*

Frequency Parameters: The exact range will be determined by the **start** and **stop frequencies** specified under the **Frequency Parameters** tab. In this example, the start frequency has been set to **10Hz** and the stop frequency is set to **10kHz**. The **points per decade** determines the number of sample points in each decade, and it has been set to **500** for this example. The higher this number, the smoother and more accurate your simulation will be.

Output Settings: Under the **Output** tab, you will select the signals that you want to be displayed on your output graph. In this case, we will just be looking at the **output voltage**, which as you can see is conveniently named V(out) because we previously named the net. We select the desired signals from what is available in the left column and click **Add**, to add the signals to the analysis column.

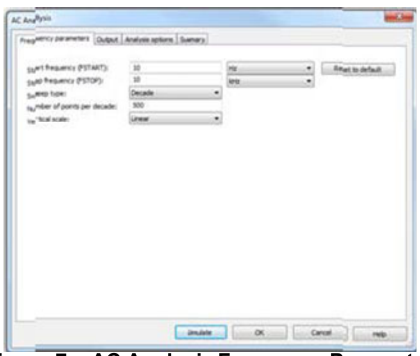


Figure 7 – AC Analysis Frequency Parameters

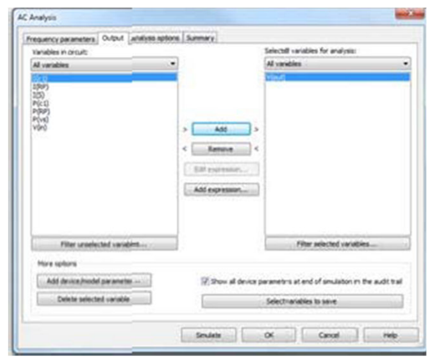


Figure 8 – AC Analysis Output Settings

4. **Run the simulation:**

The next step will be to **run** the simulation. To do this, simply click the **Simulate** button in the AC Analysis window. The following figure showing **magnitude** and **phase** will appear.

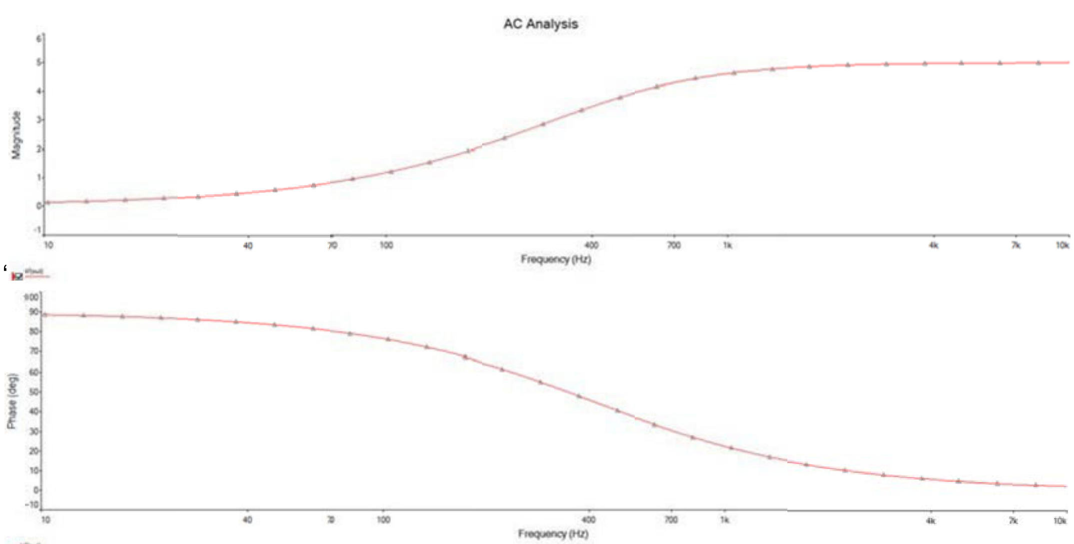


Figure 9 – Magnitude and Phase vs. Frequency

5. **Measure the values from the simulation:**

   To easily get exact values from the simulation graph, we can use the **cursors** provided Multisim. Add the cursors to the graph by going to **Cursor » Show Cursor** or by clicking the **Show Cursors** icon in the toolbar. This will add two individual cursors to the graph as well as a window showing you the values of the signal wherever the cursors are placed. You can also move the cursor to a specific location by **right clicking** on it and selecting to **Set X Value**. In the example below, **Cursor 1** has been left at the origin showing that at **10Hz**, the output voltage across R_P is **121.0414mV**. **Cursor 2** shows that at **100Hz** the output is **1.1768V**. The same can similarly be done for the Phase graph by simply clicking on it to make it the active graph, then adding the cursors again.

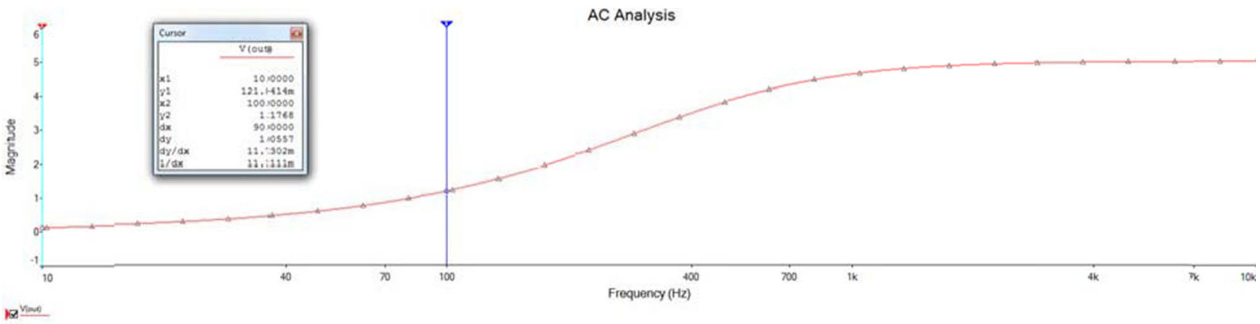


Figure 10 – Use of Cursors in AC Analysis

Using Multisim to Find AC V_{TH}, I_{SC}, and Z_{TH}

For the following circuit, we will need to find V_{TH}, I_{SC} and Z_{TH}.

1. **Find V_{TH}:**

   To find V_{TH}, **disconnect** any load resistance between nodes A and B so we are only looking at the circuit to be analyzed. At this point, we realize that the Thévenin Voltage will be equivalent to the voltage between A and B. Since no current flows through L_2 or C_2, V_{TH} is the voltage drop across the co bination of C_1, L_1, and R_2 in the middle of the circuit. In **Multisim**, we can easily measure the open-circuit voltage by performing another AC Analysis. In this example, the frequency has been swept between **50.39kHz** to **60kHz** with 500 points per decade.

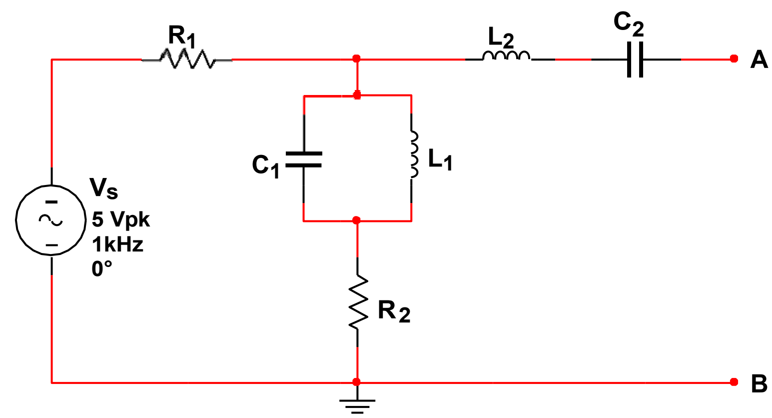


Figure 11 – Circuit for Thévenin Analysis

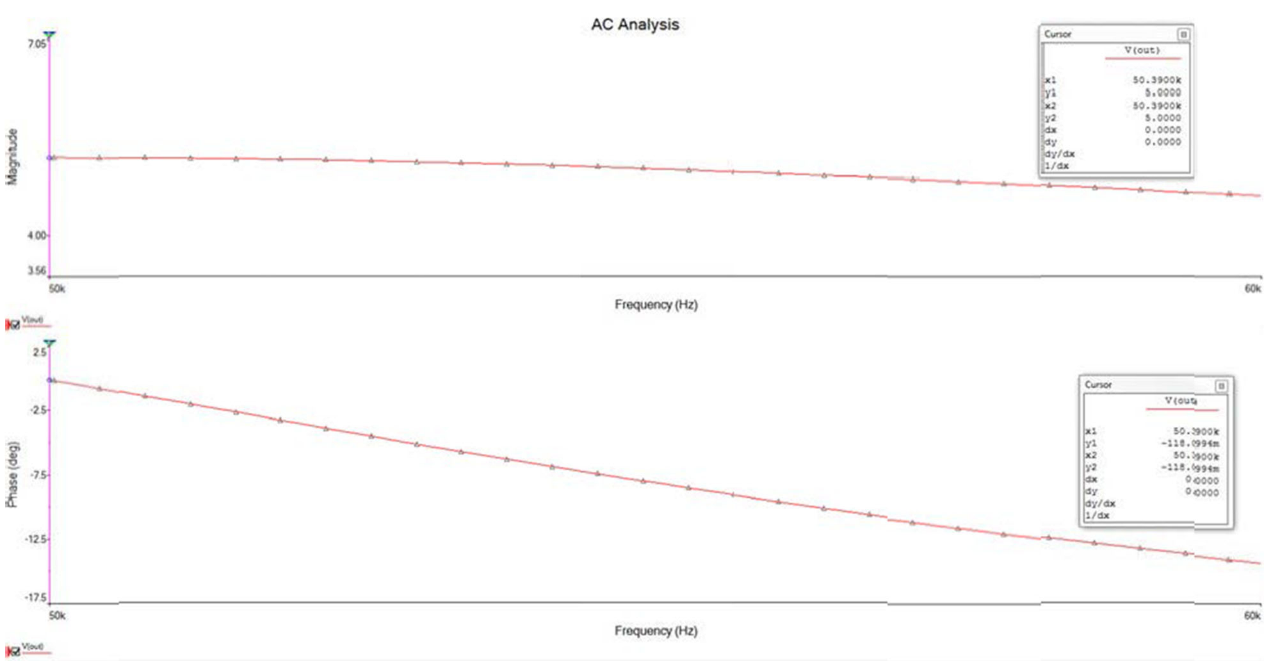


Figure 12 – AC V_{TH} Magnitude and Phase Measurement

From these graphs, V_{TH} at 50.39kHz is found to be **5.00 ∠-0.118°V**.

Note: *The x-axis of the graphs has been adjusted to directly match our desired frequency range of 50.39kHz to 60kHz, giving a more detailed view of the specified range. This can be accomplished by right clicking the axis and changing the range within AxisProperties.*

2. Find I_{SC}:

To find I_{SC}, **disconnect** any load resistance between nodes A and B, then short the load terminals with a wire as shown in **Figure 13**. The Norton Current or I_{SC} will be equivalent to the current flowing through the short. In **Multisim**, we can easily measure the short-circuit current by rerunning our AC Analysis with the shorted circuit. The frequency sweep parameters remain unchanged from **50.39kHz** to **60kHz**.

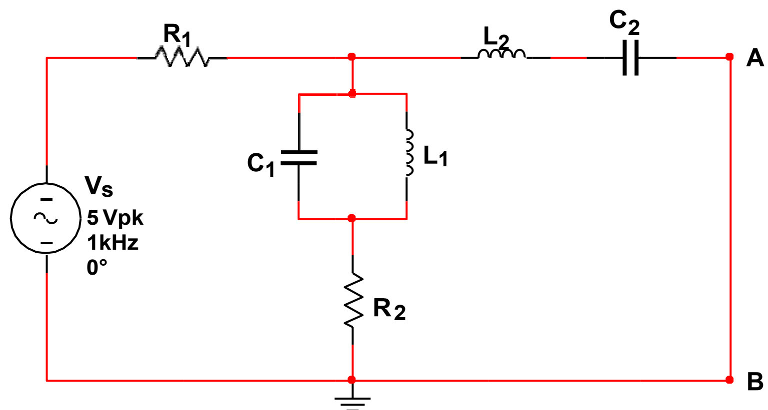


Figure 13 – Shorted Terminals A and B for I_{SC} Measurement

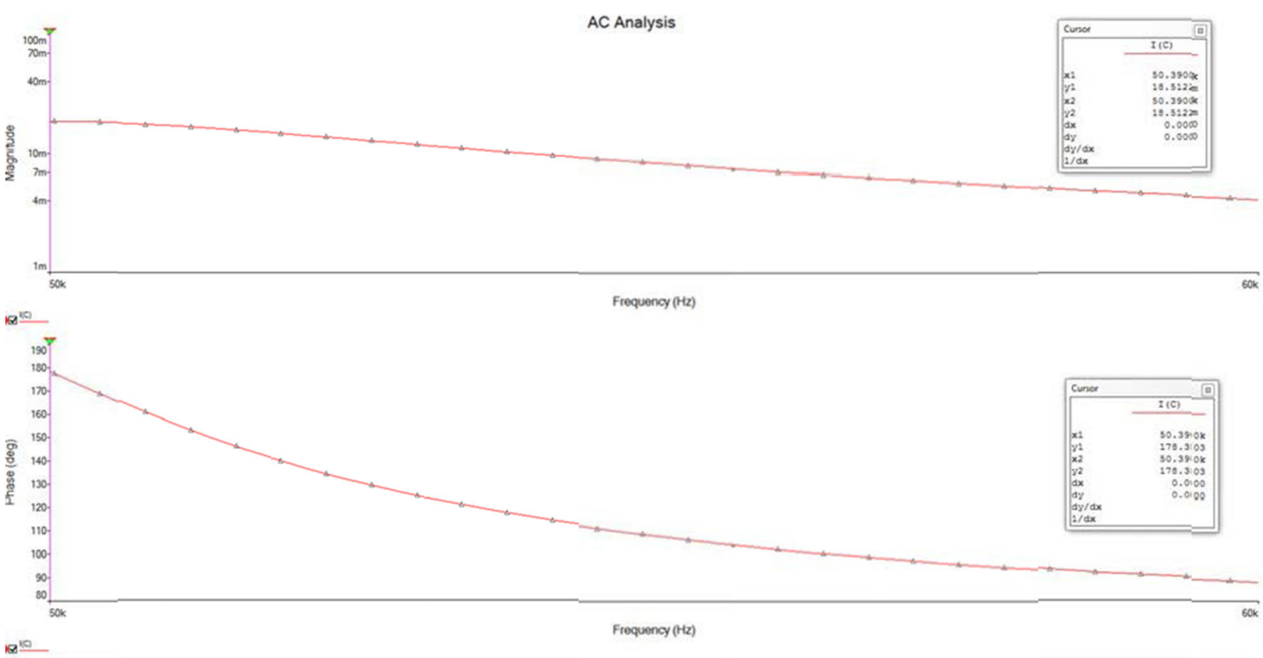


Figure 14 – AC I_{SC} Magnitude and Phase Measurement

From these graphs, I_{SC} at 50.39kHz is found to be **18.5122 ∠178.38°mA**.

3. **Find Z_{TH}:**

Finally, we want to find the Thévenin impedance Z_{TH}. In Multisim, we have two separate methods available to find Z_{TH}. The first method is to simply use Ohm's law and divide the two phasors: **V/I**. Since Ohm's Law holds for AC circuits, we know that the following must be true as well.

$$Z_{TH} = \frac{V_{TH}}{I_{SC}}$$

Equation 1 – Thévenin Impedance

Using this equation, we can perform the following calculation to find Z_{TH} from the simulated data for V_{TH} and I_{SC}:

$$Z_{TH} = \frac{V_{TH}}{I_{SC}} = \frac{5.00\angle -0.1180994°V}{18.5122\angle 178.3803°mA} = \frac{4.999 - 0.0103j}{-0.0185 + 0.00052j}$$

$$Z_{TH} = 269.999 + 7.0777j = 270.092\angle 1.502°\Omega$$

$$Z_{TH} = 269.999\Omega + jw(22.355\mu H)$$

The alternative method for finding the equivalent impedance of a circuit is to use the built-in **impedance meter** in Multisim. This device is located in Multisim in the following location: **Simulate » Instruments » LabVIEW™ Instruments » Impedance Meter**. Similar to when measuring the equivalent resistance of a circuit, we must remember to short any voltage sources and open any current sources.

Note: *The built-in impedance meter has an equivalent resistance of -50Ω and must be compensated for to obtain the proper impedance for the circuit. To compensate, we simply add a 50Ω resistor in series with the impedance meter as shown below in* ***Figure 15****.*

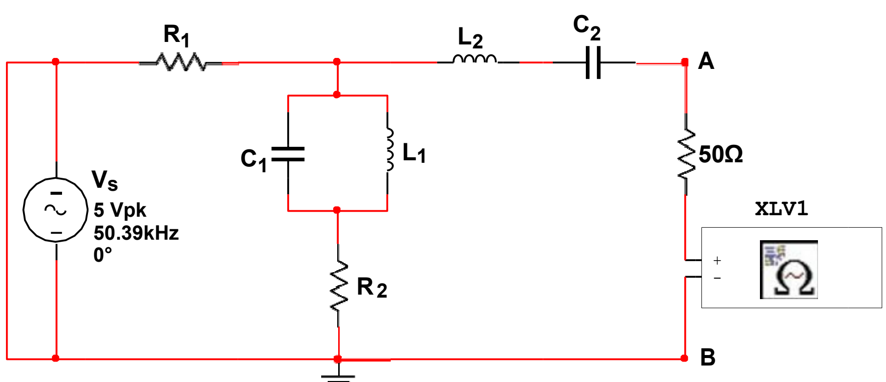


Figure 15 – Measuring Equivalent Impedance

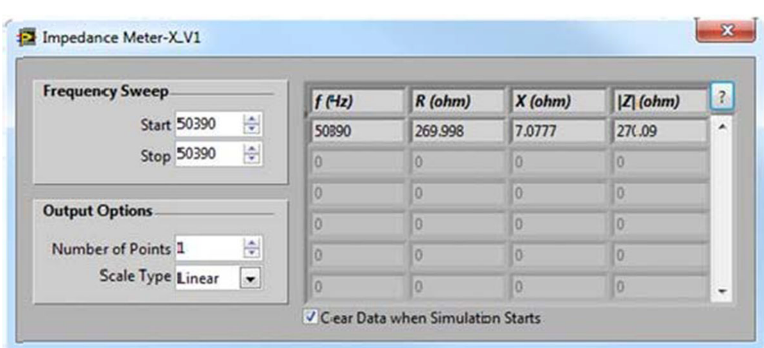


Figure 16 – Impedance Meter Results at 50.39kHz

PRELAB

Part I – AC Circuit Analysis

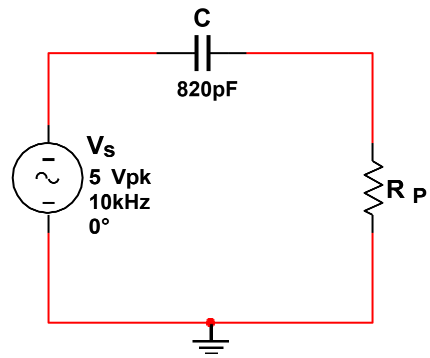


Figure P.1 – AC Analysis Circuit

1. **Derive** a general equation for the **magnitude and phase** of the voltage of R_P, V_{RP}, in terms of V_s, C, and ω. Assume the voltage source $\mathbf{V_s = A\angle 0°}$. V_s is assumed to have a zero phase φ=0 because it is the reference voltage. A is the amplitude of V_s.
2. **Label** this equation: **Equation P.1 – Equation for Finding Magnitude and Phase of V_{RP}**
3. **Substitute** the following resistors in for R_P and find the corresponding **magnitude** and **phase** of the voltage across each R_P value given:
   a. **Use C = 820pF, V_S = 5V_{pk} @10kHz.**
   b. R_P = 3.3kΩ, 6.8kΩ, 15kΩ, 22kΩ, 33kΩ, 47kΩ, 68kΩ, 110kΩ, 220kΩ, and 470kΩ.
4. **Repeat** Step 3 using Multisim to **simulate** the magnitude and phase difference of V_{RP}.
5. **Record** all of your data below in **Table P.1**.

R_P	*Magnitude of V_{RP}*			*Phase of V_{RP}*		
	Calculated	***Simulated***	***Percent Error***	***Calculated***	***Simulated***	***Percent Error***
3.3kΩ						
6.8kΩ						
15kΩ						
22kΩ						
33kΩ						
47kΩ						
68kΩ						
110kΩ						
220kΩ						
470kΩ						

Table P.1 – Magnitude and Phase Data

Part II – AC Thévenin Circuit Analysis

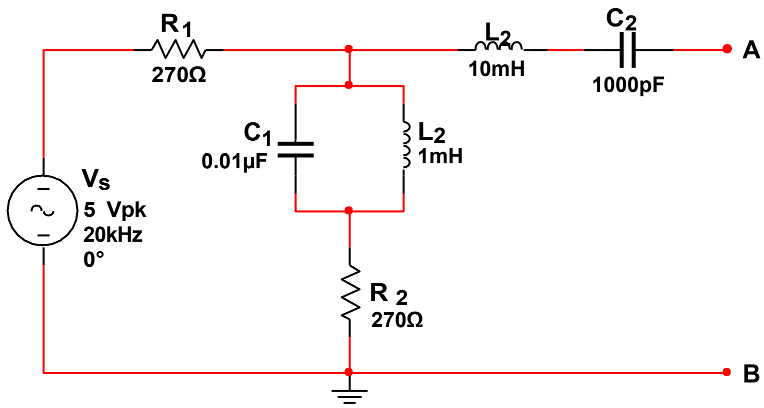


Figure P.2 – AC Analysis Circuit

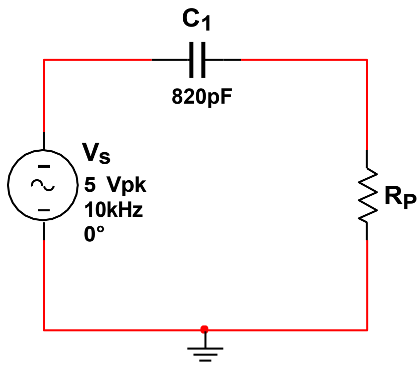
1. **Derive** general **equations** to find the Thévenin Voltage, $\mathbf{V_{TH}}$, Thévenin Impedance, $\mathbf{Z_{TH}}$, and Short Circuit Current, $\mathbf{I_{SC}}$, with respect to terminals A and B in **Figure P.2**.
   Note: *Leave the general equations in terms of the component symbol names.*
2. **Label** the three equations:
   a. **Equation P.2.1 – Equation for Finding Thévenin Voltage V_{TH}**
   b. **Equation P.2.2 – Equation for Finding Thévenin Impedance Z_{TH}**
   c. **Equation P.2.3 – Equation for Finding Short Circuit Current I_{SC}**
3. **Use** the **values** for R_1, R_2, C_1, C_2, L_1, and L_2 from the component list in **Table 2** and your equations to **calculate V_{TH}, Z_{TH},** and **I_{SC}**.
   a. **Assume V_S** is defined by **V_{pk} = 5V @ 20kHz.**
4. **Draw** the **Thévenin Equivalent circuit** for **Figure P.2**.
   a. **Label** it as **Figure P.2a – Thévenin Equivalent of Figure P.2**
5. **Simulate** the circuit in Multisim to **verify** your answers. **Keep** this circuit file so you can use it again later during lab.
6. **Calculate** the **Percent Error** between your calculated and simulated results.
   a. **Record** all of your data in **Table P.2** below.

	V_{TH}	I_{SC}	Z_{TH}
Calculated			
Simulated			
Percent Error			

Table P.2 – Thévenin Analysis Data

LAB

Part I – AC Voltage and Phase Measurement



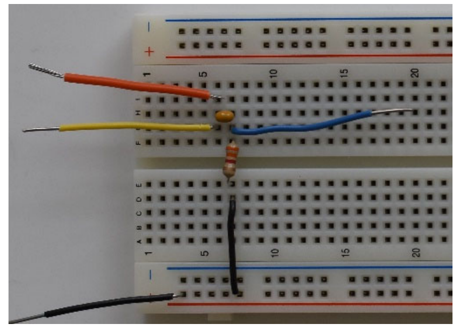


Figure 1.1 – AC Analysis Circuit

1. **Build** the circuit in **Figure 1.1** on a breadboard.
2. **Configure** the **function generator** to the following specifications:
   - **Waveform:** **Sine**
   - **Frequency:** **10kHz**
   - **Amplitude:** **5.0V$_{pp}$**
   - **Offset :** **0V**
   - **Phase:** **0°**
3. The "1+" pin is Channel One of the Oscilloscope. Use this pin to measure the input signal from the Function Generator.
4. The "2+" pin is Channel Two of the Oscilloscope. Use this pin to measure the voltage across Rp.
5. Ensure the Oscilloscope tab is open in WaveForms.
6. Set the oscilloscope to display both channels simultaneously.
7. Start with RP = 3.3kΩ in the circuit.
8. Measure the magnitude of CH2 (it is the amplitude of RP) and measure the phase difference
9. between the signals on CH1 and CH2.
10. Click on the "Measurements" tab under Scope 1 as in previous labs. Add the Peak2Peak measurement for Channel 2.
11. For Phase Angle, click "Custom Global" and then click the "Add" button that appears at the bottom of the pop up menu.
12. Record these values in Table 1.1.
13. Repeat Step 9 for RP = 6.8kΩ, 15kΩ, 22kΩ, 33kΩ, 47kΩ, 68kΩ, 110kΩ, 220kΩ, and 470kΩ.
14. Calculate the percent error between your calculated and measured results and record below.

R_P	***Magnitude of V_{RP}***				***Phase of V_{RP}***			
	Calculated	***Simulated***	***Measured***	***Error***	***Calculated***	***Simulated***	***Measured***	***Error***
3.3kΩ								
6.8kΩ								
15kΩ								
22kΩ								
33kΩ								
47kΩ								
68kΩ								
110kΩ								
220kΩ								
470kΩ								

Table 1.1 – Measured Magnitude and Phase Data

Part II – AC Thévenin Analysis with Measured Values

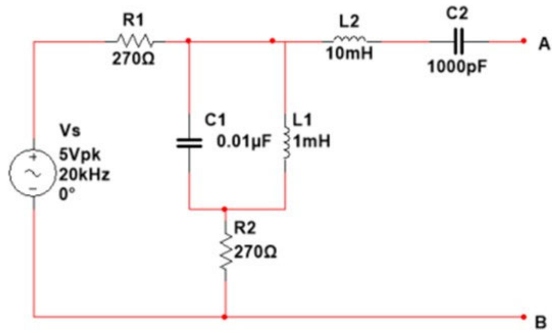


Figure 2.1 – AC Thévenin Analysis Circuit

1. ***Review the Introduction to the Impedance Tab in WaveForms before starting this portion of the lab. You must also watch the tutorial video for this portion of the lab once you reach Step 5, as the AD2 cannot perform the required tasks of this lab.***
2. **Use** the **Impedance tab** to obtain measured values of R_1, R_2, C_1, C_2, L_1, and L_2.
   a. **Record** the measured values in **Table 2.1**.
3. **Substitute** the **measured values** of your components into the equations you derived in the prelab to **recalculate** V_{TH}, I_{SC}, and Z_{TH}.
   a. **Record** your results in **Table 2.2**.
4. **Update** your Multisim simulation, replacing the nominal component values with the actual measured values.
   a. **Find** V_{TH}, I_{SC}, and Z_{TH} of your simulated circuit and **record** your results in **Table 2.2**.
5. **At this point, the AD2 can no longer complete the rest of the lab. You must watch the prerecorded video in order to get the data for the rest of this lab.**
   a. **Record the values for the following from the video:**
      1. **Thévenin voltage** V_{TH} (Voltage across terminals A and B)
      2. I_{SC}.
      3. Z_{TH} (Thévenin impedance)
6. **Compare** the measured data to the results in Part II by **calculating** the **percent error**.

	R_1	R_2	C_1	C_2	L_1	L_2
Nominal						
Measured						
Percent Error						

Table 2.1 –Measured Component Values

	V_{TH}	I_{SC}	Z_{TH}
Calculated			
Simulated			
Percent Error			
Measured			
Percent Error			

Table 2.2 – Thévenin Analysis Data with Measured Values

Post-Lab Analysis

1. **Why** do the measured values have to be used for R, L and C in the Thévenin analysis?
2. **Determine** the accuracy of your measurement data in comparison to your calculated data. **What** are the percentages of error? **Analyze** the source(s) of the errors.
3. **Why** can an Oscilloscope not measure voltage directly across a device?

References

[1] "Phasor," *Wikipedia: The Free Encyclopedia*, http://en.wikipedia.org/wiki/Phasor.