

## Experiment 1

Introduction to Instrumentation, Basic Circuit Concepts and *Capture/PSpice*

**Purpose:** The objective of this exercise is to gain some experience with the electronic test and measuring equipment and the analysis software found at each station.

Equipment Required (Please see the course and studio webpages for information and tutorials on the instruments listed. You will find the information at <http://hibp.ecse.rpi.edu/~connor/studio.html#facilities> )

- **DMM** (HP 34401A 6-1/2 Digit Multimeter)
- **Oscilloscope** (HP 54603B 2 Channel 60 MHz Oscilloscope)
- **Function Generator** (HP 33120A 15 MHz Function / Arbitrary Waveform Generator)
- **DC Power Supply** (HP E3630A or HP E3631A Triple Output Power Supplies)
- One 50 ohm resistor, two 1 M ohm resistors and two 1 k ohm resistors.
- Protoboard
- Audio speaker

**Background** Read over the specification sheets for the DMM and the 'scope until you find out what the **input resistance** or impedance is for each measuring device. Write down these resistances here. If more than one value is given, that means it is possible to change this resistance. We will use the more typical lower value. For the 'scope, you may also find that there is an input capacitance. We will not be using that parameter yet. You should also review the definition of voltage magnitude, peak-to-peak voltage, RMS voltage, frequency and period of a sinusoidal signal. For links related to sinusoidal signals and analyzing simple circuits, see the links page for this course: <http://hibp.ecse.rpi.edu/~connor/education/EILinks.html>

## Part A Sine Waves and Hearing

Many of us have studied the sine and cosine trigonometric functions in math and physics classes. A graph of the sine function is shown in Figure 1. In this exercise, a function generator will be used to produce electrical signals with various shapes, including sine waves. The electrical signals will then be applied to an oscilloscope and a speaker, which will allow us to see, hear and analyze the signals. Our objective is to learn about the basic properties of sine waves and related signals by seeing them, hearing them and analyzing them with an oscilloscope. We will also demonstrate some interesting facts about human hearing and speech.

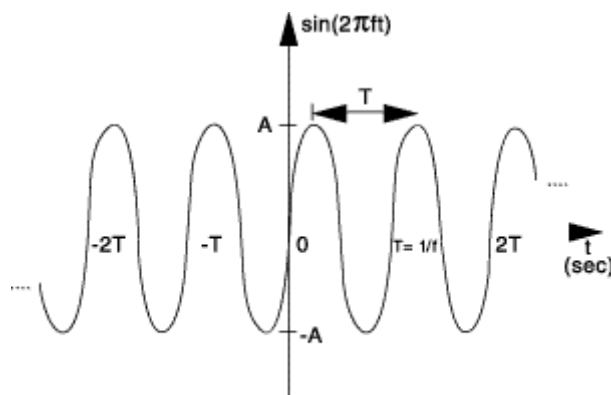


Figure 1: Sine wave with amplitude A, frequency f, and period T.

**Background About Sine Waves and Hearing** A sine wave is described by an equation of the form

$$v(t) = A \sin(2\pi ft) = A \sin(\omega t) \quad (1)$$

where the variable  $t$  represents time. Figure 1 contains a plot of a sine wave. We use the term "wave" because the shape is similar to a water wave that you might see on an ocean or a lake. A sine wave is characterized by two parameters, called amplitude ( $A$ ) and frequency ( $f$ ). The amplitude  $A$  determines the maximum value that the sine wave achieves along the vertical axis. The sine wave takes on values between  $+A$  and  $-A$  at various times.

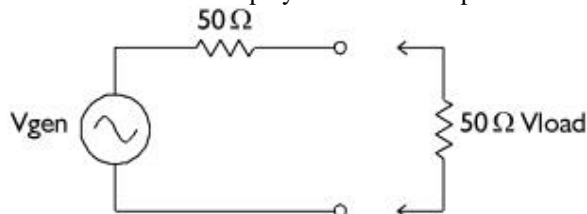
The frequency  $f$  of the sine wave can be understood as follows. Notice that the sine wave reaches its peak value of  $+A$  at regular intervals. The time between adjacent peaks is called the period of the sine wave. The period is denoted by the letter  $T$  and it is measured in units of seconds (sec). The frequency is defined as the number of times per second that the sine wave achieves the peak value of  $+A$ . Since adjacent peaks are separated by  $T$  sec, the wave achieves  $1/T$  peaks per second. Hence the frequency  $f$  is equal to  $1/T$ , and the units of frequency are  $\text{sec}^{-1}$ . Another name for the unit  $\text{sec}^{-1}$  is hertz, or Hz for short. It is usual to denote the product  $2\pi f$  as  $\omega$ , where  $\omega$  is called the angular frequency. Note that one of the most common mistakes made in this class is to confuse  $f$  and  $\omega$ .

Measurement devices do not usually give us the voltage amplitude  $A$  directly. Rather they determine  $V_{P,P}$  (the peak-to-peak voltage) or  $V_{RMS}$  (the RMS voltage). The peak-to-peak amplitude is the difference between the largest positive value of the sine wave and the largest negative value of the sine wave, so it should be nearly equal to  $A - (-A) = 2A$ . The RMS value is determined by taking the square root of the average of the square of the voltage. Since the voltages here are sinusoids,  $V_{RMS} = A/1.414$ .

We are exposed to a wide variety of sounds every day. We hear a sound after our brain processes the sensations recorded by our ears. Two attributes that are commonly used to characterize sounds are loudness and pitch. Loudness, of course, refers to how loud or intense we perceive the sound to be. Pitch refers to whether we perceive the sound to be high or low. For example, the sound of an ambulance siren has a higher pitch than the sound of a fog horn.

**Equipment and Setup** The equipment we will be using includes a function generator, an oscilloscope and an audio speaker. The function generator is used to create electrical signals with various shapes, including sine waves. The function generator can be programmed to generate waves with specified amplitude and frequency. The function generator is connected to both the speaker and the oscilloscope. The speaker converts an electrical signal to sound, which we then can hear. The oscilloscope analyzes an electrical signal and displays a picture of the signal. The combination of the oscilloscope and the speaker allows us to see with our eyes what we are hearing with our ears. We can also determine a mathematical representation of the sound that can then be used for system analysis.

The simple electrical model of the function generator is shown in the figure below. All voltage sources have an internal impedance. For the function generator, this impedance is 50 ohms. Most instruments are designed with this value to properly match coaxial cables and standard loads. We will return to this issue several times in the course. For now, it is only necessary to work with a realistic model of the function generator. Since the internal impedance of the function generator is 50 ohms, the default assumption is that the resistive load driven by this source is also 50 ohms. You may recall from previous courses that matching the internal impedance of a source to the impedance of the load gives maximum power transfer. The voltage displayed on the front panel is the voltage that appears across a matched load. If the load resistance is very large compared to 50 ohms, then the voltage that will appear across the load will be twice the value displayed on the front panel.



The combination of the internal impedance and the load forms a voltage divider. When both are 50 ohms, then half the voltage appears across the load and, thus, it is necessary for  $V_{gen}$  to be twice the value that is displayed. When the load is very large,  $V_{load}$  will equal  $V_{gen}$ . If this is not clear, it will be useful to draw the circuits for both cases and solve for  $V_{load}$  in terms of  $V_{gen}$ .

For the first experiment we wish to do here, we need a sinusoidal voltage with a frequency of 1 kHz and an amplitude of 100 mV. The front panel displays the peak-to-peak value of the signal, which should be twice the amplitude or 200 mV. Given these circumstances, how should you set up the function generator so that you obtain the desired voltage across its output, when there is nothing connected (that is, when the load impedance is infinite)? Once you have decided how to do this, set up the function generator and then connect it directly to the oscilloscope using a standard BNC-BNC coaxial cable. The output voltage of the function generator is adjusted as follows:

- Push the AMPL button on the function generator.
- Move the <> arrows keys of the function generator until the hundreds digit is blinking.
- With the knob, change the voltage up or down as desired.

The default frequency for the function generator is 1 kHz, so you should not have to change it. The appendix below contains some hints for the operation of the very nice 'scopes we have in our studio classroom. If you are reasonably well acquainted with 'scopes like these, just hook things up, turn them on and push the AUTOSCALE button on the 'scope. Once you have the signal from the function generator displayed on the 'scope, check to be sure that you have the specified sine wave output. (If you do not recall how to read the signal displayed on the 'scope CRT, the following section contains a description of how our 'scopes can do this automatically.) If you do not have the specified waveform, adjust the function generator until you do. Explain how to set up the function generator to obtain this voltage waveform (be sure that you try out your explanation on your partner):

Now look at the display on the oscilloscope. Clearly, the wave has the shape of a sine function. One way to measure the amplitude and frequency of the sine wave is to use the grid on the display of the oscilloscope. This method is somewhat tedious, as some of you may know from previous experience with oscilloscopes. (However, you should still know how to do this and can expect to be asked to do so, on at least one quiz.) The oscilloscope that we are using in this exercise is very convenient, because it automatically computes the amplitude and frequency of the sine wave. The procedure to display the amplitude of the sine wave is as follows:

- Push the Voltage button on the main panel of the oscilloscope.
- Push the "V<sub>P-P</sub>" button on the bottom of the oscilloscope display.

The peak-to-peak amplitude of the sine wave will appear in the lower left portion of the oscilloscope display. The peak-to-peak amplitude is the difference between the largest positive value of the sine wave and the largest negative value of the sine wave, so it should be nearly equal to  $A - (-A) = 2A$ , or 200 mV. Is the amplitude reported by the oscilloscope close to the expected value of 200 mV? (Note that the oscilloscope recomputes the sine wave amplitude from new measurements about twice per second, so the value that it reports is changing with time. The average of the measurements should be close to 200 mV.)

The frequency of the sine wave is measured by a similar procedure, which is as follows:

- Push the Time button on the main panel of the oscilloscope.
- Push the "Freq" button on the bottom of the oscilloscope display.

The frequency will appear next to the amplitude value in the lower portion of the oscilloscope display. The measured frequency should be close to the frequency of the function generator, which is 1000 Hz. Is this the case? (As with the amplitude measurement, the oscilloscope periodically forms a new estimate of the frequency. The average of the measurements should be close to 1000 Hz.)

**Driving a Speaker with a Function Generator** We now wish to connect the function generator, the 'scope and the speaker together to perform some simple experiments. Disconnect the BNC-BNC cable from the function generator, but leave it connected to the 'scope. Place a BNC Tee on the output of the function generator and then reconnect the cable to output via the Tee. Connect a second BNC-BNC cable to the other side of the Tee. Place a BNC to mini-grabber adapter on the other end of the second cable. Connect the mini-grabbers to the speaker terminals. If you have not already done so, turn on the function generator and the 'scope. If you have left the

function generator and 'scope on, you may notice that the signal observed on the 'scope has changed. This is to be expected since the speaker changes the load impedance for the function generator. Adjust the volume of the signal to a comfortable level following the procedure given above. By comfortable level, we mean the lowest amplitude that allows you to hear a distinct sound. There will be lots of sounds being made in the room, so it is best to keep them as soft as possible.

What is the value of the voltage amplitude that you have selected?

Next vary the frequency of the sine wave as follows:

- Push the **FREQ** button on the function generator. Note that the current frequency is 1000 Hz.
- Move the <> arrows keys of the function generator until the hundreds digit is blinking.
- With the knob, change the frequency up or down as desired.

Let us investigate how our perception of loudness changes as the frequency of the sine wave is varied. Turn on the speaker, keep the sine wave amplitude fixed at your comfortable level, and vary the frequency over the range from 100 Hz to 10,000 Hz. Try cycling through the following frequencies, without changing the speaker volume control: 100, 200, 300, 400, 500, 600, 700, 800, 900, 1000, 2000, 3000, 4000, 5000, 6000, 7000, 8000, 9000, and 10,000 Hz. Which frequency do you hear the loudest? Is there any variation among the members of your group? If you have problems discerning significant differences in loudness, try a different speaker.

Generate a tone at the frequency that appears loudest. Does the pitch of this tone seem to be one that you commonly hear in speech, music, and automobile traffic?

At this point, you will have put the function generator and 'scope through some basic tasks. Try some of the other functionality of the instruments (push buttons and see what happens). Some very interesting and annoying waves can be produced. Play around a little and then find a particular set of operating conditions that you find the most interesting. Describe how you have the instruments setup for these conditions. Be as complete and specific as possible.

Under what circumstances might you experience the sounds you have produced or generally when might you encounter a waveform like the one you have displayed on your 'scope?

**50 Ohm Load** Before going on to the next part of this experiment, we wish to check to see that the function generator works as it should with a 50 ohm load. Remove the speaker from the mini-grabber connectors and replace it with a 50 ohm resistor. Now you should observe that the function generator display and the 'scope are in agreement. Note also that the voltage seen by the 'scope will double when the resistor is removed.

## Part B Electronic Test & Measuring Equipment

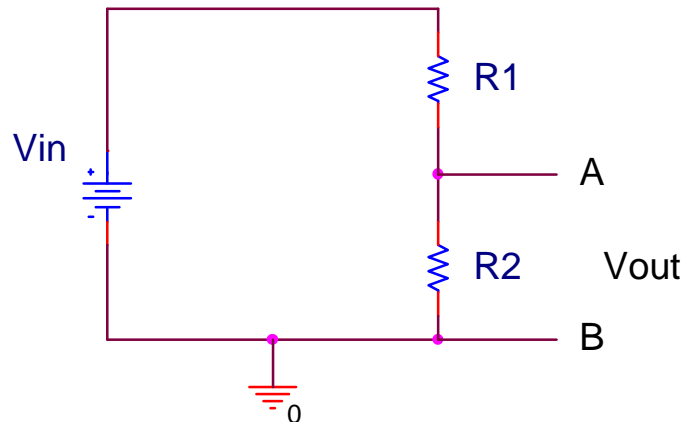
(Based on an experiment by Barry Lunt, Brigham Young University, but now does not appear in Educators Corner.)

*Note: From now on we will not be specifying exactly how to connect particular cables to instruments. You will notice that sometimes we will need BNC connectors, sometimes mini-grabbers, sometimes banana connectors, etc. You should be able to figure out what connectors and cables to use. If you are at all uncertain, check with one of your friendly course staff.*

### Some DC Measurements

- Measure the actual value of the two 1 M ohm resistors using the DMM. Record the two values using all the significant digits displayed on the DMM.

- Set one of the DC supplies to +10 Volts. Measure the voltage using the DMM. Make sure that the display on the DC supply and the DMM agree as much as possible. Write down the values displayed on each device.



- Apply the 10 Volts from either of the DC power supplies to the two 1 k ohm resistors connected in series, as in figure above. Calculate the voltage you would expect to measure across each resistor when no measuring device is connected to the circuit. After calculating the expected voltage measurement, measure each and explain the difference. Repeat with the two 1 M ohm resistors in series. You should see a larger discrepancy between the predicted and measured voltage when the 1 M ohm resistors are used. Read the note below and then explain why this is the case.

*Aside: Essentially all measurements are imperfect in that the measuring device used changes the parameter being measured. The best we can hope for is that the effect of the measurement is so small that we can neglect it. Measuring the voltage across two resistors in series will change the voltage since the measuring device (the DMM in this case) has an input resistance. When the DMM is connected across the bottom resistor, the input resistance of the DMM is then added in parallel with the bottom resistor. Adding a resistor in parallel will always lower the overall resistance, since there will be another path for the current to flow. Thus, the voltage measured in this case will be lower than the voltage without the DMM. The effect is not significant as long as the DMM resistance is much larger than the resistor where the measurement is being made. There is a very important engineering lesson to be learned here. Whenever one is dealing with some kind of an engineering system, it is essential that the entire system be considered, not just some subsystem. Once the measuring device is connected to a circuit, it becomes part of that circuit.*

### Some AC Measurements

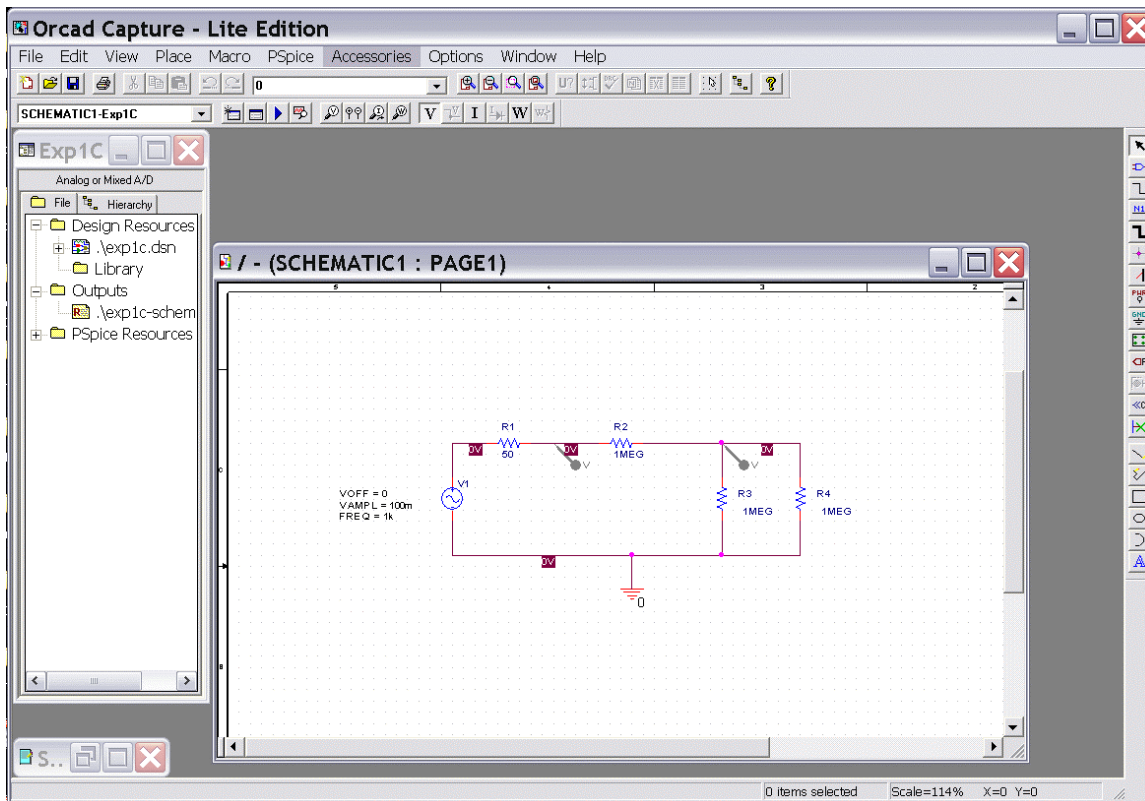
- We already know how to use the 'scope to help us create a signal with the desired amplitude and frequency. Can we do the same thing using only the DMM? Set the function generator to 1 kHz and attach it directly to the DMM. Calculate the value of  $V_{RMS}$  for a 1  $V_{P-P}$  (peak-to-peak voltage) sine wave. (Remember:  $V_{RMS} = A/1.414$ ) If we adjust the amplitude of the signal coming from the function generator so that the AC voltage on the DMM reads the same as the  $V_{RMS}$  we calculated, then we should have created a 1  $V_{P-P}$  sine wave. What happens when you try to measure the DC voltage of the signal from the function generator with the DMM? Why?
- Measure the amplitude and frequency of the 1 kHz sine wave you created using the DMM with the oscilloscope. (See the Appendix below for some basic hints on 'scope operation.) Compare these measurements to those obtained from the DMM. If the amplitude differs greatly from the expected value, check the output impedance setting on the function generator. (One more parameter to look up in the instrument specifications.) If the peak-to-peak voltage observed on the 'scope is not seen to be 1 volt, adjust the function generator until the correct value is observed. Then measure the voltage again with the DMM. On a piece of paper, sketch the signal as it is observed on the 'scope CRT. Label the vertical (voltage) and

horizontal (time) scales. Show that the peak-to-peak voltage is 1 Volt and that the period is equal to  $(1/f)$  where  $f$  is the frequency.

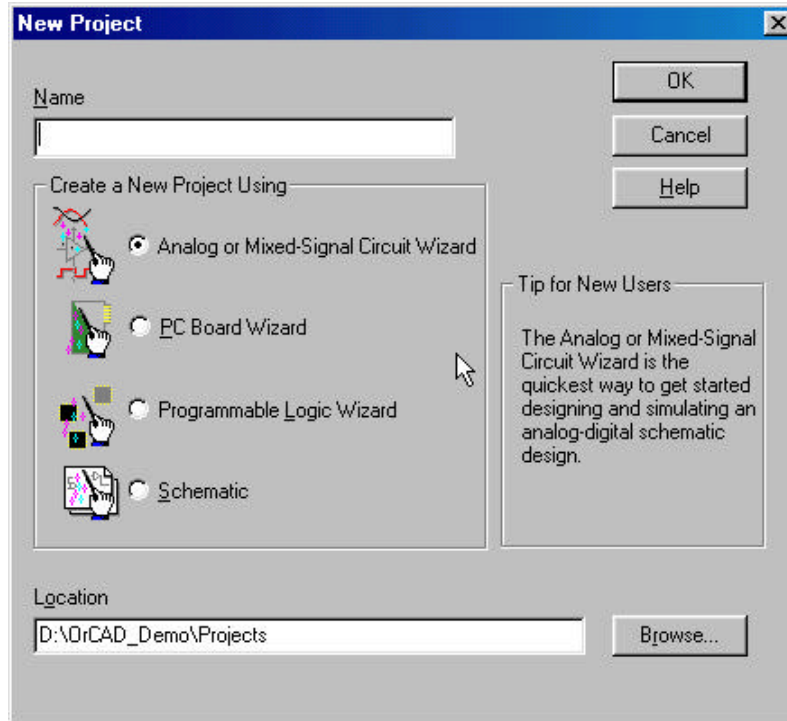
- We will now apply the sinusoidal source to the same series resistor combination we used for DC measurements. Calculate the voltage you would expect to measure across each resistor and across both resistors, when a sine wave with  $f = 1$  kHz and  $V_{P-P} = 1V$  is applied, both for the DMM and for the oscilloscope.
- Apply the above output of the function generator to the 1 M ohm resistors (remove the DC voltage). Using the DMM (on AC Voltage) measure the voltage across each resistor and across both resistors. Compare these to the expected values. Repeat with the 1 k ohm resistors. Note that the internal impedance of the 'scope is 1M ohms. How do you think this affects your expected values in each case?
- Disconnect the DMM; now using the oscilloscope, observe the voltage across both resistors, and the voltage across the resistor closer to ground. Do not try to measure the voltage across the other resistor. Compare the measured values to the expected values.

### Part C Introduction to PSpice

The software we will be using to simulate the operation of circuits in this course is called *PSpice*. Actually we will be using a combination of two programs, *Capture* and *PSpice*. We will use the first to set up the circuit problem and the second for the analysis. These are loaded on the studio computers, so one looks for something called *Capture Student* and then clicks on it. (This is located in the applications folder and on the start menu under "Pspice Student"). Once you have done this, you should eventually see a window that looks something like the one below, except that neither of the smaller windows will be showing in it. In general, we will use the command buttons to make PSpice do our bidding. By moving the mouse arrow until it is located over any button you will be able to determine its function. To get a feel for how this works, we will draw the simple circuit we have been studying: a combination of resistors and a sinusoidal voltage source.



Before we can draw our circuit, we must set up the problem architecture for *Capture*. First click on the *File* pull-down menu and select *New Project*. You will see a new window named *New Project* shown on the next page. Be sure that the *Analog or Mixed-Signal A/D* is selected, since that is the one that allows us to set up circuits for *PSpice* using *Capture*. You will use this window to give this project a name and choose a location where you wish to store it. Most of the time, you will wish to keep a copy of your projects either in your RCS directory or on a floppy disk. However, working from these directories will be **very slow**. You should, therefore, set up a directory with your name, or something you can easily remember, on the hard drive of your computer. You should not assume that the files you put there will stay there for any length of time, so you should copy them to a floppy or your RCS directory at the end of each class. For now, though, you should just put your project files in this new directory you have created. Either by typing in the name of your new directory or by using the *Browse* button, select your directory as the location indicated in the box at the bottom. In the box at the top of this window, give your project a recognizable name, such as *EXPI-C*. Once you have finished setting up the project, click on the *OK* button and you will get a window asking if you wish to use an existing project. Choose create a blank project and click *OK*. You should then see the two small windows above, but with no circuit showing in the larger one. We will next see how to draw a circuit.



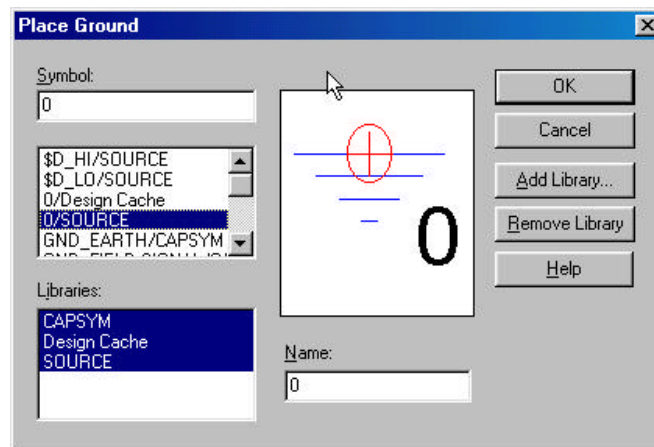
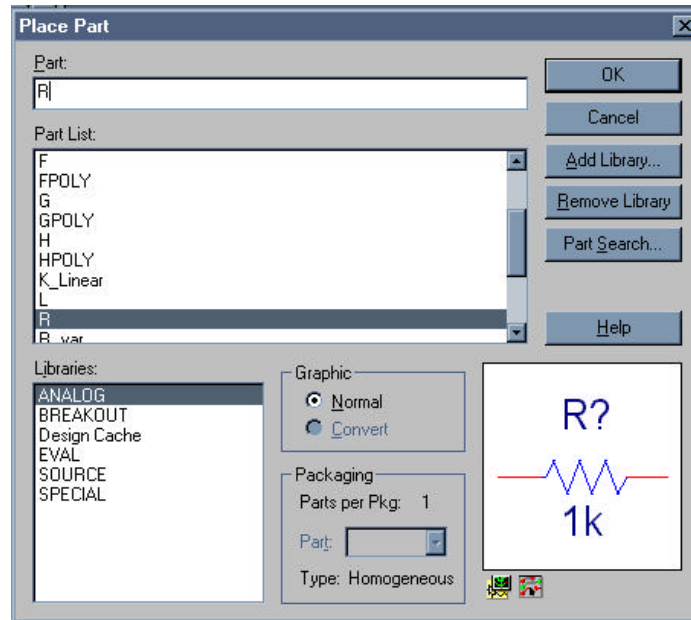
### Drawing a Circuit

In the circuit shown, we have some resistors, a sinusoidal voltage source, a ground and some wires. To create this diagram, we will use the command buttons. For the resistors and the voltage source, we will click on the second button from the top that has the device with the three little wires sticking out of it (*Place Part*). You can also do this by using the *Place* menu or by hitting shift-P. The components and their PSpice names are:

Components Name	Resistor R	Sinusoidal Voltage Source VSIN
-----------------	------------	--------------------------------

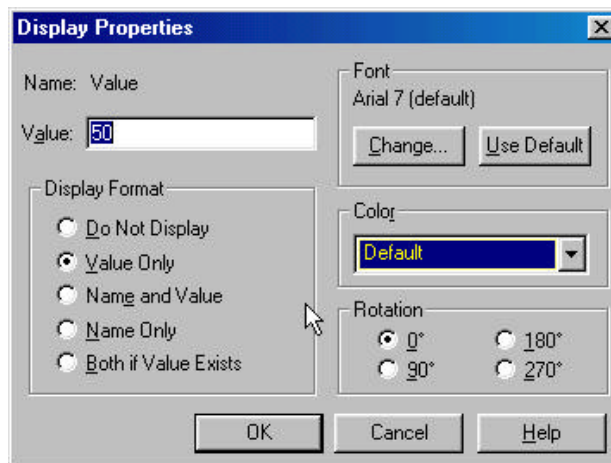
If you do not see any parts in the list, there are no libraries loaded. You can use the Add Library button and browse to add the SOURCE and ANALOG libraries, if they are not already there. To get a resistor, click on this button and then type an *R* in the space at the upper left marked *Part*, as shown below. The diagram below shows what the *Place Part* window looks like after a resistor has been selected. Then click on *OK* and you will be back to the main window and can place the resistors where you want them by moving the mouse around and clicking at the appropriate location. You will notice that there is a default condition for each component. For the resistor, the value will be 1k ohm and it will be horizontal. Since our circuit involves both horizontal and vertical resistors, you will need to rotate the symbol before placing it. To rotate the symbol, you need to use the keyboard command *R* for rotate or the right click menu. Once you have the resistor in the proper orientation, you can place in your selected position. Just place all the resistors first. We will change their values later. When you have finished placing resistors, hit the *Esc* key or just click on the *Parts* button again. Then, to get the voltage source, click on the *Place Parts* button and type *VSIN* in the parts box and place the voltage source where you want it. Again, you will have to rotate it first. Finally, all circuits need a ground reference so we know where the voltage is equal to zero. Click on the *Place Ground* button, which is about half way down on the right. You want to choose the *0/SOURCE* ground from the *Place Ground* window, since it is the only one that works with *PSpice*. If the 0 ground is not listed, you should load the PSpice/source library.



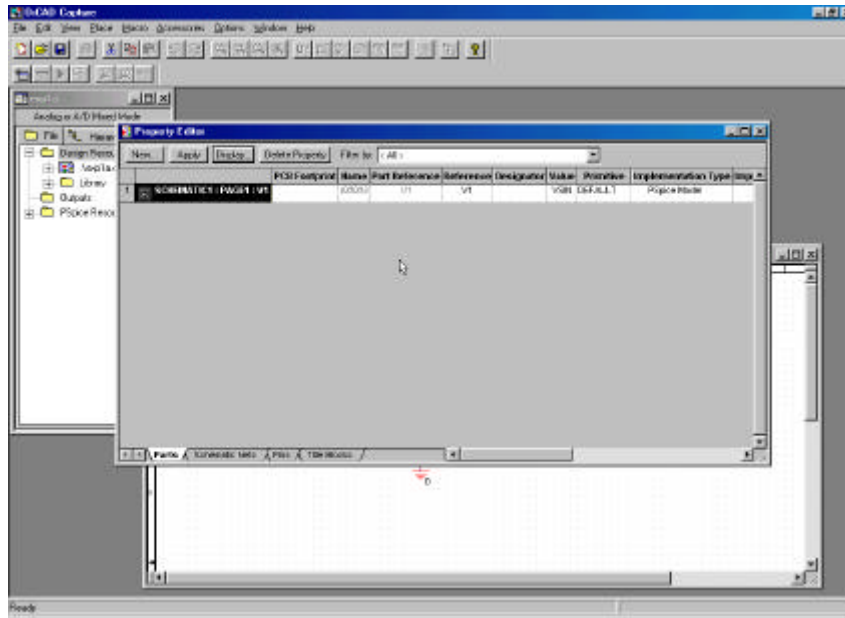


We now have all the components and must connect them with wires. For the wires we will click on the third button down on the right with the thin line that steps down on it (*Place Wire*). The mouse symbol will become a little cross. You will note that each component comes with little connecting wires on it. To connect two components together, just click the little cross near the boxes at the end of the little connecting wires. Make sure you click near the boxes. You can draw a wire almost anywhere, but you can only connect to the device at the ends of its connecting wires; be careful not to draw the wire through the components. Note that whenever you connect more than one wire at some point, a dot will appear there indicating a connection. It is possible for wires to cross one another without connecting if you choose. Then the dot will not appear. We now have drawn the circuit and need to indicate on it where we want to determine the voltage. On a physical circuit, we would connect a multimeter or an oscilloscope at these points. In our simulation we will do the same thing using the *Voltage Level Marker* button, which should be obvious, since it shows something that looks like a little probe with a *V* on it. (This button is located in the bottom row at the top of the main window.) Click on that button and place the probe to display the voltage at the upper right corner of the circuit (across the 1MEG resistor). You may want to rotate it into a convenient position. When the circuit is analyzed, a plot will be produced that shows the voltage at this location.

To complete the schematic, we have to change the component values. Each resistor was given a name in the order it was placed on the diagram. Thus, your resistors may not have the same names as shown above. However, for simplicity, they will be referred to by the name shown here. To change R1 to 50 ohms (the input impedance of the function generator), double click on the value *1k* and you should get a window with the name *Display Properties*. Change the *1k* to *50*. If you find that the number is in a hard to read position on the circuit diagram, you can single click on it and then move it with the mouse. The next resistor brings up a very important issue since it has a value of 1 M ohms. When you type the value in the *Display Properties* window, you must type *1MEG* with no spaces. Note that you have to type *MEG* since PSpice uses *M* to mean  $10^{-3}$  and *MEG* for  $10^6$ . It is not case sensitive. Note that in the schematic pictured, R1 represents the function generator impedance, R2 and R3 are the resistors in your voltage divider and R4 represents the impedance of the scope. Since the 'scope impedance is 1M, we set R4 to 1MEG. R2 and R3 can be set to the values of the resistors you used in part B. Initially, just leave them at 1k ohms.

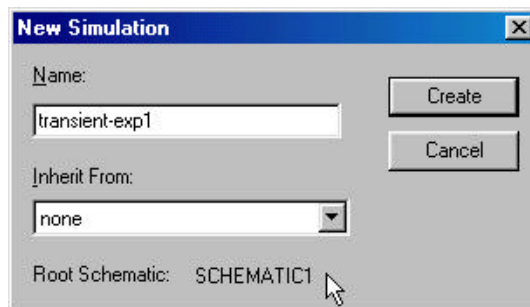


After you have changed all the resistor values and moved them to readable positions, you must set up the voltage source. To do so you must click on the part itself, not the name. You should see a spreadsheet in which all the properties of the voltage source are stored, except that there will be no values for the last six parameters. These sources can be very flexible and can be used in a variety of ways. We need an amplitude and a frequency, so everything else except the last item can be zero. We will not be using the *ID* so we can leave that blank. Set the frequency (*FREQ*) to 1k and the amplitude (*VAMPL*) to 100mV. Note again that there is no space between the number and the *m* or *k*. Since we will often be using the source for more than one kind of analysis, it might be good to get into the habit of setting the AC source (*AC*) value to be the same as the amplitude. If you always do this, it will make life a little simpler in the future. Now the schematic is complete and each symbol stands for the correct part.

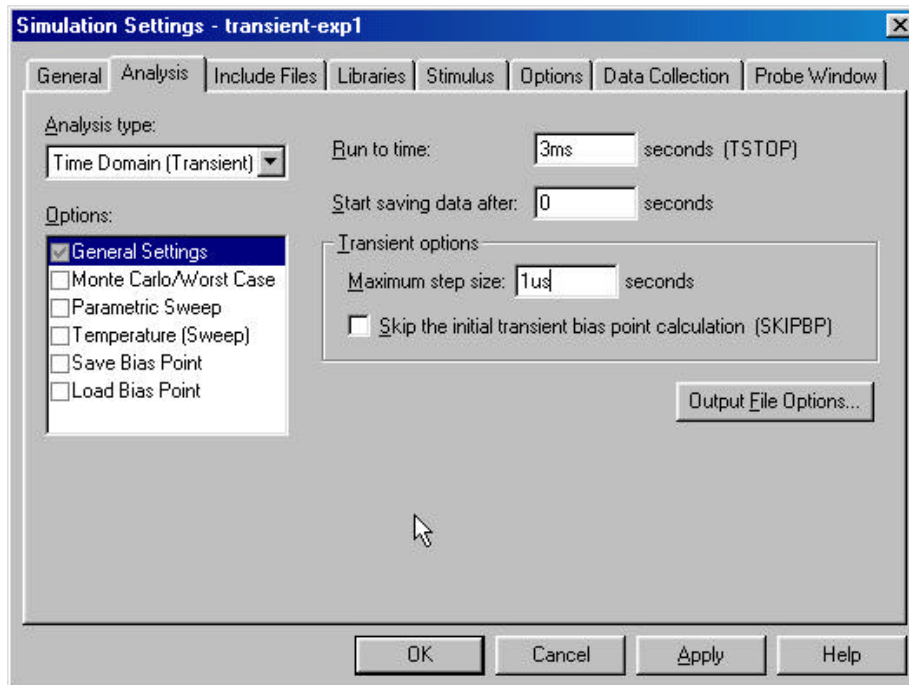


### Setting Up the Analysis

After we have defined all the components, wired them up and changed their values appropriately, we are ready to do some analysis. Find the *New Simulation Profile* button and click on it. (Hint: It looks like a little window.) You will get the window shown below, in which you must give a name to the file where the specifications for the analysis will be stored. Click on the *Create* button when you have chosen the name for your profile.



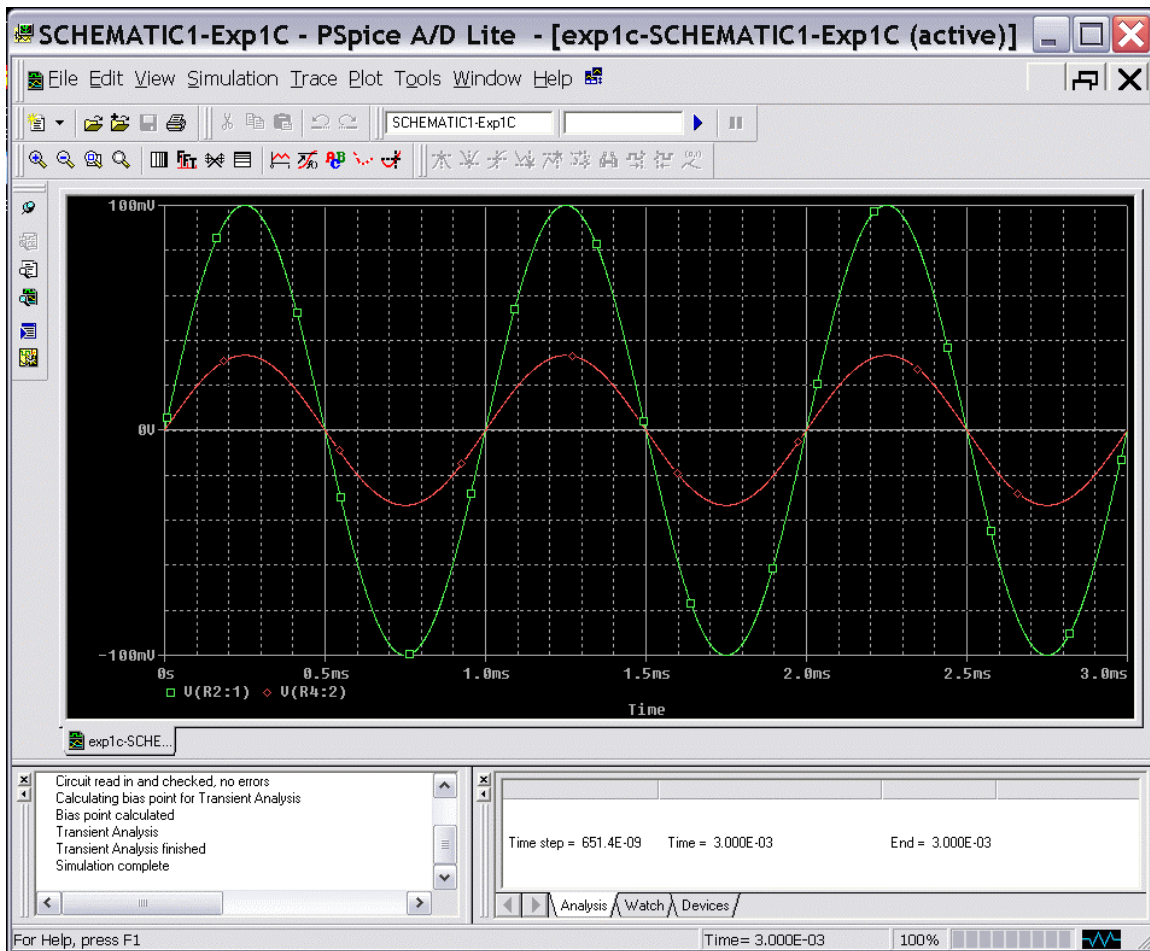
We will be doing *Transient Analysis* since that will produce a plot that is similar to what we see on the 'scope'. The analysis options are found in the pull-down menu at the upper left of the *Simulation Settings* window, as shown below. Since this is the default option, you will not have to do anything this time. There are three boxes in which times need to be indicated. At least two and usually three numbers have to be put into these boxes. The final time (*Run to time:*) has been set at *3ms* because the period of a 1kHz sine wave is 1 msec. This allows us to see three periods. The *Maximum Step Size* has been set to speed up analysis. PSpice will keep cutting its step size down until it achieves some standard for accuracy. For transient analysis, this will sometimes cause it to take a very long time to finish. The analysis will begin saving data at 0 seconds. Note again that there should never be any spaces between the number and its units. Click on the *OK* button when you are finished putting in the numbers. When you have finished, you should notice that the button that looks like an arrow in the second row at the top of the main window, now has become active. This is the *Run* button that we will use to run our simulation. All the buttons in this row should now be active.



### Transient Analysis

You are now ready, we think, to do the simulation. Click on the *Run* button. It is possible to set up many kinds of analysis using the *Simulation Settings* window. Since you have already told PSpice where you want to know the voltage, it will produce a plot with the signal you have asked for. If you do not get something that looks like what you have seen on the 'scope, ask for help. You should now go back and add another voltage level arrow at the location that represents the output of the function generator. Be sure you have the correct location. A very nice feature that *Capture* gives us, that was not available in the past, is that the voltage plots and the voltage probe markers will be the same color, so it will be easy to determine which is which. You should get something like (but not identical to) the window shown on the next page. *Print the output plot for the 1K voltage divider. Change the values of the resistors in the voltage divider to 1MEG, rerun the simulation and print this plot as well. Both plots should have two traces: the source voltage, and the voltage across the resistor closest to ground. Do the plots agree with your results from part B? What happens if you set the resistor values to the exact measured values of your resistors in part B? Are the results closer? Try varying the frequency, amplitude and offset of the VSIN source one at a time and rerun the analysis. What happens to your signal? Does it make sense based on your knowledge of sine waves and voltage dividers?*

The combination of *Capture* and *PSpice* is a very powerful simulation tool meant to address the circuit simulation needs of all engineers who must do circuit design and analysis. Thus, there are many, many opportunities to make what seem like silly mistakes that prevent the analysis from working properly. In your first attempt at using these tools, it is likely that you have already made some of these mistakes. You should also have heard about some of them in class. What mistakes did you make?



### Report And Conclusions:

The following should be included in your report. Everything should be labeled and easy to find. Partial credit will be deducted for poor labeling or unclear presentation.

#### Part A:

Answer following questions:

1. When only the 'scope is directly connected to the function generator, what does the display on the function generator read when the signal measured by the 'scope has a peak-to-peak amplitude of 200 mV? (1 point)
2. When the speaker is also connected to the function generator, what does the voltage display on the function generator read when the speaker is producing a comfortable level of sound? What does the 'scope measure for the peak-to-peak amplitude? Which of the two values is smaller and why? (2 points)
3. What is the period of the tone at the frequency that appears loudest when you scanned through the entire range of frequencies? (1 point) *Note: There is a range of acceptable answers to this question since it depends on the hearing of the person listening and the frequency response of the circuit.*

## Electronic Instrumentation

ENGR-4300

Fall 2004

Section \_\_\_\_\_

4. What frequencies make up the audio range? That is, what are the lowest and highest frequencies that humans can hear? (1 point) If you do not know this, check the frequency response of a good speaker or do a search on the web for *Audio Frequencies*. Provide the reference you use.

### Part B:

Answer following questions:

1. This exercise was intended to show that errors in measurements can be due to the measuring equipment being used. Discuss the sources of error in your measurements. Also discuss the difference in resolution between the DMM and the oscilloscope, and the advantage(s) and disadvantage(s) of both the DMM and the oscilloscope. (2 points)
2. What was the difference (in percent) between the actual measured values of the resistors and their specified values? (2 points)
3. You applied 10 Volts from one of the DC power supplies to the two 1 k ohm resistors connected in series. Calculate the voltage you would expect to measure across each resistor when no measuring device is connected to the circuit. After calculating the expected voltage measurement, explain the difference between your prediction and the voltages you measured with the DMM. (1 point) Repeat with the two 1 M ohm resistors in series. (1 point) You should see a larger discrepancy between the predicted and measured voltage when the 1 M ohm resistors are used. Explain why this is the case. (1 point) (Note that the DMM has an internal impedance of 10M when reading DC values. How does this affect the two voltage dividers?)
4. Repeat the previous question for the 1kHz AC voltage. Note the 'scope has an internal impedance of only 1M. (3 points)
5. You saw that the DMM and the 'scope can change the voltage they are trying to measure with the 1k ohm and 1M ohm resistors in series. There is quite a large range of resistance values that we could have chosen that would have seen very little change in voltage when either the DMM or the 'scope are used to make a measurement. This is the range in which these devices operate in an ideal sense. Give an example of such a resistance value. Again, there is no one right answer to this question. (1 point)

### Part C:

Include the following plots

1. Voltage divider with 2 1K resistors (two traces). (2 point)
2. Voltage divider with 2 1MEG resistors (two traces). (2 point)

Answer the following question

1. How does the signal compare with the 'scope? (1 point) What is the same and what is different. Can you explain the differences? (2 points) Did we, for example, use the correct amplitude for the voltage source?

**Summarize key points (1 pt)**

**Discuss mistakes and problems (1/2 pt)**

List problems you had hooking up and using the equipment and problems you had with Capture/PSpice.

**List member responsibilities (1/2 pt)**

**Total: 25 points for write up**

**Attendance: 3 classes (5 points) 2 classes (3 points) 1 class (0 points) out of 5 possible points**  
**No attendance at all = No grade for experiment.**

K. A. Connor  
8/29/2004

Rensselaer Polytechnic Institute

14

Revised:

Troy, New York, USA

**Appendix** Operation of the Digitizing Oscilloscope

The operation of a Hewlett-Packard digitizing oscilloscope of the type we have in the Instrumentation Studio or in the Circuits and Electronics Studio across the hall is reasonably straightforward for someone who is experienced at using a traditional analog 'scope. The scopes we use are essentially the same as those in the Auto Control lab, except that they have the usual set of buttons and knobs one usually sees on an analog 'scope. These buttons, etc. have been programmed to act like their analog counterparts. There are also functional controls that appear only as words at the bottom of the CRT display, like icons on a computer screen. In the terminology of 'scopes, these are both called KEYS.

HARDKEY denotes an actual button on the front panel of the oscilloscope which has a label printed on it.

SOFTKEY denotes one of six (6) buttons just at the bottom of the CRT; the function of each button depends on which HARDKEY you have previously pressed.

1. When starting out, press SETUP, then press Default Setup. This sets all controls to their default values, so you are starting fresh and are not stuck with whatever settings the previous user had. With this and other instruments we use in the studio, it is important to remember that you are dealing with computer controls that need to be programmed. Programming is done through the KEYS and the computer remembers the state it was in when it was last turned off. Also, like any computer, when all else fails, turn the instrument off and then back on. *Depending upon who has used your station in other courses, you may find you need to reset the 'scope before every class.*
2. There are two standard ways of connecting the scope to the circuit you are measuring. You can connect using a standard coaxial cable with some kind of clip leads on it or you can use a scope probe. The default setting for probe type is a 1X probe, which assumes you will be using the coaxial cable with clip leads. Most measurements are done with a 10X attenuating probe, in order to minimize the loading effects of the probe/oscilloscope system on the circuit under test. Since we are not especially worried about exact measurements in this course, we don't use 10X probes.
3. For many repetitive signals (when  $f$  is greater than or equal to 50 Hz, duty cycle is greater than or equal to 1%), pressing the AUTO-SCALE hardkey will give a very satisfactory initial display. In this class, since we use many mechanical systems (which are much slower in general than electrical systems), AUTO-SCALE often will not be able to pick up your signal. In many cases, it will pick up the noise instead. When using AUTO-SCALE always be careful to check if your signal makes sense and if your time scale is reasonable. You will often have to adjust the time and voltage scales by manually. You should experiment with the Vertical and Horizontal controls (Volts/Div, Time/Div, etc.) to produce the most useful display. Sometimes it is useful to view a signal at two different time scales to illustrate different features.
4. Generally in this class, it is best to use DC coupling. AC coupling removes the DC offset from your signal. If your DC offset is too large to get a reasonable display of the output amplitude of the signal, you can try switching to AC coupling (which removes the DC offset automatically). In this class, however, we are often interested in the value of the DC offset. Using AC coupling essentially erases this part of the signal and should be used with caution.