

ECE-205 : Dynamical Systems

Homework #2

Due : Monday March 18 at 5 PM

1) Problem 2.11 from the Notes

2) Problem 2.12 from the Notes

3) Problem 2.13 from the Notes

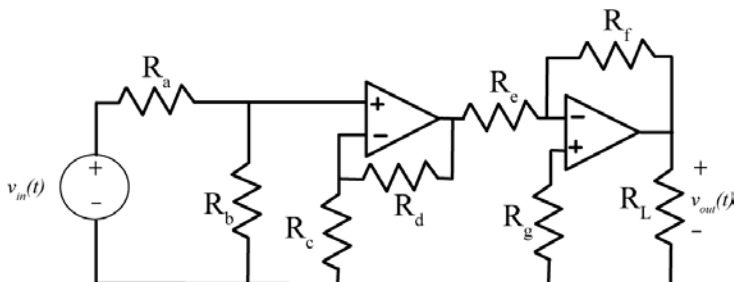
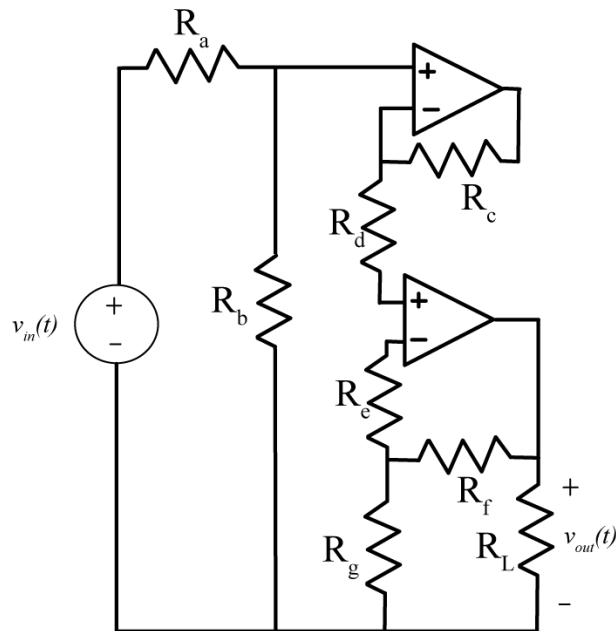
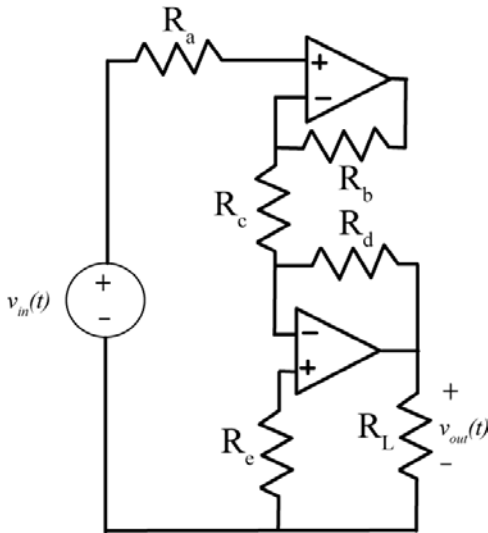
4) Problem 2.14 from the Notes

5) Problem 3.1 from the Notes

6) Problem 3.2 from the Notes

7) Problem 3.3 from the Notes

8) For the following op-amp circuits we can write $v_{out}(t) = Gv_{in}(t)$. Determine an expression for G .



Answers: $G = \frac{-R_d}{R_c}, G = \frac{R_g + R_f}{R_g} \frac{R_b}{R_a + R_b}, G = \frac{-R_f}{R_e} \frac{R_c + R_d}{R_c} \frac{R_b}{R_a + R_b}$

9) Matlab/Simulink Problem

One of the standard forms for a second order system is

$$\ddot{y}(t) + 2\zeta\omega_n\dot{y}(t) + \omega_n^2y(t) = \omega_n^2Kx(t)$$

where ζ is the damping ratio, ω_n is the natural frequency, and K is the static gain. Use this form of the standard second order system in the remainder of this problem.

We want to simulate a system described by this differential equation using a Matlab driver. Just as you did for a first order system, you need to solve for the highest power derivative (as a function of the input and lower power derivatives). If you then take $\dot{y}(t)$ and run it through an integrator you will get $y(t)$ and if you run this through an integrator you will get $y(t)$. Hence you will need two integrators and two feedback loops and one input into your summing block (click on the summing block and modify it to get three inputs). You may need to click on the gain block and then choose **flip block** to get the correct direction.

- We want to look at the step response, so the input to your system should be a step. The amplitude of the step should be controlled by the Matlab program, the length of the simulation should be controlled by the Matlab program, and the step should start at time zero.
- Your Simulink file should only contain variables (static gain, natural frequency, damping ratio, amplitude of the step, length of the simulation)
- Plot the transient output of the system and the steady state output KA on the same graph.
- If you use the parameters $\zeta = 0.1$, $\omega_n = 20$, $K = 2$, $A = 1$, and $Tf = 3$ (final simulation time), you should get results like that shown in the following figure. Print out your figure, your Matlab code, and your Simulink model and turn them in.
- Note if you get a jagged plot (not a nice smooth one), go through the following steps:

At the top of the Simulink window, click on **Simulation**, then click on **Configuration Parameters**, and then set (fill in the box) the **Max step size** to $1e-3$ (or smaller)

