Experiment 1
Signals, Instrumentation, Basic Circuits and Capture/PSpice

Purpose: The objective of this experiment is to gain some experience with the electronic test and measuring equipment and the analysis software.

Equipment Required

- Analog Discovery (With Digilent Waveforms)
- Oscilloscope Analog Discovery
- Function Generator Analog Discovery
- DC Power Supply Analog Discovery & Batteries
- DMM (Handheld multi-meters in JEC 4201)
- Two 100 Ohm resistors, two 1M Ohm resistors and two 1k Ohm resistors.
- Protoboard

Helpful links for this experiment can be found on the Links by Experiment page. Be sure to read over all required info and ask for help when you need it.

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 running Windows. This board, coupled with the Digilent Waveforms software, can produce the same functionality as each of the following pieces of equipment: a two channel oscilloscope (scope), a digital voltmeter (DVM), two DC power supplies, a two channel function generator, and a 16 channel digital IO board. The digital voltmeter (DVM) has 2 channels (Here we use the Scope Channel 1+ (Orange) and Scope Channel 2+ (Blue)). 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 PSpice), as well as to operate Analog Discovery.
In this experiment we will use the function generator, the oscilloscope, and the audio output. 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. 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. We can also determine a mathematical representation of the sound that can then be used for system analysis. The two function generators are labeled as Waveform Generator W1 (Yellow) and Waveform Generator W2 (Yellow/White). We will only need 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.

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 \( \omega \), where \( \omega \) 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 \( \omega \).
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 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 $V_{PP}$ (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 only depends only on the time-varying amplitude and not on any offset.

Impedance and resistance: You are probably 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, $\Omega$, 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 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. 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.

After correctly installing the Digilent Waveforms software and connecting the Analog Discovery, open the software and select the WaveGen feature and the Scope feature from the Digilent Waveforms window.

- 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 “Select 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 Select Channels drop down menu. We only need one Function Generator in this experiment.
  - 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.
  - Make sure the Amplitude is checked. Check that the voltage amplitude is set to 200mV.
  - Your WaveGen window should look like the screen capture below. Make sure is showing.

- On scope channel C1, select the Volts/div to 100mV, the offset to 0 V. Uncheck C2. Time/div should be set at 200us/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.
- When you are ready, press the “Run AWG1” button on the WaveGen and the “Run” button on the Scope. If you cannot see a signal on the scope, double-check to make sure all of the settings are correct.
- 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. It should look something like this:

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 using one of the four DMMs located in the classroom. There are two on the center table and one under the cabinets on each side of the room. You may also use your own multi-meter if you have one.

• 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. 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?

• 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 group? If you have problems discerning significant differences in loudness, try a different set of ear buds.

• 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? Use the website on the links page to verify this.

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.

Review the report write-up section at the end of this document. Make sure that you have all the data required to complete the write-up.

K.A. Connor, S. Bonner, P. Schoch
Rensselaer Polytechnic Institute

Revised: 30 January 2014
Troy, New York, USA
Part B – Voltage Dividers and Measuring Equipment

In this part of the experiment you will be learn that equipment isn’t ideal and that “real” behavior must be taken into account when making measurements. You will look at batteries and measure the effective internal resistance; they aren’t ideal voltage sources. You will also look at the behavior of two voltage dividers when a DC voltage and an AC voltage are applied. You will use circuit analysis to examine the behavior of these circuits.

Background

Impedance: Every piece of electrical equipment has an effect on the circuit you connect it to. Just as it is impossible to design a dynamic mechanical system without friction (that resists motion), it is impossible to design an electrical system without impedance (that resists the flow of electrons). Impedance has two effects on an electrical system. It changes its magnitude (the value of the voltage) and its phase (voltage behavior over time). If the impedance affects only magnitude, then we call it resistance. Each electrical measurement device has an internal impedance, and this is also true for the Analog Discovery. The impedances we will concern ourselves with in this class are listed in table B-1 below: (These values aren’t exactly correct, but they still can be used to make the point.)

<table>
<thead>
<tr>
<th>Device</th>
<th>Impedance (magnitude only)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Scope Analog Discovery</td>
<td>1MegΩ</td>
</tr>
<tr>
<td>DMM (DC voltage)</td>
<td>10MegΩ</td>
</tr>
<tr>
<td>DMM (AC voltage)</td>
<td>1MegΩ</td>
</tr>
<tr>
<td>function generator Analog</td>
<td>very small</td>
</tr>
<tr>
<td>Discovery</td>
<td></td>
</tr>
<tr>
<td>DC power supplies (any)</td>
<td>negligible</td>
</tr>
<tr>
<td>Batteries</td>
<td>0.4 to 32Ω</td>
</tr>
</tbody>
</table>

Table B-1

Note that presently we are only concerned about the effect of the equipment on the magnitude (resistance component) of the impedance. Also note that the devices in the studio are designed to have minimal effect on any circuit they are connected to. In this part of the experiment, we will examine how much of an effect the equipment has.

Voltage dividers: In order to analyze the effect of the equipment, we need to understand a fundamental concept of circuit analysis called a voltage divider. Basically, when a voltage in a circuit is subjected to different resistances, it divides up in a manner proportional to the resistances. That is, a larger resistance will have a larger voltage drop and that voltage drop will be proportional to the size of the resistance divided by the total resistance of a circuit.

![Figure B-1](image)

In Figure B-1 above, Vin is divided between R1 and R2. Mathematically, this can be expressed:
Note that $R_1 + R_2$ is the total resistance of the circuit. We can use a voltage divider to determine how much effect a device has on a circuit, or in this case, the effect that a circuit has on a device. In the simple electrical model of the battery shown in Figure B-2, the internal resistance of the battery depends on the battery size and chemistry. This is a simple model that ignores much of the internal chemistry including changes as the battery is discharged. The default assumption normally is that the voltage output of a battery doesn’t change with the load. We will investigate how this works in an actual circuit.

The output of the battery is measured using the Analog Discovery with and without a load resistor. Remember that $R_{bat}$ represents the internal model of the battery, **you don’t add this resistor to the circuit**. $R_{load}$ represents the load, or combined resistance of whatever circuit you place on the source. Using the voltage divider rule, we know that the voltage drop across the load is given by:

$$V_{measured} = \frac{R_{load}}{R_{bat} + R_{load}} V_{bat}.$$ 

**Series and parallel circuits:** Another fundamental concept we need to understand in order to analyze the circuits we will build is how to mathematically combine resistances. If any number of resistances are connected in series, you simply add them to find the total resistance. If any number of resistances are wired in parallel, the total resistance is the reciprocal of the sum of the reciprocals of all of the resistances. This is summarized in Figure B-3.
Note that the voltage divider rule applies only to series circuits. Any time we use our measuring devices to measure the voltage across a device, as illustrated in Figure B-4, we are combining that device in parallel with the resistance we are measuring. So just connecting the oscilloscope will affect the quantity to be measured. In this case the effective load resistance on the battery is \( R_{\text{total}} \) and it is the parallel combination of the scope impedance (1\( \text{Meg}\Omega \)) with the resistance of the load resistor \( R_{\text{load}} \). This results in total load resistance, \( R_{\text{total}} \).

![Figure B-4.](Image)

Once you have the total load resistance, \( R_T \), you can use the voltage divider rule to find the internal resistance of the battery. Note that, since the voltage drop across any number of resistors in parallel is the same, \( V_{R_{\text{total}}} \) is equal to \( V_{R_{\text{load}}} \).

**Other basic circuit components:** There are two other basic circuit components: capacitors and inductors. To combine capacitors in series take the reciprocal of the sum of the reciprocals. To combine capacitors in parallel, simply add the capacitances. [Note: This is the opposite of combining resistors.] Inductors combine like resistors. To combine inductors in series, you add them. To combine them in parallel, you take the reciprocal of the sum of the reciprocals.

\[
\text{series } \frac{1}{C_n} = \frac{1}{C_1} + \frac{1}{C_2} + \cdots + \frac{1}{C_n} \quad L_T = L_1 + L_2 + \cdots + L_n
\]

\[
\text{parallel } C_T = C_1 + C_2 + \cdots + C_n \quad \frac{1}{L_n} = \frac{1}{L_1} + \frac{1}{L_2} + \cdots + \frac{1}{L_n}
\]

**Experiment**

**B.1) Some DC Measurements**

We will look at what happens when we apply a load to a battery. We will be using batteries extensively in this course, so understanding their basic electrical properties is critical. We will be making DC measurements, like we do with a typical multi-meter. For this section, shut off the scope, go to the main Waveforms window and select Voltmeter from the ‘More Measurements’ drop down menu. When this is enabled, it will use the inputs for the scope channels, but it is better to have the scope off to avoid confusion.

- Measure the voltage of a 9V “Heavy Duty” battery without any load. Simply connect the battery to the protoboard and connect the leads from the protoboard to the \( 1+ \) (Scope Channel 1 Positive (orange)) and \( 1- \) (Scope Channel 1 Negative (orange-white)). Note that, when we make most measurements in this course, they will be single-ended (referenced to ground). Then you only need to touch the \( 1+ \) wire to the point of interest. To do this, **the negative input 1- and GND must be connected.** When we make what are called differential measurements, we use the two wires the way we are here. We will return to that in a future experiment. In this case the load is an open circuit (infinite resistance) because we have added no load to the battery; the input resistance of the Analog Discovery is also so large compared to the range of battery resistance listed above, that it can also be ignored. Record the value of the voltage you measure. (It will also be useful to check your measurement with a multi-meter if you have one. This extra step is not required.)

- Now add a load to the battery, as in Figure B-5. The load is two 100\( \Omega \) resistors in series. We will discuss why two resistors are used a little later. Set up the circuit so that you can add and remove the load quickly, leaving it disconnected unless you are making a measurement. This just means wire it so that it is easy to pull out and
reinstall one end of one resistor. You should only connect the load to the battery for a short moment (a second or two) long enough to make the measurement. If you leave the resistors connected, your battery will drain down quickly and will definitely not last a full semester. Record the voltage displayed in the Voltmeter window of the Analog Discovery, with the resistive load disconnected. Then connect the resistors, quickly record the new voltage, and quickly disconnect the load. You may want to repeat this a few times to find the typical change of voltage with and without the load. Remember to record the unloaded battery voltage as well as the change in voltage. In the figure, A1+ is 1+ and A1- is 1-. The A reminds us this is an analog measurement.

Use the results from this experiment to determine the value of Rbat.

- Repeat the experiment with a different battery from a plastic case the TAs will place on the center table. Choices are a 9V Alkaline, a pack of 2 AA alkaline batteries in series, and a watch battery. If you do a low voltage battery it may be wise to load the battery with only one 100Ω resistor.
- Share data with other teams so that you have numbers for at least 4 battery types.

### B.2) Some AC Measurements

The part above showed that the load can effect the equipment, in this case a battery. Now we will look at how the instrument can affect the circuit. The Analog Discovery oscilloscope, can load the circuit and affect the circuit to be measured. Now you can shut off the Voltmeter and turn on the Scope again.

- Use the function generator, W1, of the Analog Discovery to put an AC signal on a resistor divider circuit shown in Figure B-6. Set the Function Generator to 1kHz and Amplitude to .5 V (since the goal is 1V_p-p). Use R1 = 1kΩ and R2 = 1kΩ. Take data and plot the output using Excel.

Make all the connections on the protoboard. In the circuit above, A1+ is analog input + which is 1+ for Analog Discovery. A1- is 1-, A2+ is 2+, A2- is 2-. GND is Ground. The Function Generator output is W1. Only one ground connection has to be made from the Analog Discovery because the Function Generators (W1 & W2) are connected internally to ground.

- Calculate the ratio of the voltage measured on 2+ to the voltage on 1+.
- Repeat the experiment using R1 = R2 = 1MegΩ resistors. Again create a plot of the voltages and calculate the ratio of the voltages.
- The more exact model of this measurement is given in Figure B-7, were RA1+ and RA2+ represent the effective internal input resistances of the analog input channels of the Analog Discovery. The effective input resistance of A1+ can be ignored (Do you know why?), but the input resistance of A2+ affects the measurement. Using the measurements above, estimate the value of RA2+. 
Figure B-7.

Useful Hints:
- You can copy an image of the plot to Microsoft Word by “Edit – Copy as Image”
- You can save the data in a csv file, which you can open in Excel, using “Export.”
- You can change the thickness of the line segment by: Right click on the plot – select and change the plot thickness.

B.3) Power Calculations and Impedance Matching

Now we will look at the power associated with the battery circuit

- Power Ratings: In part B.1 you used two resistors in series. The effective resistance of resistors in series is simply the sum of the resistances. So why use two 100Ω resistors in series when we could use one 200Ω resistor? Power rating is the answer. \[ P = IV = I^2 R = V^2 / R \] where \( P \) is the power, \( V \), \( I \), and \( R \) are the voltage, current, and resistance of the load. The power is in Watts if you use Volts, Amps and Ohms. Our resistors have a power rating of \( \frac{1}{4} \) watt.
  - Calculate the total power out of the battery for part B.1 for just the 9V battery measurements.
  - Calculate the power per resistor. Ask for help if it isn’t now clear as to why we used 2 resistors rather than one for this measurement.
  - Calculate the total power out of the battery for part B.1 for the 2 AA battery pack.

- Impedance matching: Impedance matching is important with weak signals, not with batteries. Even so, the concept can be demonstrated using our circuits. Don’t wire this circuit; it would cause excessive heating and a rapid discharge of the battery.

For this part assume that you have a 9V battery with an internal resistance of 30Ω. Using Figure B-2, calculate the voltage that would be measured across the load if the load resistance is 100Ω, 60Ω, 30Ω, 20Ω, and 15Ω. For each load resistance, determine the power that would be dissipated in the load resistor. Plot the power dissipated vs. the load resistance.

If you did this correctly, you will see that the maximum power in the load occurs when the load resistance is equal to the internal battery resistance. This is call impedance matching.

Summary

You should now understand how to calculate the effective resistance of resistors in series and/or in parallel. You have an appreciation of AC and DC signals, and that the load and/or the equipment affects the voltages and currents in the circuit. Lastly you should be comfortable with using the Analog Discovery, including the function generator, the oscilloscope and the voltmeter.
Part C – Introduction to Capture/PSpice

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. The download is located at with the other information on Experiment One the Links by Experiment page. 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 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. Capture is a windows program that provides a visual interface that lets you enter circuits. It translates your diagrams into information that PSpice can understand. Figure C-1 shows a simple circuit created with Capture with some useful buttons defined. PSpice takes circuit information and analyzes how it will behave. It displays an output similar to what you would see if you hooked the circuit to an oscilloscope. Figure C-2 contains a sample output. For simplicity, we will generally refer to both programs as PSpice.

![Capture/PSpice Interface]

Figure C-1.
Opening a New Project in Capture

In this part of the experiment, we will use Capture to draw the simple circuit we have been studying, a combination of resistors and a sinusoidal voltage source.

- Run the “OrCAD Capture CIS Demo” program found under Cadence in the start menu.

- Capture will open with no current project. Click on the File pull-down menu and select New Project. You will see a new window (Figure C-3) named New Project.
Be sure that the Analog or Mixed-Signal A/D is selected. [ALWAYS select this option since this tells Capture that you wish to do a PSpice simulation.] You will use this window to give this project a name and choose a location where you wish to store it. In the box at the top of this window, give your project a recognizable name, such as EXP1-C. For a location, it is recommended that you create a directory in which you store all of the files you will generate in this course from Capture/PSpice, Analog Discovery, Word, Excel, etc. Capture and PSpice create around 20 files per project, so a folder for each project is a good idea. Once you have finished setting up the project, click on the OK button.

- Next you will get a pop-up window asking if you wish to use an existing project. Choose create a blank project and click OK.
- Now you should see the main Capture screen. You are ready to draw a circuit. If you don’t see the Capture screen but see a file tree structure then: click on the file with the .dsn, click on SCHEMATIC1, and double click on PAGE1. This will be the Capture screen.

Drawing a Circuit
Figure C-4 is a picture of the Capture main screen with the circuit we will be drawing. Note that this is the circuit used in Figure B-7 including the input resistance of one channel of the Analog Discovery.

![Figure C-4](image_url)

In the circuit shown in Figure C-4, we have some resistors, an AC 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
button that looks like an integrated circuit chip or . You can also do this by using the Place menu or by hitting shift-P. You should bring up a screen similar to the one pictured in Figure C-5.

![Figure C-5.](image)

• For the circuit we are analyzing, the components and their PSpice names are:

<table>
<thead>
<tr>
<th>Components Name</th>
<th>Resistor</th>
<th>Sinusoidal Voltage Source</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>R</td>
<td>VSIN</td>
</tr>
</tbody>
</table>

• If you do not see any words in the Libraries 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. You should add every library but you will need at least these two.

• To get a resistor, click on the Place Part button and then type an R in the space at the upper left marked Part. Then click on OK and you will be back to the main window.

• You 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 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 it 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.

Finally, all circuits need a ground reference so we know where the voltage is equal to zero. Click on the Place Ground button, or . You want to choose the 0/SOURCE ground from the Place Ground window shown in Figure C-6, since it is the only one that works with PSpice. If the 0 ground is not listed, you should use Add Library to 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 Place Wire button: or . 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.

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 R3 to 10Meg Ohms, double click on the value 1k and you should get the window shown in Figure C-7 with the name Display Properties. Change the 1k to 10MEG. 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. When you type the value in the Display Properties window, you must type 10MEG 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 and R2 are the resistors in your voltage divider and R3 represents the impedance of the scope. Since the scope impedance is 10M, R3 should be 10MEG. R1 and R2 are already 1k Ohms, you don’t need to change them from their default values.
After you have changed all the resistor values and moved them to readable positions, you must set up the voltage source. You can set the values of each of the voltage parameters next to the source by double clicking on them. Set the value of VAMPL (voltage amplitude) to 100m. Set the value of VOFF (voltage offset) to 0. Set the value of FREQ (frequency) to 1k. Note again that there is no space between the number and the m or k. 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 in the top toolbar menu and click on it. You will get the window shown in Figure C-8, 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.

Now we can set up the simulation. 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 shown in Figure C-9.
There are three boxes in which times need to be indicated. Usually two of the three values in the text boxes need to be set. The final time (Run to time:) has been set at 3ms because the period of a 1kHz sine wave is 1msec. This allows us to see three periods. The Maximum Step Size 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.

- 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.

- Before we run the simulation, we need to indicate on the circuit 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 10MEG 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.

**Transient Analysis**
You are now ready to do the simulation.

- Click on the Run button. It is possible to set up many kinds of analyses 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 Figure C-10, 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 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.

Figure C-10.

- You should get something like (but maybe not identical to) the window shown above.
- You can print this plot directly. However, it is also useful to know how to copy plots and paste them into word. Under the Window menu in PSpice, click on “copy to clipboard”. This will bring up a window. Choose “change all colors to black” or “change white to black” and click OK. 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.

Note that the sine wave lines on the plot are a bit thin and hard to read. You can change the data display on the plots generated by PSpice to make them easier to read. This is worth doing since it makes reports much easier to work with. Go back to the PSpice data display (Figure C-10) and right click on the symbol for one of the traces.
(it will be a diamond or square and will be the color of the trace). A menu will appear. Select *Trace Property*. This will allow you to change the color, pattern, thickness, symbol … of the trace.

![Trace Properties dialog box](image)

Select width and make the trace somewhat wider. When you copy the data onto the clipboard, you will see a thicker line, as shown below. Only the green trace has been changed.

![Graph showing changes in trace properties](image)

- 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? 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?

**Summary**

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?
Checklist and Conclusions

Provide the following packet. Include the cover/signature page attached to the end of this handout. The signatures are required for all statements with a signature line next to it. They must be signed by TAs or Professor(s) after seeing the results on the computer screen while the experiment or simulation is running. Give all required results and answer the questions concisely. It is intended that these be quick to write.

The following should be included in your experimental checklist. Everything should be labeled and easy to find. Partial credit will be deducted for poor labeling or unclear presentation. ALL PLOTS SHOULD INDICATE WHICH TRACE CORRESPONDS TO THE SIGNAL AT WHICH POINT and the key information contained in the plot should be labeled so the reader can fully understand the data without referring to the report text.

Part A: Plots for Sine Waves and Hearing (14 points)

Part A1: Setting up a Sine Wave on the Function Generator
1. Printed output plot of signal measured by the scope with a peak-to-peak amplitude of 400mV (Amplitude of 200mV) (TA MUST see this live on your computer screen to sign checklist) (5 pt)
2. On the plot, mark the period and amplitude and denote the calculated frequency (1 pt)
3. Briefly comment on any differences between the FG settings, the Measurement window results, and the results from the oscilloscope plot (1 pt)

Part A2: Using the Audio Output from Analog Discovery
1. Write down the measured resistance of each channel of your ear buds. (1 pt)
2. When listening to the audio, what does the scope measure for the peak-to-peak amplitude when the speaker is producing a comfortable level of sound? (3 pt)
3. What is the period of the tone at the frequency that appears loudest when you scanned through the entire range of frequencies? 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. (3 pt)

Part B: Voltage Dividers and Measuring Equipment (48 points)

Part B1: DC Measurements
1. Create a table of data for all four battery types. Remember that you only need to measure 2, and then collect data from other groups to complete the table. The table must have a) battery type, b) unloaded battery voltage, c) loaded battery voltage, d) the total load resistance of the test, and e) the calculated value of Rbat. Show the formula you used to calculate the value of Rbat. (20 pt)
2. Find one reference that states the expected internal resistance of one of the batteries used. In most cases we assume you will find one using a web search. Battery company web sites might be a place to look. Give the source and the value stated. Compare it to the measured value. (4 pt)

Part B2: AC Measurements
1. Printed Analog Discovery scope plot or the plot copied to a Word doc of input and output from the 1k voltage divider. (TA MUST see this on your computer screen to sign checklist) (4 pt)
2. Printed Analog Discovery scope plot or the plot copied to a Word doc of input and output from the 1Meg voltage divider. (4 pt)
3. Calculate the value of RA2+. (4 pt) Show your work. (4 pt)

Part B3: Power Calculations and Impedance Matching
1. List the power out of the 9V battery and the power per resistor. Also state the power out of the AA battery pack. (4 pt)
2. Plot the predicted power into the load vs. Rload for a 9V battery assuming the battery has an internal resistance of 30Ω. Note: There should be a maximum power (matched impedance) (4 pt)
Part C: Introduction to Capture/PSpice (14 points)

Part C1: AC Measurements

1. Printed PSpice transient plot of voltage divider with 2 1k resistors (two traces). (4 pt)
2. Printed PSpice transient plot of voltage divider with 2 1MEG resistors (two traces). (TA MUST see this on your computer screen to sign checklist) (4 pt)
1. How does the signal you generated with PSpice using the 1k voltage divider compare to the one you generated with the Analog Discovery scope? (3 pt)
2. How does the signal you generated with PSpice using the 1Meg voltage divider compare to the one you generated with the Analog Discovery scope? (3 pt)

Responsibilities (4 points)

List group member responsibilities. (4 pt) Note that this is a list of responsibilities, not a list of what each partner did. It is very important that you divide the responsibility for each aspect of the experiment so that it is clear who will make sure that it is completed. Responsibilities include, but are not limited to, reading the full write up before the first class; collecting all information and writing the report; building circuits and collecting data (i.e. doing the experiment); setting up and running the simulations; comparing the theory, experiment and simulation to develop the practical model of whatever system is being addressed, etc.

Total: 80 points for experiment packet
+20 points for attendance
100 points

Attendance (20 possible points)

2 classes (20 points), 1 class (10 points), 0 class (0 points)
Minus 5 points for each late.
No attendance at all = No grade for this experiment.

Notes:

There will be an additional task in all subsequent experiments to identify practical examples of the content from the experiment that one encounters as a consumer, an engineer, a physician, etc. Since this is the first experiment, it is not necessary to address this issue. However, one of the key circuit configurations we will see over and over and over (could really go on forever here) is the voltage divider. You should be on the lookout for it in every experiment and project because almost all circuits involve some combination of components in series. Make sure that you fully understand the voltage divider in the context of the engineering design process as represented graphically by the diagram. That is, what is the basic theory (background info)? How do you set a divider up to analyze it through simulation and experiment? What are the practical issues associated with it? For the last question, you should be able to explain, from the results of this experiment, how adding a load to a voltage divider can significantly change its performance. We will revisit this issue many times, but especially in the experiment on Op-Amps.

Engineering Design Process
Experiment 1
Electronic Instrumentation

Section: _____
Report Grade: _____

__________________________ Name
__________________________ Name

Checklist w/ Signatures for Main Concepts

PART A: Plots for Sine Waves and Hearing
A1: Setting up a Sine Wave on the Function Generator
   1. Signal Vp-p 400 mV
   2. Mark plot
   3. Comment
A2: Using the Audio Output from Analog Discovery
   1. Measured resistance ear buds
   2. Comfortable sound peak to peak value
   3. Loudest frequency: value of period

PART B: Voltage Dividers and Measuring Equipment
B1: DC Measurements
   1. Battery table
   2. Internal resistance reference
B2: AC Measurements
   1. 1k voltage divider plot
   2. 1 Meg voltage divider plot
   3. Calculated RA2+
B3: Power Calculations and Impedance Matching
   1. Power of batteries
   2. Predicted power plot 9V battery

PART C: Introduction to Capture/PSpice
C1: AC Measurements
   1. PSpice transient 2 1k resistors
   2. PSpice transient 2 1Meg resistors
   3. Comment comparison scope/ PSpice 1k
   4. Comment comparison scope/ PSpice 1 Meg

Group Responsibilities