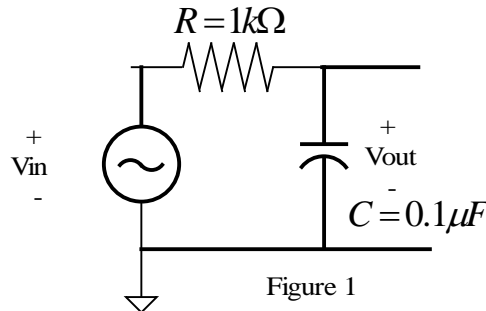


Laboratory 1

Frequency Response Analysis, Measurement, and Simulation

Our objective in this lab is to do a thorough investigation of the *frequency response* of the very simple (and useful!) RC circuit shown in Figure 1. Our investigation will include analysis, measurement, and simulation with PSpice. We will use Matlab to create a Bode plot of the analytical and measured results, and we will compare with the PSpice results. Most of the analysis is contained in Sections 9-1 and 9-3.1 in the textbook, but I will ask you to work out all of the computations on your own to make sure you understand all of the steps.

Please work in groups of two in lab.



The response of a circuit as a function of frequency may be classified by the way that it “passes” sinusoidal signals with different frequencies. For example, a circuit that has a larger output for *low* frequencies than high frequencies is a *low-pass* filter. A circuit that has a larger output for *high* frequencies is a *high-pass* filter (it “passes” the high frequency signals). Other filter types include *band-pass* and *band-stop* filters.

As you work through this lab, please collect the following items to document your work. Each group will submit these items as an informal lab report (the due date will be announced later). Some instructions are given on page 2 of this handout, and we will provide individual help during lab.

- **Analysis:** Derive the complex frequency response $\mathbf{H}(\omega)$ and the magnitude and phase angle functions, $M(\omega) = |\mathbf{H}(\omega)|$ and $\phi(\omega) = \angle \mathbf{H}(\omega)$.
 - Determine the cutoff (or corner) frequency ω_c in terms of the R and C values.
 - Use Matlab to plot $M(\omega)$ and $\phi(\omega)$ versus ω , using linear and logarithmic scales for the frequency axis (Matlab functions are `plot` and `semilogx`). Also, use Matlab to make a Bode plot of $M(\omega)$, which is dB versus log frequency. (The plots will display the frequency in Hz (f) rather than rad/sec ($\omega = 2\pi f$)).
 - What is the advantage of displaying the magnitude gain $M(\omega)$ on a Bode plot?
- **Measurement:** Use the signal generator and oscilloscope to measure the voltage gain and phase shift at several frequencies, and use Matlab to plot the measured values on the same graph as your analytical plots. Compare the analytical and measured results. Tips:
 - Make the plots versus log frequency, and use dB for the voltage gain (Bode plot).
 - Measure the values for R and C and use these values in your analytical formula.
 - Compare the cutoff (or corner) frequency obtained by analysis and measurement.
 - For the voltage gain, be sure to measure both the input and output voltage amplitudes. (Why does the input voltage magnitude vary with frequency?) For high frequencies (where the gain is small), it helps to use larger input amplitudes.
- **PSpice Simulation:** Use PSpice to simulate the circuit and produce Bode plots of $M(\omega)$ and $\phi(\omega)$ (versus frequency in Hz), and compare with the Matlab results.
[Do PSpice at the end, after completing the analysis, measurements, and Matlab plots.]

Instructions: Please ask the instructor and TA plenty of questions! Take your time and try to understand every step that we perform. Record notes in a lab notebook.

1. Analytically determine the *frequency response* of the circuit for both the amplitude gain and the phase difference between the input and output signals at sinusoidal steady-state. Express the amplitude gain formula in decibels (dB) = $20 * \log_{10}$ (amplitude gain). Keep R and C as variables in your analysis, then later we will substitute the particular values of R and C to produce plots in Matlab. What is the cutoff (or corner) frequency in terms of R and C?
2. Provide a rough plot of the amplitude gain (in dB) and the phase shift versus frequency using a logarithmic scale on the frequency axis (this is a Bode plot). Label the “-3 dB cutoff frequency” on your plots and relate this frequency to the R and C values.
3. Use Matlab to produce plots of the amplitude gain and phase shift versus frequency. Use the measured value of your resistor to produce the plots (as opposed to the nominal value based on the color code). Refer to the sample Matlab script at http://www.linux.bucknell.edu/~kozick/elec22612/lab1_freq.m Make plots using linear and logarithmic scales for the frequency axis (Matlab functions are `plot` and `semilogx`), and make a Bode plot of $M(\omega)$ with dB versus log frequency. What are the advantages of logarithmic frequency scale and dB scale for amplitude gain?

Matlab tutorials are posted on the [ELEC 226 labs page](#).

4. Build the circuit in Figure 1, and experimentally measure the the magnitudes of v_{out} and v_{in} and the phase difference between v_{in} and v_{out} at five frequencies: 16 Hz, 160 Hz, 1,600 Hz, 16 kHz, and 160 kHz.
Be sure to display both v_{in} and v_{out} on the oscilloscope when you take your data.

Set the function generator for sine-wave output with an amplitude of 2V peak-to-peak to begin, and increase the input amplitude at the higher frequencies (why?). Connect Channel 1 of the scope to measure the output of the function generator (v_{in}) and connect Channel 2 to measure the voltage across the capacitor (v_{out}). Set the scope to trigger from Channel 1. (Does the amplitude of v_{in} change with frequency?) Find the frequency at which the voltage v_{out} is 0.707 times v_{in} (the cutoff frequency). Use the cursors (or functions) on the oscilloscope to measure the phase difference in degrees.

Plot your measured data with Matlab on the same Bode plots as your analytical curves. You will need to add a plot of your measured phase data. Then measure data at additional frequencies to check the fit between analysis and measurement. Add the additional data points to your Matlab plots of gain and phase response.

Keep a copy of the Matlab scripts so that you can run them to produce plots for your lab report. Make sure your plots have the axes labeled (with units), a title, and a legend.

5. Simulate the circuit using PSpice, and refer to the PSpice tutorials on the [ELEC 226 labs page](#). Do the PSpice simulation after you have completed the above steps.

In Simulation Settings, set the Analysis type to AC Sweep/Noise to generate the frequency response. For the AC Sweep, select a start frequency of 16 Hz, end frequency of 160 kHz, 100 Points/Decade, and an AC Sweep Type of “Logarithmic / Decade.” Plot the gain and phase results as a Bode plot (use the “Add Trace” button, then the functions $20*\text{LOG}10()$ or $\text{DB}()$ to compute dB and $\text{P}()$ for phase).