Class #4: Experiment
Circuit Simulation with LTspice IV

Purpose: The objectives of this experiment are to gain some experience with the tools we use (i.e. the electronic test and measuring equipment and the analysis software) and to gain some fundamental understanding of voltage dividers.

Background: Before doing this experiment, students should be able to
- Determine the values of series and parallel combinations of resistors
- Identify the audible frequency spectrum in humans
- Identify the value of standard, low wattage resistors from the color and pattern of their stripes
- Download and install software on a Windows machine

Learning Outcomes: Students will be able to
- Simulate a simple resistive voltage divider and demonstrate conditions under which measurement devices (e.g. DMM or oscilloscope) significantly affect the operation of the circuit. Then, use the changes in voltages caused by the measurement devices to determine the resistance of the measurement device.
- Be able to set up the numerical analysis of simple resistive circuits driven by constant and periodic voltage sources.
- Simulate and display the voltages in a simple resistive circuit driven by a sinusoidal voltage source.
- Fully annotate voltage plots obtained from simulated experiments, including such signal characteristics as frequency (both types), period, amplitude, average or DC offset, etc. and identify where on a standard circuit diagram the voltages are found.
- Articulate a series of questions posed about simple circuits and answer the questions using fully annotated data obtained both from physical and simulated experiments.
- Develop the circuit model of a physical battery using an ideal voltage source and an ideal resistor.
- Calculate the power delivered by a battery and dissipated in a resistor.

Equipment Required
- None

Pre-Lab

Required Reading: Before beginning the lab, at least one team member must read over and be generally acquainted with this document and the other **required reading** materials listed on the course website.

Hand-Drawn Circuit Diagrams: Before beginning the lab, hand-drawn circuit diagrams must be prepared for all circuits either to be analyzed using SPICE or physically built and characterized using your Analog Discovery board.

Part A – Sine Waves and Hearing

In this exercise, a function generator will be used to produce electrical signals with various shapes, including sine waves. Our objective is to learn about the basic properties of sine waves and related signals by seeing them, hearing them and analyzing them with the oscilloscope and audio output capabilities of the Analog Discovery. You will need a set of ear buds or something similar to hear the audio. We will also demonstrate some interesting facts about human hearing and speech.

Background

Equipment: What formerly would require the use of an entire workbench of equipment can now be accomplished using the Analog Discovery (see **Figure A-1** below) and a laptop computer. This board, coupled with the Digilent Waveforms software, can produce the same functionality as each of the following pieces of equipment (and more): a two channel oscilloscope (scope), a digital voltmeter (DVM), two DC power supplies and a two channel function generator. The digital voltmeter (DVM) has 2 channels (Here we use the Scope Channel 1+ (Orange) and Scope Channel 2+ (Blue)). (In Wavesforms 2015, use Logger) The scope is a measuring device that lets you view a plot of a voltage signal vs time. The DC power supplies generate constant DC voltage signals (like a battery). The function generator creates a voltage signal that varies with time. The PC is an integral part of the equipment setup. You use it to simulate many of the circuits you will build (using a SPICE program), as well as to operate Analog Discovery.
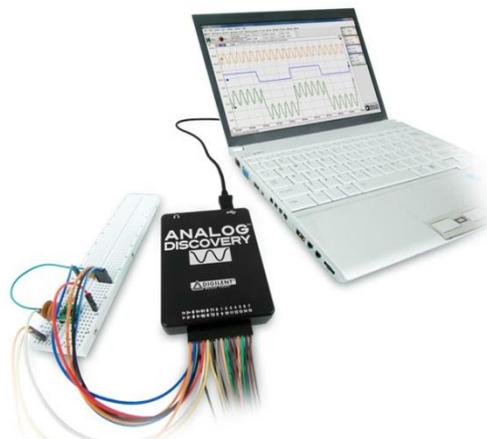
The function generator can be programmed to generate waves with specified amplitude and frequency. Ear buds and speakers convert an electrical signal to sound that we then can hear. The oscilloscope analyzes an electrical signal and displays a picture of the signal. The combination of the oscilloscope and audio output allows us to see with our eyes what we are hearing with our ears. The two function generators are labeled as Waveform Generator W1 (Yellow) and Waveform Generator W2 (Yellow/White). We will start with only one of the function generators in this experiment (Waveform Generator W1). See **Figure A-1**.

The sine wave equation: All of us should have studied the sine and cosine trigonometric functions in math and physics classes. A sine wave is described by an equation of the form $v(t) = A \sin(2\pi ft) = A \sin(\omega t)$, where the variable t represents time. We use the term "wave'" because the shape is similar to a water wave that you might see on an ocean or a lake. As shown in **Figure A-2**, 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.
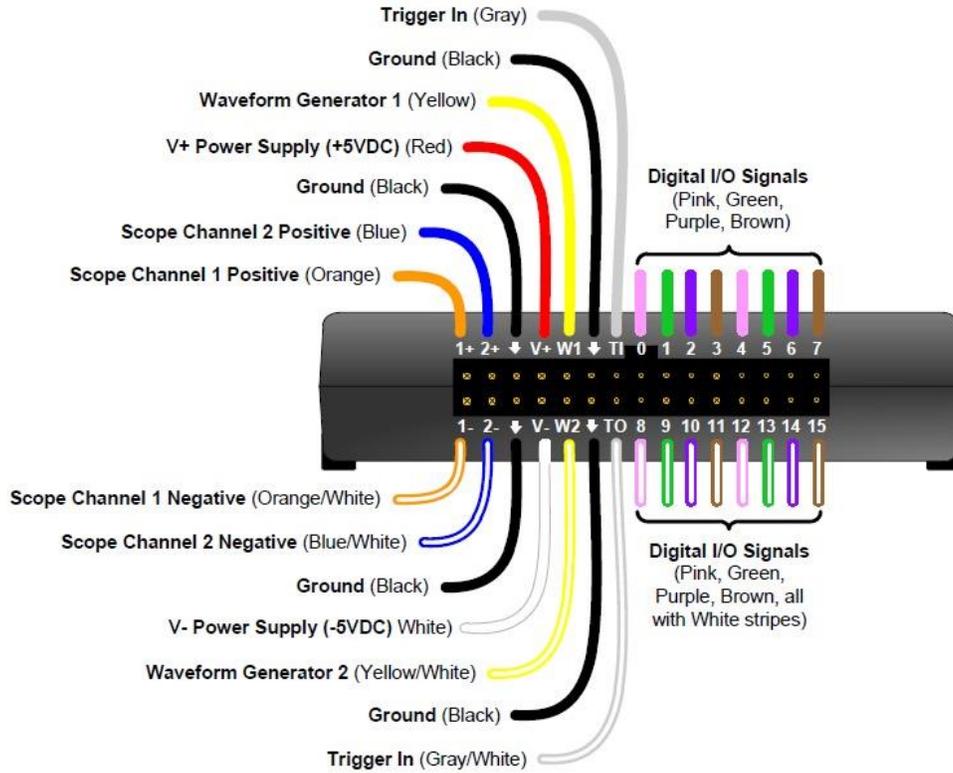
Figure A-1 Analog Discovery & Pin-Out Diagram

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 sec^{-1}. Another name for the unit sec^{-1} is Hertz, or Hz for short. It is usual to denote the product $2\pi f$ as ω, where ω is called the angular frequency in electronics. (In physics, this is the rate of change of the angle in a rotating system, called angular velocity.) Note that one of the most common mistakes made in this class is confusing f and ω.
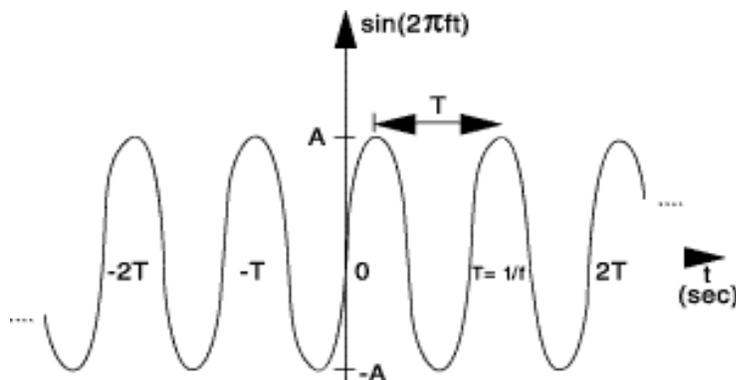

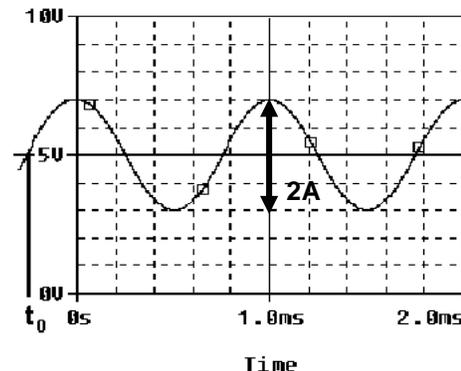**Figure A-2. Sine wave with amplitude A, frequency f, and period T.**   **Figure A-3**

Adding a DC offset: If we add a DC offset voltage to the sine wave signal, as shown in **Figure A-3**, it moves the wave such that it is centered around V_{DC}, the DC offset. The equation becomes v (t) = A sin($2\pi ft$)+V_{DC}. In electronics, the AC and DC parts of a signal can be treated as two mutually exclusive entities.

Scalar measurement of sine waves: 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 $V_{RMS} = V/\sqrt{2} = V/1.414$. Note that in electronics the RMS voltage depends only on the time-varying amplitude and not on any offset.

Impedance and resistance: You should be familiar with the term *resistance*. It is a measure of the degree to which a resistor resists the flow of electrons. Circuits that have a combination of components (some of which are not resistors) also affect the flow of electrons. However, the behavior of these circuits is more complicated because it varies with the frequency of the signal. We call this complicated response "*impedance.*" Both resistance and impedance are measured in Ohms (Ω) and the terms are often used interchangeably.

Human hearing: 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 (amplitude) and pitch (frequency). 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. Keep in mind that your ear is a biological system. It is designed to hear certain pitches better than others even though, technically, they have the same loudness.

Experiment

A.1) Setting up a Sine Wave on the Function Generator

For the first experiment, we need to set up a sinusoidal voltage. Set up Analog Discovery to generate a signal from function generator 1 (W1, GND) and measure it with scope channel 1 (1+, 1-). This can either be done by connecting the wires directly (Orange to Yellow and Orange/White to Black) or using a protoboard. Both options are shown in **Figure A-4**. Note that the connection pins that come with Analog Discovery can be broken apart, but it is helpful often to have some remain in pairs or other combinations.
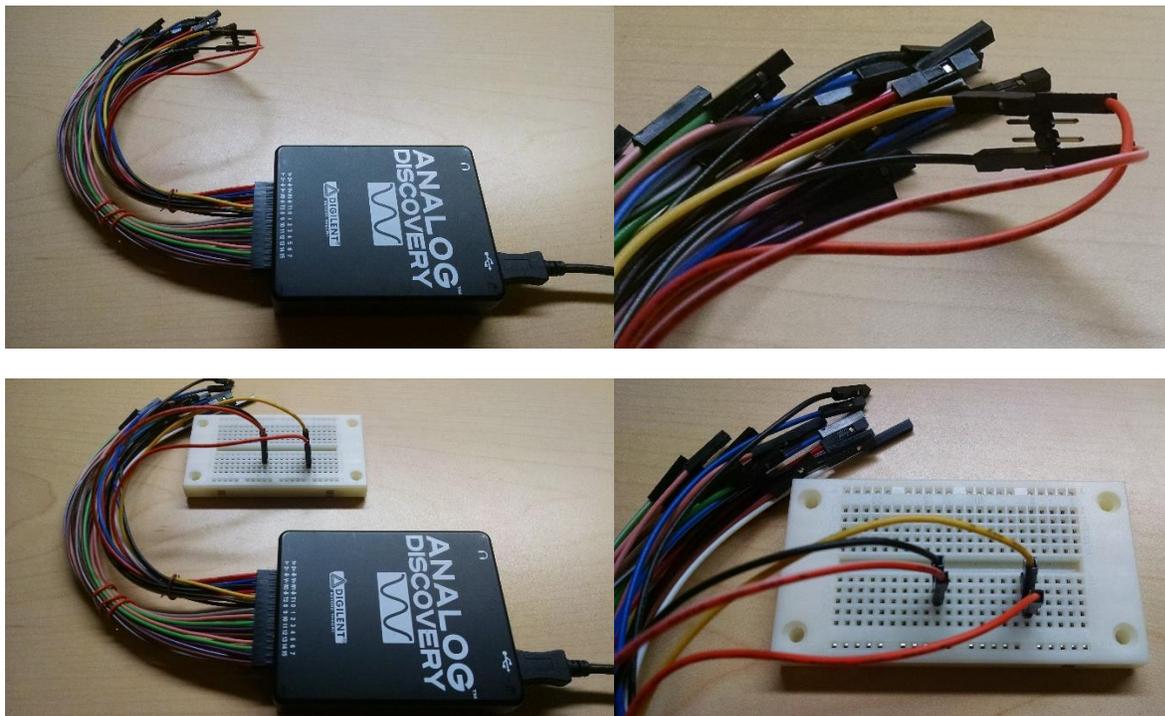

Figure A-4

After correctly installing the Digilent Waveforms software and connecting the Analog Discovery, open the software and select the WaveGen (for the function generator) and the Scope (for the oscilloscope) from the Digilent Waveforms window. You should see the window similar to that shown in **Figure A-5** (with 'Basic' selected) for the function generator and the window shown in **Figure A-6** for the scope. The values for various parameters shown in both windows will likely be different. We will set those up next.

Function Generator:
- First we will set the frequency. The frequency of the function generator is adjusted as follows:
  - Make sure that you choose the channel or channels you are using in the "Channels" menu. The default choice when Waveforms starts is usually Channel 1 (AWG1). If this is the case, you do not need to use the Channels drop down menu. We only need one Function Generator in this experiment. Note that we will use Function Generator and Arbitrary Waveform Generator (AWG) pretty much interchangeably. This system can produce any time-varying signal, so it is an AWG, but we mostly use it to produce sine, triangular and square waves like a function generator.
  - Make sure that the "Frequency" box is checked and AWG configuration mode is "Basic". Select the "Frequency" box or drag the "frequency bar" for Ch.1. Set it to display 1kHz.
- Amplitude: Make sure the Amplitude is checked. Set the voltage amplitude to 200mV.
- Your WaveGen window should look like **Figure A-5**. Make sure ‾Enabled‾ is showing.
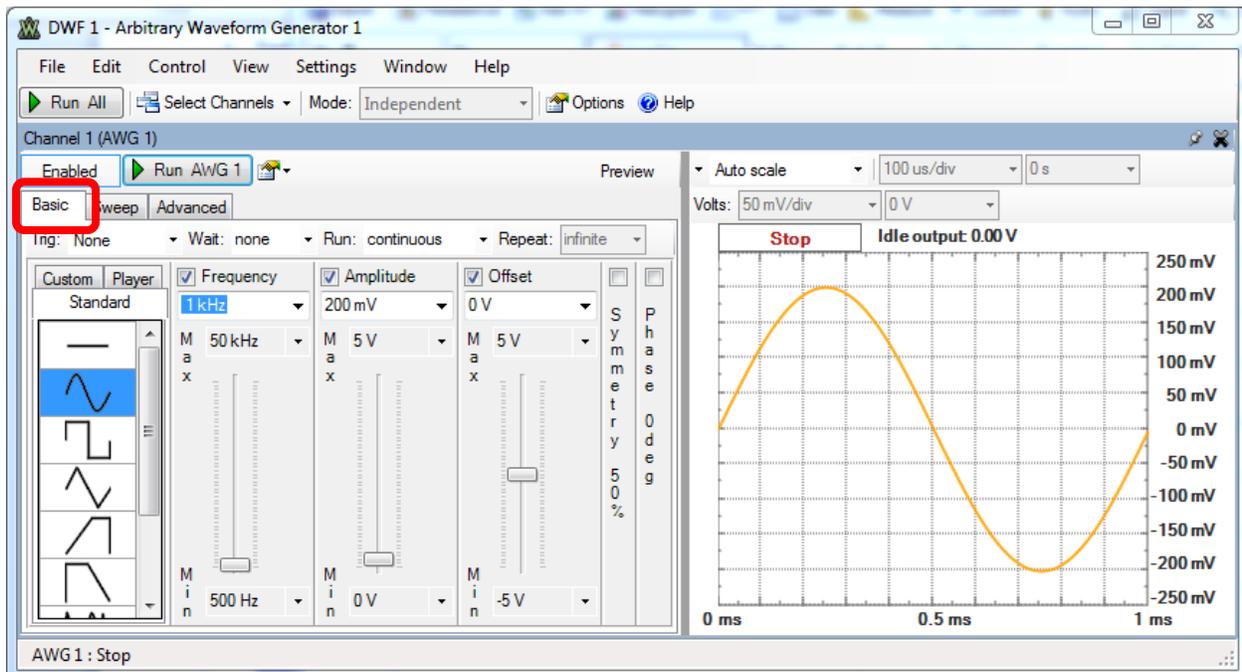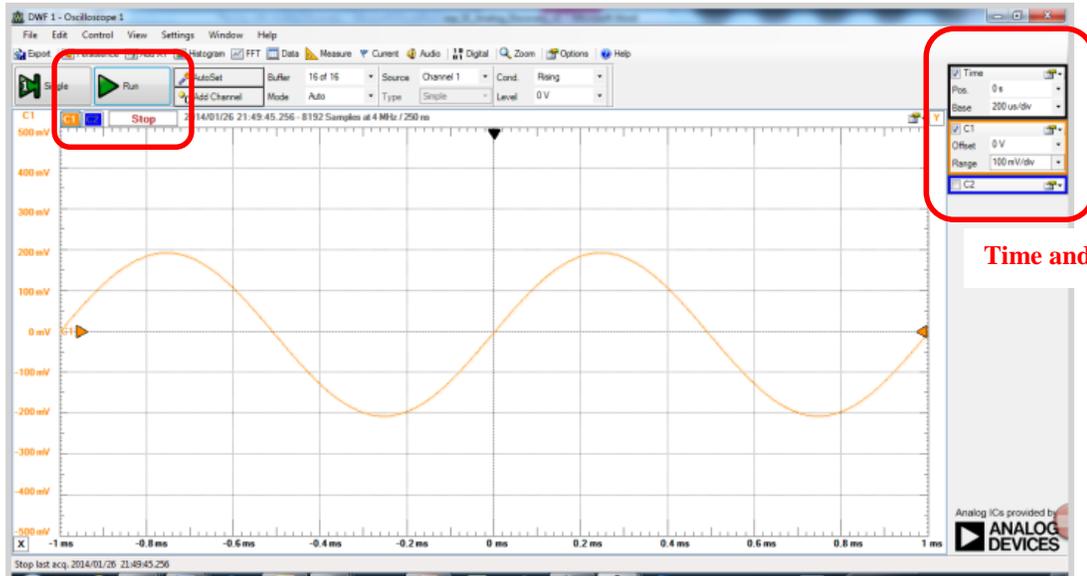

Figure A-5

Scope:
- First we set up the vertical and horizontal scales for the display. On scope channel C1, select the Volts/div to 100mV, the offset to 0 V. Uncheck C2, since we will not be using channel 2. The Time/div should be set at 200μs/div. The voltage and time scale settings are found on the right hand side of the scope window.
- To make a measurement, connect the source (W1) to scope input (1+) and scope input (1-) to ground. Make sure the connections you made above are good.
- When you are ready, press the "Run AWG1" button on the WaveGen and the "Run" button on the Scope. These are the 'On' buttons for the function generator and scope, respectively. If you cannot see a signal on the scope, double-check to make sure all of the settings are correct.
- It is useful to know how to copy plots and paste them into word. In the upper left hand corner, under the *Files* menu, select Export. Select the Images tab and you can save the Oscilloscope screen to a file or use the Copy to Clipboard option to put the image directly into your laboratory report. Save a copy of the Oscilloscope plot for your report.

- Change the frequency up or down as desired. How does this change the signal on the scope? The purpose of this step is to see what kind of signals this setup can produce. You should play around a little with different frequencies, voltage amplitudes, signal shapes, etc.
- Set WaveGen again so the display reads 1kHz and the amplitude is 200mV with no offset. Use the 'Copy Window as Image' option in the Edit drop down menu on the Scope and paste the image in your report document. Clearly label both the amplitude and period of the signal you have measured.


Run Button

Time and voltage scales

Figure A-6

A.2) Using the Audio Output from Analog Discovery
We now wish to connect the function generator, the scope and earphones to perform some simple experiments.

- Start by measuring the resistance of each channel for your ear buds or earphones (from left or right to ground or common) using a desktop or handheld DMM. We only need an approximate value for the resistance, so any device can be used for this purpose. There are typically two types of connectors used for earbuds, depending on whether or not a microphone is included. Both are shown below.

1. Left
2. Right
3. Common
4. Mic input

Left
Right
Ground
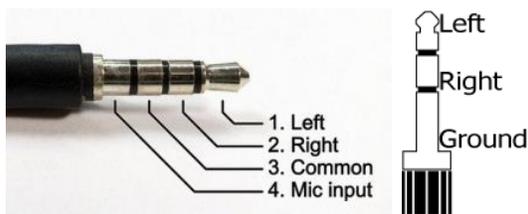
- Plug your ear buds into the audio output on the Analog Discovery. Do not do this with the ear buds in your ears. The volume may be too high. It is prudent to turn on the volume with the ear buds away from your ears and bring them closer until you are sure the volume level is comfortable. You should hear only one channel. If you use both AWG sources you will hear both channels.
- Adjust the volume of the signal to a comfortable level by changing the amplitude of the signal. By comfortable level, we mean the lowest amplitude that allows you to hear a distinct sound. What is the value of the voltage amplitude that you have selected?
- Let us investigate how our perception of loudness changes as the frequency of the sine wave is varied. With the sine wave amplitude fixed at your comfortable level, vary the frequency over the range from 100Hz to 10,000Hz. Try cycling through the following frequencies, without changing the signal amplitude: 100, 200, 300, 400, 500, 600, 700, 800, 900, 1000, 2000, 3000, 4000, 5000, 6000, 7000, 8000, 9000, and 10,000Hz. Which frequency do you hear the loudest? Is there any variation among the members of your team? If you have problems discerning significant differences in loudness, try a different set of ear buds.

Experiment with the Equipment

At this point, you will have put the function generator and scope through some basic tasks. Experiment with the other features of the function generator 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. 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?

Summary

You should now know how to set up voltage signals with the function generator feature, connect the function generator output to the scope input and display them using the oscilloscope feature. You should understand the pitch/frequency and amplitude/volume relationships, and know how these relate to human hearing.

Part B – Introduction to *LTspice IV*

In this section we will learn about the circuit analysis software we will use as our primary simulation tool. You should download and install this software on your laptop. It is recommended that you install the latest version.

Background

The software we will be using to simulate the operation of circuits in this course is called *LTspice IV* which takes information on a circuit and analyzes how it will behave. When placing parts in the circuit, resistors, capacitors, inductors, diodes and wires all have their own symbols in the menu. All other parts are accessed through the "Place Parts" button. Once the parts are places, their values must be specified. This is usually done by right clicking your mouse on the circuit element or one of its parameters. Once the circuit is complete, the simulation must be setup. This is done through the 'Simulate' drop down menu. The example below shows a transient simulation that runs from 0 to 5ms with a maximum step size of 1µs. Once the simulation is run, it displays an output similar to what you would see if you hooked the circuit to an oscilloscope. Figure B-2 contains a sample diagram.
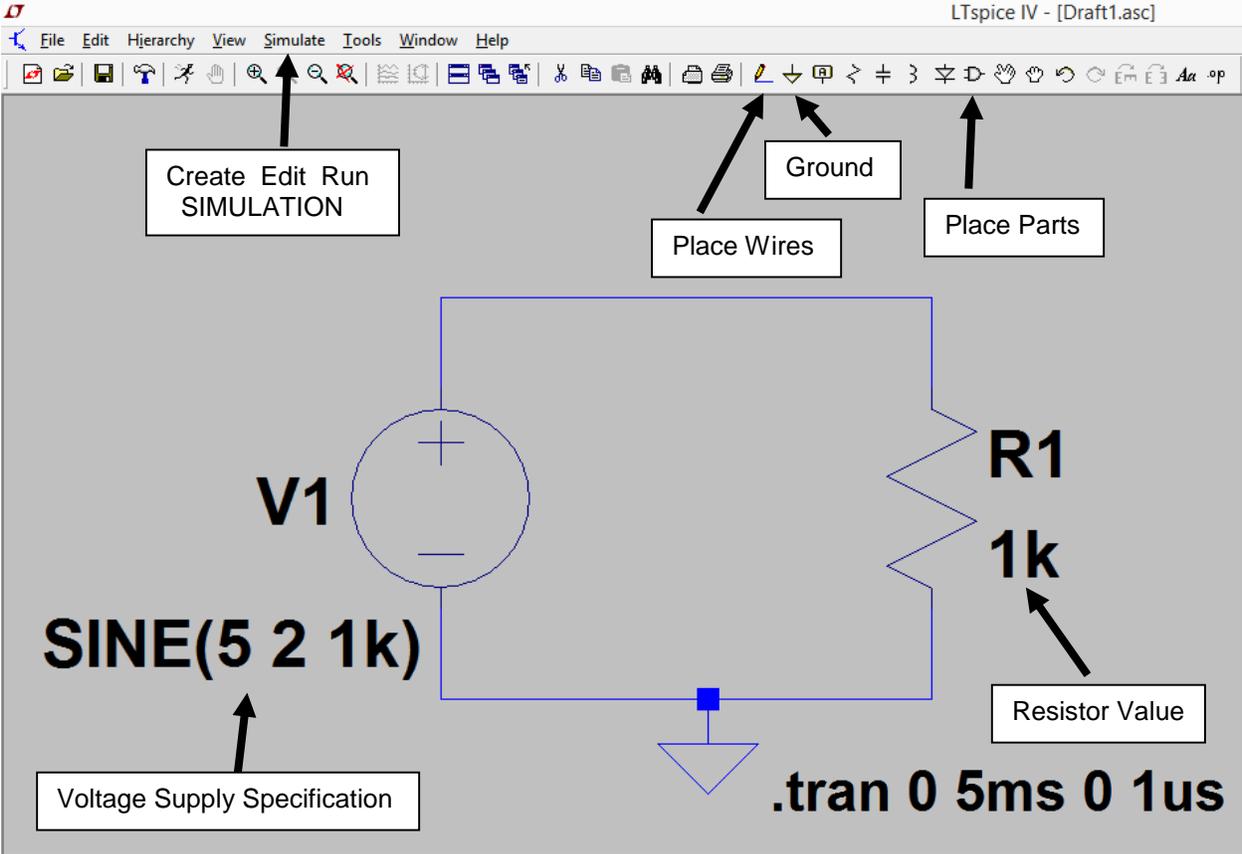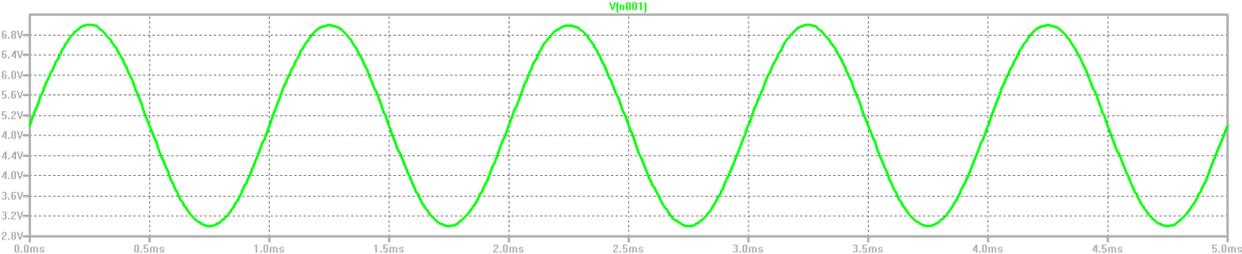

Figure A-1.

Figure B-2.

Experiment (Simulation)

Opening a New Simulation
In this part of the experiment, we will draw the simple circuit we have been studying, a combination of resistors and a sinusoidal voltage source representing a resistive divider with voltages measured by Analog Discovery.

- Run the "*LTspice IV*" program

- It will open with no existing simulation. Click on the *File* pull-down menu and select *New Schematic*. You will name your project when you save it. You should also decide where you want to save schematics. You will have to choose a directory when you save your work.

Drawing a Circuit
Figure A-3 is a picture of the *LTspice IV* main screen with the circuit we will be drawing. Note that this is the circuit used in Figure B-7 from the previous experiment including the input resistance of one channel of the Analog Discovery as R3.
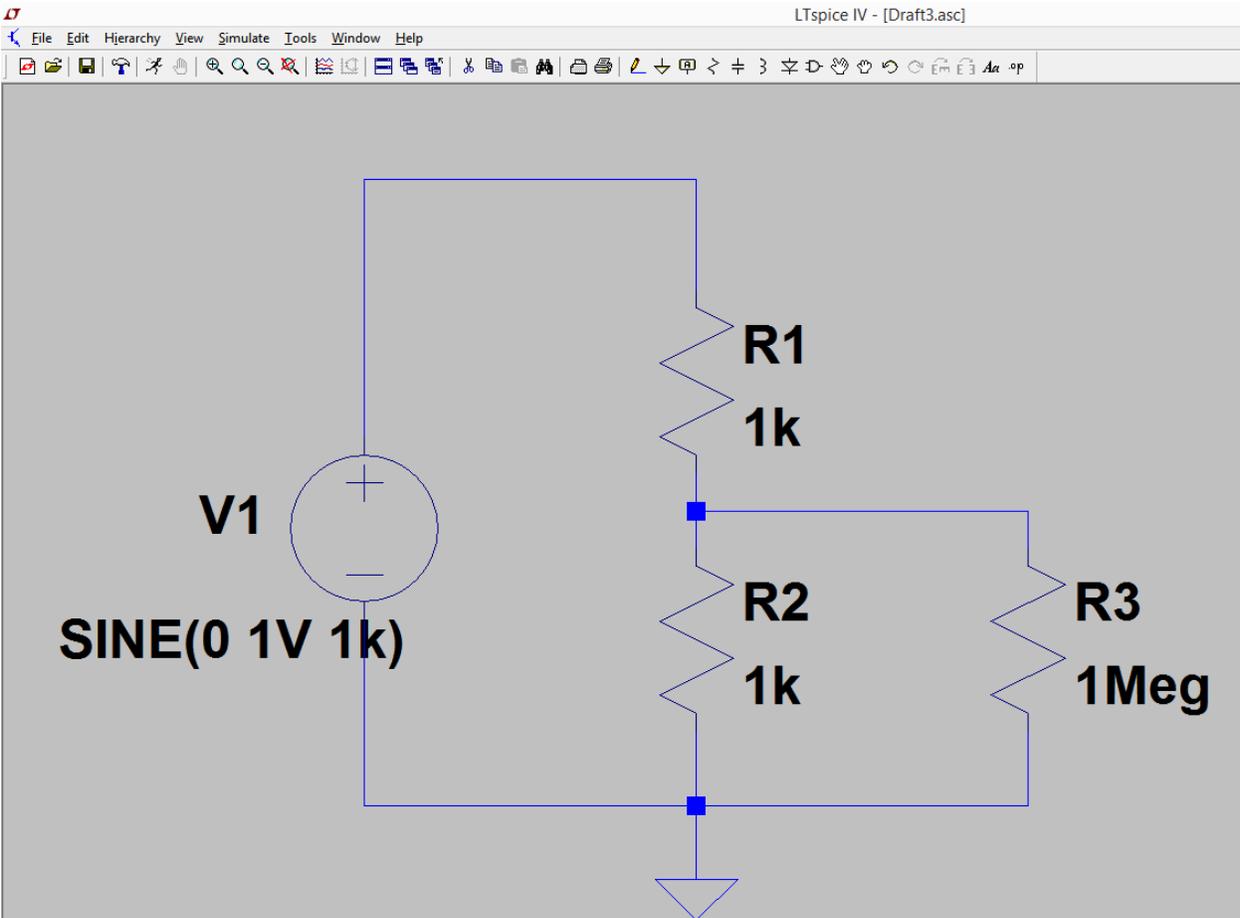

Figure B-3.

In the circuit shown in Figure B-3, 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 we will click on the button that looks like a resistor. You can also do this by using the *Edit* menu or by hitting the *R* key. You should see a resistor symbol that you are free to place anywhere in the schematic. The *Edit* menu also shows you what name goes with every symbol. For the ground, select the ground button, use the *Edit* menu or hit the *G* key. For the voltage source and most other

components, you have to use the component button ⬡ (which can also be identified from the *Edit* menu) and then pick *Voltage* from the many options offered.
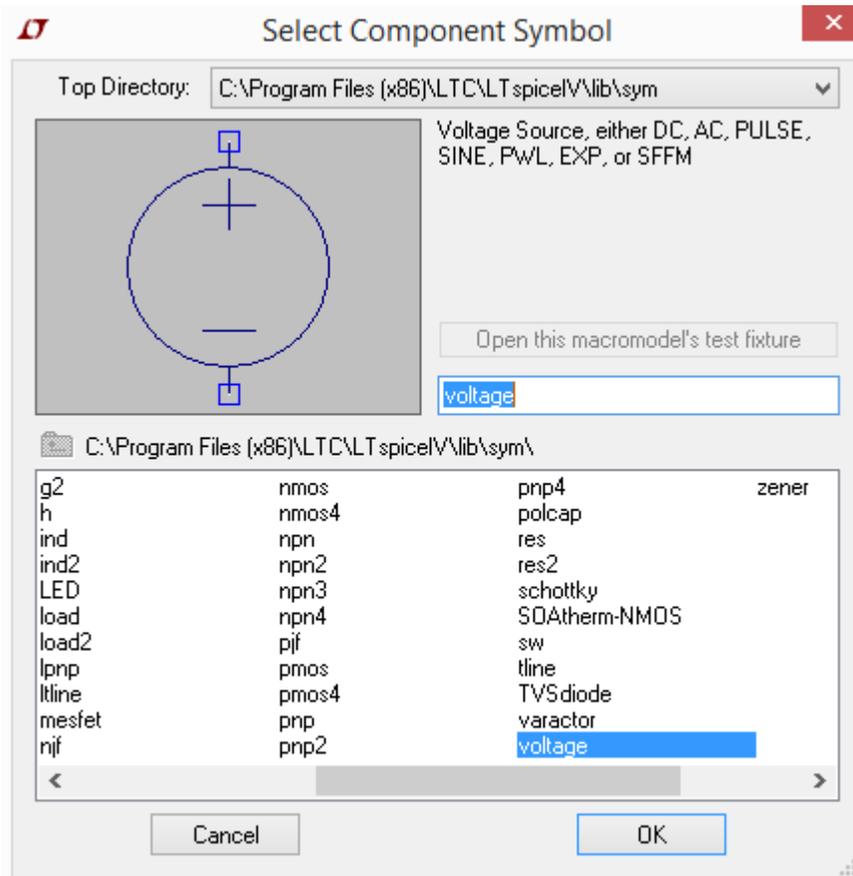

Figure B-4.

- When you have finished placing resistors or doing just about anything else, hit the *Esc* key to get back to the basic diagram.
- When you have placed all resistors, the voltage source and the ground, connect everything with wires. Access wires by clicking on the symbol that looks like a pencil, next to the ground button.
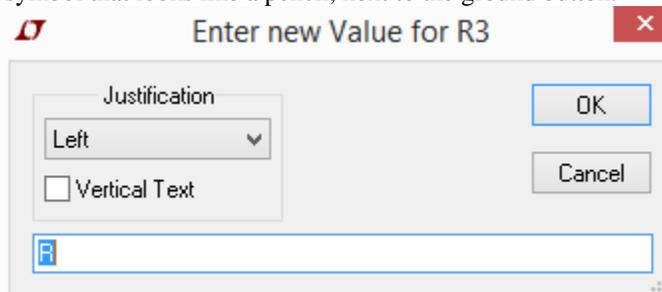

Figure B-5.

- 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 and you can change the names if you wish the same way their parameter values are changed. To change R3 to 1Meg Ohm, right click on the bottom *R* and you should get the window shown in Figure B-5. Change the *R* to *1MEG*. If you find that the number or anything else is in a hard to read position on the circuit diagram, move it with the mouse by first clicking on the *Move*

button that looks like a hand ✋. When you type the value of the resistance, you must type *1MEG* **with no spaces**. Note that you have to type *MEG* since *SPICE* uses *M* to mean 10^{-3} and *MEG* for 10^6. It is not case sensitive. Note that in the schematic pictured, R1 and R2 are the resistors in your voltage divider and R3 represents the impedance of the scope. Since the scope impedance is 1M, R3 should be 1MEG. R1 and R2 should be 1k Ohms. .

- After you have changed all the resistor values and moved them to readable positions, you must set up the voltage source. In *LTspice IV*, voltage sources are DC by default. To specify the source as sinusoidal, right click on the voltage source symbol and select the *Advanced* button. The window in **Figure B-6** will appear. Select *SINE* and then set the values shown for offset, amplitude and frequency. Once the symbol is labeled as *SINE*, these parameters can be changed by right clicking on the symbol. The values are used to label the voltage source as shown in **Figure B-3**.
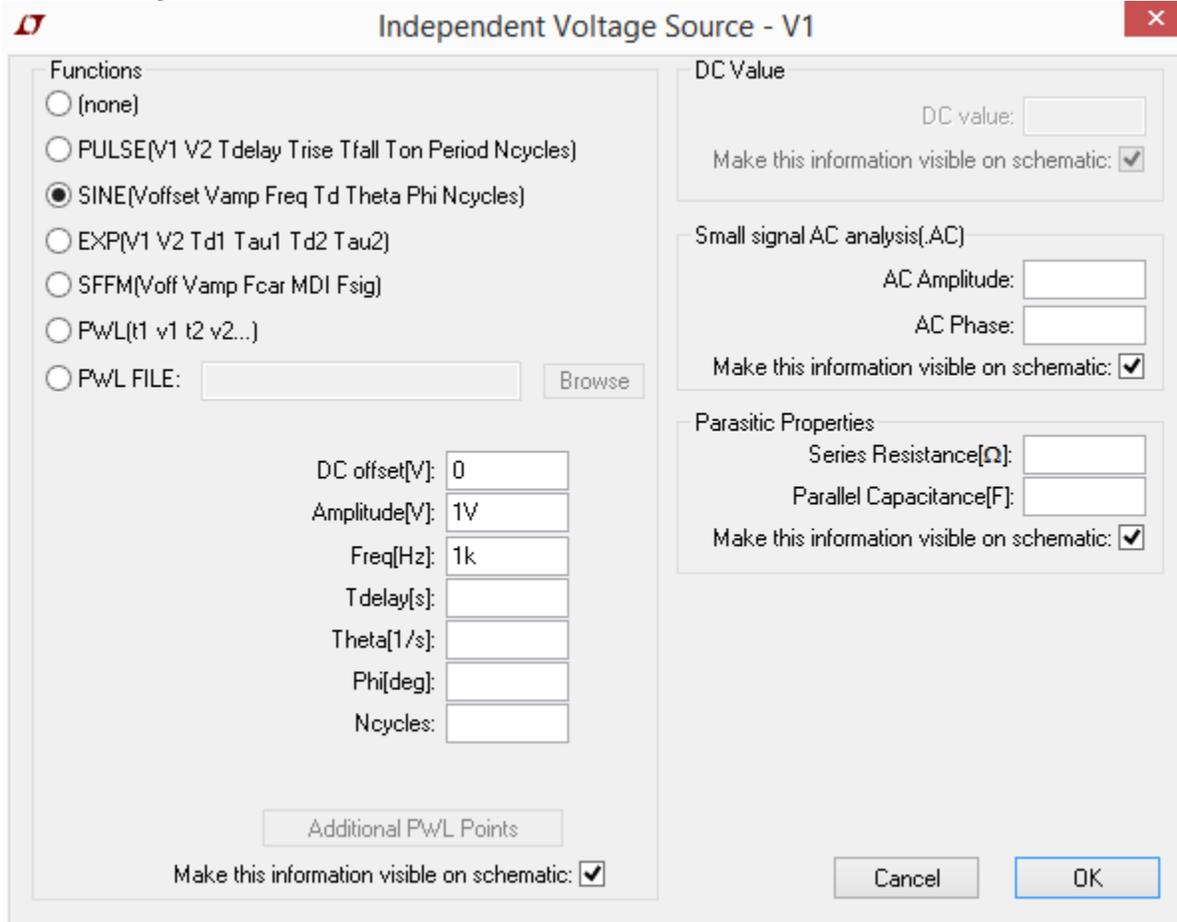

Figure B-6.

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 *Simulation* drop-down menu and click and select *Edit Simulation Cmd*. You will get the window shown in Figure B-7. Select the tab for the type of analysis you wish to perform. In this case it is *Transient* since we are interested in output that is time-dependent like a scope. The simulation will begin at time t = 0, but we can choose to start saving data after that. Thus, we specify the *Stop Time* (the end time of the simulation), the *Time to Start Saving Data*, and the *Maximum Timestep* (the resolution of our simulation). Here we choose, 5ms, 0 and 1μs, respectively. Note that there is no μ so we have to use the letter u.
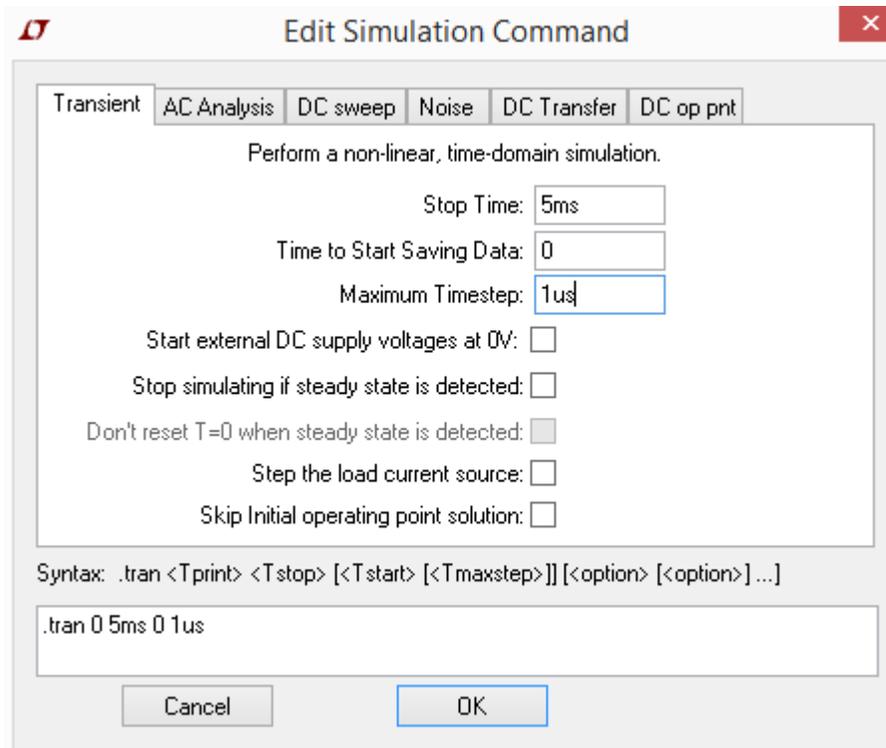
Figure A-7.

The final time *(Stop Time:)* has been set at 5*ms* because the period of a 1kHz sine wave is 1msec. This allows us to see five periods. The *Maximum Timestep* needs to be set so that you get a reasonable representation of the output. A step size that is too small will take a long time to run, a step size that is too big will give you an under-sampled representation of the output. A step size between 1/100 to 1/1000 of the run time is reasonable. 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. You will be able to place the information on the simulation anywhere on your circuit. Choose a space where it is easy to read and does not block anything else.

- When you have finished, you can run the simulation either by pushing the button that looks like a stick person running or select *Run* from the *Simulation* drop-down menu.

Transient Analysis
You are now ready to do the simulation.

- Click on the *Run* button. A set of axes with no signals shown will appear. You can now add voltages where ever you wish by left-clicking with your mouse. You will see that whenever you hover over a wire with your mouse, a symbol like a voltage probe will appear. Click where you want to know the voltage. If you want to plot current, hover over a device symbol and your mouse symbol will change to a current probe. If you click on a device, the current through it will appear on the plot. In the plot shown in **Figure B-8,** the input and output voltages for the voltage divider are displayed. Note that the ratio between the two is what you should expect for this simple circuit.
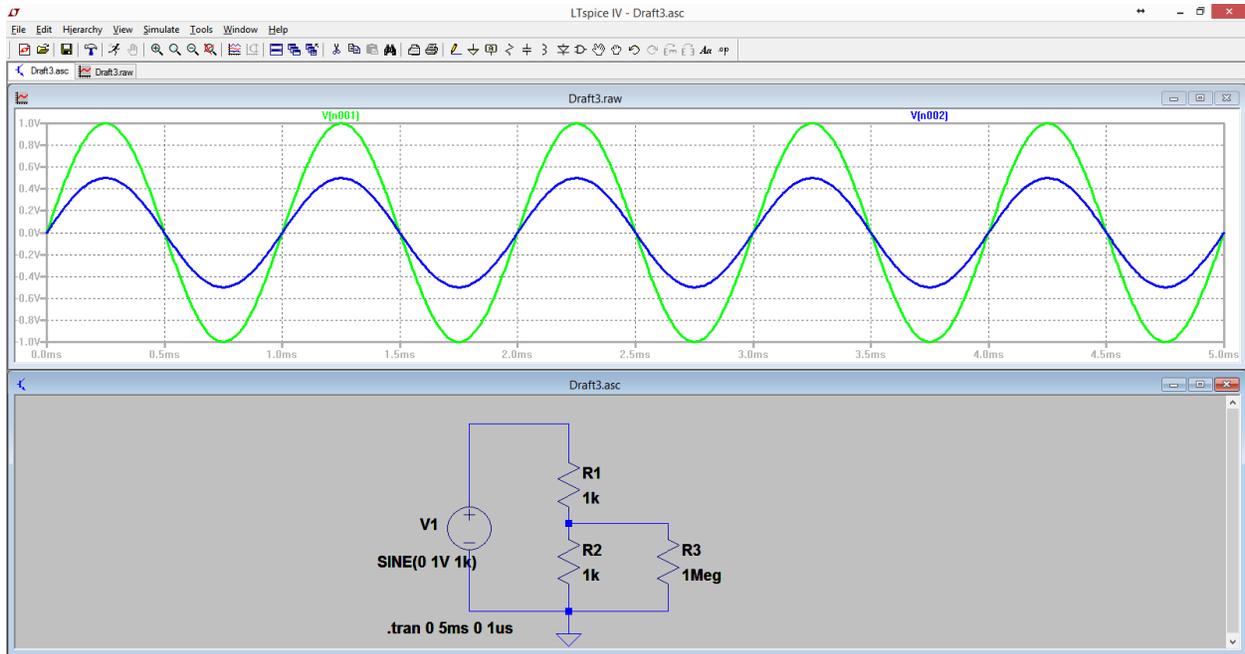
Figure A-8.

- You should get something like (but maybe not identical to) the window shown above. The voltage divider expression we used in the previous lab can be applied to sinusoidal signals as well. For sinusoidal signals, the amplitude of the sine wave changes based on the voltage divider ratio. We can then perform the same comparison between source voltage amplitude and resistor voltage amplitude as we did in Laboratory 3 where we compared the source DC voltage and the resistor DC voltage. For the 1kΩ voltage divider circuit, do we see the same ratio of voltage amplitudes as seen for DC voltages in Laboratory 3?

- It is also useful to know how to copy plots and paste them into word. Under the *Tools* menu, click on "copy to clipboard". Now there is a bitmap in the clipboard that you can paste into any application. Open word and paste the bitmap in. *Save this file or print the output plot for the 1k voltage divider directly.* You should see a plot like the one shown below. If you get the circuit diagram instead, click your mouse on the voltage plot and try again.
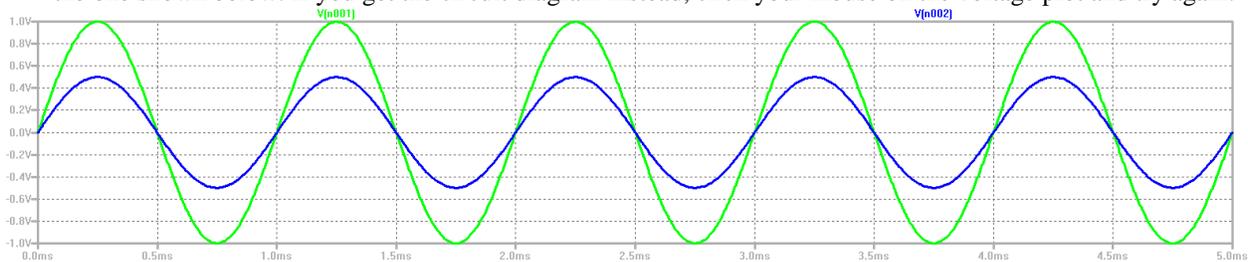
Note that the sine wave lines on the plot can be a bit thin and hard to read. You can change the data display on the plots generated by selecting *Control Panel* from the *Tools* menu and then selecting *Waveforms.* Select *Plot data with thick lines.* They will be much easier to see.

- Change the values of the resistors in the voltage divider to 1MEG and rerun the simulation. *Save or print this plot as well.* Both plots should have two traces: the source voltage, and the voltage across the resistor closest to ground. For the 1MegΩ voltage divider circuit with the 1MegΩ probe resistor, do we see the same ratio of voltage amplitudes as seen for DC voltages in Laboratory 3? Try varying the frequency, amplitude and offset of the VSIN source one at a time and rerun the analysis. What happens to your signal? Is the amplitude ratio still the same? Does it make sense based on your knowledge of sine waves and voltage dividers?

  Show your results for the 1MEG simulation to a TA or instructor and have them sign your hand-drawn circuit diagram for this simulation.

For this and other signatures you obtain in this course, you must explain:
- *the purpose of the data (using your hand-drawn circuit diagram),*
- *what information is contained in the plot and*
- *why you believe that the plot is correct.*

Any member of your group can be asked for the explanation so present your information together..

Summary

LTspice 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

For your report, follow the template proved on the course website. Be sure to include your hand-drawn circuit diagrams and fully annotate all plots so that the questions you are addressing can be easily answered using the plot alone rather than reading the accompanying text.