ELEC 4700 Assignment-4 Circuit Modeling

Due: Sunday, Apr. 11, 2021 23:59

Submission

The assignment submission should be two files, a .zip of the repo and code, and a .pdf of the report. cuLearn will accept only two files on your submission so make sure they are these.

The code should be organized such that the master branch of the repo contains the submission version and the entry point script (the one that creates the report or runs all the other code) is titled as the repo. Any additional code should be stored in a folder titled code.

In order to create a repo for your assignment, you can follow these instructions:

1. On your local machine, make sure you do git init in your assignment folder.
2. In the same directory as your assignment code and PDF commit all changes you've made to your project with first git add −A and then git commit −m "MyMessage"

As mentioned before, the submission should have the format of:

1. The report
2. A zip folder name "code" containing the code used to create the report

Your code will be checked using the latest version of MATLAB as found on DOE computers. Please ensure that your code clearly indicates which question and which part it is answering.

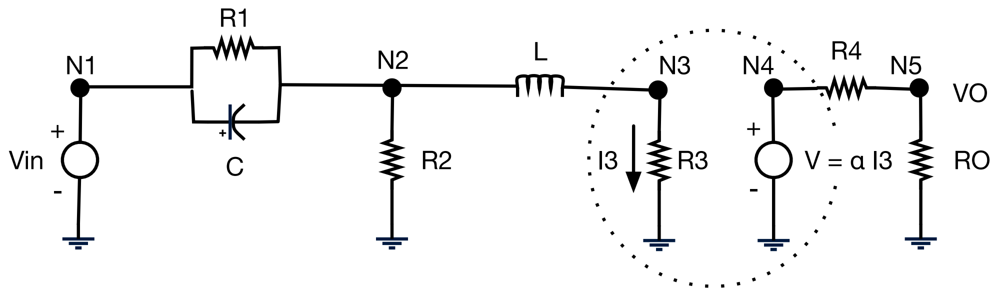

Figure 1: Circuit: $R_1 = 1$, $C = 0.25$, $R_2 = 2$, $L = 0.2$, $\alpha = 100$, $R_4 = 0.1$ and $R_O = 1000$.

For the circuit in Figure 1 use the work done in assignment 3, and PA 9 on MNA building and circuit simulation.

1. Using a fixed bottled neck value and the simulation code for assignment 3, do a voltage sweep of the device from 0.1 V to 10 V and model the current-voltage characteristics.
2. Use a linear fit to determine the resistance value of the device and use this value as R_3
3. Report on the work done in PA 9.
   (a) Formulation:
      i. Write out the differential equations that represent this network in the time domain using KCL ($\sum I = 0$ at each node).
      ii. Now write them in the frequency domain $\frac{dY}{dt} \rightarrow j\omega Y(\omega)$
      iii. Write down the matrices **C**, **G**, and the vector **F** that can be used to describe the network using:

$$\mathbf{C}\frac{d\mathbf{V}}{dt} + \mathbf{G}\mathbf{V} = \mathbf{F}$$

or

$$(\mathbf{G}+j\omega\mathbf{C})\mathbf{V} = \mathbf{F}(\omega)$$

   (b) Programing:
      i. For the DC case sweep the input voltage V_1 from -10V to 10V and plot V_O and the voltage at V_3.
      ii. For the AC case plot V_O as a function of ω also plot the gain $\frac{V_O}{V1}$ in dB.
      iii. For the AC case plot the gain as function of random perturbations on C using a normal distribution with $std = .05$ at $\omega = \pi$. Do a histogram of the gain.
4. Transient circuit simulation.

   The circuit can be represented in the time domain as:

$$\mathbf{C}\frac{d\mathbf{V}}{dt} + \mathbf{G}\mathbf{V} = \mathbf{F} \tag{1}$$

(a) By inspection what type of circuit is this?

(b) What sort of frequency response would you expect?

(c) Derive a formulation using finite difference for the numerical solution of this equation in the time domain:

(d) Write a Matlab program that can simulate the circuit using this formulation

   i. Simulate for 1s and use 1000 steps.

   ii. Use three input signals:

      A. A step that transistions from 0 to 1 at 0.03s.

      B. A $\sin(2\pi f t)$ function with f = 1/(0.03) 1/s. Try a few other frequencies. Comment.

      C. A guassian pulse with a magnitude of 1, std dev. of 0.03s and a delay of 0.06s.

   iii. Plot the input V_{in} and output V_O as the simulation progresses

   iv. After the simulation is complete use the *fft()* and *fftshift()* functions to plot the frequency content of the input and output signals.

   v. Increase the time step and see what happens. Comment.

Include in Report (a) C and G matrices (b) Plot of DC sweep (c) Plots from AC case of gain ($\frac{V_{out}}{V_{in}}$) (d) Plot of V_{in} and V_{out} from numerical solution in time domain (e) Fourier Transform plots of Frequency response

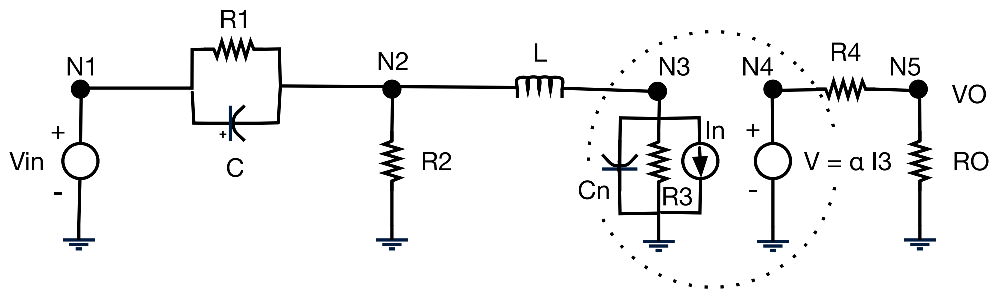

Figure 2: Circuit with noise source.

5. Circuit with Noise:
   (a) Add a current source I_n in parallel with R_3 to model the thermal noise generated in the resistor R_3. Refer to Figure 2.
   (b) Also add a capacitor $C_n = 0.00001$ in parallel with the resistor to bandwidth limit the noise. This will change your **C** matrix.
   (c) Simulation:
      i. Use this circuit to model the V_o signal with noise using the gaussian excitation.
      ii. Use a random number function to simulate the value of I_n and add this to your simulation.
      iii. Use a gaussian distribution for the random numbers. In order to be able to see the noise on the plot set the magnitude of noise current $I_n = 0.001$
      iv. Plot the new output V_o.
      v. Use the 'fft' function in Matlab to calculate the spectrum on the output. Plot this.
      vi. Vary C_n to see how the bandwidth changes. Comment on your results.
      vii. Vary the time step and see how that changes the simulation.

Include in Report (a) Updated C matrix (b) Plot of V_{out} with noise source (c) Fourier Transform plot (d) Plot of fourier transform (e) 3 Plots of V_{out} with different values of C_{out} (f) 2 Plots of V_{out} with different time steps

6. Non-linearity:

   (a) If the voltage source on the output stage described by the transconductance equation $V = \alpha I_3$ was instead modeled by $V = \alpha I_3 + \beta I_3^2 + \gamma I_3^3$ what would need to change in your simulator?

   (b) Describe the changes and the implementation.

   (c) **Bonus: (10pts)** Implement it!

   Include in Report (a) Description of steps needed to implement it and changes that would need to be made to simulator (b) (For Bonus) V_{out} plot and fourier transform plots of new system