



Class #26: Steady State AC Signals

Purpose: The objective of this experiment is to become familiar with and how to handle the phase shift between signals in circuits and engineering in general.

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

- Analyze simple circuits consisting of combinations of resistors.
- 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 M1K board and Alice software.
- Review the background for the previous experiments.

Learning Outcomes: Students will be able to

- Determine and validate mathematical expressions for steady-state sine and cosine voltage and current waves including offset, 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.

Equipment Required:

- M1K Board (with Alice tools) or Analog Discovery (with Waveforms Software)
- Part Kits
- LTspice
- Matlab

Helpful links for this experiment can be found on the course website under Class #26.



Part A – Sinusoidal Signals

Background

We have looked at sinusoidal signals in previous laboratories. Mathematically, we have written the expression as

$$f(t) = A \sin(\omega t + \theta)$$

There are three parameters of interest for any sinusoidal signal

- 1) **A is the amplitude** of the signal which corresponds to the maximum positive value (and minimum negative value) of the signal as it oscillates with time. When implementing a source in LTspice or setting Alice AWG configuration to a sinusoidal signal, the amplitude is one of the parameters that needs to be set.
- 2) **ω is the radial frequency** of the signal. Previously, we have related the radial frequency of the sinusoid to the frequency using the expression $\omega = 2\pi f$. Like amplitude, frequency, f in Hz, is one of the parameters set in both LTspice and Alice AWG controls. It is important to recognize that the mathematical expression of the sinusoid includes the radial frequency term while the frequency is the parameter that is specified. Also, the frequency is related to the period of the signal, with the expression $f = \frac{1}{T}$. Substitution gives the

expression relating radial frequency to period, $\omega = \frac{2\pi}{T}$.

- 3) **θ is the phase of the signal**. Phase is a more subtle concept when analyzing sinusoidal signals. In the mathematical expression we can consider the phase as a shift of the sinusoidal signal forward or backward in time (shifting the plot to the right or the left). Figure A-1 provides an example of where the sinusoidal in blue is ‘shifted to the right’, which corresponds to a phase delay.

**Important note: Shifts to the right correspond to negative phase shifts.
Shifts to the left correspond to positive phase shifts.**

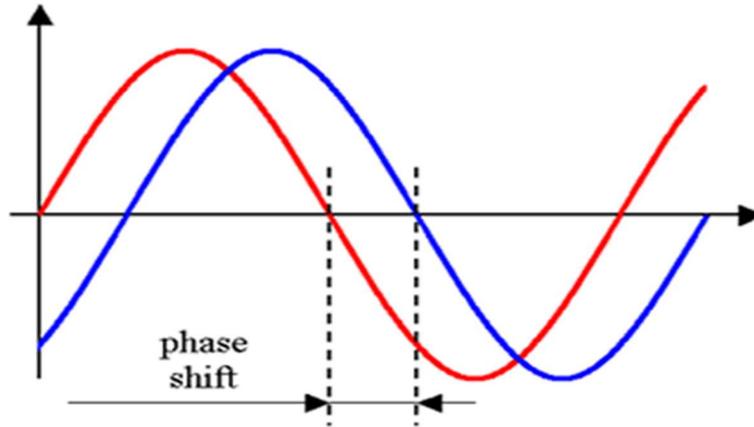


Figure A-1 Sinusoidal Signals with a Phase Difference

Phase Differences

In circuit analysis, we are more interested in phase difference measurements. For example, in a circuit with multiple components and one source, we are interested in the phase of the voltage (or current) across those components relative to the phase of the source. Typically, the source has zero phase, but that does not need to be the case. Also, we are sometimes interested in the phase difference between current and voltage. To determine the phase difference between two signals using measured data, we use the following process,

- 1) Identify the reference waveform (for example the source voltage).
- 2) Pick two corresponding points on each waveform. In Figure A-1, the points where the waveforms cross zero was chosen. However, if there is a DC offset, these points will not be a good choice. Picking points where the voltages are at a maximum or minimum is safer, though, a little bit of a rougher approximation.
- 3) Determine the time difference between those two points, including the sign. The expression should be $\Delta t = t_1 - t_2$, where t_1 is the point on the reference signal and t_2 is the point on the other signal
- 4) Determine the period, T , of the waveform, using methods in previous experiments.
- 5) Determine the phase difference, using the expression.

$$\theta_{rad} = 2\pi \frac{\Delta t}{T}, \text{ giving the phase in radians.}$$

- 6) You can convert the phase to degrees, using the relationship, $2\pi \rightarrow 360^\circ$.

$$\theta_{deg} = \theta_{rad} \frac{360}{2\pi}$$

Mathematical notations

There are number of ways to mathematically write the expression for sinusoidal signal. The familiar expression at the top of the page is only one example. Another form is to use the **phasor form**, which includes only the amplitude and phase terms. In this form, it is assumed that the frequency is known. This form is very convenient when performing mathematics with sinusoidal signal, as you will see in Circuits and Signals and Systems.

Sinusoidal form: $f(t) = A \sin(\omega t + \theta)$

Phasor form: $\tilde{f}(t) = A \angle \theta$

where the amplitude and phase are the same for both expressions. **For the phasor form, the frequency is implied but not written in the expression.**

Exercises

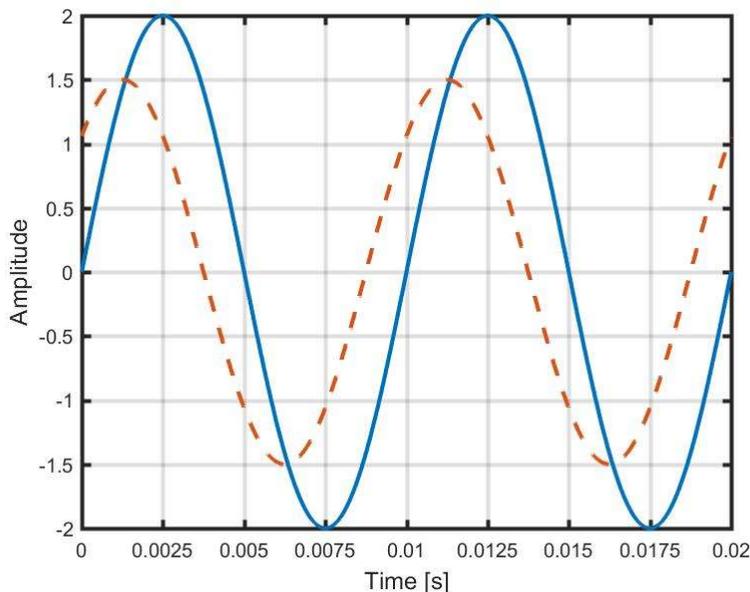


Figure A-2 Exercise Problem

The solid blue line is the reference signal, with zero phase.

- Determine period of both wave forms.
- Determine the frequency of both wave forms.
- Determine the radial frequency of both wave forms.
- Determine the amplitude of the reference signal.
- Write the reference signal in sinusoidal form (amplitude, frequency and phase) and in phasor form (amplitude and phase).
- Determine the amplitude of the other signal.
- Determine the phase difference between second signal and the reference signal, including sign. Express your result in both radians and degrees.
- Write the other signal in sinusoidal form (amplitude, frequency and phase) and in phasor form (amplitude and phase).

Part B – RC Circuits in Steady-State

LTSpice simulations

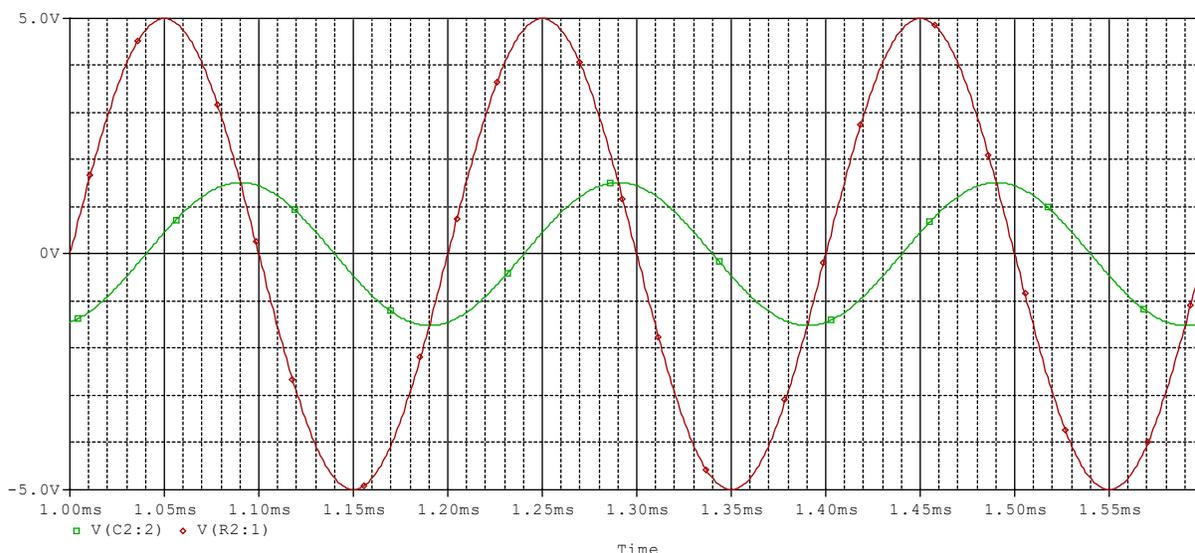


Figure B-1: Examples of AC Plots in Spice

One of the issues with LTSpice is that it includes the transient results when the source turns on at $t = 0$ (similar to observations in the previous four experiments). To avoid including the transient effects in our simulations, we want to start our plots after the transients have attenuated to zero. Typically, several periods will be sufficient. In the above case, a start time of 1ms corresponds to 5 periods after $t = 0$, which is sufficient to ignore the effects of the initial transients. In this example the run time is 1ms to 1.6ms, with a maximum step size of 10us, allowing 3 full cycles to be displayed. The start time, run time, and step size values depend on the frequency and circuit.

Experiment

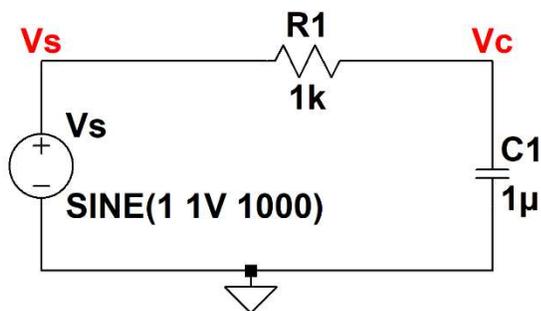


Figure B-2 RC Circuit

We begin with a very simple experiment with an RC circuit driven by a 1kHz sinusoidal voltage. Build this circuit on your protoboard using a 1k Ω resistor and a 1 μ F capacitor. The voltage source shown is to be provided by CH A of M1K board.

- Connect Ain to CH A (to measure the source voltage) and use Bin to measure voltage across capacitor.
- Plot both measurements using Alice desktop application (adjust volts/div and ms/div to display signals appropriately and at least 3 full cycles)
- Determine the amplitude of the capacitor voltage.
- Using the source as a reference, determine the phase difference between the capacitor voltage and the source voltage (including sign).
- Write the capacitor voltage in sinusoidal form (amplitude, frequency and phase) and in phasor form (amplitude and phase).

- Use a Math channel (CA V – CB V) to now display the voltage across the resistor. Again, determine the amplitude of the signal and the phase difference relative to the source.
- Write the resistor voltage in sinusoidal form (amplitude, frequency and phase) and in phasor form (amplitude and phase).
- For the capacitor only, i.e. measure the voltage across the capacitor CB V, repeat the previous measurements for frequencies, 10 Hz, 100 Hz, 500 Hz, 2kHz, 10kHz and 20kHz. Write your results in phasor form.
- In Matlab, plot the amplitude of the capacitor voltage as a function of the logarithm of frequency, $\log(f)$, (including 1kHz). What observations can you make about capacitor voltage in this RC circuit as frequency increases.
- In Matlab, plot the phase difference of the capacitor voltage as a function of the logarithm of frequency, $\log(f)$, (including 1kHz). What is the phase difference between the low frequency result and the high frequency results?
- Implement the circuit in LTspice, measuring the voltage across the source and the capacitor. You can label the nodes, as shown in Figure B-2.
- Obtain the capacitor voltage and phase difference at each of the same frequencies. For each frequency, you will need to change the source frequency and adjust the simulation profile using the Edit Simulation Cmd so that you plot 4-5 periods of the voltage measurements.
- Again, in Matlab, plot the amplitude and phase vs. $\log(f)$ (separate plots). How do your simulation results compare to your measured results?

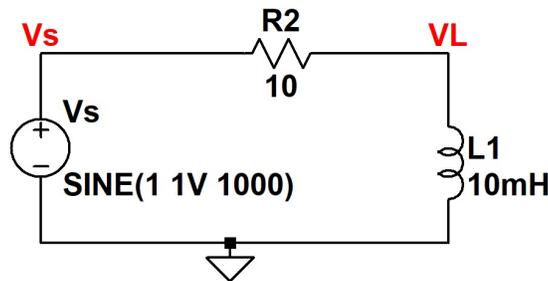


Figure B-3 RL Circuit

- Repeat the same tasks for the RL circuit in Figure B-3.
- Implement the RL circuit in LTspice.
- Obtain the inductor voltage and phase difference at each of the same frequencies. For each frequency, you will need to change the source frequency and adjust the simulation profile using the Edit Simulation Cmd so that you plot 4-5 periods of the voltage measurements.
- Again, in Matlab, plot the amplitude and phase vs. $\log(f)$ (separate plots). How do your simulation results compare to your measured results? Based on your observations in previous laboratories about inductor properties, what do you think causes the difference between the simulation and measurement results?