

Name \_\_\_\_\_ Date \_\_\_\_\_ Lab Section \_\_\_\_\_

This is a calculation/simulation-oriented lab to be completed individually.

**Problem 1**

Determine  $z_{11}$ ,  $y_{21}$ , and  $h_{21}$  for the circuit shown below, both by hand calculation (must show work) and Pspice simulation.

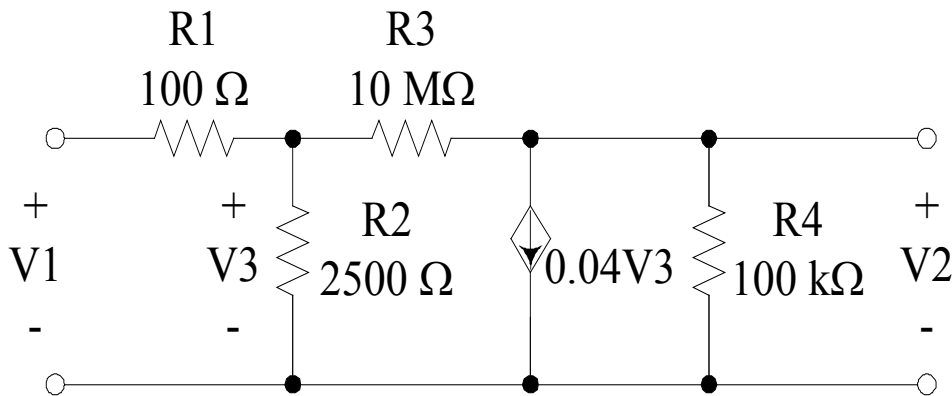
**Problem 2**

Add a capacitor,  $C_2 = 6\text{pF}$ , in parallel with  $R_2$ , and a capacitor,  $C_3 = 2\text{pF}$  in parallel with  $R_3$ . Use Pspice to plot the magnitude of the voltage gain  $|V_2/V_1|$  vs. frequency from 10Hz to 10MHz.

**Results**

Submit all of your results using this sheet as a cover. Your report show include your hand calculations, a copy of your Pspice circuits, your Probe plots, and your results. Clarity and organization counts.

**Remember:** A probe plot that is not clearly marked with appropriate data points provides nothing of value to the reader. **Comment your work well.**



**Review**

Chapter 26 in your Boylestad textbook (available in a PDF file located on the course materials – ECET 2113 web page) will provide you with a good review of two-port networks. Note that the circuit of this laboratory exercise contains a voltage-controlled current source. A voltage-controlled current source is modeled in PSpice with a symbol of “G” (be sure to set the gain attribute of “G” to 0.04). Each chapter has a Pspice example included, and chapter 14 has a good example for Time Domain (Transient) analysis setup.