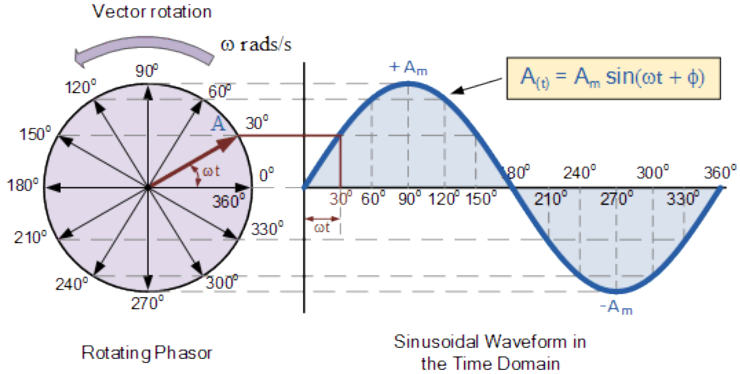
# Class #14: Experiment
# Phasor Analysis of Steady-State Circuits




**Purpose:** The objective of this experiment is to begin to become familiar with and how to analyze AC (steady-state) circuits using Phasors.

Background: Before doing this experiment, students should be able to

- Analyze simple circuits consisting of combinations of resistors, especially voltage dividers.
- Measure resistance using a Multimeter.
- Do a transient (time dependent) simulation of circuits using LTspice
- Build simple circuits consisting of combinations of resistors, inductors, capacitors, and op-amps on protoboards and measure input and output voltages vs. time.
- Make differential voltage measurements using Analog Discovery and Waveforms.
- Determine and validate mathematical expressions for steady-state sine and cosine voltage and current waves including amplitude, frequency and phase.
- Use the mathematical expressions that describe voltage and current signals obtained both experimentally and using simulation to validate and understand the relationships between voltage and current for inductors and capacitors.
- Review the background for the previous experiments.

Learning Outcomes: Students will be able to

- Determine complex impedance for R, L and C
- Analyze RC and RL circuits using phasor analysis and approaches applicable to standard voltage dividers.
- Convert phasor voltages and currents back to time dependent forms.

Equipment Required:

- **Analog Discovery** (with Waveforms Software)
- **Oscilloscope** (Analog Discovery)
- **Function Generator** (Analog Discovery)
- **Protoboard**
- Resistors, Capacitors, Diodes, 9V battery and connector
- LTspice

Helpful links for this experiment can be found on the course website under Class #14.

## Pre-Lab

*Required Reading:* Before beginning the lab, at least one team member must read over and be generally acquainted with this document and the other **required reading** materials.
*Hand-Drawn Circuit Diagrams:* Before beginning the lab, hand-drawn circuit diagrams must be prepared for <u>all</u> circuits either to be analyzed using LTspice or physically built and characterized using Analog Discovery.

**Due:** At the beginning of Class #16

## Part A – Phasor Analysis of RC Circuits in Steady-State

### Background

To prepare for the following experiments, review the material in the slides, especially the steps in applying phasor analysis. Summarizing the method:

- Write sources in phasor form
- Label all circuit components with their complex impedance
- Analyze the circuit using the same methodology as for resistive circuits. For example, apply the same approach used for a voltage divider, for circuits consisting of a resistor in series with a capacitor or inductor. {See the 4th video on voltage dividers: http://youtu.be/mIJknFwBhQU.}
- Once the voltage or current of interest has been determined, multiply it by $e^{j\omega t}$ and take the Real part of the expression to convert the phasor back to time dependent form.
- Check the solution against either an experiment or simulation results, if available.

### Experiment

We begin with a very simple simulation with an RC circuit driven by a 10kHz sinusoidal voltage. Build this circuit in LTspice using a 1kΩ resistor and a 0.1μF capacitor.

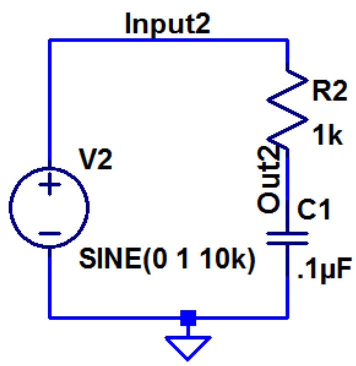



**Figure A-1**

Before running the simulation, we need to first put things in context by answering a few questions.

1. What type of filter is this circuit? (e.g. high pass, low pass …)
2. What is the corner frequency for this circuit? Is the 10kHz input signal above, below, near or far from the corner frequency?
3. Is the output voltage likely to be about the same, smaller or much smaller than the input signal, given its frequency?
4. By convention, the phase of the source is defined to be zero. Given its amplitude and frequency, write out the mathematical expression for the source voltage as a function of time. That is, write it out in the form $V(t) = V_o \cos\omega t = \mathrm{Re}\left(V_o e^{j\omega t}\right)$. Do this in spite of the fact that LTspice produces a *sinωt* voltage and not one that depends on *cosωt.* We are really only concerned with the phase difference between the input and the output, not on exactly where the input starts.
5. From your answer to question 4, identify the phasor representation of the source. TA/Instructor ________________________________________ (questions 1-5)

The goal of the LTspice simulation is to obtain some information that can be used to verify that phasor analysis will give us the correct solution for this and similar circuits. To that end, in the LTspice circuit, replace the source frequency with the corner frequency $f_c$. Set up transient analysis to display 5 periods after about 20 periods. That is, start saving data after about 20 periods. Note that you should choose these times to be easily read from your plots, and not make them exactly 20 or 25 periods because the corner frequency is unlikely to be a simple number. Run the simulation and obtain a plot of both the input and output voltages (the latter being the voltage across the capacitor. Following the method used in the previous experiment, determine and label the amplitude and phase of the output voltage. Also label the amplitude of the input voltage. Save your plot and make sure it is fully annotated. Repeat your simulation at 10 times the corner frequency $10f_c$ and 1/10 the corner frequency $0.1f_c$.

6. What is the phase shift between the input and output at the corner frequency?
7. Approximately describe the ratio of the output to input magnitudes at the two other frequencies.
8. Approximately describe the phase shifts at that two other frequencies.

Now build the circuit on your protoboard and measure the input and output voltages using Analog Discovery. Obtain plots for all three frequencies, save and annotate them in your report. Do not forget to do your hand-drawn circuit diagram for this and all other experiments and simulations. Also export your data for use in Excel.

9. Compare your experimental and simulated results.

**Summary**

When there are resistors, capacitors and inductors in a circuit, there will be differences between both the magnitude and relative phases of voltages at the input and output of circuits. There will also be a phase difference between voltage and current. It is possible to accurately predict these results using phasor analysis, as we will see in the next section.

## Part B – Phasor Analysis of the RC Circuit

Now you will see how phasor analysis works for this circuit.

**Analysis**

Write out the general forms of the R and C impedances and the transfer function for the RC circuit using phasor notation. Do not plug in any values for R and C at this point. Remember that the source representation should not include the $e^{j\omega t}$ term. You should be able to do this by applying what you know about voltage dividers. Note, in the following four questions, you do not need to use the actual values for R and C.

10. What is the general form of the transfer function for this RC circuit?
11. Simplify the transfer function at the corner frequency.
12. Simplify the transfer function at 10 times the corner frequency.
13. Simplify the transfer function at 0.1 times the corner frequency.
14. Now that you have the transfer function, in simplified form, at the three frequencies, determine the magnitude and phase of the output voltage and fill in the following table. Express the phase in both radians and degrees.

| Frequency | Calculated Magnitude | Calculated Phase |
|---|---|---|
| $0.1f_c$ | | |
| $f_c$ | | |
| $10f_c$ | | |

15. Convert your phasor output voltages to standard time varying form (multiply by $e^{j\omega t}$ and take the real part of the expression).
16. Using the information you obtained from simulation and experiment, fill in the following table. You can express the phase in either degrees or radians.

| Frequency | Measured Magnitude | Measured Phase |
|---|---|---|
| $0.1f_c$ | | |
| $f_c$ | | |
| $10f_c$ | | |

It is unlikely that you will have obtained perfect agreement, but the two tables should be very similar. Discuss the similarities and differences in the two tables in your report.
TA/Instructor ______________________________________

17. If you have time, use Excel to plot the data obtained experimentally and add a column with the output voltage obtained by converting the phasor result to standard time varying form.

**Summary**

We have now verified that phasor analysis can give results that are very accurate and allow us to study RLC circuits without having to solve differential equations.

## Part C – Task Checklist

The following list summaries the tasks from Parts A and B

- Answer the five general questions about the RC circuit (similar to the ones at the beginning of the previous experiment).
- Simulate the RC circuit in LTspice and determine the output voltage and phase relative to the input voltage at the three frequencies: $f_c$, $0.1f_c$ and $10f_c$.
- Set up the RC circuit on a protoboard and measure the input and output voltages using Analog Discovery.
- Find the general form of the complex RC transfer function and simplify it at the three frequencies
- Using phasor analysis, determine the magnitude and phase of the output voltage at three frequencies and enter the values in a table.
- Convert your phasor output voltages to standard time varying form.
- Fill in a similar table with your measured results from Part A.