

## Investigation of RC circuits driven by ac currents

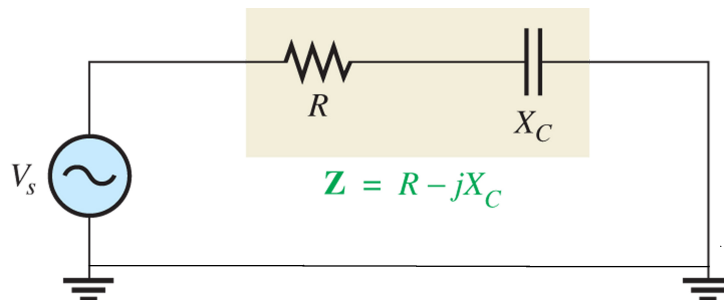
### AC potential divider: series RC circuit

#### Theory

The impedance of a resistor and a capacitor in an ac circuit are given by:

$$\mathbf{Z}_R = R \angle 0^\circ \Omega$$

$$\mathbf{Z}_C = X_C \angle -90^\circ \Omega = \frac{1}{\omega C} \angle -90^\circ \Omega = \frac{1}{2\pi f C} \angle -90^\circ \Omega$$



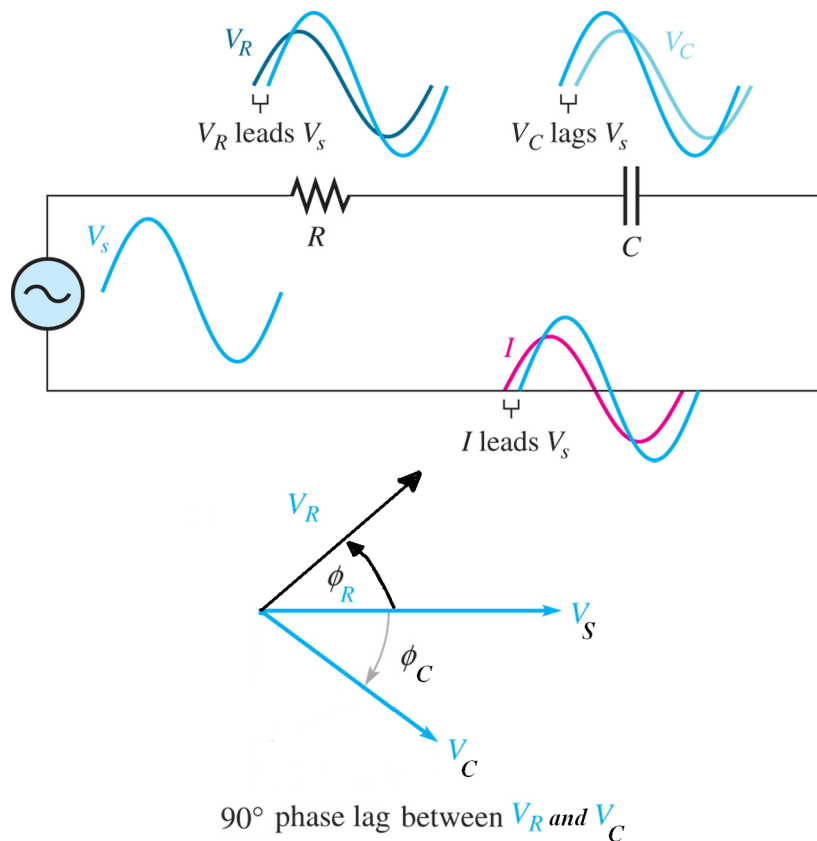
#### For a series RC circuit:

1. The current is constant  $I = I_R = I_C$
2. The total impedance  

$$\mathbf{Z}_T = \mathbf{Z}_R + \mathbf{Z}_C = R \angle 0^\circ \Omega + X_C \angle -90^\circ \Omega = R - jX_C$$
3. The total source voltage is equal to the sum of the two potential difference values  $V_s = V_R + V_C$
4. The two components form a potential divider across the source i.e.

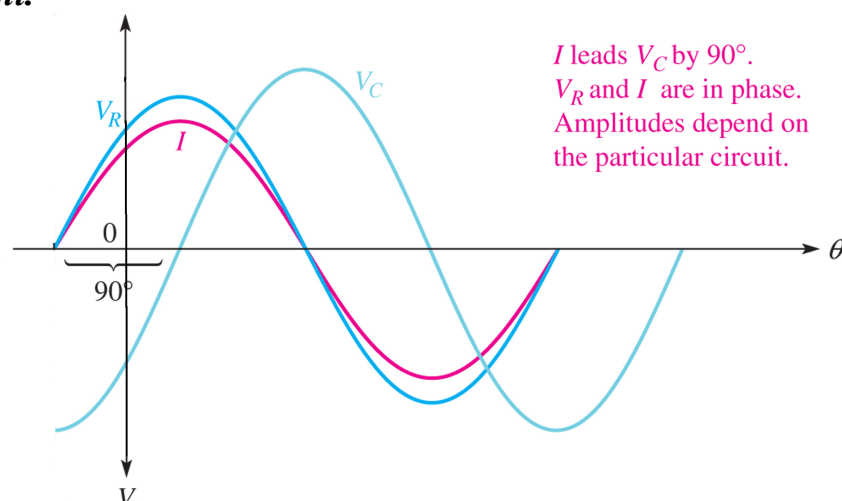
$$V_C = \left( \frac{\mathbf{Z}_C}{\mathbf{Z}_C + \mathbf{Z}_R} \right) V_s \quad \text{and} \quad V_R = \left( \frac{\mathbf{Z}_R}{\mathbf{Z}_C + \mathbf{Z}_R} \right) V_s$$

The relative phases of the voltages  $V_R$  and  $V_C$  must also be considered for the circuit. For the series RC circuit  $V_R$  is in phase with the current,  $I$  and  $V_C$  is  $\pi/2$  (90 degrees) out of phase with  $I$ . Thus  $V_R$  and  $V_C$  may be represented by two vectors which are  $\pi/2$  out of phase with each other. Taking  $V_s$  as the reference voltage  $V_R$  'leads'  $V_s$  and  $V_C$  'lags'  $V_s$ .



The impedance of the resistor  $R$  and the capacitor  $X_C$  are at an angle of  $90^\circ$  to each other. Since the reactance  $X_C$  of a capacitor depends on the frequency of the source voltage. As the frequency increases,  $X_C$  decreases,  $Z$  decreases, and  $\theta$  decreases. Therefore the total impedance depends on frequency and the ratio of the source voltage across each of the two components also varies as a function of that frequency,  $f$ .

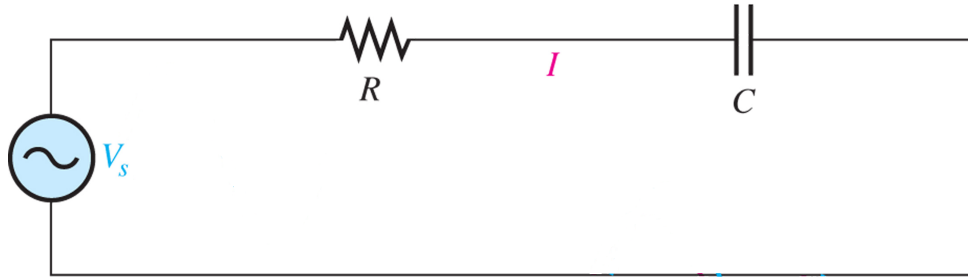
***The amplitude and the phase angle of  $V_R$  and  $V_C$  are both frequency dependant.***



By measuring  $V_R$  and  $V_C$  as a function of frequency this dependence of  $X_C$  on  $f$  can be confirmed.

## Experimental Procedure

Set up the circuit shown below:



1. Set the voltage output from the function (signal generator) to be approximately 2V ( $V_s \angle 0^\circ$ ). (Use CRO)
2. Set the frequency of the function generator to **1 kHz** (Use CRO)
3. Choose appropriate R and C values for the circuit.

Use the two channels of the **CRO** to measure the voltage values.

Remember to ground each of the cables (the black lead of the cathode ray oscilloscope must be earthed at all times and therefore must be connected to the ground of the function generator). When measuring voltage values one channel of the CRO should measure  $V_s$  at all times therefore  $V_R$  and  $V_C$  cannot be measured simultaneously. In the arrangement above as the circuit first measure  $V_s$  using Channel 1 and  $V_R$  using Channel 2. To measure  $V_C$  switch around the two components (**R** and **C**) on the protoboard, then  $V_C$  can be measured on Channel 2.

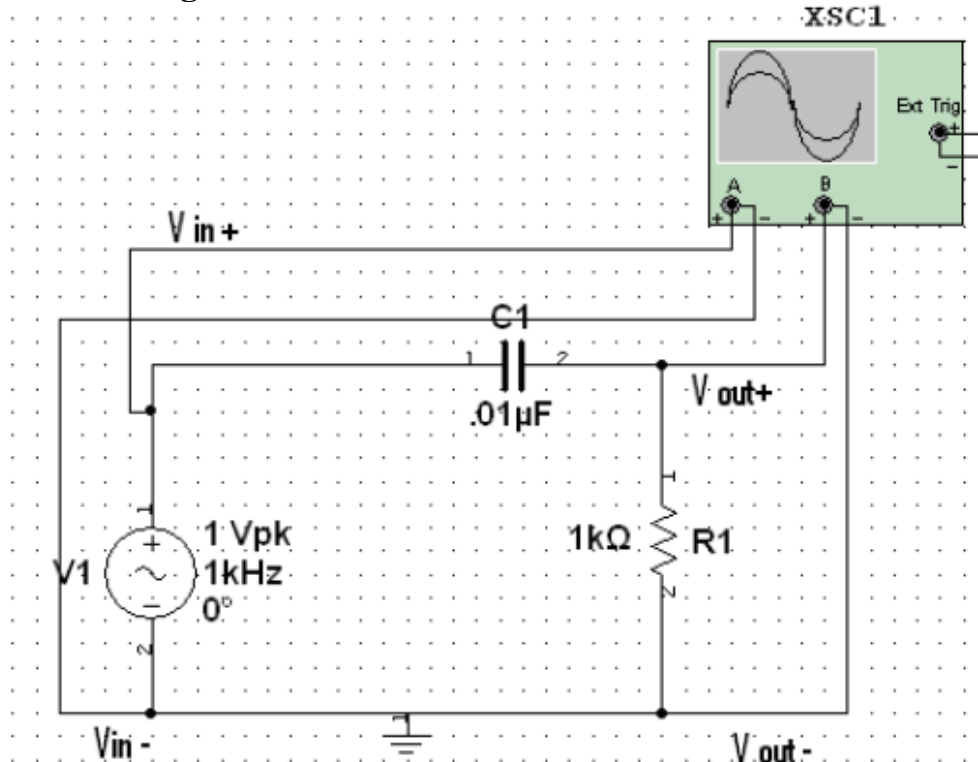
4. Measure and record the peak values of  $V_s$ ,  $V_R$  and  $V_C$  in table 1.
5. Measure and record the time lag  $\tau_R$  and  $\tau_C$ , with the total voltage as a reference value, in table 1. Note whether this value is positive or negative.
6. Repeat the measurements at a frequency of **5 kHz** and record the values in table 1.

$V_s(V)$	$V_R(V)$	$V_C(V)$	$\tau_R$	$\tau_C$	$T (ms)$	$f (Hz)$

**Table 1**

## Part 2: Introduction to using Multisim: Capture and simulate an RC Circuit

Consider the following RC series circuit

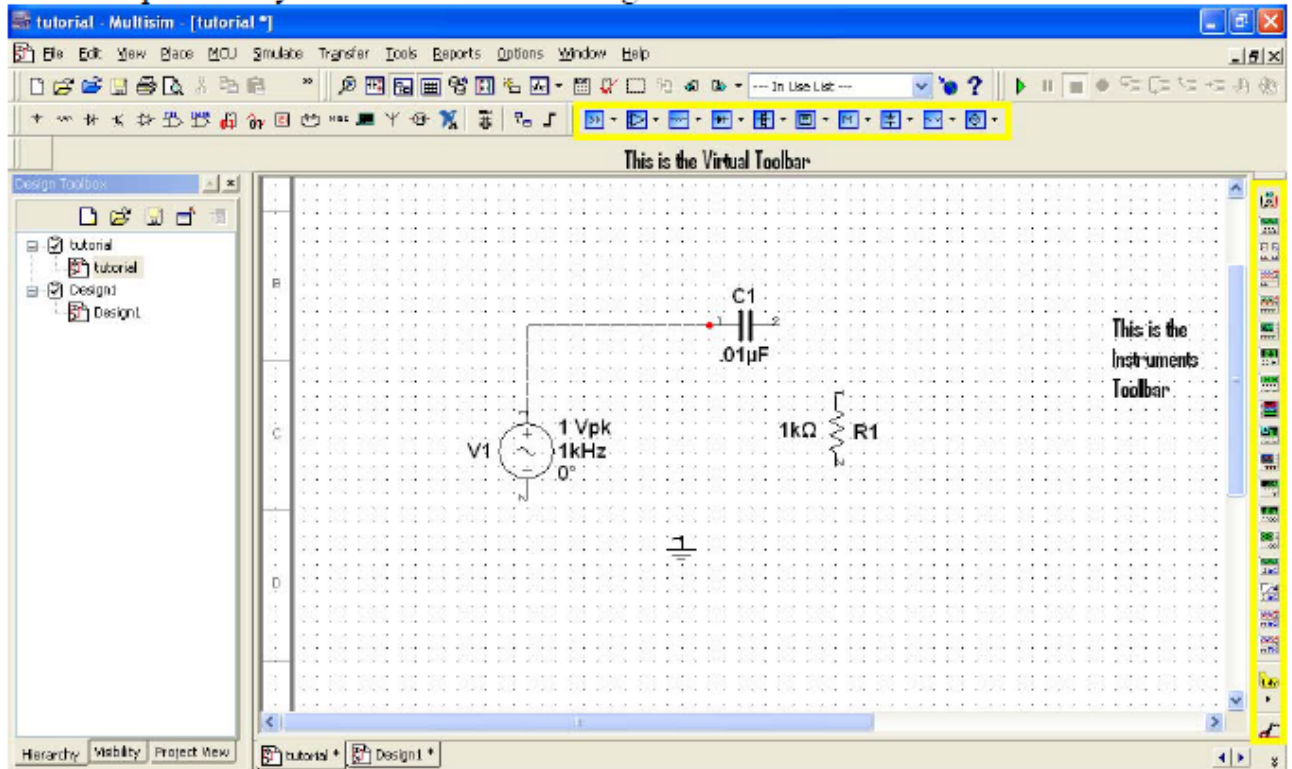


Open Multisim by clicking on Start -> Programs -> National Instruments -> Circuit Design Suite 11.0 -> Multisim 11.0 . Create a new file with File-> New-> Design.

First we need to find the components. There are a few ways: **Place -> Components** (Ctrl + W) or right-click a blank spot and go to Place Component. The AC source is in the “Sources” group (top left), “Signal Voltage Sources” family and is called “AC Voltage” select it and click on OK. A ghost component is now pinned to the cursor. Choose an appropriate spot and click to place the component. Components, and the associated labels, can be dragged after placement. Rotate components with **Ctrl+R**

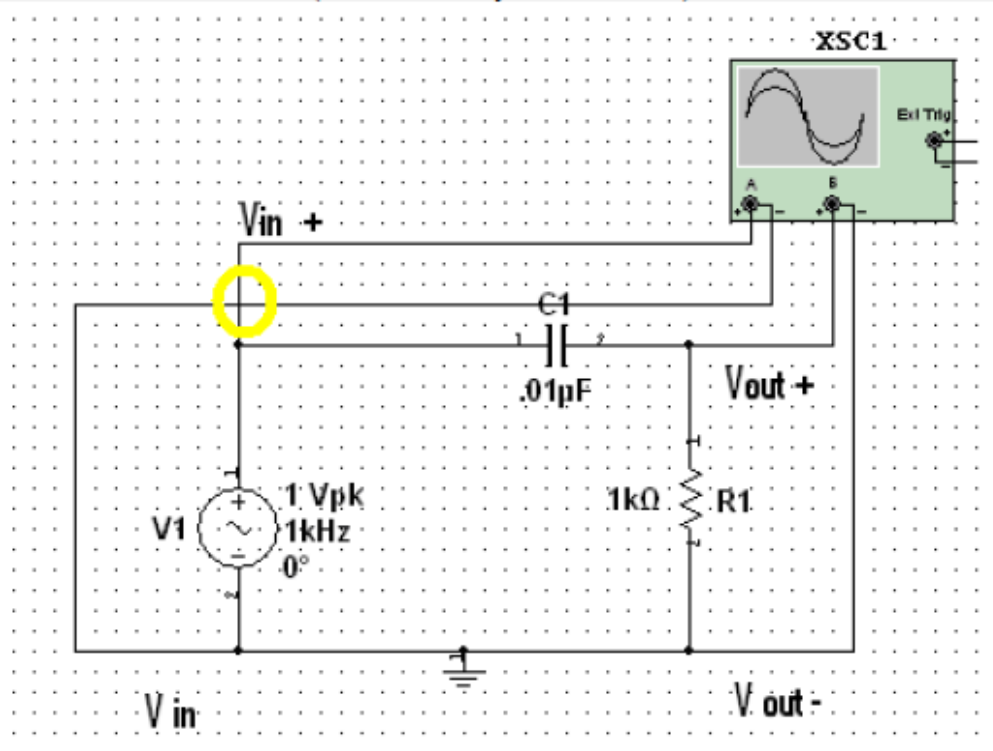
For the resistor let's use the Virtual Toolbar (shown below) to quicken the process. (If you don't see a toolbar with 9 blue buttons, right click any of the toolbars and check that "Virtual" is on.) Click on the "Basic" group (with a resistor icon) and choose "Place Virtual Resistor" (notice that you could have found the capacitor in the same place). Place the resistor and make sure it is the right value if it is not, double click it and change the value.

Up to now you must have something like this:



Wire the circuit by clicking on the end points of the components (do not worry about the "1" and "2"'s written on the ends of the components). If you need to make multiple connections at the same point you can add a Junction by going to **Place->Junction (Ctrl+J)**. You might be afraid that the connection has not been made since MultiSim does not really show a connection but as long as you click on the red dot that appears near the terminals the connection will be made. To connect a wire to a component you only need to click once near the red dot however to leave it loose you need to double click (the red dot is visible in the picture above.) A quick way to make sure a connection exist is by dragging the component and seeing whether the wire comes along.

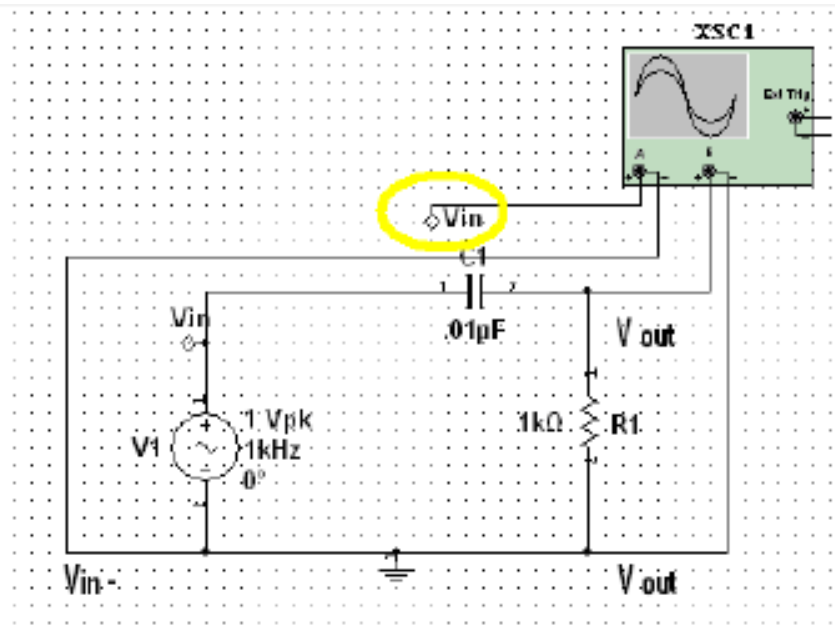
Now choose an oscilloscope from the right hand toolbar (the Instruments toolbar). You have four choices; for most circuits the generic 2-channel oscilloscope does the job. So click on the one that is simply called "Oscilloscope" and stick it in your favorite spot on the board. Channel 1 is normally used to look at the input of the circuit, and channel 2 is used for the output. After placing the oscilloscope and wiring it to the input and output terminals it should look like (without the yellow circle):



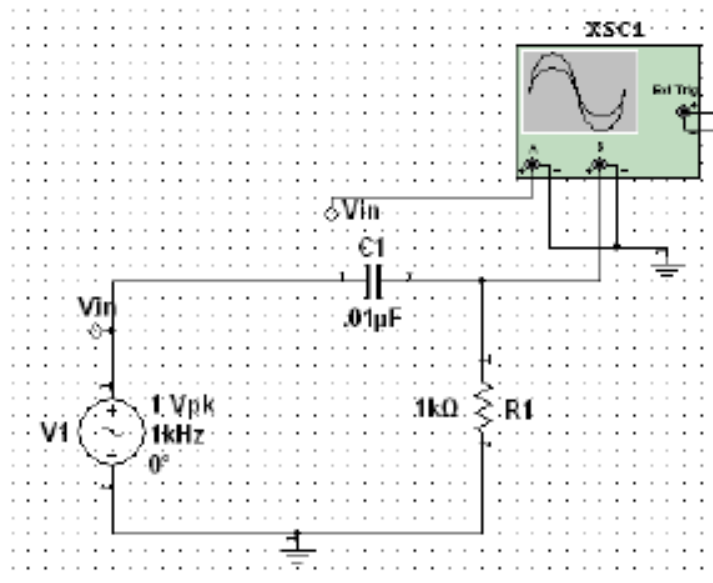
Notice that when two wires simply pass over each other there is no connection (inside the yellow circle) but where the wires are actually connected the connection is represented by a dot.

If the wiring is messy you can use an On Page Connector (in this circuit it might be overkill but it will be useful later), to use the On Page Connector go to **Place-> Connectors-> On Page Connector** (or **Ctrl+Alt+O**). In the box that appears name the connector something useful (e.g.  $V_{in}$ ) and place it somewhere so you can connect it to the positive side of the channel A of the oscilloscope. An On Page Connector behaves like a wire those with the same name are electrically connected so get another connector and in the window that appears double-click on the name that you chose earlier and put the connector near the positive side of the AC source as shown below.





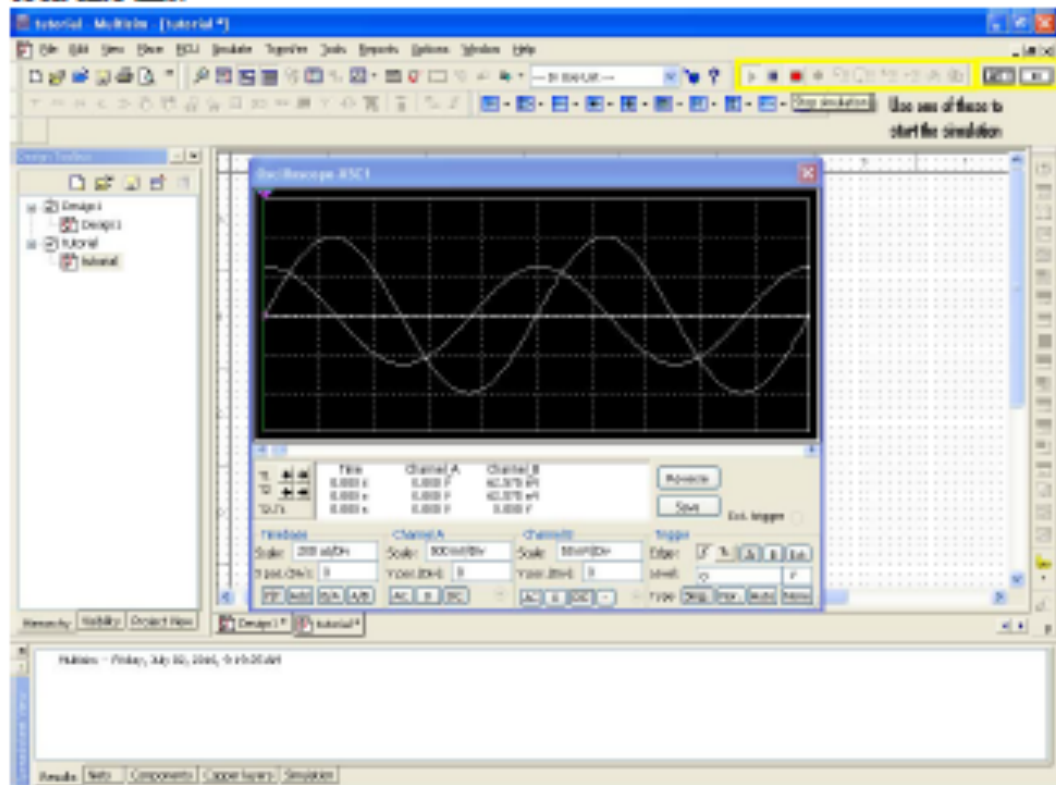
We can make the circuit yet a bit more organized. Since the negative inputs of the oscilloscope are connected to the ground we don't have to have the wire go around the circuit and we can just connect these two wires to a ground closer to the scope.



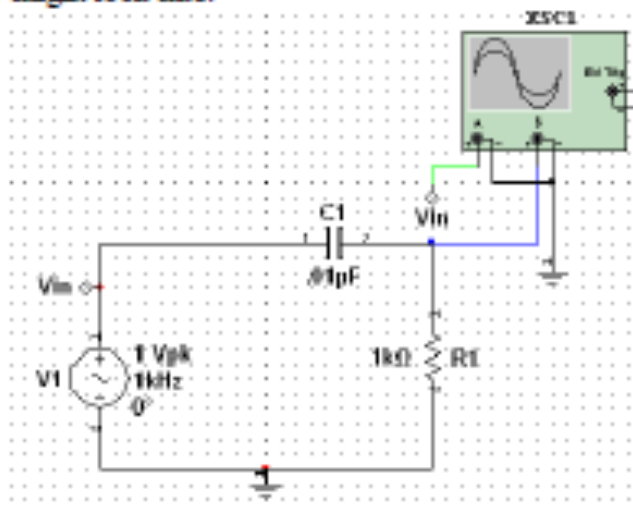
### Simulation:

Click the green play button in the Simulation toolbox, or click the toggle switch in the upper right corner of the page (or you could go to **Simulate->Run**). Double-click the oscilloscope and set the horizontal and vertical scales appropriately. The trace should

look like this:



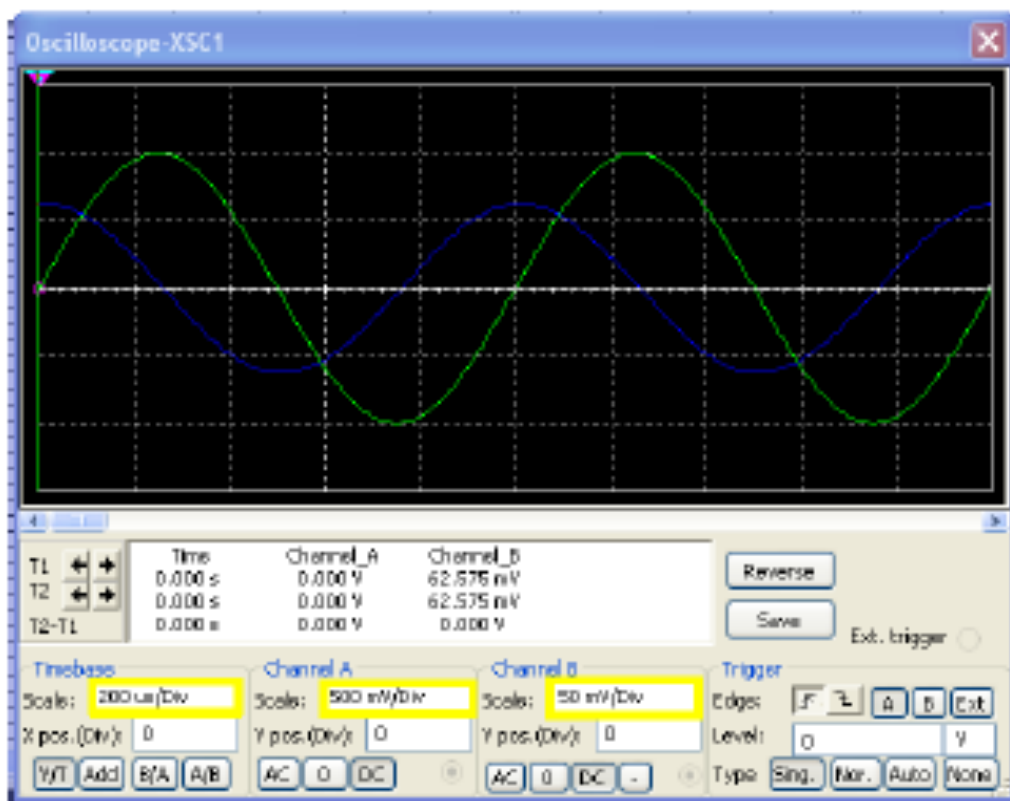
It is hard to see which graph corresponds to which input, to solve this problem, we will change the color of the wires that go into the oscilloscope and that would change the color of the graphs. To change the color of the wires right click on them and choose "Color Segment" then choose the color and click ok it is good practice to use the colors that you would have used if you were to physically set up the circuit. The new wiring might look like:



(I have pulled the oscilloscope a little to the right to make room for our next task)

Run the simulation and double click on the oscilloscope again change the divisions to the one shown below to get a similar graph.





Change the input frequency (or any other parameter) on the Value page found by double-clicking the source to see their effect on the graph. Most components can be modified through the double-click menu however you must be sure that the simulation is not working because otherwise the value will not effect the circuit until you restart the simulation.

1. Design the circuit in Multisim, taking the total voltage as the reference.
2. Measure the voltage and time lag values using Multisim and record the values in table 2.

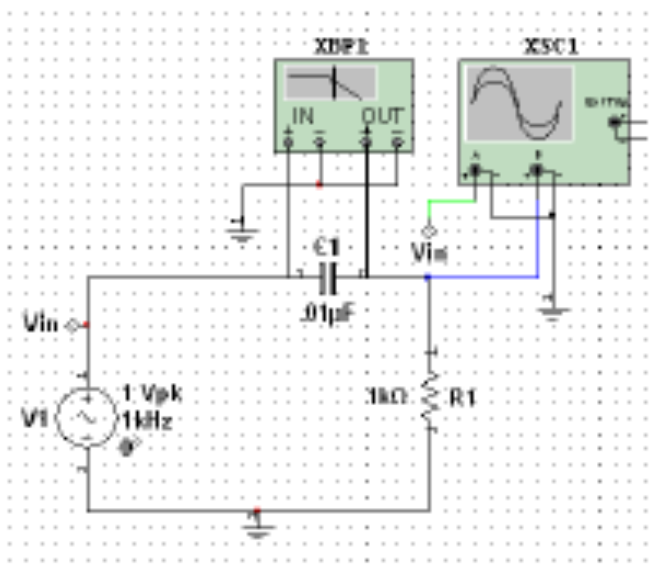
$V_S(V)$	$V_R(V)$	$V_C(V)$	$\tau_R$	$\tau_C$	$T (ms)$	$f (Hz)$

**Table 2**

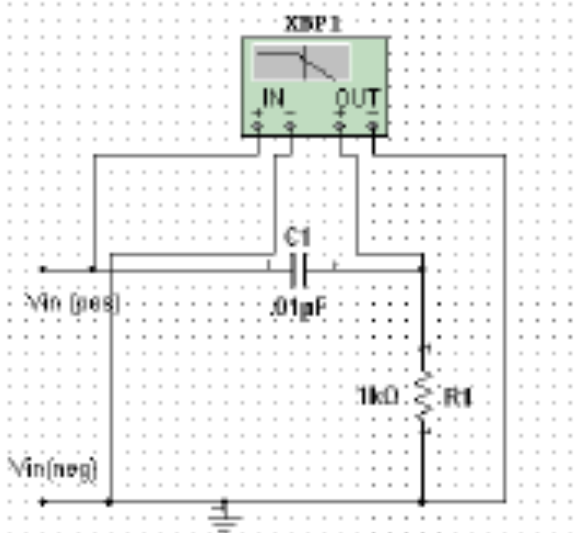
3. Plot the waveforms for each of the voltages  $V_R$ ,  $V_C$  and  $V_S$ .

### ***Bode Plotter***

To learn more about the circuit we want to see the transfer function. If one wants to do this manually one must feed the circuit different signals and measure the response of the circuit, however, MultiSim has an instrument that does exactly that. It is called **Bode Plotter**. Bode plots are traditionally done in log axis and are most useful to determine the role of frequency on the circuits and as you might have found out this circuit is strongly frequency dependent. To get a bode plot, place a Bode Plotter from the instruments toolbar and wire it as shown below.

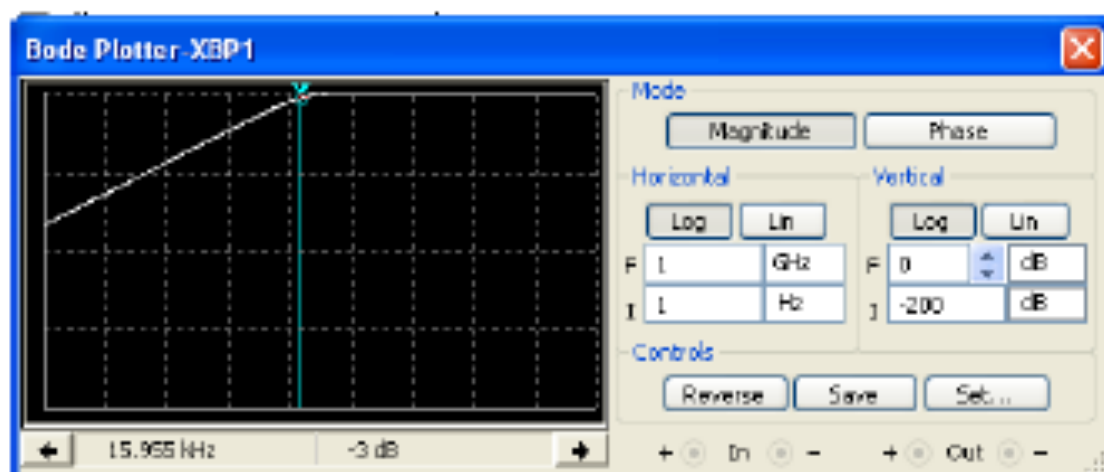


To make sense of the connections it might help to see the circuit from the Bode plotter's eyes. The Bode plotter disables the AC source (to use Bode Plotter you must be sure you have an AC source in your circuit) so to the Bode plotter the above circuit and connections (without the oscilloscope) look like:



Because the V- are connected to the ground it makes the wiring more organized if you just place a ground near the plotter and connect the circuit to it.

Double click on the bode plotter. You should see a figure similar to the following figure.



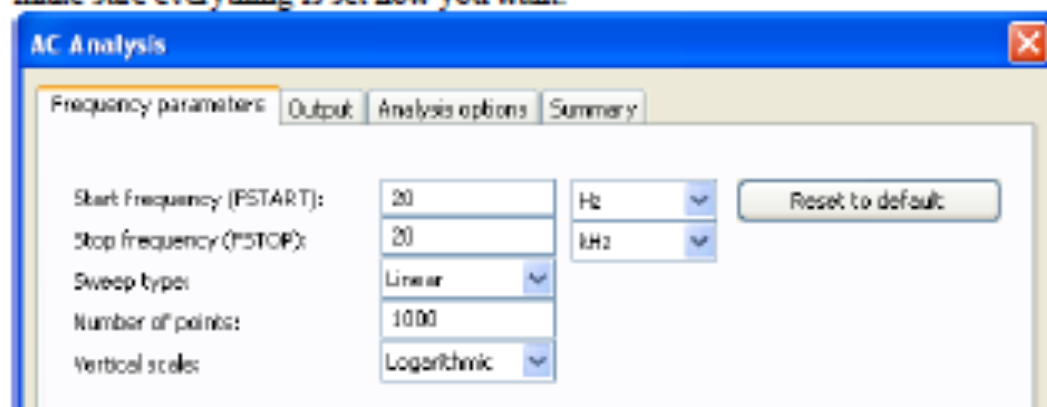
The bode plot is especially useful because it helps us determine the cutoff frequency of the filter. To do so drag the cyan cursor to the top left of the plot to -3 dB or right click on it choose "set Y value =>" and type in -3 then the frequency will represent the roll off frequency of the circuit. The bode plot tells us that the frequency is about 15.955 you should be getting the same value if you used the same components. Using the formula we get:

$$\frac{1}{2\pi RC} = \omega = \frac{1}{2\pi * 1000 * 10^{-8}} = 15915 \text{ not bad at all!}$$

You can also see the phase change caused by the circuit if you click on phase button.

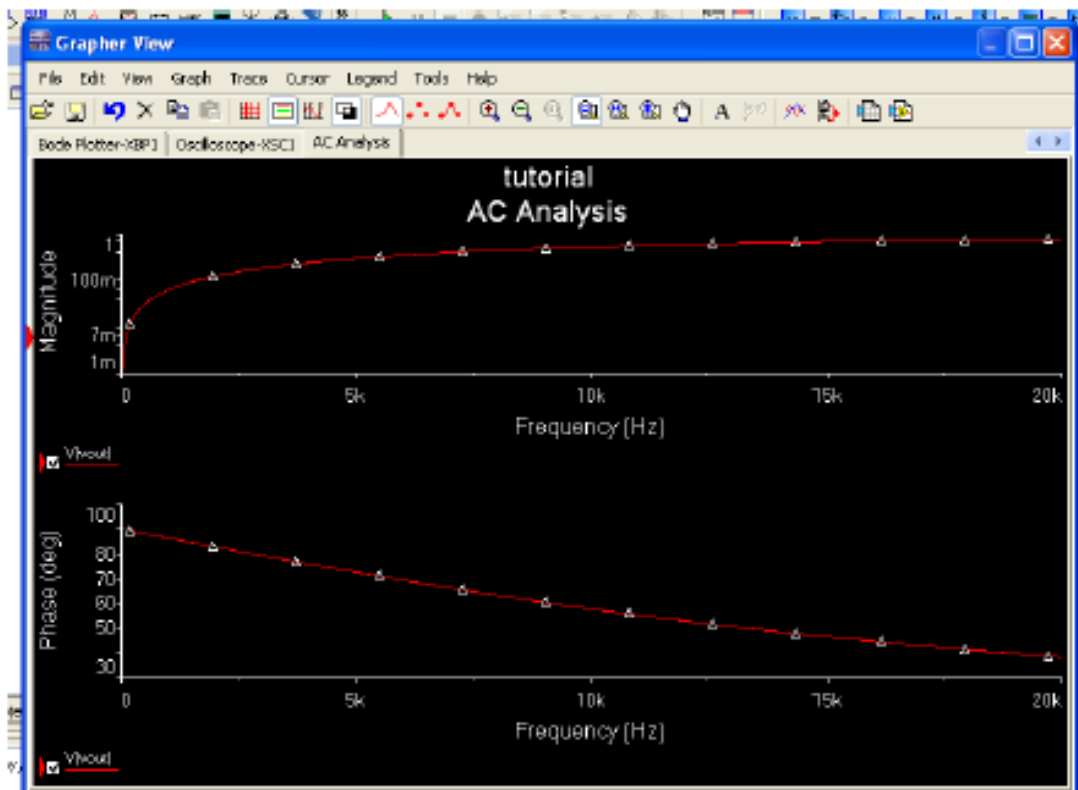
#### Analysis:

There are many types of analysis that are easy in Multisim. Let's find the transfer function again, to check our results. Go to **Simulate->Analyses->AC Analysis** and make sure everything is set how you want.



Now look at the output tab. You must add any quantity that you want to analyze. If you don't recognize the variables you want, rename them. Cancel the analysis and double-click on any wire you want to monitor. Type something related for the *Preferred Net Name* field (e.g. Vout).

When the analysis is finished Grapher automatically opens. Your analysis results should look something like:



The scales of any axis can be changed, along with other properties, by right-clicking the axis and selecting Properties. You may also use the toolbar at the top to navigate the graphs.

You'll notice that the information displayed on any instrument in your schematic shows up on separate tabs in the Grapher window. To see Grapher without running an analysis, go to **View-> Grapher**. Old graphs can be deleted by clicking the "X" in the top toolbar, next to the Undo button, twice.

When something doesn't work the way you think it should, National Instruments has a helpful forum where many common questions are answered:

<http://forums.ni.com/t5/Circuit-Design-Suite-Multisim/bd-p/370>