University of New Hampshire
Department of Mechanical Engineering

ME 747 – Lab # 1
(6 Sept 2017)

Time and Frequency Response

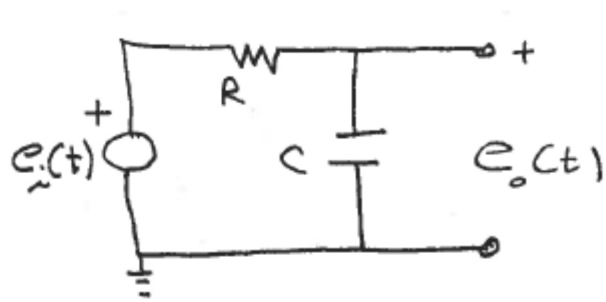
Purpose:

In this lab, you will investigate first and second order electrical systems to determine their responses to step inputs (step response) and sinusoidal inputs (frequency response). The first order is an RC circuit, and the second order is an RLC circuit.

Please note that each section of the lab has major topics denoted by **1.1**, **1.2**, etc., and under each of these topics are numerals signifying work to be done during lab. The letters beneath each numeral refers to work to be done, but that can be accomplished outside of lab. In other words, you must finish all the work described under a numeral before you leave lab.

1 First Order System

The figure below describes a first order RC-circuit. The circuit is powered by a periodic input (provided by the NI PXI-5142 function generator). The voltage output is the voltage across the capacitor C.



1.1 Step Response of a First Order System

1. Record your electrical board and station number shown on the NI frame.

2. Measure the resistor in your circuit using the digital multimeter (NI PXI-4065) and the DMM Soft Front Panel software on your computer Desktop. Alternatively, you may also use the hand held multimeter at your lab station. Make sure that when you take your measurement, you isolate the resistor so that there are no parallel current paths.

3. Connect the NI function generator to the digital scope and the circuit input $e_i(t)$ (use a t-junction bnc), and adjust for a square wave of 2.0 volts amplitude (-2 to +2 volts).

4. Connect the circuit output $e_o(t)$ to the scope and adjust the frequency of the square wave input so that a complete step response is observed (i.e., to steady state). Note that a trigger is recommended here. Set trigger options such that you have an optimal view of the entire step response. Use the same scale and origin for both input and output curves.

5. Sketch both the input and output curves, and save the data for later analysis.

6. Re-adjust the square wave amplitude to approximately a -3.0V to +5.0V output. For this, make sure your DC offset (using the drop down menu) is set to 0V. Then, set the amplitude to 4V with the Amplitude p-p drop down menu. (Recall that this amplitude is amplitude of oscillation and not peak-to-peak amplitude.) Finally, use the DC offset option to set the DC offset to 1.0V. Once again, sketch both the input and output curves and save this data. You should note any differences/similarities between the 2 output curves.

a) For each output curve, calculate the time constant two different ways (i.e., the initial slope method and the 63.2% method).

b) Compare the time constants and note/explain any differences.

c) Derive the differential equation of the system.

d) Calculate the capacitor value of your circuit.

e) Use Matlab/Simulink to simulate the theoretical step response for a -2.0 to +2.0 step input. Use your calculated capacitance value. Plot **<u>both</u>** the experimental and theoretical step responses on the same graph and compare them. You may have to shift/scale your theoretical time axis appropriately for a good comparison.

f) Comment on similarities/differences in your findings in step 6 compared to that of step 5.

1.2 Frequency Response of a First Order System

In this part of the lab, you will input a sine wave (at a constant frequency) to your system and measure the resulting output amplitude (ratio) and phase shift. You will repeat this for a number of input frequencies.

1. Adjust the FGEN input to a sine wave with an amplitude of 2 volts (<u>no dc offset</u>). Check this with the scope and note the exact amplitude of the input signal. Leave this amplitude constant for the remainder of the frequency response testing. Make sure both the input and output signals of the circuit are connected to the scope for measurement.

2. For a input frequency close to the breakpoint (give in Hz), use the NI Scope Soft Front Panel software to show the sample output and input responses on the same graph. Sketch a sample output and input and save this data. (*It would be best to calculate the breakpoint frequency from part 1.1a priori.*)

3. For a range of input frequencies, record the output amplitude and phase shift. Note that it is recommended that you use cursors to obtain this data. Make sure you take enough data <u>especially around the breakpoint frequency</u>. Get enough data (minimum 12 points) for 3 decades of frequency, (e.g. 10 Hz to 10 kHz).

4. Sketch the frequency response (magnitude ratio vs. frequency, and phase vs. frequency) as you collect your data.

a) How would a dc offset (voltage bias), or a voltage input amplitude change, affect the system Bode plot?

b) Derive the transfer function from the differential equation of section 1.1 (c). Use Matlab/Simulink and the capacitor/resistor values from section 1.1 to find the theoretical frequency response plot (Bode plot), and then plot both the experimental data on the same graph (amplitude and phase). Compare both plots.

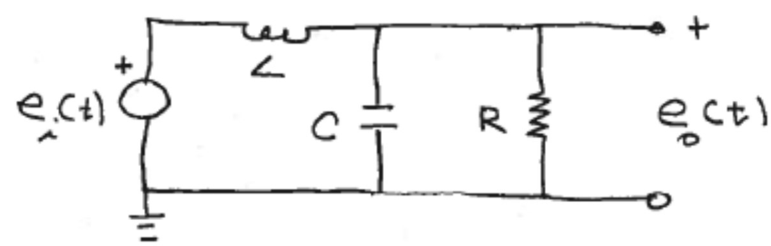
c) Calculate the experimental time constant and the resulting capacitor value from the frequency response data.

d) Compare the capacitor value of section 1.1 (d) to the capacitor values of 1.2 (c).

e) At the breakpoint frequency, what is the decibel drop in amplitude ratio?

2 Second Order System

The figure below describes a second order RLC circuit. The input $e_i(t)$ is supplied by the NI function generator. The output voltage $e_o(t)$ is the voltage across the capacitor C.



2.1 Step Response of a Second Order System

1. Measure the resistor in your circuit using the NI DMM, or the hand held multimeter at your lab station.

2. Connect the NI function generator to the NI scope and to the circuit input $e_i(t)$, and adjust for a square wave of 2 volt amplitude. Also, connect the output voltage of the circuit $e_o(t)$ to the scope. Once again, observe the input voltage $e_i(t)$ before and after it is connected to the second order circuit and note any differences.

3. Adjust the frequency of the square wave so that a complete step response is observed (i.e. to steady state) on the NI scope (between 1-10 Hz). *Note that a trigger is again recommended.* Use the same scale and origin for both input and output curves.

4. Sketch both the input and output curves, and save the data.

5. Adjust the input for a -3 V to +5 V square wave. Once again, sketch both the input and ouput curves and save the data. You should note any differences/similarities between the two output curves. *Note: if the peaks are clipped, you may need to reduce the input voltage amplitude*.

a) Derive the differential equation of the system.

b) Compute the experimental damping ratio and the undamped natural frequency for both input cases and discuss any differences. Use the log decrement method.

c) Calculate L (H) and C (μF) using the computed natural frequency and damping ratio from b).

d) Use Matlab/Simulink to simulate the theoretical step response of the system. Use the L and C values calculated above. Plot your experimental data on the same graph (as the theoretical) and compare the curves. Again, shift/scale the theoretical response appropriately to compare to the experimental curve.

2.2 Frequency Response of a Second Order System

This part of the lab is similar to that for your first order system. Your input to your system will be a sinusoidal input voltage of varying frequencies. You will find the amplitude ratio and phase at a low frequency, at the breakpoint frequency, and at a higher frequency.

1. Adjust the FGEN to create a sine wave of 2.0 V amplitude. Measure the exact input amplitude using the scope.

2. Sweep the frequency of the sine wave over a broad range of frequencies (10 Hz to 10 kHz) and note how the output changes. Find and record the frequency which gives the maximum output amplitude (very close to the break frequency). Make sure you record the phase shift and amplitude at this frequency

3. At input frequencies of 1/20, ¼, ½, 3/2, 2, and 4 times the break frequency, find and record the output amplitude and phase shift.

a) Compute the amplitude ratio in decibels, and phase angle in degrees for each frequency measurements taken in section 2.2.

b) Derive the transfer function of the second order system from the differential equation of part 2.1(a). Use Matlab/Simulink and the L and C values of part 2.1(c) to find the theoretical frequency response (Bode plot) for the transfer function. Then plot your experimental data on the same Bode plot and comment on the comparison.

c) Determine the damping ratio and undamped natural frequency from the measurements and compare to that obtained from the step response, part 2.1 (b).

d) Calculate L and C values from the measurements and compare to those from part 2.1 (c).

3 Frequency Response Using LabView

Using the NI equipment and LabView, you will find the frequency response (Bode Plot) for the first and second order systems. (An alternative approach is to use a dedicated spectrum analyzer such as the HP we have in the lab). The input will be a broadband noise (random signal), generated by the NI/LabView system. Both input and output signals will be used by LabView to find the amplitude ratio and phase of the system. Often, the cross power spectral density divided by the input power spectral density is used to determine the frequency response.

Use the directions provided at the end of this document (Mike DeLeon 9/3/14) to obtain Bode plots for the first and second order electrical systems.

3.1 Frequency Response of a First Order System Using LabView

1. Use LabView to obtain the Bode plot of the first order electrical system.

2. Use the cursor to find (and record) the system break frequency.

3. Sketch the Bode plot and save your data.

a) Using Matlab, plot the theoretical frequency response ON the experimental Bode plot and compare the two plots.

b) Use the experimental Bode plot to find the system time constant. Compare this to the experimental values found in section 1.1 and 1.2.

c) Calculate the capacitor C value from the LabView Bode plot and compare these values to that found in part 1.2 (d).

3.2 Frequency Response of a Second Order System Using LabView

1. Use LabView to find the Bode plot of the second order electrical system.

2. Use the cursor to find (and record) the system break frequency.

3. Sketch the Bode plot and save your data.

    a) Using Matlab, plot the theoretical frequency response ON the experimental Bode plot and compare the two plots.

    b) Using the experimental Bode plot, find the experimental damping ratio and natural frequency. Compare these values to the experimental values of section 2.1 and 2.2.

    c) Calculate L and C values from the experimental Bode plot and compare to those from part 2.2 (d).

Using LabView for Frequency Responses

M.H. deLeon
September 3, 2014
edited M. Thein
September 18, 2015

1. Find LV-SE project file "FreqResp_ DAQnn.seproj" in the "LabViewSE Templates" directory on the PC's "Desktop". Open it in LV-SE. Make sure you pick the one with name/number (nn) associated with your NI/DAQ number.

2. Connect your filter board's input to the NI DAQ via analog output (ao0) terminals 12 (+/red) and 14 (-/black). Use BNC adapter and cable. (This LV-SE template generates broadband noise and outputs it to the ao0 output on the DAQ.)

3. Exit the ScopeSoft program on the PC to enable the LV-SE program to access the NI digitizer (PXI-5142) via the "NI-SCOPE Acquire" step.

4. Make the following connections:
   - the noise output of DAQ Ch. 0 to the filter board input.
   - the filter board input monitored to 'scope Ch. 0.
   - the filter output monitored to 'scope Ch. 1.

5. Once all connections have been made , click on the "Data View" tab to look at both the magnitude and phase response plots. *Note that you need to follow this procedure precisely. (i.e., You can NOT switch the channels.)*

6. Press the "Run" button near the upper left corner and watch the averaging algorithm compute and plot the frequency response of the circuit.

7. Hit the "Stop" button when the averaging produces smooth curve (converging on a solution). You might have to try a few times until you obtain a good curve.

8. There are two quick ways to analyze these plots:
   (a) Use the zoom tools in LV-SE to estimate the break frequencies.
      i. You can use the zoom tool (magnifying glass symbol) below each plot, then the hand tool to pick up and move the graph so that the magnitude or phase of the portion of interest is displayed directly below the left axis. Try lining up a resonant peak of the magnitude plot to the left edge of the window, for example; or try moving the phase response plot so that the curve intersects either the -45 or -90 degree points on the y-axis.

(b) Export Plot Data to MS Excel:

   i. Hover the cursor over the magnitude (upper) plot and right-click to get pop-up menu. Select "Export To" then "Microsoft Excel". This will open a MS Excel spreadsheet and dump the plot's coordinate data (x, y pairs) into it. Do the same for the phase (lower) curve. Now you should have two sets of data (in two separate worksheets). Save this file to your local folder.

   ii. From these two data sets, you can plot the curves and/or study the data directly. To plot, highlight the frequency and magnitude (or phase) column pairs. For these 1st- and 2nd-order, passive circuits, it is recommended that you select the very lowest frequency and only include a moderate frequency above the break frequencies (e.g., 1000-2000 Hz. should suffice). Once the data are highlighted, select Insert-Scatter (under "Charts") and "Scatter with Smooth Lines" to insert the plot into the spreadsheet. You can then hover the cursor over the inflections or peaks in the magnitude curve to estimate the break frequency. The phase plot can similarly be analyzed for break frequencies by knowing at what phase angles the breaks occur. (TIP: You will probably want to change the default x-axis format to "log", as it is much easier to apply Bode analysis techniques to identify break frequencies this way. Hover the cursor over the x-axis on the plot, right-click, and edit axis properties to select "log". The y-axis should be kept linear, as the data is already saved in logarithmic units of decibels.)

9. Also open the Snipping Tool and highlight the entire pair of frequency response plots; save the screenshot image to your own folder with an appropriate file name.