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	Agilent 33522A Function/Arbitrary Waveform Generator
(1) Digital Multimeter (DMM)	Agilent 34460A (DMM)
(1) Digital Oscilloscope	Keysight DSOX1024G Digital Oscilloscope
(1) RCL Meter	Philips PM6304 Programmable Automatic RCL Meter
(1) Breadboard	Prototype Breadboard
(1) BNC T-Connector	One input to two output BNC connector
(1) Test Leads	Banana to Alligator Lead Set
(1) Test Leads	BNC to Alligator Lead Set
(1) BNC Cable	BNC to BNC Cable

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 RCL Meter

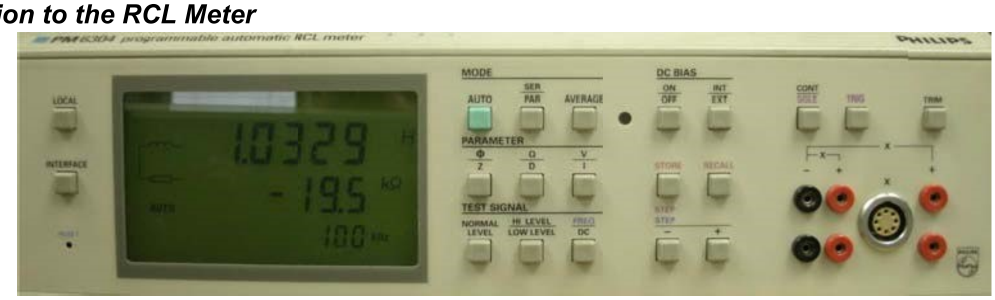

Figure 1 – Philips PM6304 RCL Meter

The Philips PM6304 Programmable Automatic RCL meter is used for precise measurements of resistance, capacitance, and inductance. The components to be measured can be connected to the instrument via a separate measurement pad or by plugging the component directly into the positive and negative, red and black, terminals directly on the device shown in **Figure 2**. The measurement results are shown directly on the display on the left. When measuring, we first connect the measurement pad to the connector on the front panel. Also, since the equipment is frequency-driven, we need to set the frequency with the buttons shown in **Figure 3**.

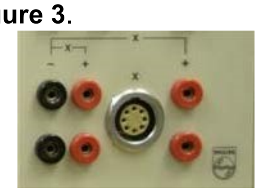

Figure 2 – RCL Meter Input Terminals

Figure 3 – RCL Meter Frequency Controls

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.

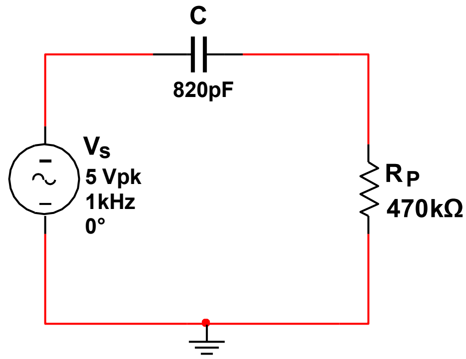

Figure 4 – AC Circuit

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.*

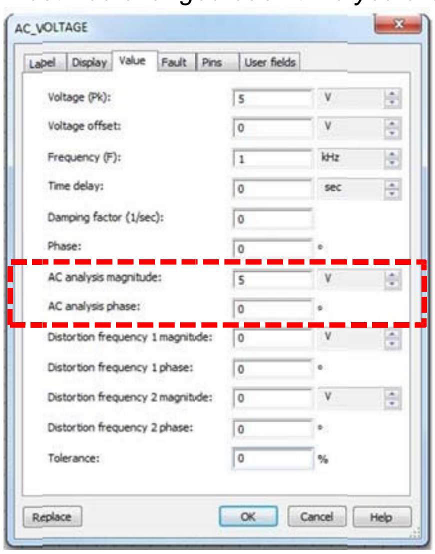

Fig re 5 – AC_Voltage Source Parameters

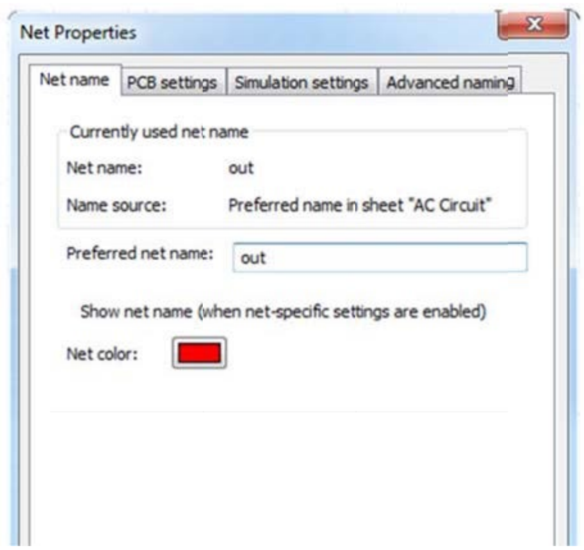

Figure 6 – Net Properties

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 on. 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 on. To actually 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 signal 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 » 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 is, 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.

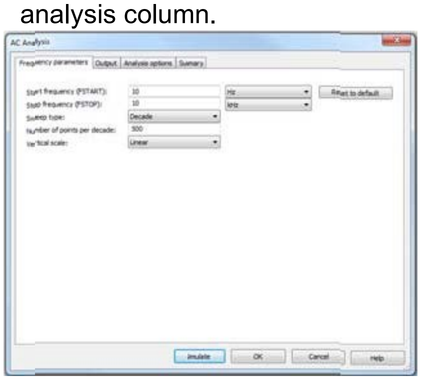

Figure 7 – AC Analysis Frequency Parameters

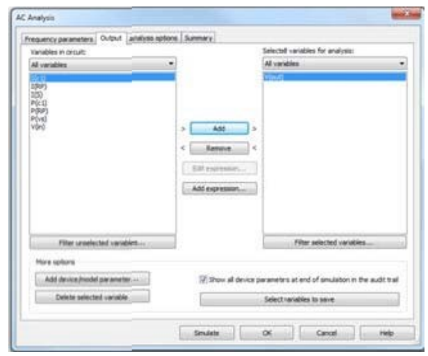

Figure 8 – AC Analysis Output Settings

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.

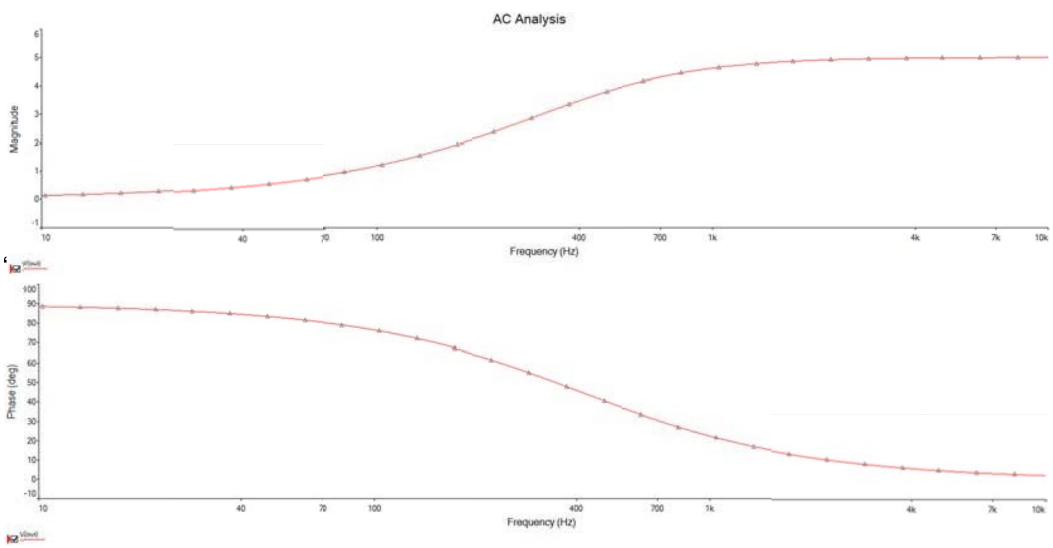

Figure 9 – Magnitude and Phase vs. Frequency

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

   To easily get exact values from the simulation graph, we can use the **cursors** provided in Multisim. Add the cursors to the graph by going to **Cursor » Show Cursors** 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 at **10Hz**, and 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.

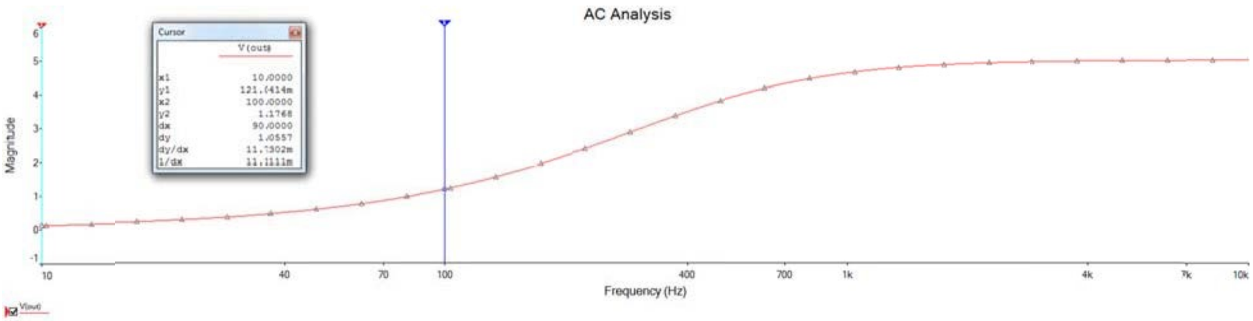

Figure 10 – Use of Cursors in AC Analysis

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 combination 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.

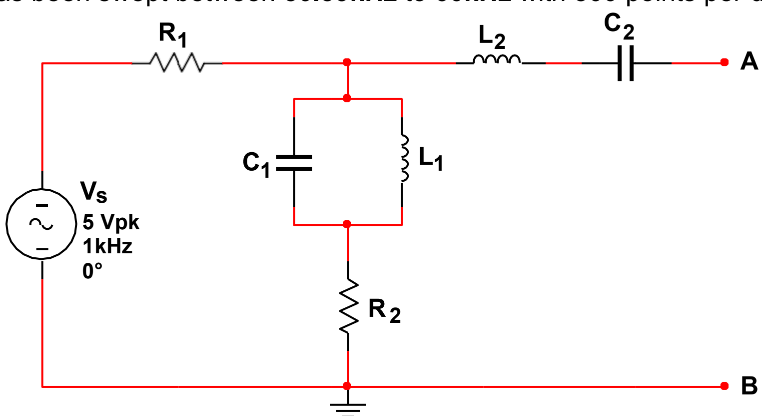

Figure 11 – Circuit for Thévenin Analysis

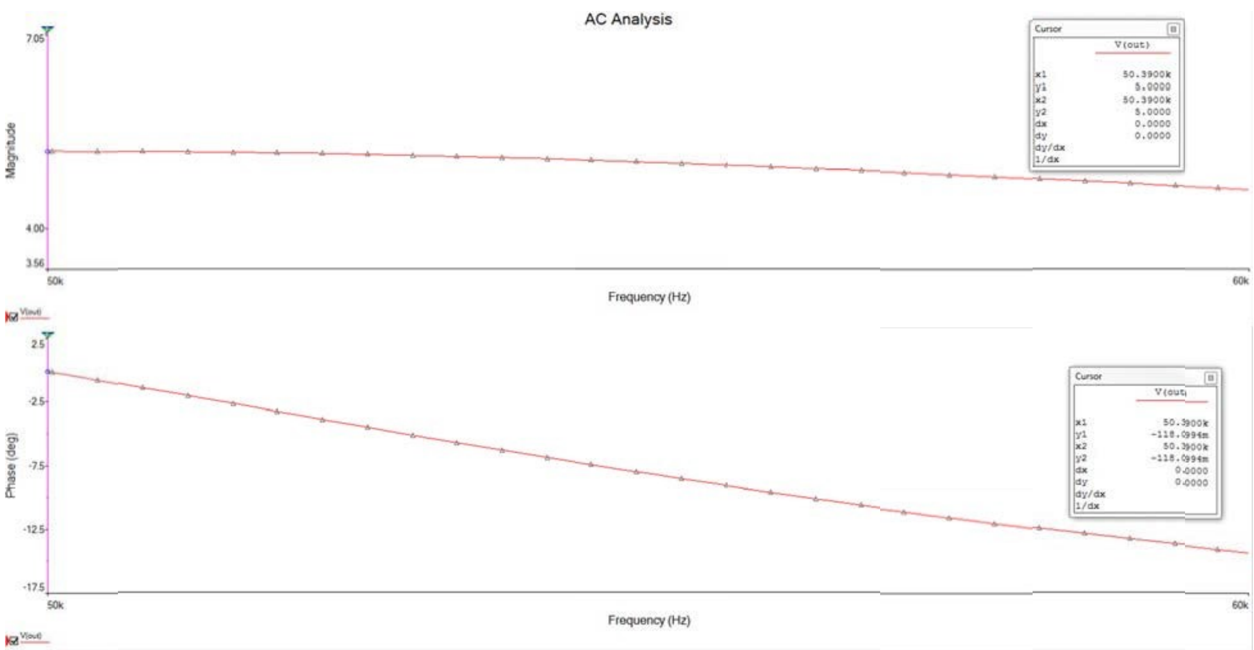

Figure 12 – AC V_{TH} Magnitude and Phase Measurement

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 Axis Properties.*

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**.

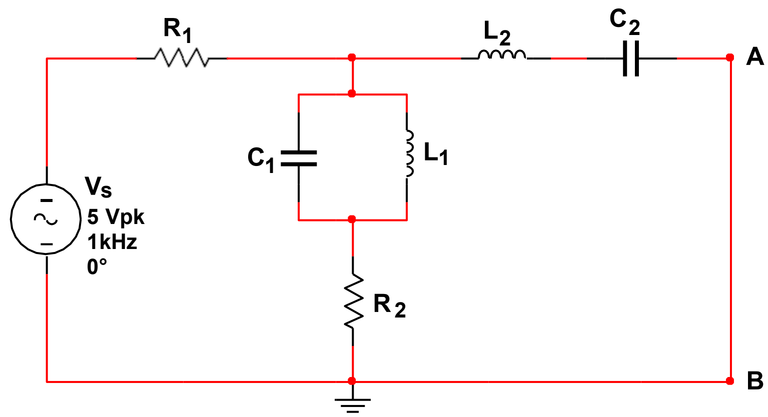

Figure 13 – Shorted Terminals A and B for I_{SC} Measurement

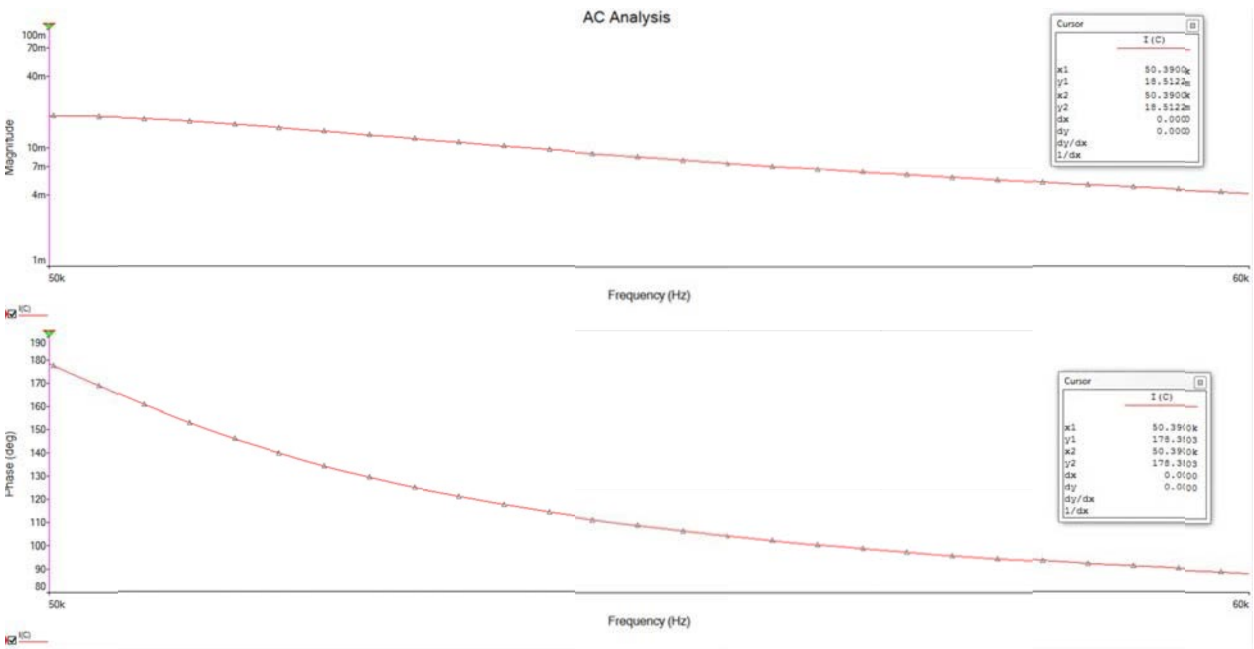

Figure 14 – AC I_{SC} Magnitude and Phase Measurement

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.

$$Z_{TH} = \frac{VTH}{ISC}$$

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}:

$$ZTH = \frac{VTH}{ISC} = \frac{5.00\angle -0.1180994°V}{18.5122\angle 178.3803°mA} = \frac{4.999 - 0.0103j}{-0.0185 + 0.00052j}$$

$$ZTH = 269.999 + 7.0777j = 270.092\angle 1.502°\Omega$$

$$ZTH = 269.999\Omega + jw(22.355uH)$$

The alternative method for finding the equivalent impedance of a circuit is to use the built-in **impedance meter** in Multisim. This device can be 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****.*

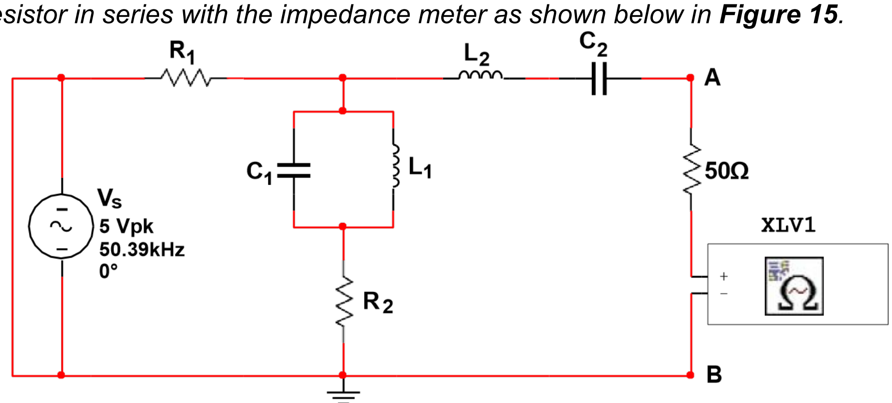

Figure 15 – Measuring Equivalent Impedance

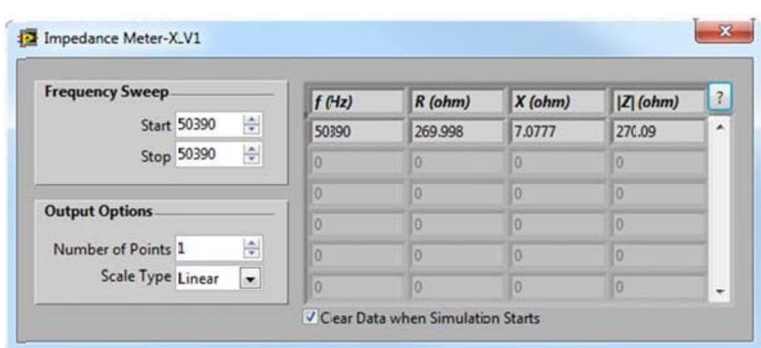

Figure 16 – Impedance Meter Results at 50.39kHz

PRELAB

Part I – AC Circuit Analysis

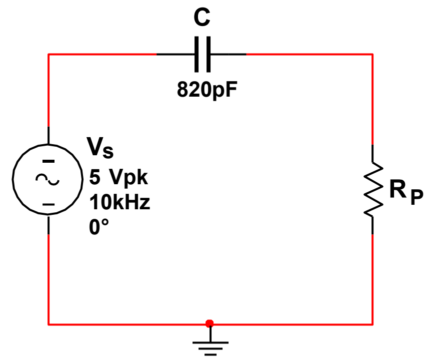

Figure P.1 – AC Analysis Circuit

1. **Derive** a general equation for the **magnitude and phase** of the voltage of RP, VRP, in terms of Vs, C, and ω. Assume the voltage source **Vs = A∠0°**. Vs is assumed to have a zero phase φ=0 because it is the reference voltage. A is the amplitude of Vs.
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

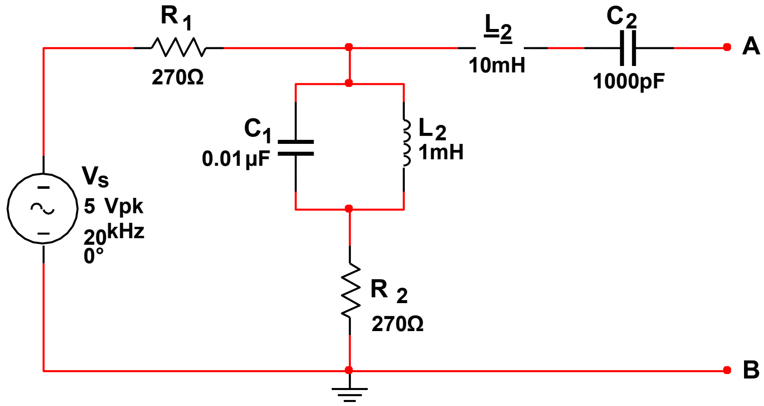

Figure P.2 – AC Analysis Circuit

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** $\mathbf{V_{TH}}$, $\mathbf{Z_{TH}}$, and $\mathbf{I_{SC}}$.
   a. **Assume** $\mathbf{V_S}$ is defined by $\mathbf{V_{pk}}$ **= 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

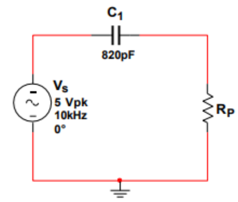

Figure 1.1 – AC Analysis Circuit

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. **Connect** the **BNC T-connector** to the output of the function generator.
4. **Connect** a **BNC to BNC cable** from the BNC T-connector to CH1 of the digital oscilloscope.
5. **Connect** a **BNC to mini-grabber** test lead from the open end of the BNC T-connector and connect the mini-grabber ends as V_S of the circuit in **Figure 1.1**.
6. **Use** another **BNC to mini-grabber** test lead to measure the voltage across R_P on CH2.
7. **Set** the oscilloscope to display both channels simultaneously.
8. **Start** with **R_P = 3.3kΩ** in the circuit.
9. **Measure** the **magnitude** of CH2 (it is the amplitude of R_P) and measure the **phase difference** between the signals on CH1 and CH2.
    a. **Record** your data in **Table 1.1**.
10. **Repeat** Step 9 for **R_P = 6.8kΩ**, **15kΩ**, **22kΩ**, **33kΩ**, **47kΩ**, **68kΩ**, **110kΩ**, **220kΩ**, and **470kΩ**.
11. **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

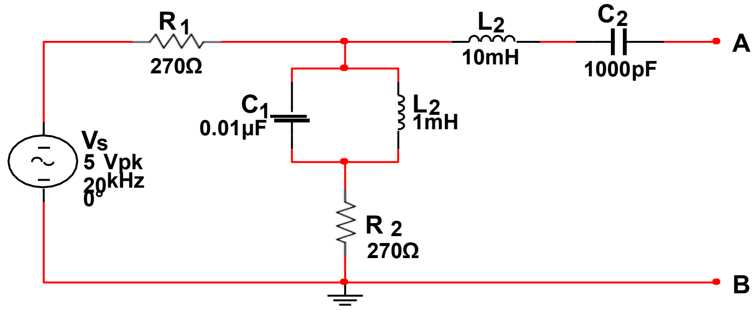

Figure 2.1 – AC Thévenin Analysis Circuit

1. ***The GTA will give you a short demonstration on how to use the Philips PM6304 RCL Meter.***
2. **Use** the **RCL meter** 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**.

   Note: *You must sign out the RCL meter measurement pad from Room 304.*
3. **Substitute** the **measured values** of your components into the equations you derived in the prelab to **recalculate** $\mathbf{V_{TH}}$, $\mathbf{I_{SC}}$, and $\mathbf{Z_{TH}}$.
   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** $\mathbf{V_{TH}}$**,** $\mathbf{I_{SC}}$, and $\mathbf{Z_{TH}}$ of your resimulated circuit and **record** your results in **Table 2.2**.
5. **Build** the circuit in **Figure 2.1** on a breadboard.
6. **Measure** the **Thévenin voltage** $\mathbf{V_{TH}}$ (Voltage across terminal A and B with R_L removed).
7. **Measure** $\mathbf{I_{SC}}$ by shorting points A and B.
8. **Measure** $\mathbf{Z_{TH}}$ (Thévenin impedance) using the RCL meter.
   a. **Disconnect** the source from your circuit and short the nodes where it used to be
   b. **Attach** the RCL meter between points A and B. **Measure** the **Thévenin impedance**.

   Note: *Be sure to properly set the frequency of the RCL meter before measuring* $\mathbf{Z_{TH}}$*.*
9. **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.