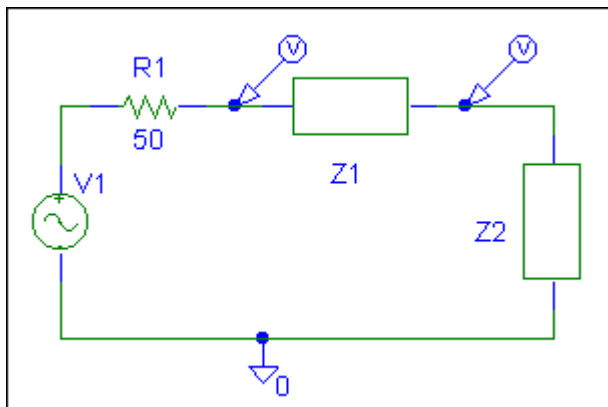


**Practical Quiz #3
PSpice Simulation**



Using PSpice, you are to simulate the operation of this circuit through the range of frequencies from 1Hz to 15MHz. Z1 and Z2 are resistors, capacitors or inductors. Possible values for these components are:

R1 = 100 Ω , R2 = 1k Ω , R3 = 10k Ω
 C1 = .001 μ F, C2 = .01 μ F, C3 = .1 μ F
 L1 = 6mH, L2 = 80mH, L3 = 125 μ H

Obtain the values of the two components you are to simulate from the TA or instructor.

Setup and run the simulation for this circuit. Be sure that your simulation is in the most useful possible form.

Plot (and print) the **magnitude** and **phase** of the two voltages shown vs. frequency.

Explain why your output makes sense. Your explanation must be accepted by the TA or instructor to obtain credit for this quiz.

Show your results and explanation to a TA or instructor and have them sign this sheet below.