

# EE 324 LAB 3

Modeling Transfer Functions Using MATLAB scripts and Simulink

In this lab, you will be working toward building a connection between the Laplace transform of a system, the frequency domain representation of the system, and the time-domain features of the system's response. To do this, you will practice predicting behavior using Laplace analysis and test using both scripts and Simulink models.

## Prelab:

Consider the circuits (systems) in Figure 1.

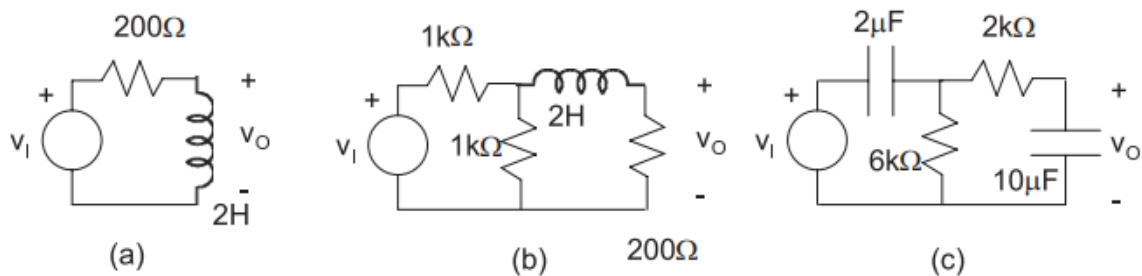


Figure 1

We want to understand what these systems do, and how they are expected to behave. Because they are “physical systems” they are causal.

For each of the circuits:

1. Use the time-domain integration or differentiation properties of the Laplace transform to derive the impedance of the capacitors and inductors (recall,  $v = L \frac{di}{dt}$ ;  $i = C \frac{dv}{dt}$ ). Since both capacitors and inductors are causal systems, what are the regions of convergence of their Laplace transforms?
2. Derive the transfer function (from input voltage to output voltage, in Laplace representation) of each circuit. Compute its pole(s) and zero(s), and the region of convergence. What is the effect of resistors on the poles?
3. Recalling that a capacitor acts as open circuit at low frequency and short circuit at high frequency, while an inductor does the opposite, intuitively determine the output at low ( $\omega \rightarrow 0$ ) and high ( $\omega \rightarrow \infty$ ) frequencies.
4. Substitute  $s = j\omega$  explaining why this is possible, and compute the magnitude response. Also verify analytically your intuition about how each system reacts to low frequency condition of the input signal, and to very high frequency condition.

5. If the input is a unit step function  $u(t)$ , what are the main features you will expect to see in the output? Justify by formally deriving the time-domain response of each system to a unit step (you will use a partial fraction expansion, and the inverse Laplace transform for each fraction term).

### Laboratory Assignment:

For each of the circuits in Figure 1:

1. Use the MATLAB functions `tf`, `pole`, `zero`, `residue`, `impulse`, `step`, and `bode` (for a description of each of the functions, use “MATLAB Help” or the `help` command) to verify the correctness of your analytical results from the prelab.
2. Using the Simulink block, “Transfer Function”, build the equivalent Simulink diagram of the system.
3. Test your systems with different impulse and step signals. Observe the corresponding output, and verify that it is consistent with both your prelab analysis.  
(Impulse may be approximated using the steps:  $N \left[ u(t) - u\left(t - \frac{1}{N}\right) \right]$ , with  $N \cong 10^6$ ).
4. Test circuit (a) of Figure 1 with different sinusoids at various frequencies. Observe the corresponding outputs noting both the amplitude gains/attenuations and phase shifts at various frequencies, and verify that it is consistent with both your prelab analysis. *Be careful when you simulate response to sinusoidal inputs, wait for the transient to vanish before you make conclusion on the frequency response. Why?*

In your report, include all your answers to the questions, describe your analytical reasoning and conclusions, and support with your simulation data and plots.

## Lab Tips:

1. For circuit 1, your coefficients should be [1 0] in the numerator –  $\mathbf{1}(s^1) + \mathbf{0}(s^0)$ . This is a common mistake. (Q1)
2. For part 3, the step and impulse responses should match up with part 1. (Q3)
  1. The only exception is the impulse response for circuit 1, where most students were not able to get the correct output. This may be a MATLAB issue.
3. For part 4, if you get many ripples in your sinusoidal output, adjust your Simulink settings to fixed-step and set the time step. (Q4)
  1. Some students saw this at low frequencies ( $\omega < 1$  rad/s) even when changing the step settings – you can skip the simulation/data collection for this frequency since you will not be able to get a time shift.
4. In some cases, you may need to adjust the simulation time (for part 4) or the scope axes (CTRL + A) to see the correct output so be sure to try that.
5. If any of your plots do not appear as expected, explain what you expected to see and what may have gone wrong.

## Report Requirements:

### Part 1

For each system:

1. System parameters
  1. Pole (returned by pole() or residue())
  2. Zero (returned by zero())
  3. Residue (R, returned by residue())
  4. Constant (K, returned by residue())
  5. Gain (returned by zero())
2. Plots (3x3 = 9 plots)
  1. Impulse response
  2. Step response
  3. Bode plot

### Part 2

1. Block diagrams
  1. You can include all 3 or just 1 since they are all the same.
  2. If you only include one block diagram, specify the coefficients for each circuit used in the Transfer Fcn block.

### Part 3

For each system:

1. Plots (2x3 = 6 plots)
  1. Impulse response
  2. Step response

#### Part 4

For system 1:

1. Data table (frequency, gain, phase, and others if you want).
2. Plots (2 plots) - Use loglog() or semilogx()
  1. Frequency (x) vs. magnitude (y)
  2. Frequency (x) vs. phase (y)