

**Experiment 1  
Signals, Instrumentation, Basic Circuits 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

- **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)
- Brand X function generator
- One 50 ohm resistor, two 1 M ohm resistors and two 1 k ohm resistors.
- Protoboard
- Audio speaker

Helpful links for this experiment can be found on the links page for this course:

<http://hibp.ecse.rpi.edu/~connor/education/EILinks.html#Exp1>

**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 an oscilloscope. The electrical signals will then be applied to an oscilloscope and a speaker, that will allow us to see, hear and analyze the signals. We will also demonstrate some interesting facts about human hearing and speech.

**Background**

*Equipment:* Each station in the studio has seven major pieces of equipment: an oscilloscope (scope), a digital multimeter (DMM), two DC power supplies, a function generator, a counter, and a PC. The DMM is a measuring device that lets you take measurements of voltage, current, or resistance. The scope is a measuring device that lets you view a plot of a voltage signal. The DC power supplies generate constant DC voltage signals (like a battery). The function generator creates a voltage signal that varies with time. The counter is a digital device that we do not use in this class. The PC is an integral part of the equipment setup. Not only will you use it to simulate many of the circuits you will build, but it is also connected directly to the scope. You will use it in this class to take pictures of and data from voltage signals you generate with your circuits.

In this experiment we will use the function generator, the oscilloscope, and a 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 that 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 Sine Wave Equation:* Many of us 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-1, 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.

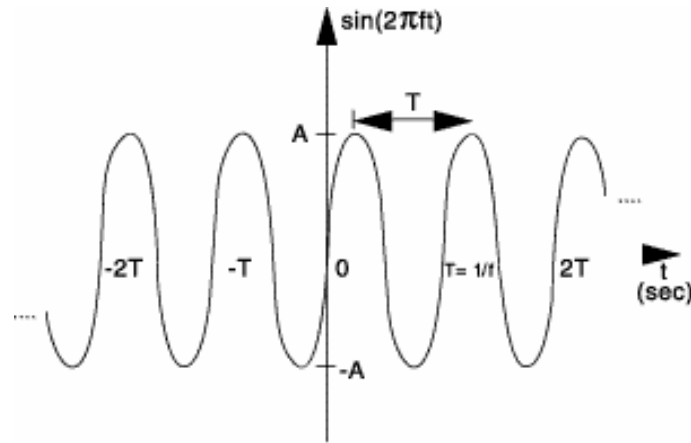


Figure 1: Sine wave with amplitude  $A$ , frequency  $f$ , and period  $T$ .  
Figure A-1

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 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 to confuse  $f$  and  $\omega$ .

*Adding a DC offset:* If we add a DC offset voltage to the sine wave signal, as shown in figure A-2, it moves the wave such that the center is 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.

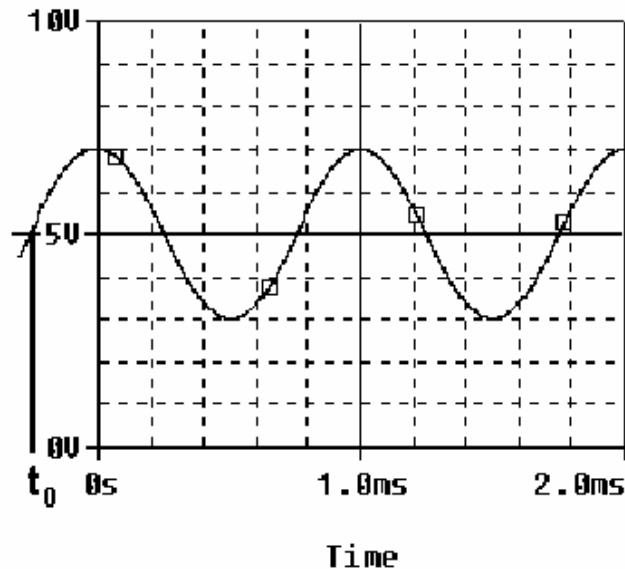


Figure A-2

# Electronic Instrumentation

## Spring 2006

ENGR-4300

Experiment 1

*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,  $V_{RMS} = A/1.414$ . Note that in electronics the RMS voltage does not take into account the DC offset even when there is one on the signal, so the RMS voltage will always depend only on the amplitude.

*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 resist the flow of electrons. However, the resistive behavior of these circuits is more complicated because it varies with the frequency of the signal. We call this complicated type of resistance "*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

*Setting up a sine wave on the function generator*

For the first experiment we wish to do here, we need set up a sinusoidal voltage.

- First we will set the frequency. Set up the function generator and then connect it directly to the oscilloscope on channel 1 using a standard BNC-BNC coaxial cable. The frequency of the function generator is adjusted as follows:
  - Push the FREQ button on the function generator.
  - If you cannot see a signal on the scope, hit the AUTOSCALE button. This will locate the signal.
  - With the knob, change the frequency up or down as desired. How does this change the signal on the scope?
  - Set it so the display reads 1K Hz.
- Now we can set the output voltage.
  - Push the AMPL button on the function generator.
  - If you cannot see a signal on the scope, hit the AUTOSCALE button. This will locate the signal.
  - Move the  $\triangleleft$  arrows keys of the function generator until the hundreds digit is blinking.
  - With the knob, change the voltage up or down as desired. How does this change the signal on the scope?
  - Set it so the display reads 200 mVp-p.
- What is the frequency of your signal? One way to measure the frequency of the sine wave is to use the grid on the display of the oscilloscope. The scale for frequency appears at the upper right of the screen. Use the scale to estimate the period of your wave. The frequency is the inverse of the period. Fortunately, our scopes provide us with an easier way to measure frequency.
  - Push the Time button on the main panel of the oscilloscope.
  - Push the "Freq" soft key on the bottom of the oscilloscope display.Is the frequency around 1000 Hz? (Note that the oscilloscope re-computes the frequency 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 1K Hz.)
- What is the amplitude of your signal? One way to measure the amplitude of the sine wave is to use the grid on the display of the oscilloscope. The scale for voltage appears in the upper left hand corner of the screen. Use

the scale to estimate the peak-to-peak voltage of your wave. You can use the scope to measure the voltage, as well.

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

Did you notice something wrong? Even though the function generator display says the peak-to-peak voltage is 200mV, the scope says it is about 400mV. What is happening? This discrepancy is the result of something called impedance matching. We will look at this in detail in the next section of the experiment. For now, it is sufficient to remember that the function generator display is not a reliable way to set voltage levels.

**IMPORTANT:** *If you want a signal with specific properties, you should always set it by looking at the scope.*

#### *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.

- Start by measuring the impedance of your speaker using the DMM. (A speaker is not a simple resistive device, so it has an impedance. We will use the same feature on the DMM to measure this impedance as we would to measure resistance, since both are measured in ohms.)
  - Connect the red lead (HI) from the DMM to one end of the speaker and the black lead (LO) to the other. A good way to do this is to connect the dual banana to BNC connector to the DMM. The dual banana connector has a polarity. The tab should point toward the negative connection (LO). (ALWAYS plug into the DMM using the connections furthest to the right.) Check to make sure you plug into the connectors designed to measure  $\Omega$ .
  - Plug a BNC cable into the dual-banana connector and a mini-grabber into the BNC cable. You can then connect the leads of the mini-grabber to the speaker and record the impedance.
- We need to hook up the speaker, but we want to do it in such a way that we can still see the input signal on the scope.
  - Disconnect the BNC cable from the function generator, but leave it connected to the scope.
  - Place a BNC tee connector on the output of the function generator and then reconnect the cable to the tee.
  - Connect a second BNC cable to the other side of the tee. Place a mini-grabber adapter on the other end of the second cable.
  - Connect the mini-grabber to the speaker terminals.

You will notice that the signal observed on the scope has changed, but the function generator is still happily displaying 200mV peak-to-peak. This is caused by the impedance matching problem again. The signal has changed because you added the speaker to the circuit. The scope is still displaying the correct signal, but it is now a different signal because you have changed the load circuit. The "load" is a term for the combined impedance of everything in your circuit. This determines how hard the source device (DC power supply, function generator, battery) has to work to supply your circuit with current.

- 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 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? 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. Try some of the other functions 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 set up 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?

**Summary**

You should now know how to set up voltage signals with the function generator and display them using the oscilloscope. You should understand the pitch/frequency and amplitude/volume relationships, and know how these relate to human hearing. You have also uncovered a problem with the display on the function generator. We will explore why this happens in the next section.

**Part B – Voltage Dividers and Measuring Equipment**

In this part of the experiment you will learn how to handle the discrepancy between the display on the function generator and the actual voltage displayed on the scope. 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 like 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 device in the studio has an internal impedance. This is generally written somewhere on the equipment. The impedances we will concern ourselves with in this class are listed in table B-1 below:

<b>Device</b>	<b>impedance (magnitude only)</b>
Scope	1 Meg $\Omega$
DMM (DC voltage)	10 Meg $\Omega$
DMM (AC voltage)	1 Meg $\Omega$
DMM (current)	negligible
DMM (resistance)	negligible
function generator	50 $\Omega$
DC power supplies	negligible

Table B-1

Note that 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.

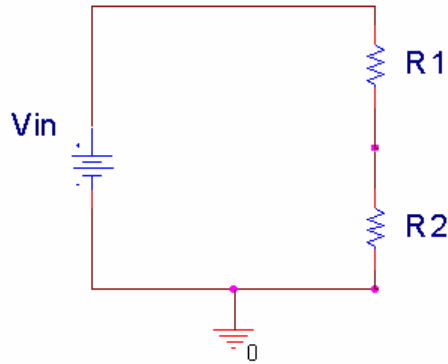


Figure B-1

In figure B-1 above,  $V_{in}$  is divided between  $R_1$  and  $R_2$ . Mathematically, this can be expressed:

$$V_{in} = V_{R_1} + V_{R_2} \quad V_{R_1} = \frac{R_1}{R_1 + R_2} V_{in} \quad V_{R_2} = \frac{R_2}{R_1 + R_2} V_{in}$$

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. In the simple electrical model of the function generator shown in figure B-2, the internal impedance of the function generator is 50 ohms, and the voltage that the function generator puts out internally is  $V_{gen}$ . The default assumption is that the resistive load driven by this source is also 50 ohms. Matching the internal impedance of a source to the impedance of the load gives maximum power transfer. We will investigate how this works in this part of the experiment.

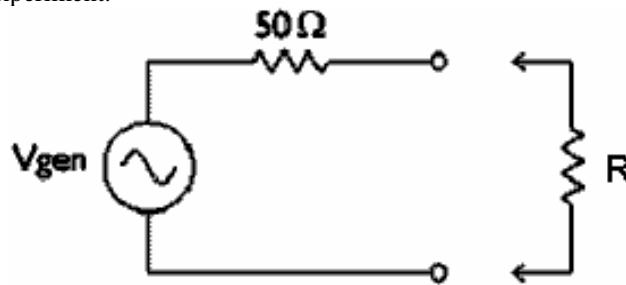


Figure B-2

The function generator (with its internal impedance of 50 Ω) is wired in series with a resistor,  $R$ .  $R$  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,  $R$ , is given by:  $V_R = \frac{R}{50 + R} V_{gen}$ .

*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.

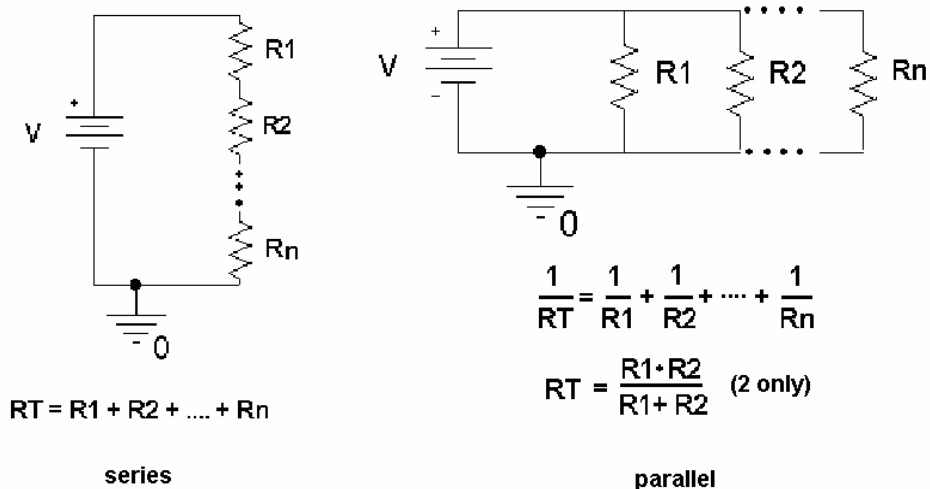


Figure B-3

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 measuring the resistance of a single resistor attached to the function generator would involve combining the scope impedance (1Meg  $\Omega$ ) with the resistance of the resistor ( $R$ ) to find the total load resistance,  $R_T$ .

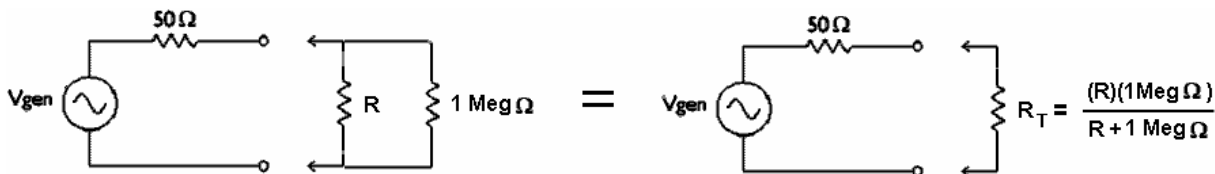


Figure B-4

Once you have the total load resistance,  $R_T$ , you can use the voltage divider rule to find the voltage drop across the load. Note that, since the voltage drop across any number of resistors in parallel is the same,  $V_{RT}$  is equal to  $V_R$ .

*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.

$$\begin{array}{ll}
 \text{series} & \frac{1}{C_n} = \frac{1}{C_1} + \frac{1}{C_2} + \dots + \frac{1}{C_n} & L_T = L_1 + L_2 + \dots + L_n \\
 \text{parallel} & C_T = C_1 + C_2 + \dots + C_n & \frac{1}{L_n} = \frac{1}{L_1} + \frac{1}{L_2} + \dots + \frac{1}{L_n}
 \end{array}$$

Experiment

*Some DC Measurements*

We will look at what happens when we apply a DC voltage to a simple voltage divider and measure the voltage across each of the two resistors.

- Measure the actual resistance value of the two 1K ohm resistors using the DMM. You will need to attach one pole (red is positive and black is negative) of the DMM to each lead of the resistor and press the  $\Omega$  button on the DMM. Record the two values using all the significant digits displayed on the DMM. (Note that when measuring resistance, the effect of the DMM is negligible, so we can assume these values are accurate.)
- Set one of the DC supplies to +10 Volts. Measure the voltage using the DMM. You will need to connect like poles (red and black) of the two devices together and hit the DC V button on 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 the DC power supply to the two 1 k ohm resistors connected in series, as in figure B-5:

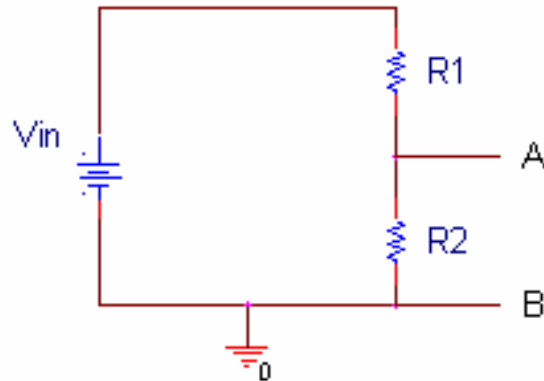


Figure B-5

This circuit is made by connecting the red (+) pole of the power supply to one end of one resistor, connecting the two resistors together end to end, and connecting the black (-) pole of the power supply to the remaining free end of the resistor pair. Do not make these connections by winding the resistor leads together. You will not get good results with bad connections. Use the alligator clips at your station or a protoboard.

- Calculate the voltage you would expect to measure across each resistor when no measuring device is connected to the circuit. (This is the theoretical voltage.) After calculating this voltage, measure the voltage across each resistor. Remember that your circuit now contains the impedance of the DMM (10 Meg ohms), as shown in figure B-6.

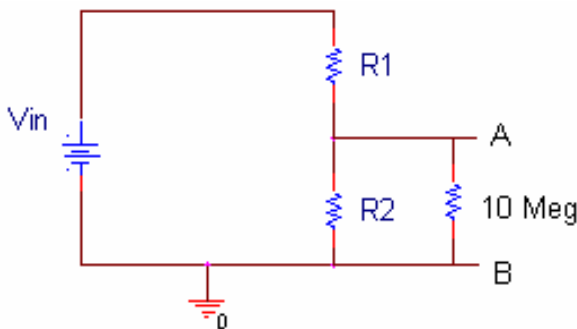


Figure B-6



## Electronic Instrumentation

ENGR-4300

Spring 2006

Experiment 1

In this circuit the resistor closest to ground is being measured. (Note that the internal impedance of the DC source is negligible, so we do not need to include it in our circuit). Can you explain the difference between the theoretical values for the voltages and those you actually measured by analyzing the circuit above?

- Repeat the above three steps with the two 1 Meg ohm resistors. You should see a larger discrepancy between the predicted and measured voltage when the 1 Meg ohm resistors are used. You should be able to see why by analyzing the circuit mathematically.

### *Impedance Matching*

Now we will look at the function generator and impedance matching.

- We will first set up the function generator. Connect a tee to the output of the function generator and connect one end of the tee to the scope with a BNC cable. Hit AUTOSCALE.
- Now we need to set up a signal on the function generator. We want this signal to have a frequency of 1K Hz and an amplitude of 100 mV. Hit the VAMPL button on the function generator. Set the function generator display to 200mV peak-to-peak. (In an ideal world, this should give you the 100mV amplitude that you want, but we know from part A that this will probably not be the case.) Hit the FREQ button on the function generator. Set the frequency to 1K Hz. Your circuit currently looks like figure B-2, with R equal to the 1Meg ohm impedance of the scope:
- What is the frequency of your signal? What is the amplitude of your signal? If you believe the function generator display, your signal should have an amplitude of 100mV (and a peak-to-peak voltage of 200mV). Does it? You can verify that the peak-to-peak voltage is *incorrect* by using the Voltage button and the "V<sub>P-P</sub>" soft key. Your scope should display a peak-to-peak voltage of around 400mV. We know from the previous section that this is caused by impedance matching. What is impedance matching?

As you observed when you connected your speaker, the output of the function generator changes depending upon what load is placed on it. Since the device is not clairvoyant, it cannot know ahead of time what load you are going to use. It picks its favorite load impedance and displays what its output would be if you put on a circuit with that particular load impedance. All function generators choose their own internal impedance as their favorite impedance because this choice maximizes power to the circuit. This is called impedance matching. Since the internal impedance of our function generator is 50 ohms, *its display will only read correctly when you attach a circuit with a 50 ohm impedance to it.*

The circuit you currently have connected (the scope) has an impedance of 1Meg ohms, so the function generator is confused. It is putting out the number that is correct for the situation when you have placed a 50 ohm load on it, but you have placed on a much larger load of 1 Meg ohms. You have not placed the function generator's favorite load on the circuit, so it won't display the correct voltage at the output.

- Let's look at a circuit that has the correct load impedance. Take a resistor of about 50 ohms (47 or 51) and hook it to the function generator. You don't need to remove the scope connection, simply connect a BNC cable with a mini-grabber connector between the other end of the tee and a 50 ohm resistor. Your circuit now looks like figure B-4.

Calculate the value of the combined resistance of the scope and the 50 ohm resistance,  $R_T$ . You should find that  $R_T$  is approximately 50 ohms. Now look at your peak-to-peak voltage on the scope. Is it now about equal to the peak-to-peak voltage displayed on the function generator? You should now observe an amplitude of about 100mV (200 mV peak-to-peak). Designing a circuit with 50 ohm impedance is known as impedance matching. You have designed your circuit to *match* the 50 ohm internal impedance of the function generator.

What is going on inside the function generator? In order to get the desired output (200mV peak-to-peak) with a 50 ohm matched impedance, the function generator must set its internal output voltage ( $V_{gen}$ ) to 400mV peak-to-peak. Using a voltage divider, we can see that the voltage measured at the output, point A, will be half of  $V_{gen}$  ( $V_A = 50/100 * V_{gen}$ ). You cannot read the voltage  $V_{gen}$  directly. (It is inside the function generator.) You

can only read the voltage at point A (after the 50 ohm impedance inside the function generator). The function generator assumes you have placed that 50 ohm resistance on it and displays only half of the  $V_{gen}$  that it puts out internally. In other words, the function generator is only guessing what the output at point A is. It will never be absolutely correct unless the impedance of your circuit matches the 50 ohm impedance of the function generator.

Does that mean that every circuit we design has to have a 50 ohm impedance? No. We don't use impedance matching in this course. What we do instead is ignore the display on the function generator and use the scope to set our signals to the desired voltages.

- Remove the 50 ohm resistor from the circuit. The peak-to-peak voltage should jump back up to 400mV on the scope. Now adjust the function generator so that the peak-to-peak voltage *on the scope* reads 200mV peak-to-peak. The function generator display is incorrect (It should read about 100mV peak-to-peak.), but the signal we are using is correct. We know this because we can see it on the scope. Note: Most of the circuits that we build have very high impedance, so most of the time, the display on the function generator will read about  $\frac{1}{2}$  of the desired output voltage. Be careful to think about when you can and cannot use this shortcut.

#### Some AC Measurements

We will look at what happens when we apply an AC voltage to a simple voltage divider and measure the voltage across one of the two resistors.

- Set the function generator so that it produces an output signal with an amplitude of 100mV and a frequency of 1K Hz. Make sure it is correct by verifying it on the scope. Now we can apply this signal to our voltage divider. Take your two 1K ohm resistors and connect the voltage divider to the function generator at the tee. Use a minigrabber to connect the resistor closest to ground to the scope on channel 2. The black lead should be connected to ground and the red lead to the voltage you are reading. Your circuit now looks like figure B-7.

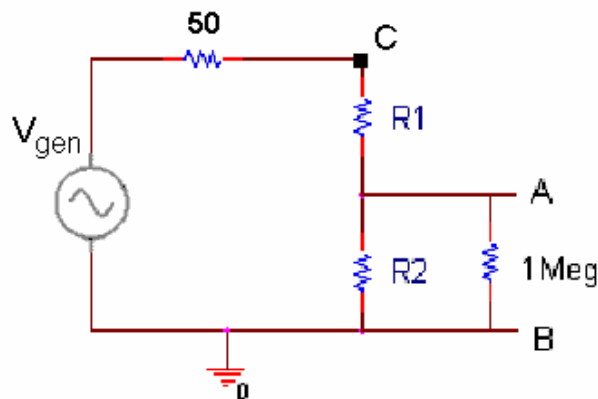


Figure B-7

The scope should have two signals, the input voltage (between C and B) and the output voltage (between A and B). What is the relationship between the two amplitudes? Is it what you would expect? Analyze the circuit and see.

- Now we will add a DC offset to our signal and see how the voltage divider behaves. Press the Offset button on the function generator and turn the dial until the display reads 50mV. Your input sine wave should now be centered around 100mV. (Recall that the function generator is only guessing the voltage and is displaying half of  $V_{gen}$ . Yet, because of the high impedance of your circuit, you are seeing  $V_{gen}$  at the output.) What has happened to your output wave? Is the amplitude the same? What is the DC offset? Is this as expected?
- We will now take a picture of the signals you have created. There is a special toolbar installed in Word and Excel on the PCs in the studio that allows us to do this. Run the version of Word located in the Applications folder on the desktop of the PC. The toolbar for the scope should appear on the screen. Press the camera button

## Electronic Instrumentation Spring 2006

ENGR-4300

Experiment 1

and place an image of the scope screen directly into Word. Save this picture as you will need to include it your report.

- Repeat the previous three steps for the 1 Meg ohm resistors. Don't forget to take a picture of the resulting output. Is this the output you would expect? Has the measuring device had an effect on the circuit again?
- In the final step of this experiment, we will use the DMM to set up a sine wave. This time we will try and set the amplitude to 1V peak-to-peak. Disconnect the voltage divider function generator (you will not need the circuit for this part of the experiment). Connect one end of the tee from the function generator directly to the DMM. Set the function generator to 1k Hz. Calculate the value of  $V_{RMS}$  for a 1V<sub>P-P</sub> 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<sub>P-P</sub> sine wave. Once you have adjusted VAC on the DMM to  $V_{RMS}$ , check the scope to see if the signal is correct. Press the VDC button on the DMM. What happens when you try to measure the DC voltage of the signal from the function generator with the DMM? Why?

### Summary

You should now understand what impedance matching is and how to set up a specific signal correctly using the function generator and the scope or the function generator and the DMM. You should also understand what a voltage divider is, how it works in theory, and how to use it to predict how much adding measuring equipment to a circuit modifies the signal. You should now realize the importance of choosing components in a range that minimizes the effect of the equipment on the circuit.

### Part C – Introduction to *Capture/PSpice*

In this section we will learn about the circuit analysis software we have installed on our computers in the studio. You can download this software to your laptop. A link to the download is located on the links page.

### 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 pictures into information that *PSpice* can understand. Figure C-1 shows a simple circuit created with *Capture* with some useful buttons defined. *PSpice* is an older piece of software that takes circuit information and analyzes how it will behave. It has been modernized so that it displays an output similar to what you would see if you hooked the circuit to a real scope. Figure C-2 contains a sample output.

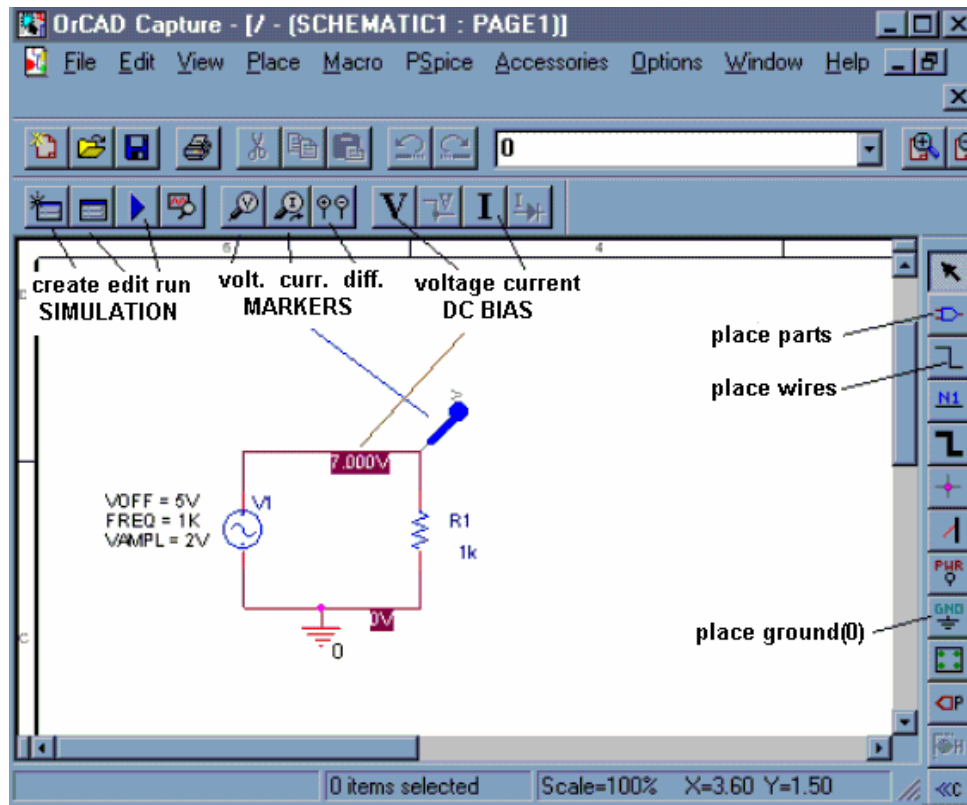


Figure C-1

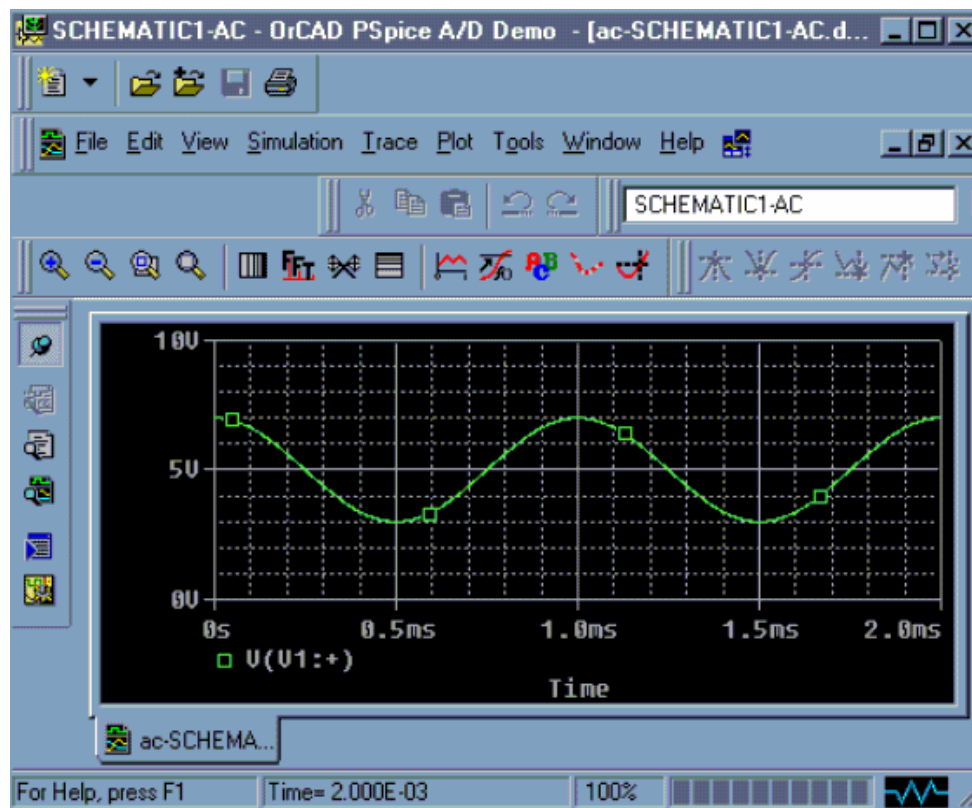


Figure C-2

## Experiment

### *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 “*Capture Student*” program located in the applications folder or on the start menu under “PSpice Student”.
- *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*.

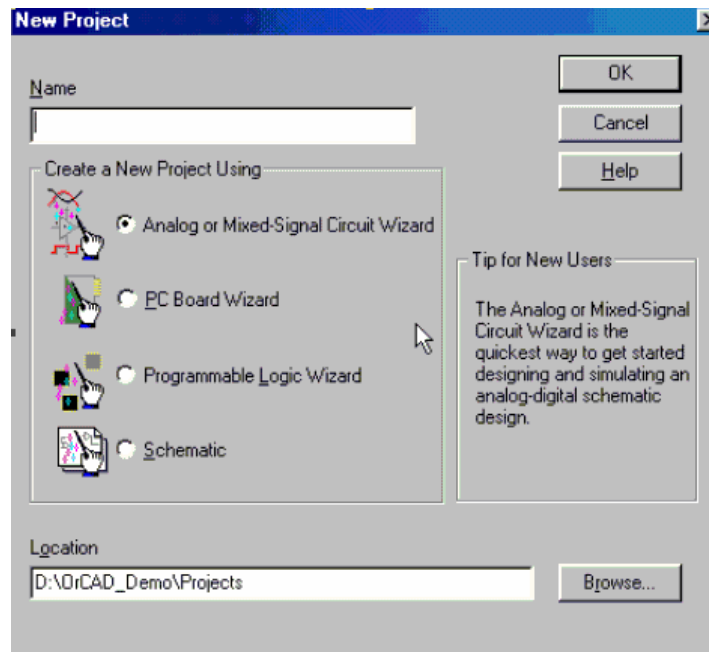


Figure C-3

Be sure that the *Analog or Mixed-Signal A/D* is selected. [ALWAYS select this option.] 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 *EXPI-C*. For a location, it is recommended that you create a temporary folder on the Desktop (or the C drive) for your projects. *Capture* and *PSpice* create around 20 files per project, so a folder for each project is a good idea. When the class is over, you should upload any new or modified projects to your RCS account. The computers in the studio do not save the contents of the desktop when you sign off. An image of a clean C drive is also periodically reloaded. There is a good chance that all your files will be erased before the next class. **WARNING:** If you try to run *Capture* using files already stored on the server, it will be agonizingly slow. 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. Always choose create a blank project and click *OK*.
- Now you should see the main *Capture* screen. You are ready to draw a circuit.

*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 our voltage divider connected to the function generator (complete with impedance). The 1Meg ohm resistor is the scope measuring the voltage over the second resistor.

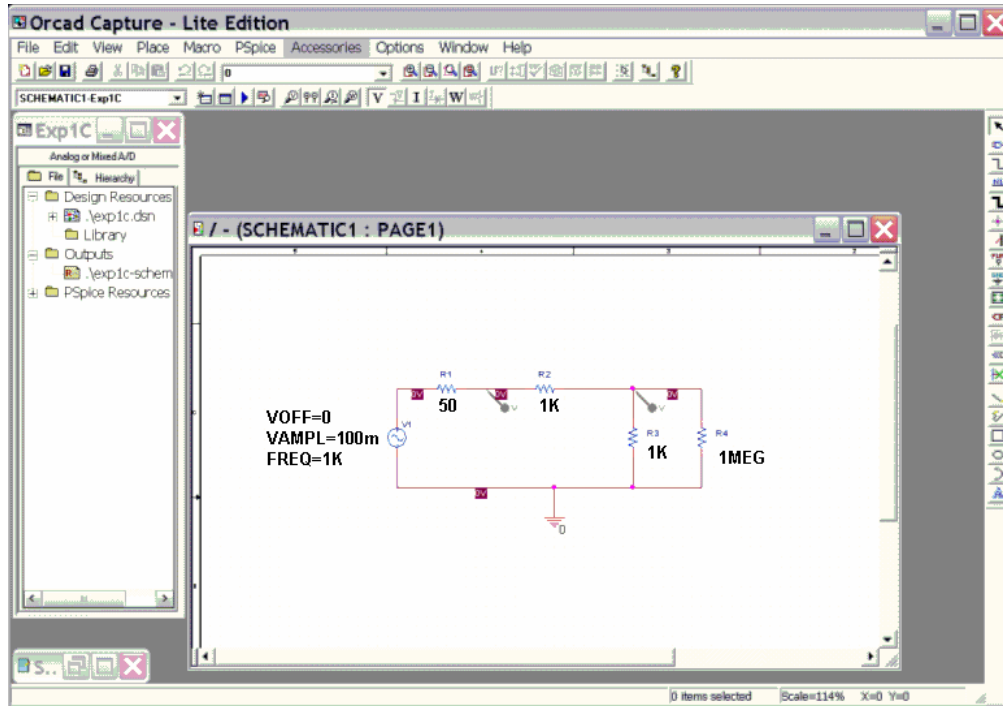



Figure C-4

In the circuit shown in figure C-4, 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 . 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.

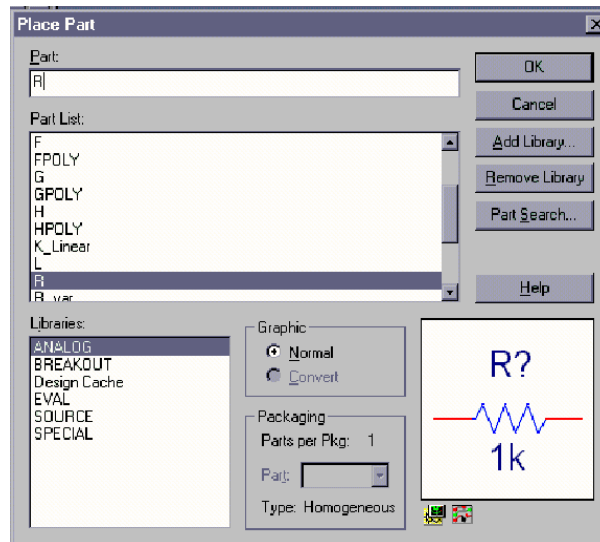



Figure C-5

- The components and their *PSpice* names are:

Components	Resistor	Sinusoidal Voltage Source
Name	R	VSIN

- 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.
- 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 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, that is about half way down on the right . 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.

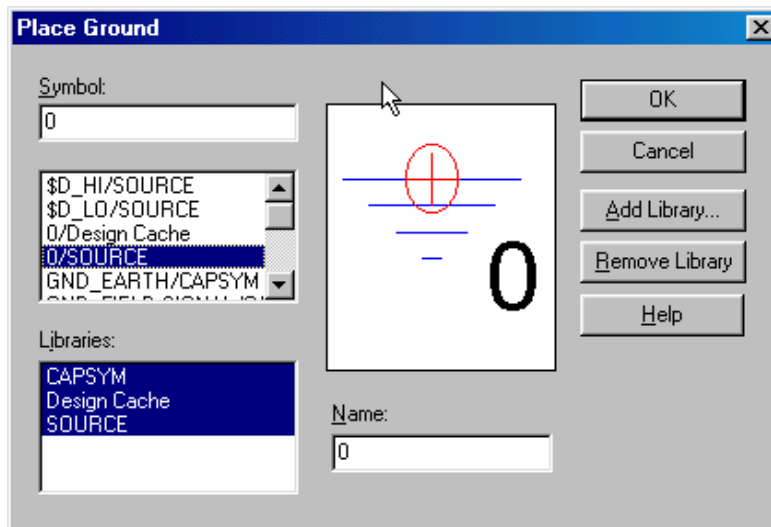




Figure C-6

- 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 . 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 the window shown in figure C-7 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 are 1k ohms, you don't need to set them since they are already 1K by default.

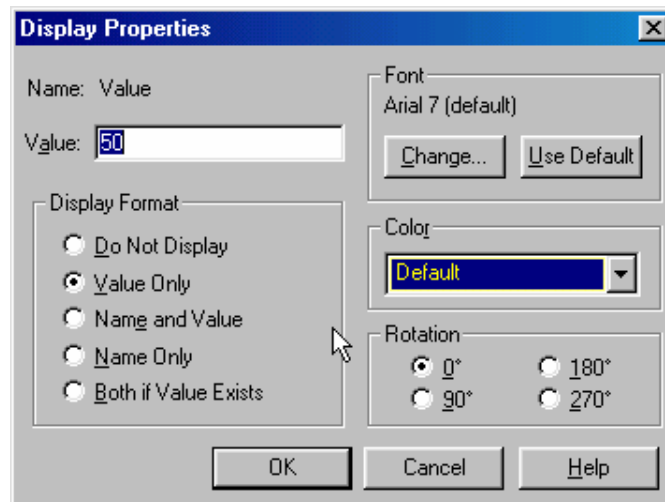



Figure C-7

- 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.

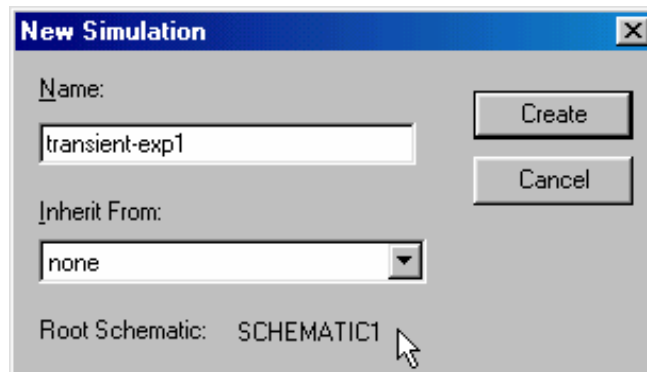


Figure C-8

- 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.

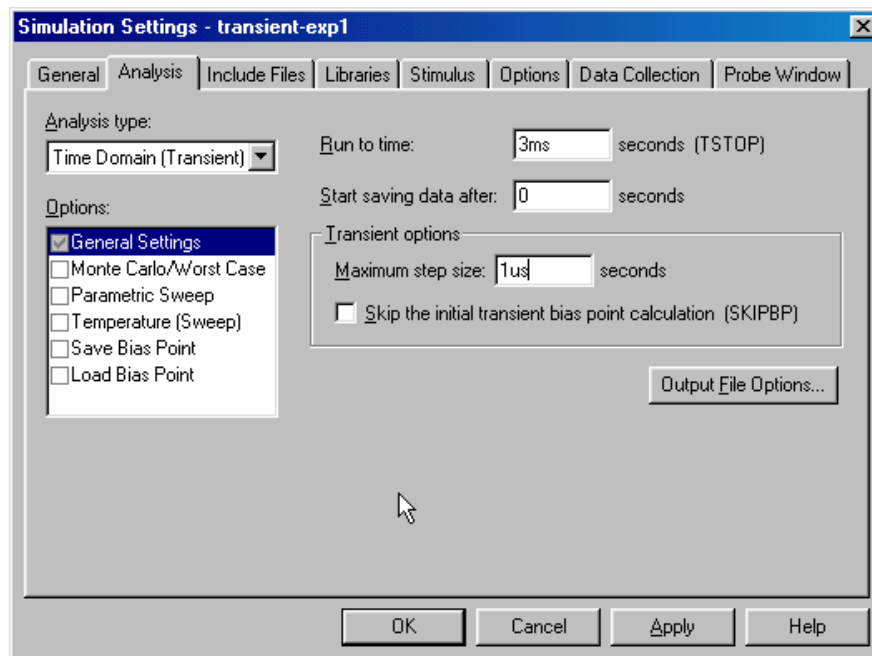



Figure C-9

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 big will take a long time

to run, a step size that is too small 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.

### 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.

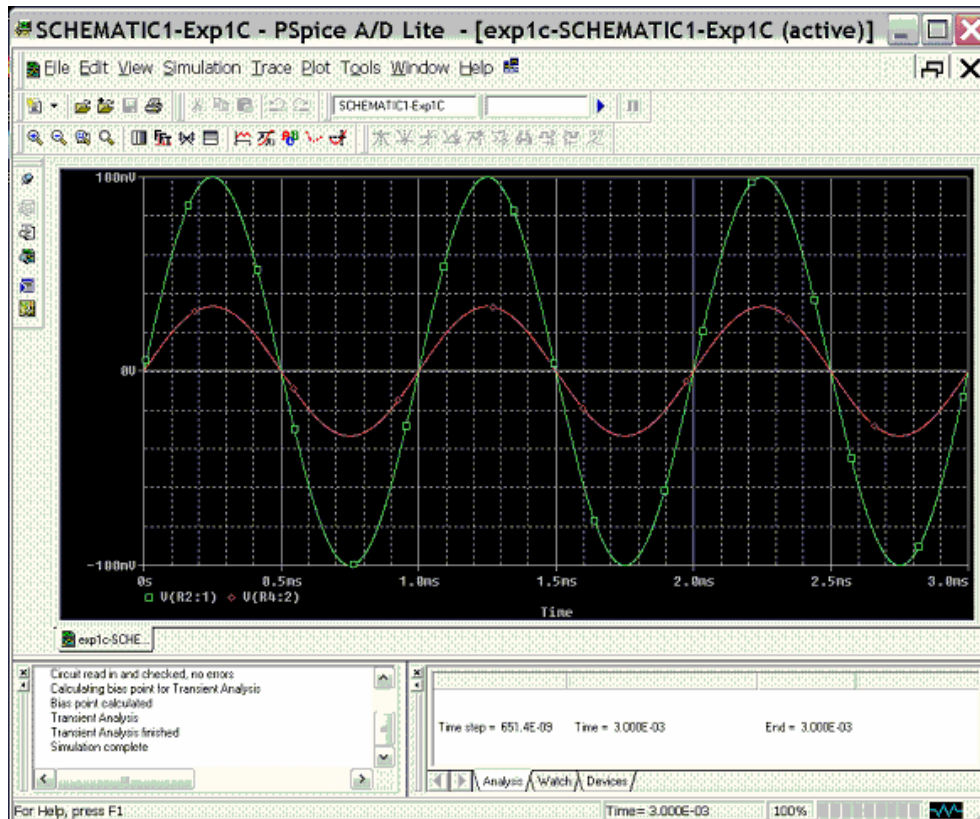


Figure C-10

- You should get something like (but 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.

- 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?

### Part D – Comparing Sine Waves

In this part of the experiment we will use the oscilloscope to compare two sine waves. First we will use Lissajous figures to compare sine waves with similar amplitudes and proportional frequencies. Then we will use subtraction to create two identical signals.

### Background

*The Cathode-ray Tube:* This is one of the most versatile pieces of laboratory test equipment that you will use. It is really a type of analog voltmeter with an arbitrary zero. It can read DC voltages as an offset voltage and AC voltages by displaying the true wave form. Most modern oscilloscopes are capable of measuring AC signals over a wide range of frequencies. The range of frequencies that our scopes can handle (also called bandwidth) is 10 Hz to 100Meg Hz. The heart of any oscilloscope is the cathode-ray tube (CRT) shown in figure D-1.

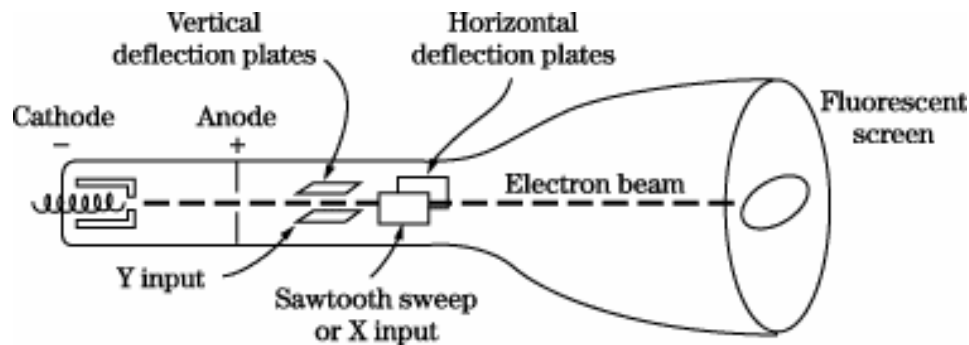


Figure D-1

Electrons are released from the hot cathode and accelerated toward the screen by the use of a positively charged anode. An appropriate grid arrangement then governs the focus of the electron beam on the screen. The exact position of the spot on the screen is controlled by the use of the horizontal and vertical deflection plates. A voltage applied on one set of plates produces the x deflection, while a voltage on the other set produces the y deflection. Thus, with appropriate voltages on the two sets of plates, the electron beam may be made to fall on any particular spot on the screen of the tube. The screen is coated with a phosphorescent material, that emits light when struck by the electron beam. If the deflection of the beam against a known voltage input is calibrated, the oscilloscope may serve as a voltmeter. Since voltages of the order of several hundred volts are usually required to produce beam

deflections across the entire diameter of the screen, the cathode-ray tube is not directly applicable for many low-level voltage measurements, and amplification must be provided to bring the input signal up to the operating conditions for the CRT.

*The Cathode-ray Oscilloscope:* An oscilloscope is comprised of a CRT, a horizontal amplifier, a vertical amplifier, a sweep circuit, and a synchronization circuit, as illustrated in figure D-2.

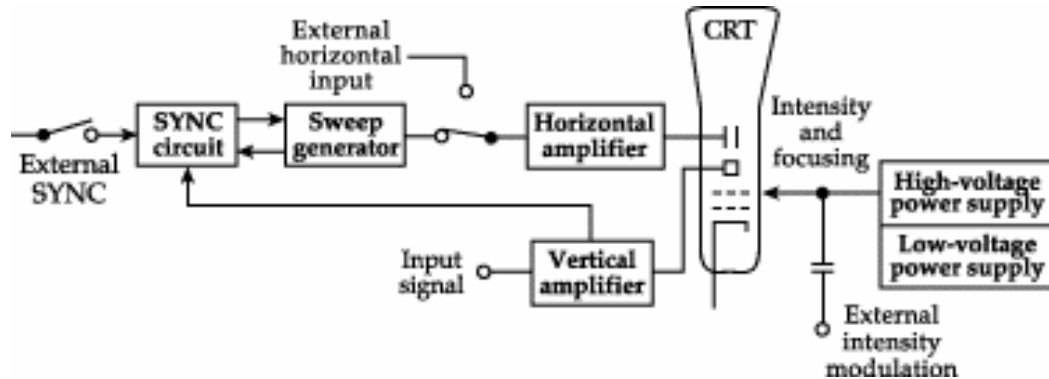


Figure D-2

The gain of the vertical amplifier is controlled by the *Volts/Div* knob found above the two BNC input connectors on the HP54603B two channel scope. The sweep generator produces a sawtooth wave that may be used to provide a periodic horizontal deflection of the electron beam, in accordance with some desired frequency. This sweep then provides a time base for transient voltage measurements by means of the vertical deflection. Oscilloscopes provide internal circuits to vary the horizontal sweep frequency over a rather wide range as well as external connections for introducing other sweep frequencies. The frequency of the sweep generator is controlled by the *Time/Div* knob found above the *Power* button. If you slow the sweep down (by increasing the time per sweep) you will be able to see the electron beam spot move across the CRT screen. Internal switching is also provided. It enables the operator of the scope to lock the sweep frequency onto the frequency impressed on the vertical input. (This is more generally called sync.) This is what the *AUTO SCALE* button does for you when you push it. If we use the function generator to provide the input to our device under test, we can connect the *SYNC* output it provides to the *External Trigger* input on the scope and then select *External* for the trigger source. This is generally the best way to connect things, if we have enough cables. It also avoids the problem of accidentally confusing the *SYNC* with the *OUTPUT* BNC on the function generator.

*Two Channel Scopes:* A dual-beam oscilloscope provides for amplification and display of two signals at the same time, thereby permitting direct comparison of the signals on the CRT screen. It is a very good idea to always use both inputs, so that you can display both the signal into and out of the device you are studying. It is common to put the input signal (coming directly from the source device) on one channel and the output signal (a signal you want to observe in your circuit) on the other channel. This way you can observe what your circuit does to the input by comparing the input signal to the signal generated by the circuit. When we compare two signals we generally look at two things. The first is the change in amplitude between the input and the output. You observed this when you built the voltage divider in part B. The output can also be shifted in phase relative to the input. This means that two sinusoids are not synchronized along the time axis. These two conditions are illustrated in figure D-3.

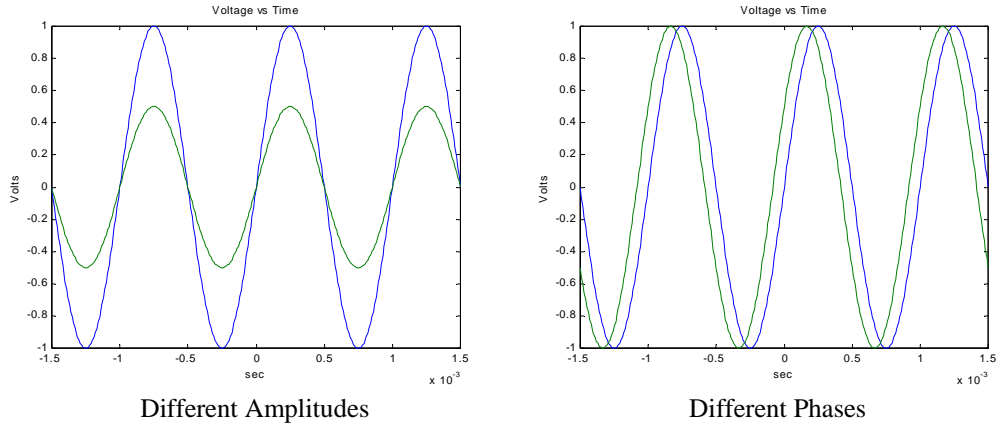


Figure D-3

The phase difference can be found using the following equation:

$$\phi = -\omega t_0 = -2\pi f t_0$$

where  $\omega$  is the angular frequency of the two sine waves and  $t_0$  is the time between the start of the output wave and the start of the input wave. (Recall from the discussion of sine waves in part A that  $\omega=2\pi f$ .)

*Lissajous Figures:* The HP54603B scope offers a convenient means of comparing signal frequencies through the use of Lissajous diagrams. Two frequencies are impressed on the scope inputs, one on the horizontal input and one on the vertical input as shown in figure D-4.

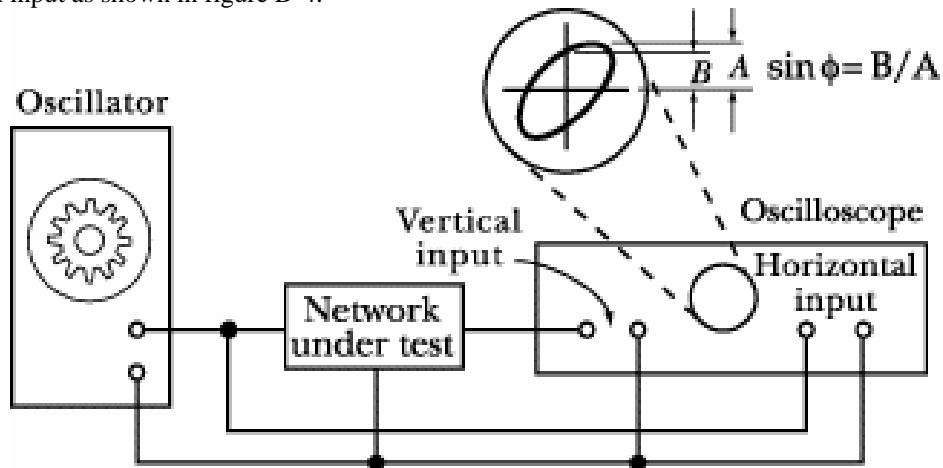


Figure D-4

One of these frequencies may be a known frequency as obtained from a variable frequency oscillator or signal generator. If the two input frequencies are multiples of each other then the patterns that are displayed on the CRT screen are called Lissajous diagrams. The frequency ratio is related to number of vertical and horizontal maxima of the signals. The shape of the output depends on the phase difference between the two signals. Figure D-5 shows Lissajous figures for some simple frequencies ratios and phase differences.

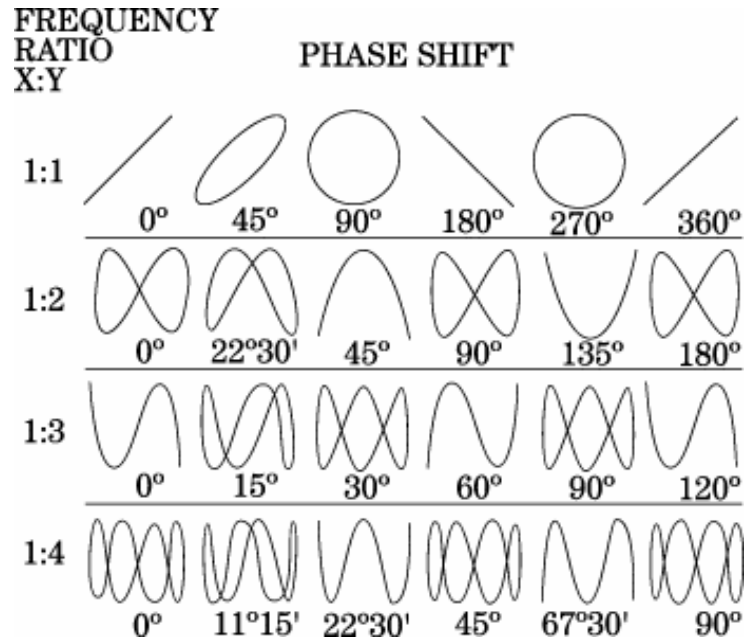


Figure D-5

Lissajous figures may also be used to measure phase shift in an electronic circuit. An oscillator is connected to the input of the circuit under test. The phase-shift angle  $\phi$  may be determined from the relation,

$$\phi = \sin^{-1} \frac{B}{A}$$

where B and A are measured as shown in figure D-4. If both signals have the same input frequency, the shape of the figure varies depending upon the phase difference between the two signals. For zero phase shift, the ellipse will become a straight line with a slope of 45° to the right; for 90° phase shift it will become a circle; and for the 180° phase shift it will become a straight line with a slope of 45° to the left. The top row of figure D-5 shows how the Lissajous figure should look at different phase differences when the frequency ratio between the two signals is 1:1.

## Experiment

### Create a Lissajous Pattern

Under normal conditions, the scope displays voltage signals on the y axis against time on the x axis. It has a special function that allows one to plot one signal on the x axis and the other on the y axis.

- Connect the HP function generator to one channel of your scope and a Brand X function generator to the other channel of your scope. Begin with both function generators set for 200 mV peak-to-peak amplitude and a frequency of 1 kHz. Use the scope to ensure that both function generators are producing the same signal. Note: It may be necessary to choose a different amplitude for the two signals, if they are to be the same. The Brand X generator may not have the same range as the HP.
- You should see two nearly identical sine waves once you hook up both function generators. Remember that the sine wave is defined by three parameters – amplitude, frequency and phase. Check to see that the voltage scale for both channels is the same. (Use the Volts/Div knob to adjust this if needed.) If the amplitudes are not the same, the sine waves will not be the same amplitude. If the phases are not the same, the sine waves will not line up horizontally. If the frequencies are not the same, the wave the oscilloscope is triggered on will be stationary, while the other wave will move (slowly we hope) to the left or right. Figure D-3 shows signals that vary in amplitude and frequency.
- If you cannot get the two waves to stabilize on the scope, the following hints will help.

**Electronic Instrumentation**  
**Spring 2006**

**ENGR-4300**

**Experiment 1**

- Use the soft keys to enable the Vp-p and Frequency measurements for both channels. (as described in part A)
  - Set the signal on the brand X generator to about the correct frequency and amplitude.
  - You can now adjust the HP generator to match. Use the > and < keys to change the digit you want to adjust. Set it so that the two amplitudes and frequencies match closely.
  - Your signals should now be fairly close. One will move either left or right relative to the other. Find the point at which the direction switches and keep going down in orders of magnitude (with >) until the wave slows down. You may never be able to get it to stop completely.
  - To stop the wave at the point you want to take a picture, hit the STOP key on the scope. You can hit RUN to resume.
- 
- We can now generate a Lissajous pattern by clicking on one of the scope buttons. Click on the *MAIN/DELAYED* button to produce a new set of functions for the soft keys at the bottom of the CRT. Then click on the soft key labeled *XY* to combine the two signals into a single Lissajous pattern. Compare what you see to the examples of Lissajous figures. Use the Agilent software to take a picture of the image. Your TA or instructor can help you with this. Include this picture with your report. [Hint: Use the STOP button on the scope to freeze your figure at the point you wish to take a picture.]
  
  - You should now play with the equipment and produce several other representative patterns. Create one pattern that you find particularly interesting. Take a picture of it with the Agilent software.

**Electronic Instrumentation**  
**Spring 2006**

**ENGR-4300**

**Experiment 1**

*Subtracting two signals*

Next, we will do a different kind of comparison of the two sine waves, one that will prove to be very important in the development of measurement techniques. In this measurement, we will compare two signals to see how close they are to one another by subtracting one from the other.

- Click on the *MAIN/DELAYED* button again in the *HORIZONTAL* section. Change back to the *MAIN* display by clicking on the left most soft key. This should bring back the two individual sine waves. Adjust them again so that they are as identical as possible. (Try to get them displayed on top of one another.)
- Click on the channel math button. Click on the soft key labeled *1-2*, which will produce a third trace that is the difference between the two channels. If you have adjusted the two sine waves to be identical, their difference should be zero. How well did you do? Note that the amplitude, the frequency, and the phase must be identical to make the difference zero.
- Now make the two signals as identical as possible by adjusting the difference signal away. What did you have to do?

**Summary**

From this exercise, you should see the value of comparing one signal against some kind of reference. It is possible to tune a guitar, for example, by comparing the tone a string makes with an electronic tone. This permits a perfectly tuned instrument, even when the player has less than perfect pitch. You have also seen how we make what are called *differential measurements*. There are many advantages to making differential over absolute measurements. You have seen one of the key reasons since differential measurements allow you to focus on smaller quantities since you are working with the difference between two signals

**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 (12 points)**

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 pt)
2. What is the measured impedance of the speaker? (1 pt)
3. When the speaker is also connected to the function generator, what does the scope measure for the peak-to-peak amplitude when the speaker is producing a comfortable level of sound? (1 pt)
4. Draw a circuit with the scope and the speaker connected to the function generator. Calculate the load on the function generator for this situation. (The load is the combined resistance of the scope and the speaker in parallel.) (5 pt)
5. 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.* (2 pt)
6. What frequencies make up the audio range? That is, what are the lowest and highest frequencies that humans can hear? *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.* (2 pt)

**Part B (32 points)**

Include the following plots:

1. Agilent plot of input and output from the 1K voltage divider. (2 pt)



**Electronic Instrumentation**  
**Spring 2006**

**ENGR-4300**

**Experiment 1**

2. Agilent plot of input and output from the 1Meg voltage divider. (2 pt)

Answer the following questions:

1. Why is there a discrepancy between the peak-to-peak voltage displayed on the function generator and the signal displayed on the scope when they are connected directly? Which is correct -- the function generator or the scope? (1 pt)
2. 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 pt)
3. What was the difference (in percent) between the actual measured values of the resistors and their specified values? (1 pt)
4. You applied 10 Volts from one of the DC power supplies to the two 1 k ohm resistors connected in series. Calculate the theoretical voltage you would expect to measure across each resistor when no measuring device is connected to the circuit. What values did you measure with the DMM? Draw and analyze the circuit of the voltage divider with the DMM connected to demonstrate that your circuit behaves as it should taking into account the effect of the equipment. (*Note that the DMM has an internal impedance of 10M when reading DC values.*) (5 pt)
5. Repeat question 4 above with the two 1 Meg ohm resistors in series. (5 pt)
6. You created a 1k hertz signal with a 100 mV amplitude and applied it to two 1K resistors in series. Calculate the theoretical voltage you would expect to measure across each resistor when no measuring device is connected to the circuit. What amplitude did the voltage you measured across the second resistor have? Draw and analyze the circuit of the voltage divider with the scope connected to demonstrate that your circuit behaves as it should taking into account the effect of the equipment (*Note the scope has an internal impedance of only 1Meg*). (5 pt)
7. Repeat question 6 above with the two 1 Meg ohm resistors in series. (5 pt)
8. You should see a larger discrepancy between the predicted and measured voltage when the 1 Meg ohm resistors are used. Explain why this is the case. (2 pt)
9. You saw that the DMM and the scope can change the voltage they are trying to measure with the 1k ohm and 1Meg 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 a range of reasonable resistance values. There is no one right answer to this question. (2 pt)

**Part C (12 points)**

Include the following plots

1. *PSpice* transient plot of voltage divider with 2 1K resistors (two traces). (3 pt)
2. *PSpice* transient plot of voltage divider with 2 1MEG resistors (two traces). (3 pt)

Answer the following questions

1. How does the signal you generated with *PSpice* using the 1K voltage divider compare to the one you generated with the 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 scope? (3 pt)

**Part D (12 points)**

Include the following plots:

1. Agilent plot of the Lissajous comparison of the signals from two function generators. Indicate on the plot the frequency you used for the X signal and the Y signal. (2 pt)
2. Agilent plot of one additional Lissajous pattern. (2 pt)

Answer the following questions:

1. When you were displaying the two sine waves separately, the scope triggered on only one of the waves. Which way did the other wave move when its frequency was lower, right or left? (2 pt)

**Electronic Instrumentation  
Spring 2006**

**ENGR-4300**

**Experiment 1**

2. Discuss which of the Lissajous patterns you were able to generate and what you had to do to create them. (2 pt)
3. Qualitatively describe your efforts to make the difference between the two signals equal to zero using the channel math button to display channel 1 minus channel 2. (2 pt)
4. If you wished to display three cycles of a sinusoidal voltage wave with a frequency of 10kHz, at what minimum value *on the scope* should you set the horizontal time division (Time/Div)? What is the minimum value would you need to display at least three cycles at 5kHz? (2 pt)

**Summary (12 points)**

1. Summarize key points (8 pts)
2. Discuss mistakes and problems (2 pts)
3. List member responsibilities (2 pts)

**Total: 80 points for write up + 20 for attendance = 100 points**

**Attendance: 3 classes (20 points) 2 classes (10 points) 1 class (0 points) out of 20 possible points  
Minus 5 points for each late.  
No attendance at all = No grade for experiment.**