

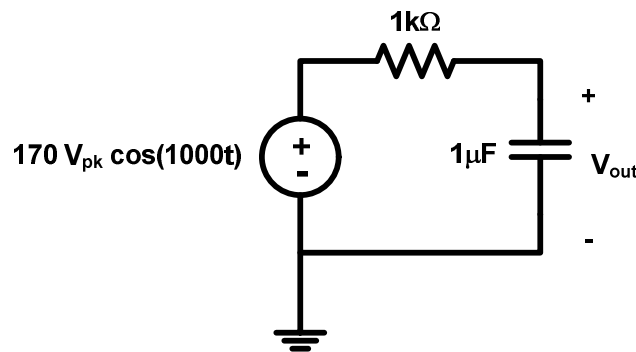
## MULTISIM Tutorial

EE320

Fall 2009

We will be building several AC circuits in this course to both confirm an analysis and to expedite the solution of a more complex network. Since EE320 concentrates on power system analysis and uses phasor analysis, we will focus on this application of Multisim and not revisit DC analyses.

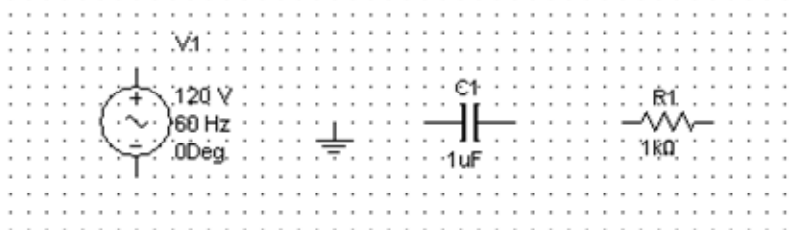
First, start the program by clicking on *ElectronicsWorkbench* - > *Multisim8*. This will open a blank circuit sheet that we can start loading in the various parts of our circuit. Let's assume that we wish to assemble the following simple circuit



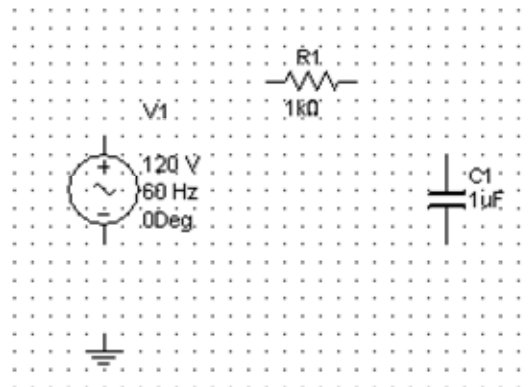
The bulk of our components can be found in the following two libraries accessed at the menu bar at the top of the sheet.



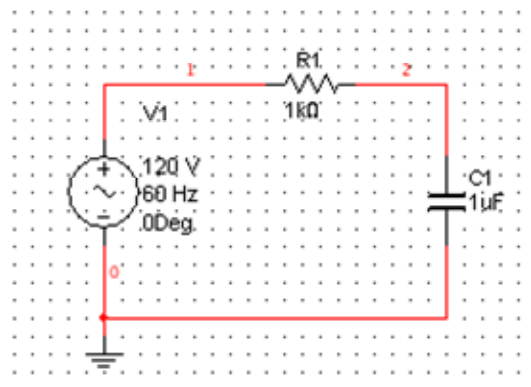
**NOTE EVERY CIRCUIT IN MULTISIM MUST HAVE A GROUND NODE.** The GROUND element is located in the SOURCES library. The voltage source is found in the POWER\_SOURCES sub-library (as AC\_POWER). The resistor and capacitor can be most conveniently specified by using the BASIC – BASIC VIRTUAL sub-library and selecting RESISTOR\_VIRTUAL and CAPACITOR\_VIRTUAL (as these components enable us to assign arbitrary values). Drag and drop these components onto the sheet.



Note, we can drag these components to different positions and right click on them to rotate them.



To wire up the circuit, we can left click on one component terminal, hold down, and drag the connection to a second terminal, then left click a second time. You may also connect from a component terminal to an existing circuit path. The completed circuit is then as shown

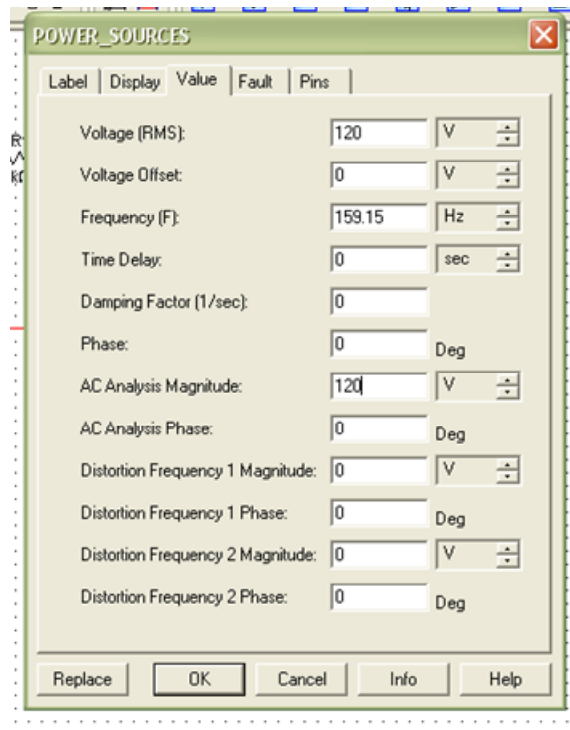


To get the node names to be automatically shown on the figure, select the following:

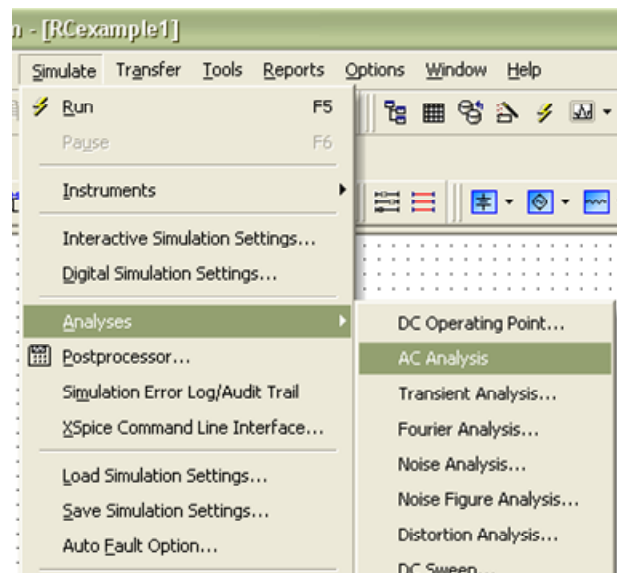
OPTIONS – Sheet Properties – Circuit – Net Names: Show All

Note, node 0 will always be ground. Knowing the node numbers will help with plotting and specifying outputs. To change the value or parameters of a component, double left-

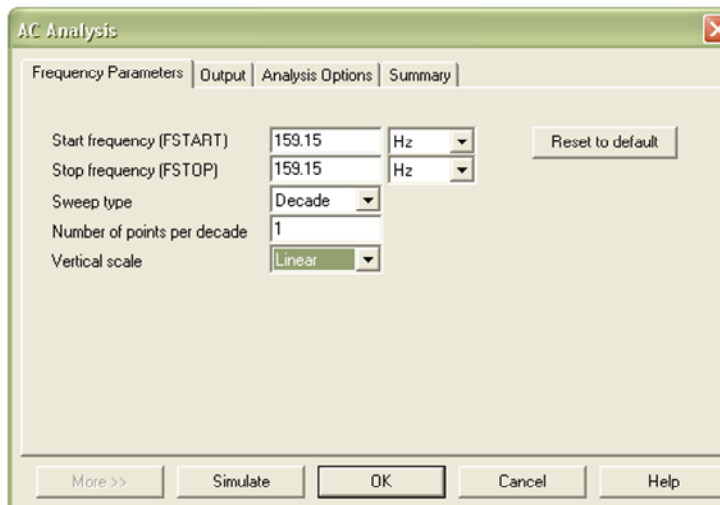
click the circuit symbol. In our case, we need to change the frequency of the source to  $\frac{1000\text{rad/s}}{2\pi} = 159.15\text{Hz}$ . We also change the AC Analysis Magnitude to 120V



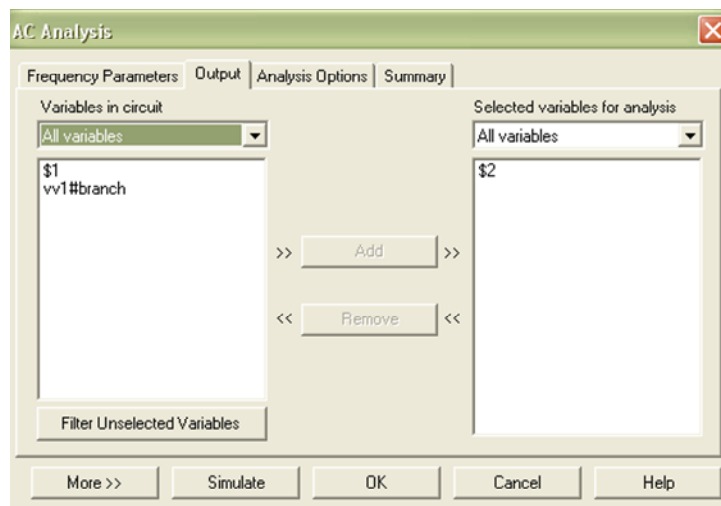
Since the resistor and capacitor values are fine, we are ready to consider performing sinusoidal steady-state analysis. Save the circuit. Next, choose Simulate –Analyses – AC Analysis as shown below



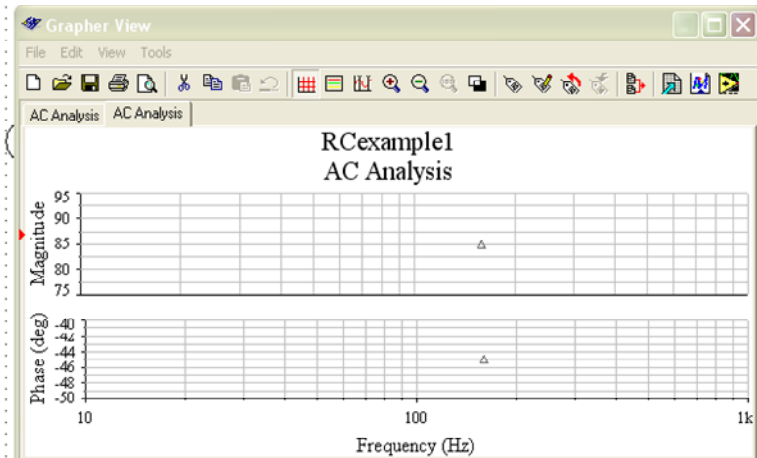
This will open a window with some options. For the Frequency Parameters menu, since we are only interested in the response at a single frequency, you will want to specify Number of Points as one and the starting frequency, the frequency of interest as shown



For the Output tab for the circuit shown above, add to the output variables \$2, meaning node voltage 2 (referenced to ground). The symbol vv1#branch is the current through the AC voltage source.



Then press "Simulate." This will open up a plot window. At the top of the window, you can change the background color of the plots and add a grid to make them more legible.

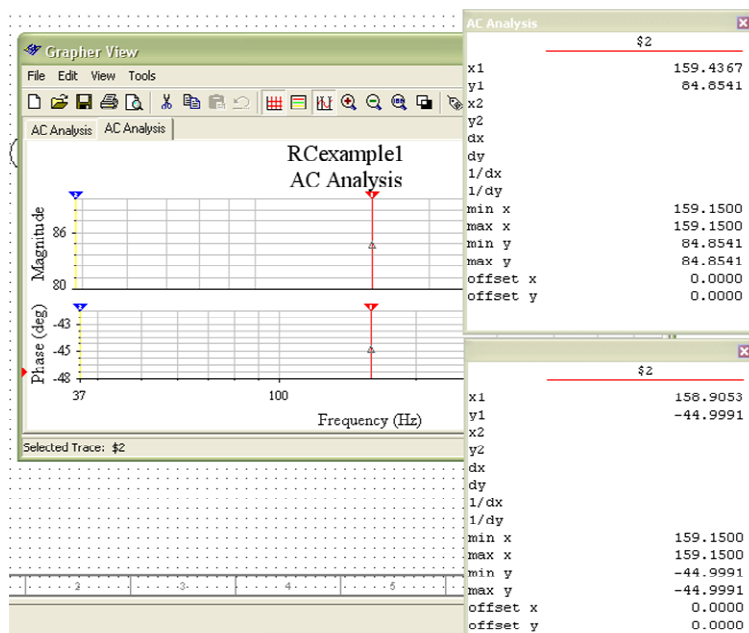


If we wish to hone in on the data points, first click on the desired plot (magnitude or phase), then activate the cursors as shown next

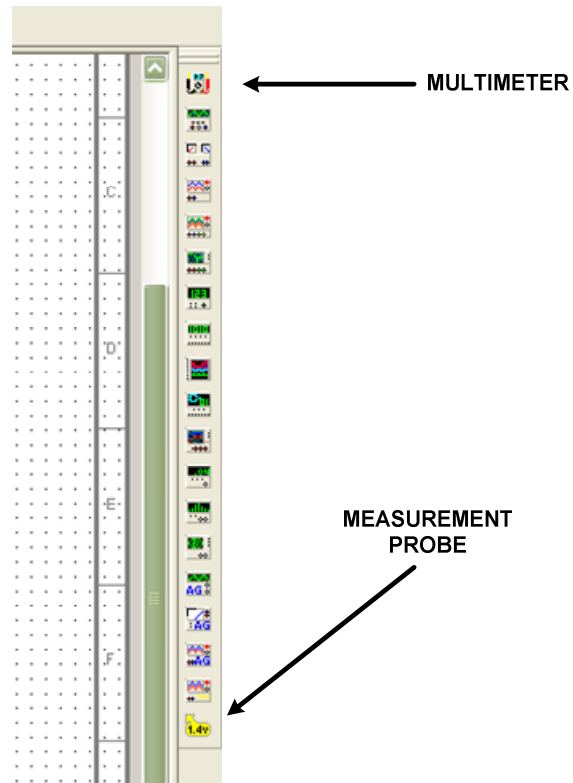
**CURSORS**



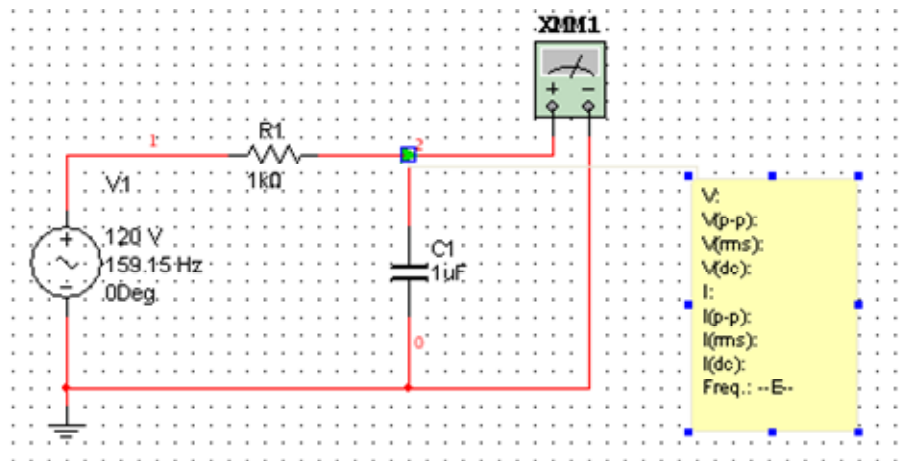
The cursors will enable you to zoom in and read the data point. As shown next, the magnitude is equal to 84.8541 and the phase angle is -44.9991 (which matches well with theory).



If you would wish to get a quick measurement of the rms magnitude of the output voltage, you can either use the measurement probe (located at the bottom of the right-most menu) or the multimeter (at the top of the same menu). We will show both

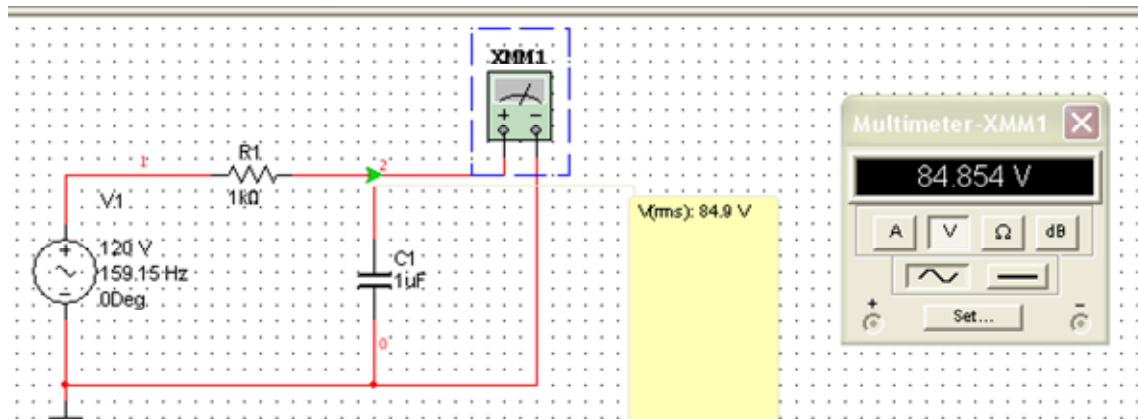


Attach the measurement probe to node 2 and then attach the multimeter across nodes 2 and 0 as shown next



We've repositioned the probe output window to view both conveniently. If you double-click the probe window, you can select its parameters and de-select everything except for the rms value. For the multimeter, make sure that it is set to AC voltage. If you then hit

the lightning bolt at the top menu, you start the simulation and both indicators confirm our previous analysis



Finally, let's assume that we want to measure the real power going into the circuit. Stop the simulation by pressing the lightning bolt again. Delete the meter and probe for convenience. Select the Wattmeter from the meters in the right-most menu. Connect the ammeter in series with the source, then place the voltmeter in parallel with the source. You need to double-left-click the wattmeter in order to display its reading. If we re-start the simulation by pressing the lightning bolt, we get the following output, indicating that about 7.2W is being transmitted to the resistor.

