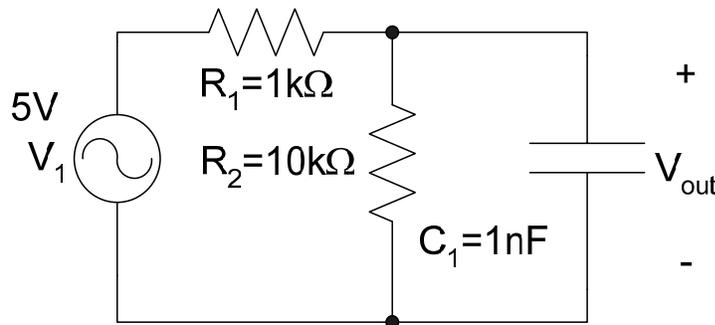


Experiment 8

Objective: To understand the principles of a simple AC circuit by calculating the frequency response and verifying the results with a simulation in PSpice

Procedure: Start by reviewing the supplied PSpice experiment to understand how the software works. Prepare a Prelab to show the analytical amplitude and phase frequency domain responses of the two discussed circuits. In the lab, you will verify your analytical answers using PSpice.

Example: Determine the frequency response and draw the linear approximation to the magnitude Bode Diagram for the circuit. Also classify the circuit (i.e. Type of filter). Verify the approximate Bode Plot (response in DB) using an AC analysis in PSpice.



Analysis:

To find the frequency response we can replace Capacitors and Inductors with their respective impedances and use node analysis to determine the ratio V_{out}/V_1 . We can then use a linear approximation to the Bode plot to approximate its shape.

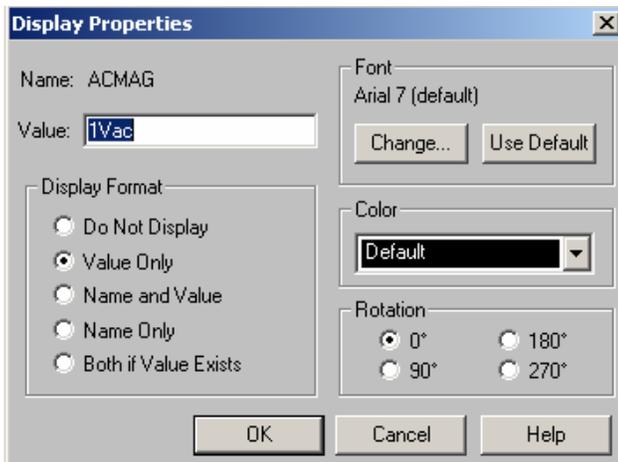
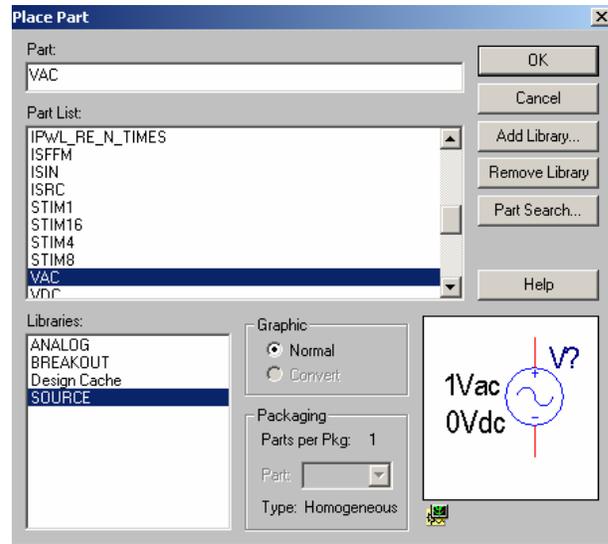
Show that the ratio of the output voltage to the input voltage is given by

$$H(j\omega) = \frac{V_{out}}{V_1} = \frac{R_2}{R_1 + R_2} \cdot \frac{1}{1 + j\omega/\omega_0} \quad \text{where } \omega_0 = \frac{R_1 + R_2}{C_1 R_1 R_2}$$

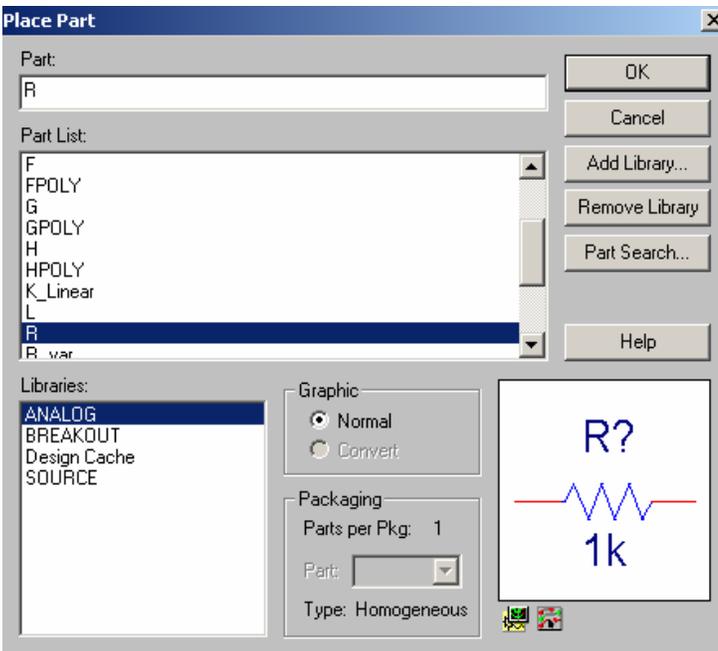
What is the type of this response?. Determine the magnitude and phase of this transfer function for the following frequencies: $0.1\omega_0$, ω_0 , $2\omega_0$, and $5\omega_0$.

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.

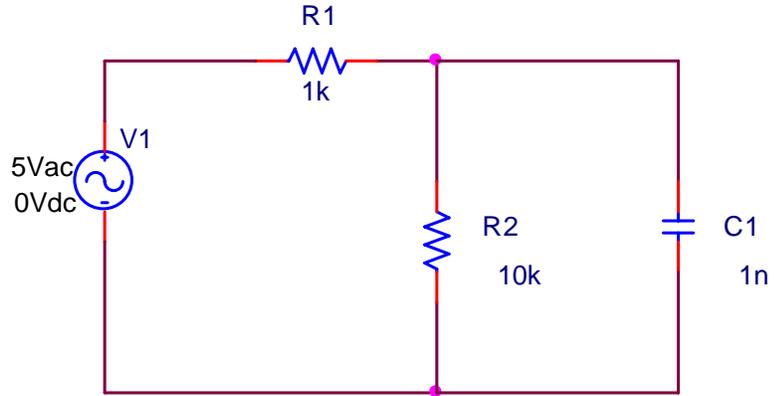


Next, two resistors must be added. We press the *Place Part* button on the right hand toolbar. We are looking for the resistor *R* which can be found in the *Analog library*.

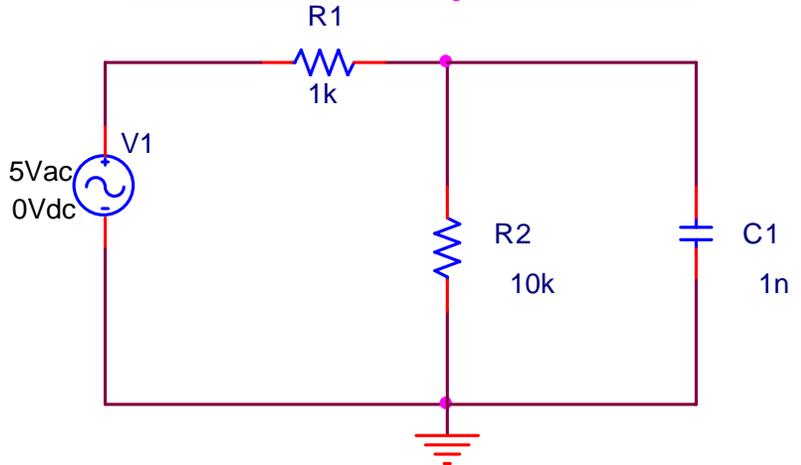
Place two resistors on the schematic and again select *End Mode*. Arrange the components such that they appear as shown.

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

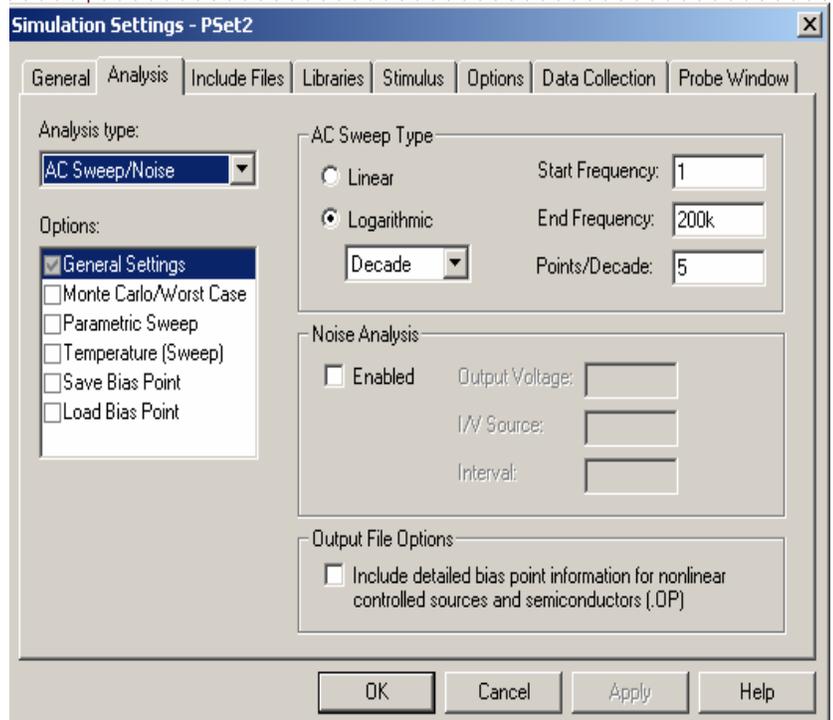
Use the place wire button to connect the elements to form the completed circuit. Do not forget to set the resistance and capacitance values for each circuit element (by right clicking on each component). In the end, the circuit should look similar to the one shown to the right.



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.

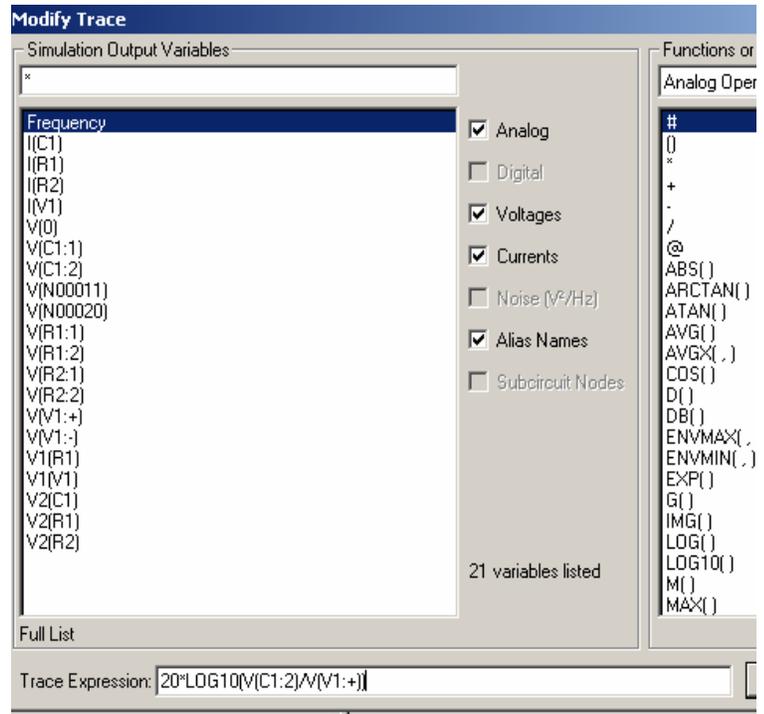
We are now ready to run our AC simulation. Click the *Run PSpice* button on the top menu toolbar. A black plotting area appears. Notice the x-axis is the logarithmic scale of frequency. We must plot the gain of the circuit in decibels (dB). To do this click the *Add Traces* button.

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} 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*.



Results: Compare your PSpice results with your analytical results. What is the type of this filter?. Repeat all the previous steps after switching C_1 and R_1 ? What is the type of this new filter?. Obtain the frequency response analytically and compare it with your PSpice results.