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 – 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 A-2 contains a sample diagram.

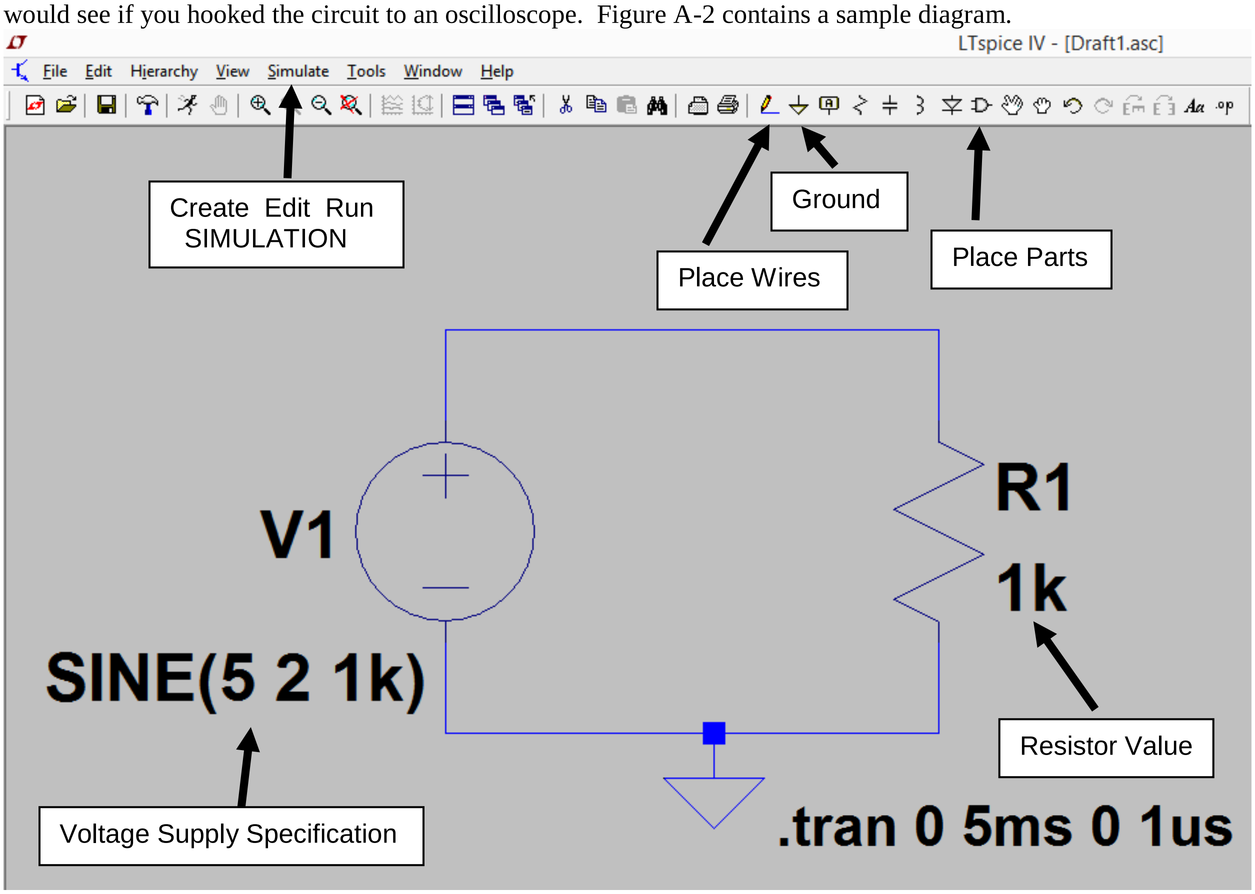


Figure A-1.

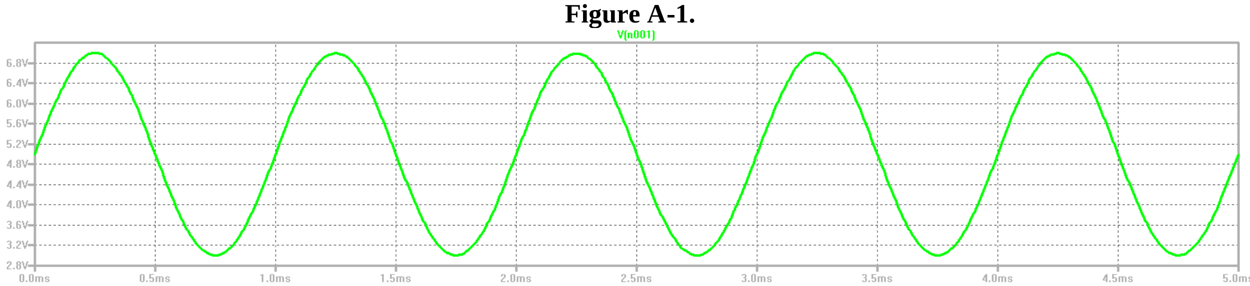


Figure A-2.

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.

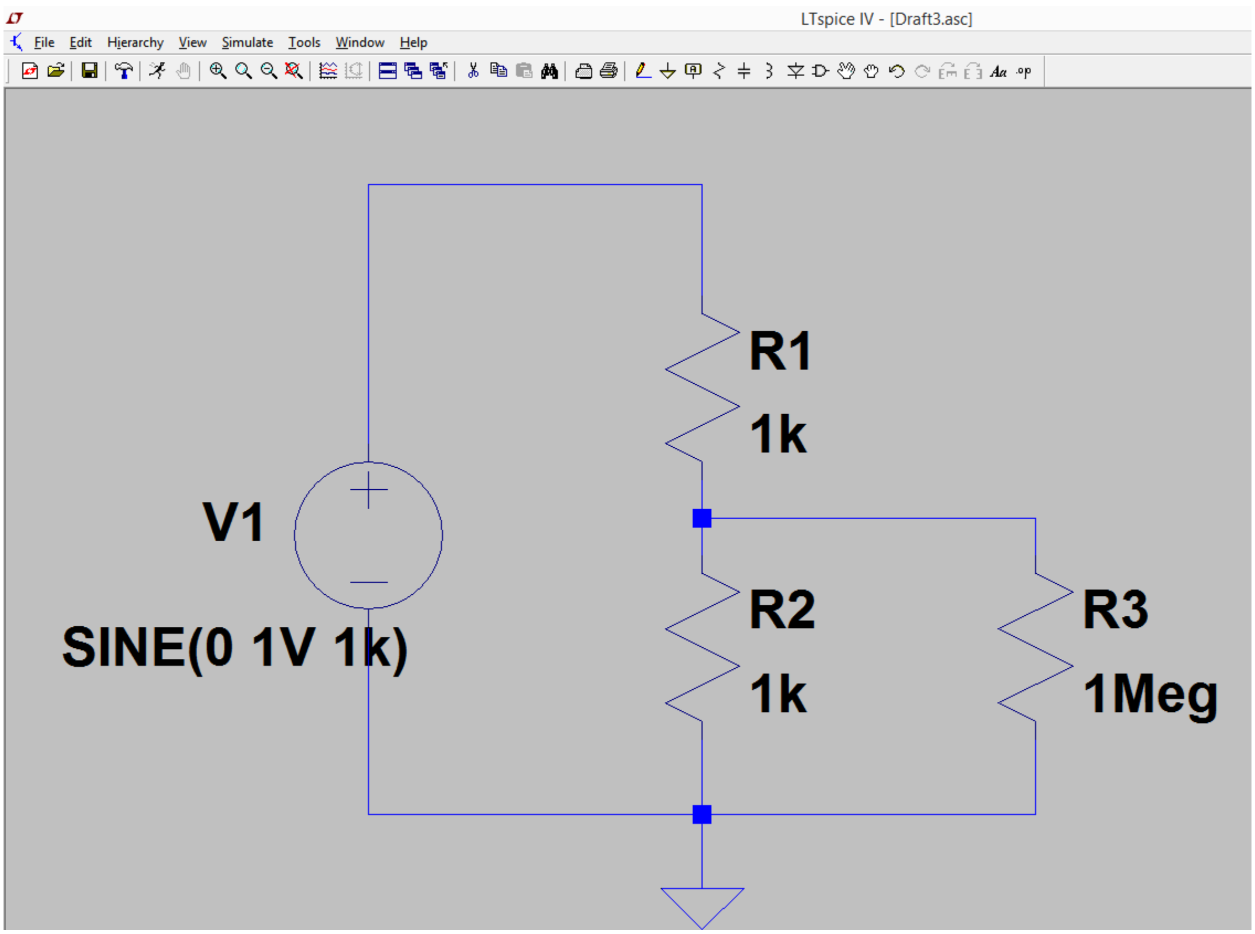


Figure A-3.

In the circuit shown in Figure A-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.

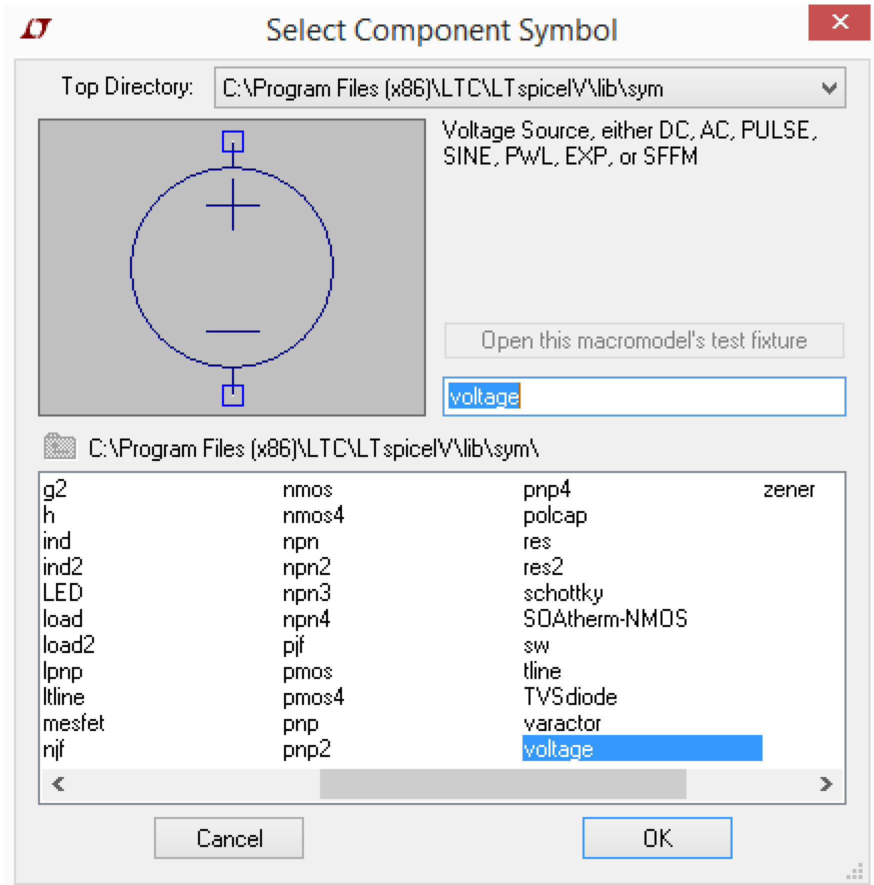


Figure A-4.

- 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.

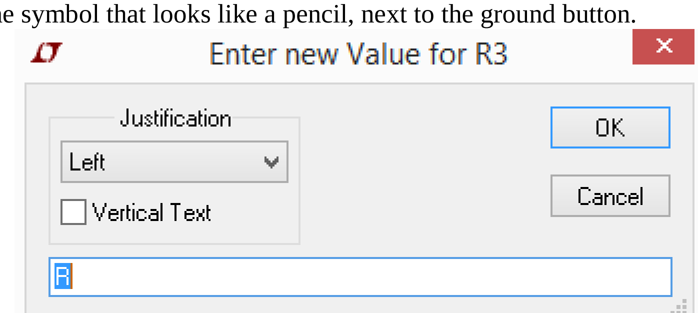


Figure A-5.

- 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 A-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 A-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 A-3**.

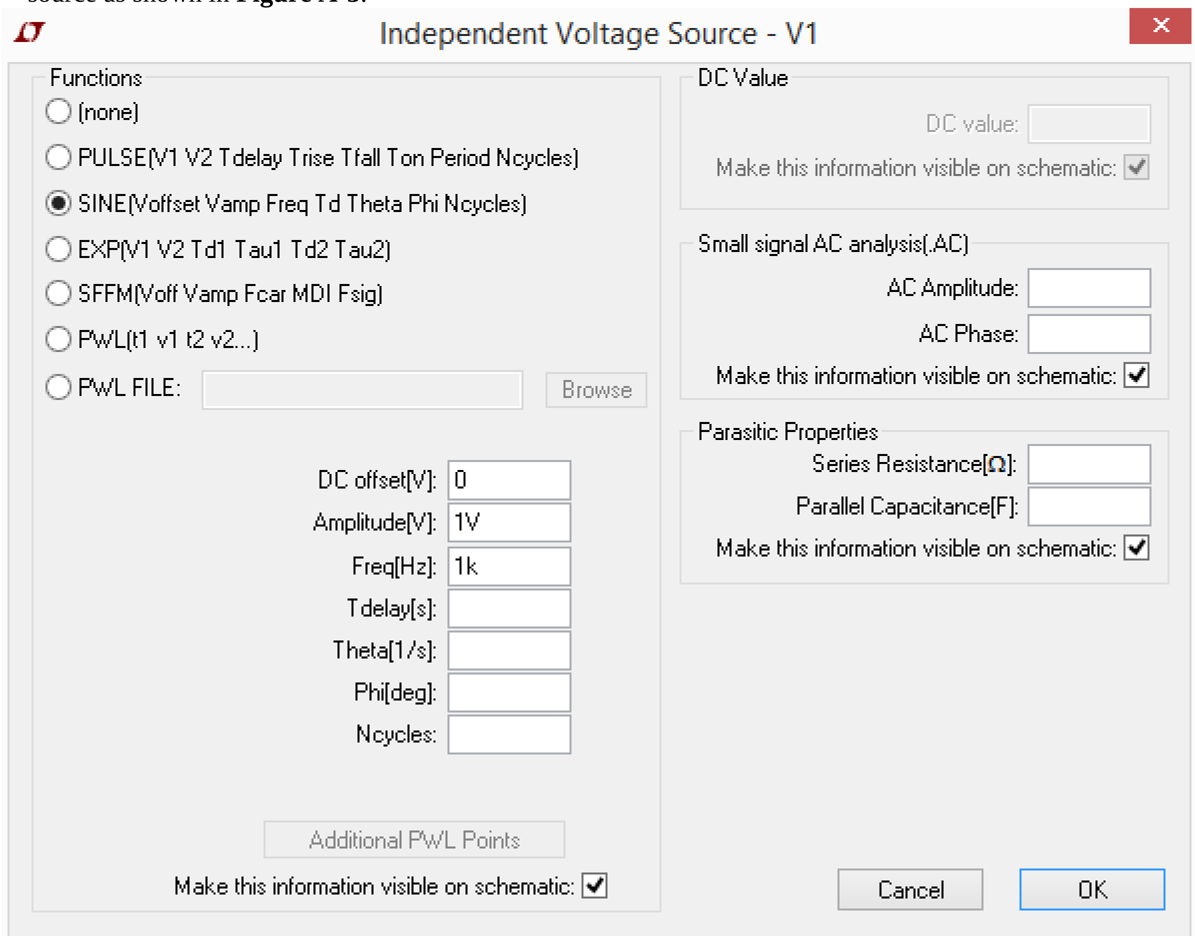


Figure A-6.

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 A-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.

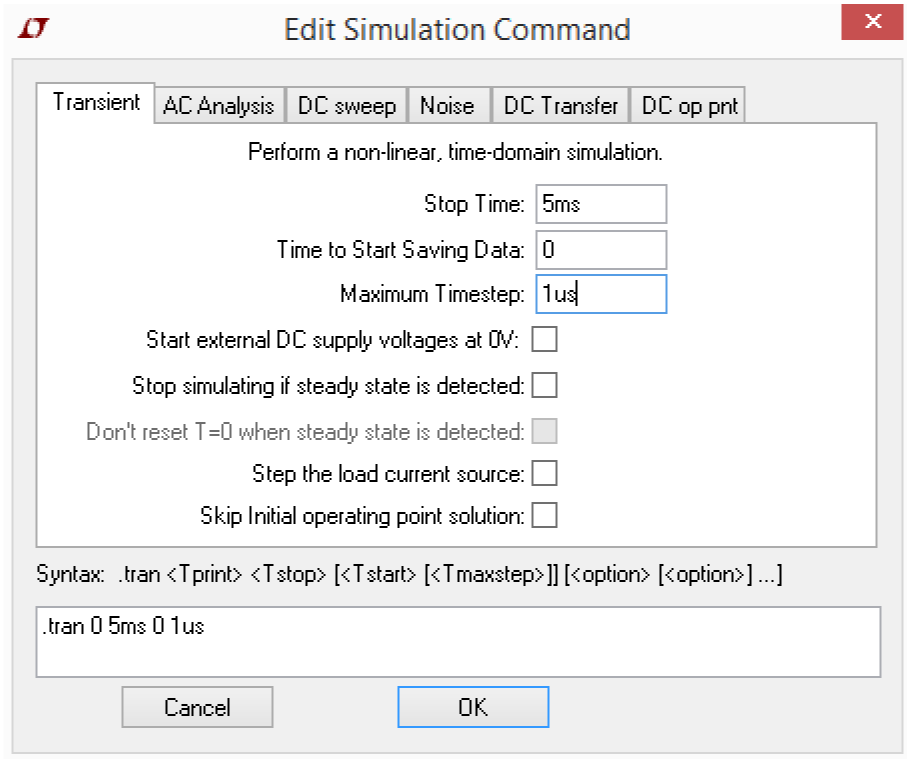


Figure A-7.

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 A-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.

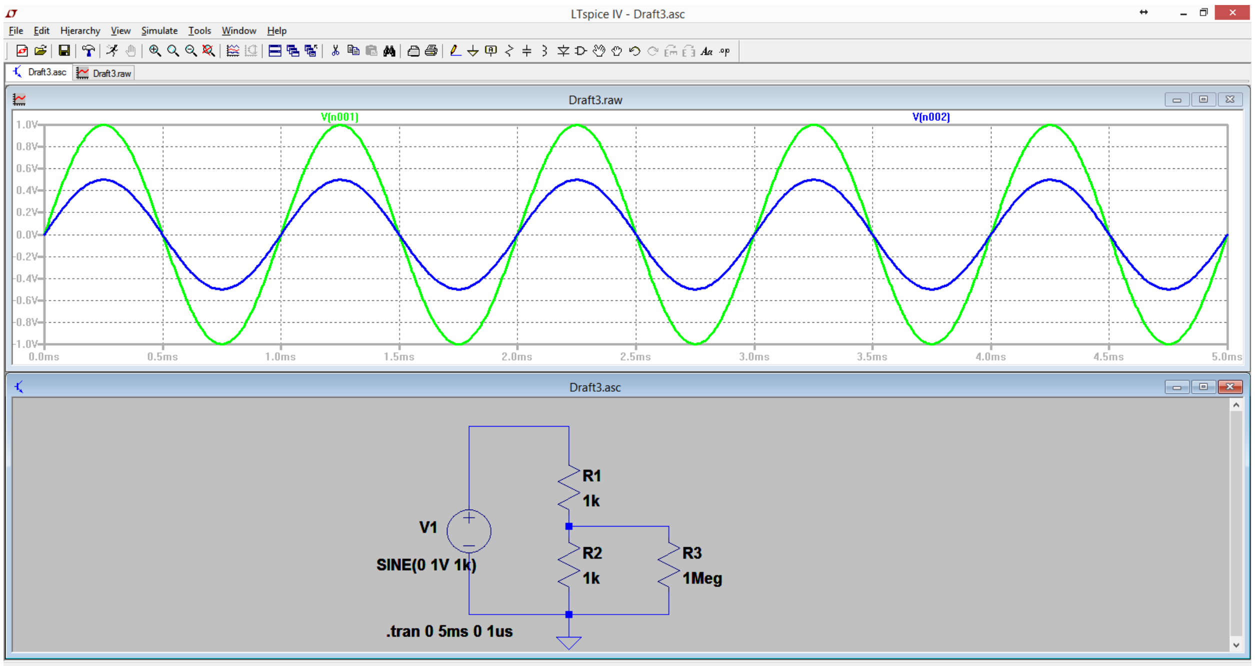
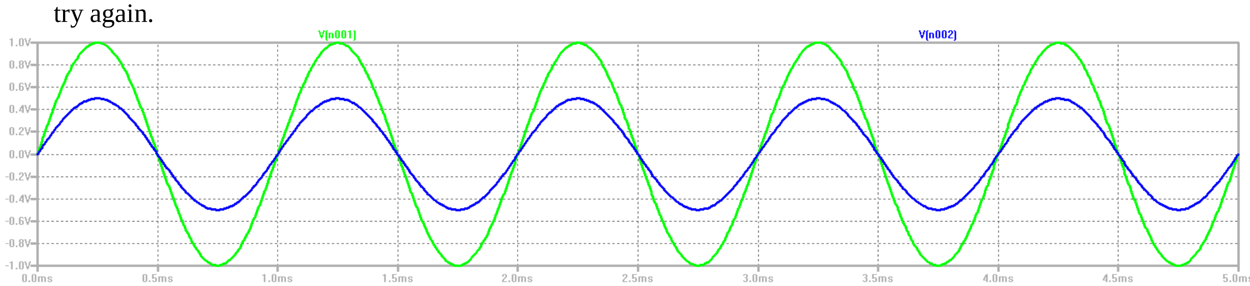


Figure A-8.

- You should get something like (but maybe not identical to) the window shown above.
- 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. Do the plots agree with your results from part B of the previous experiment? 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?

  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 LMS site. 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.