

EE2CI5 Lab 6: Introduction to PSpice

1 Objective

The objective of this laboratory is to investigate the use of the industry-standard circuit simulation tool, PSpice, in the analysis of the frequency response of a simple circuit.

2 Equipment

PSpice software

3 Pre-Lab Exercise

3.1 Circuit Analysis (6 marks)

Consider the circuit model in Figure 1.

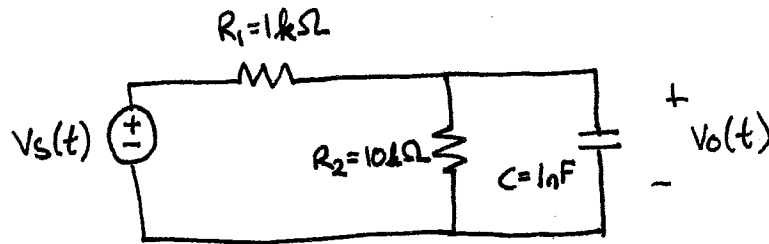


Figure 1: Circuit model for Lab 6. Here $v_s(t) = 5 \cos(\omega t)$.

Let \mathbf{V}_s denote the phasor for the source voltage, $v_s(t)$, and let \mathbf{V}_o denote the phasor for the output voltage, $v_o(t)$.

3.1.1 (2 marks) Show that the ratio of the phasor for the output voltage to the phasor of the input voltage is

$$H(\omega) = \frac{\mathbf{V}_o}{\mathbf{V}_s} = \frac{R_2}{R_1 + R_2} \cdot \frac{1}{1 + j\omega/\omega_0}, \quad (1)$$

where $\omega_0 = \frac{R_1 + R_2}{CR_1R_2}$.

3.1.2 (1 mark) What is the magnitude of the frequency response, $|H(\omega)|$? What are its characteristics and low and high frequencies? Explain these characteristics in terms of the properties of the capacitor.

3.1.3 (1 mark) Determine the magnitude and phase of the frequency response for frequencies:

$$\omega = \omega_0/10, \quad \omega_0/\sqrt{10}, \quad \omega_0/2, \quad \omega_0, \quad 2\omega_0, \quad \sqrt{10}\omega_0, \quad \text{and} \quad 10\omega_0. \quad (2)$$

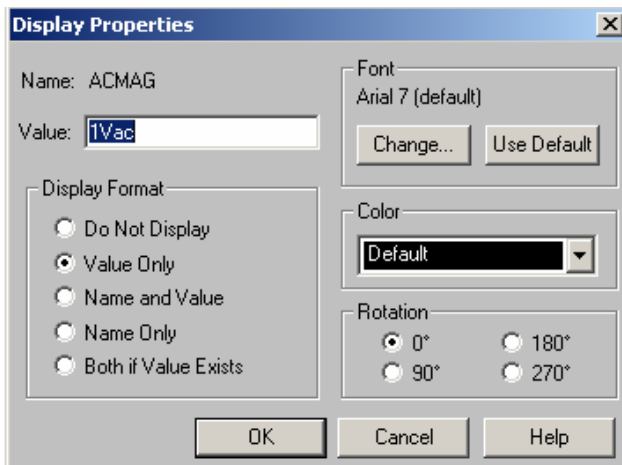
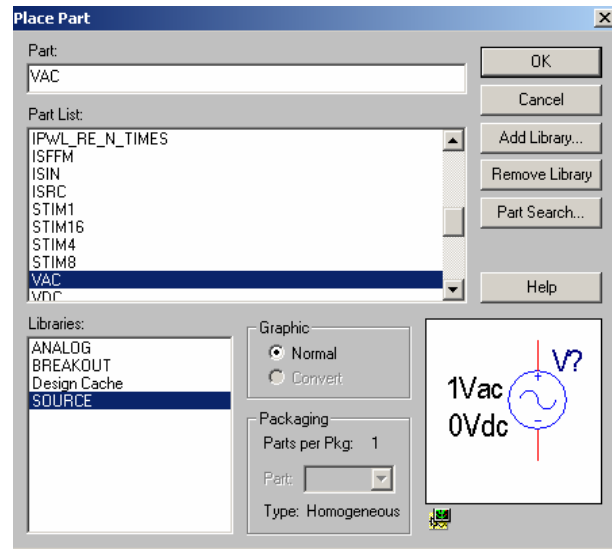
3.1.4 (1 mark) If $v_s(t) = 5 \cos(10\omega_0 t)$, what is $v_o(t)$?

3.1.5 (1 mark) Plot the magnitudes calculated in Step 3.1.3 on a graph of $20 \log_{10}(|H(\omega)|)$ versus $\log_{10}(\omega)$, and try to approximate that graph using two straight lines.

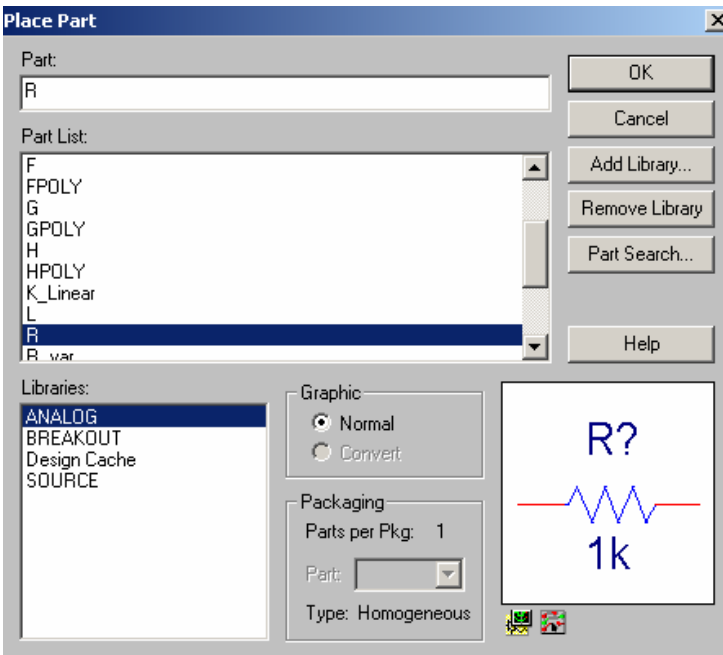
The remainder of the pre-lab exercise is to read the instructions on the following pages. These instructions describe how you can use PSpice to assist you in the analysis of this circuit.

Building the circuit: Create a new project in OrCAD Capture. We use the *Analog Digital A/D* option. Save the files to a folder of your choice.

We first require an AC Voltage Source. Click on the *Place Part* button and in the source library locate the AC voltage source VAC and place it on the schematic.



The source has a 0V DC bias which is already set by default. The AC component must have an amplitude of 5V. This value is changed by right clicking on the circuit element and selecting *Edit Properties...* In the VACMAG column change the value to 5Vac. Alternatively, one could double click on the VAC caption beside the circuit element to bring up the display properties dialog also in which the value can be changed.

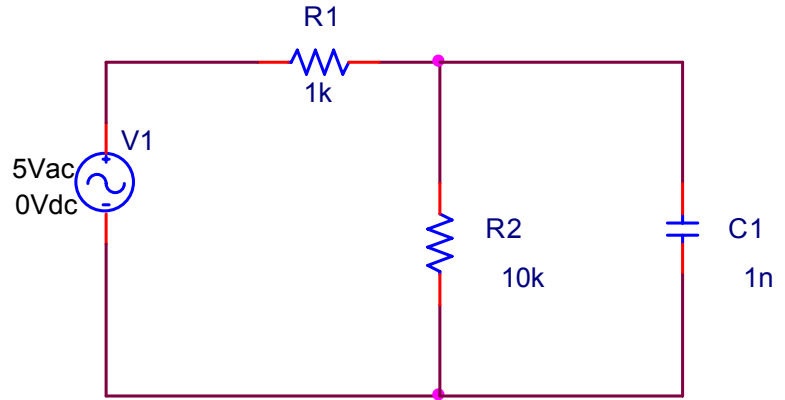


Next, two resistors must be added. We press the *Place Part* button on the right hand toolbar. We are looking for the resistor *R* which can be found in the *Analog library*.

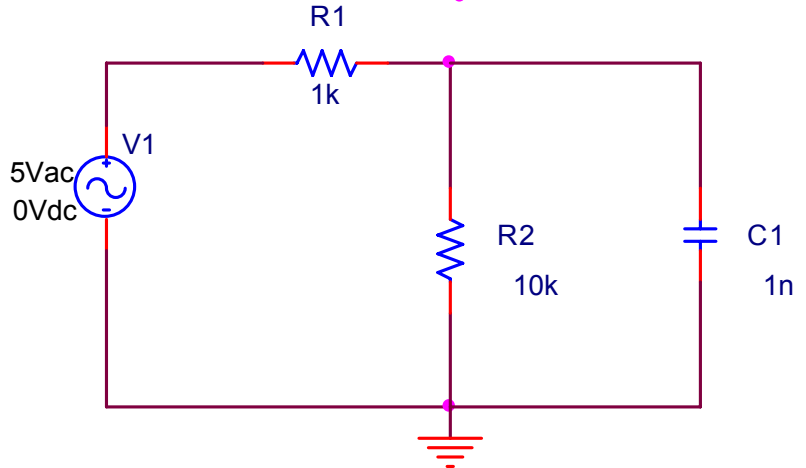
Place two resistors on the schematic and again select *End Mode*. Arrange the components such that they appear as shown.

Place the capacitor *C* in the same way. We are looking for the capacitor *C* which can be found in the *Analog library*

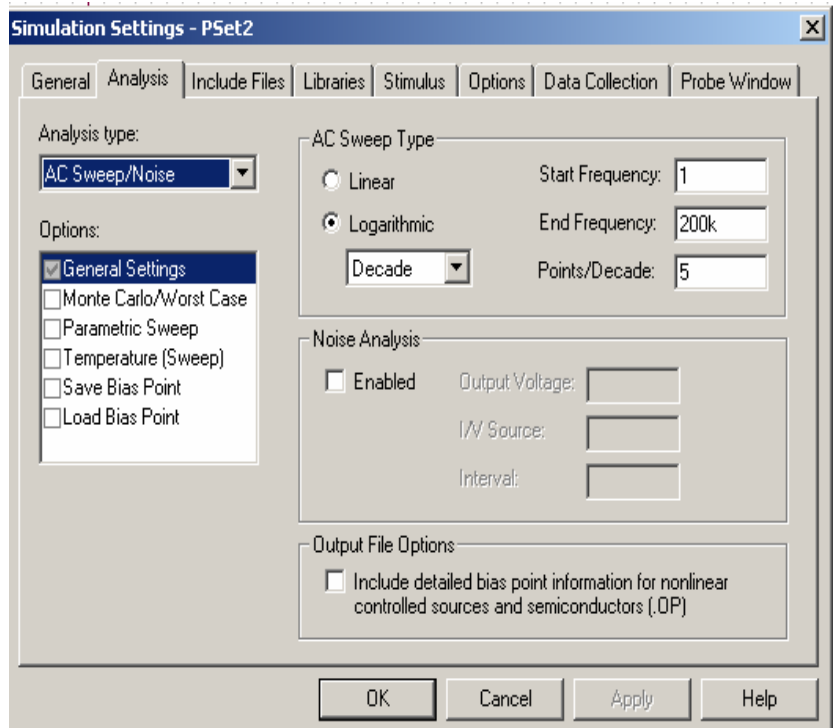
Use the place wire button to connect the elements to form the completed circuit. Do not forget to set the resistance and capacitance values for each circuit element (by right clicking on each component). In the end, the circuit should look similar to the one shown to the right.



It is easy to forget to set the reference node - but PSpice does not allow floating nodes and all nodes must have a path to ground. Set the bottom node to be ground – the reference node. Also edit the name of this node to be “0” (by right clicking on this node). The completed circuit looks as shown:



Click *PSpice* --> *New Simulation Profile* and give it a name of your choice. This simulation profile tells PSpice about the simulation type and parameters. This will bring up the *Simulation Settings Dialog*. In this experiment, we perform an AC Sweep.



A logarithmic AC sweep with a frequency range that includes 1Mrad/sec must be performed. PSpice uses Hertz as units rather than radians. So remember to use the formula $\omega = 2\pi f$ to convert 1Mrad/sec to units of Hertz. $f = \frac{1M}{2\pi} \cong 160kHz$
 Choose a frequency range of 1Hz – 200kHz. Ensure the settings look as they do on the screen to the right and click OK.

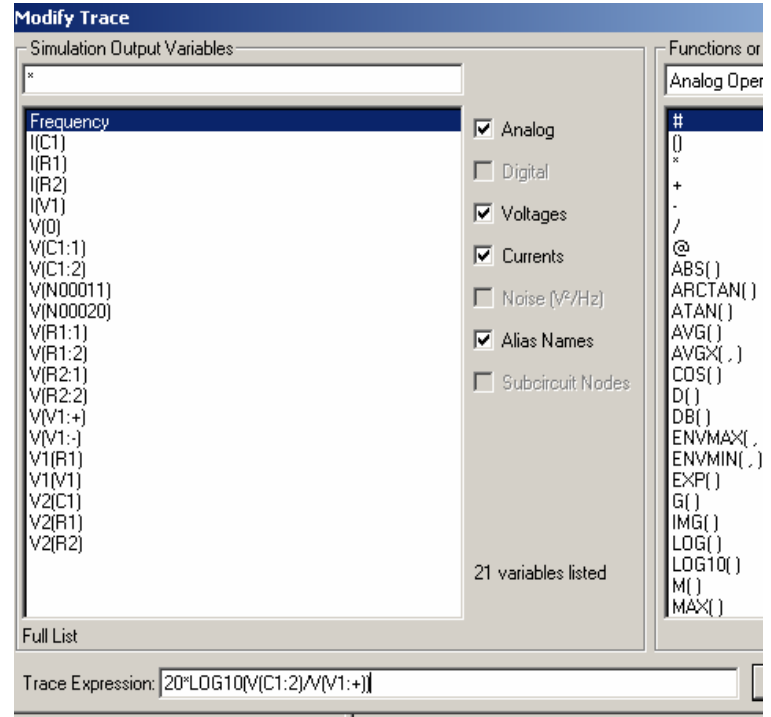
We are now ready to run our AC simulation. Click the *Run PSpice* button on the top menu toolbar. A black plotting area appears. Notice the x-axis is the logarithmic scale of frequency. We must plot the gain of the circuit in decibels (dB). To do this click the *Add Traces* button.

The *Add Traces Dialog* appears. Last time we plotted a simple voltage or current. This time we must graph the frequency response and thus need to plot the gain(transfer function) $H(j\omega)$ in dB. The gain can be expressed in dB as

$$20\log_{10}\left[\frac{V_{out}(j\omega)}{V_1(j\omega)}\right] \text{ dB. In our circuit } V_{out} \text{ is the}$$

voltage at the node above of the capacitor, V(C1:2). Thus enter the formula for the gain as shown to the right and click OK. Note that the voltage may need to be taken as V(C1:1) depending on the orientation of the capacitor!.

The frequency response is plotted on the black graphing area. This can be exported to another program easily by clicking *Window-->Copy to Clipboard*.



4 Experiment (4 marks)

- **(2 marks)** Following the instructions on the previous pages, use PSpice to plot the curve of $20 \log_{10}(|H(\omega)|)$ against $\log_{10}(\omega)$ for the circuit in Figure 1.
- **(2 marks)** Compare the accuracy of your straight line approximation to the actual curve. Where is the straight line approximation quite accurate? Where is it slightly inaccurate? Can you think of a way to adjust your straight line approximation in a way that might make it a little more accurate?