

Experiment 2 Basic Circuits and Electronics

Purpose: In the following exercises, we continue gaining experience with some of the equipment in the Instrumentation Studio and work with some basic circuits and electronics concepts.

Equipment Required:

- **Oscilloscope** (HP 54603B 2 Channel 60 MHz Oscilloscope)
- **Function Generator** (HP 33120A 15 MHz Function/Arbitrary Waveform Generator)
- **Function Generator** (Brand X Function Generator)

Background The following material is taken from *Basic Electronics*, a lab exercise written by James A. Harrington of Rutgers University. Some of the material on the working of the oscilloscope is taken from J.P. Holman, *Experimental Methods for Engineers*, McGraw-Hill, New York, 1989. The following paragraphs describe the analog oscilloscope. Most new oscilloscopes today are digital, not analog, and offer substantial benefits over the analog scope. Hewlett-Packard (now Agilent) stopped introducing analog scopes over 20 years ago, but most conventional TV sets have worked like analog 'scopes.

The Cathode-ray Oscilloscope

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 as well as AC voltages by displaying the true wave form. Most modern oscilloscopes are capable of measuring AC signals over a wide range of frequencies. Check your oscilloscope for its frequency response. Find the minimum and maximum frequency. Note – You can get this information from the oscilloscope link in the *Facilities* section of the course webpage. You should find that there are two ranges listed, one for DC coupling and one for AC coupling. Either answer is fine.

The heart of any oscilloscope is the cathode-ray tube (CRT), which is shown schematically in Fig. 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, which 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.

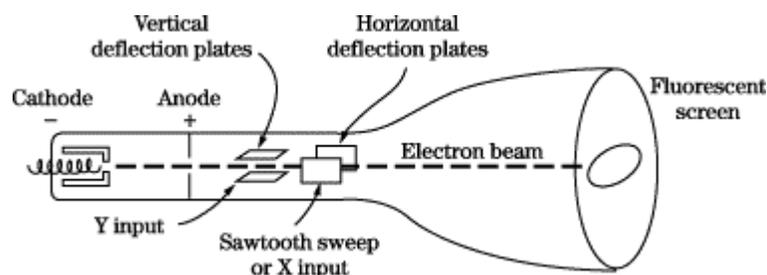


Figure 1 - Schematic diagram of cathode-ray tube (CRT)

A block schematic of the oscilloscope is shown in Fig. 2. The main features are the CRT, as described above, the horizontal and vertical amplifiers, and the sweep and synchronization circuits. The gain of the vertical amplifier is controlled by the *Volts/Div* knob found above the two BNC input connectors on the HP 54603B 2 Channel 'scope. (Here you should either look at the 'scope or the picture on page 4 of this write up.) The sweep generator produces a sawtooth wave which 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, which 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.

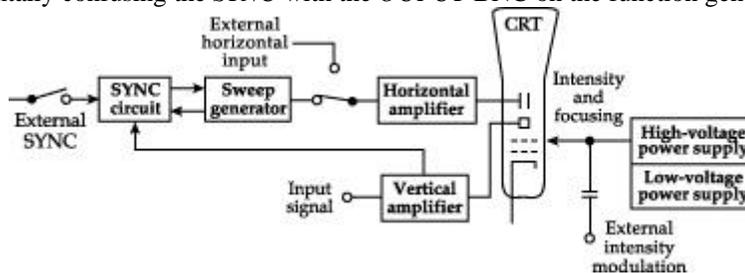


Figure 2 – Block schematic diagram of an oscilloscope

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. On the HP 54603B 'scope, we can use one of the two inputs as the horizontal input shown on the diagram above.

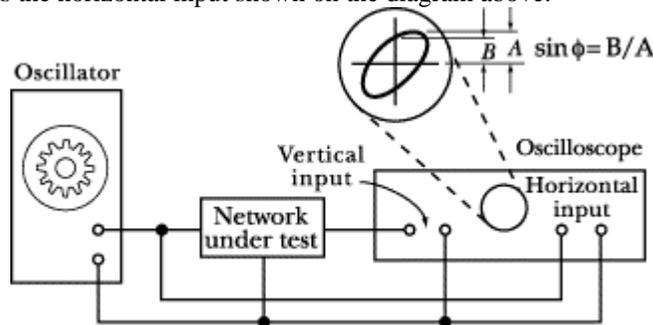


Figure 3 – Use of oscilloscope for phase measurements

The 'scope may be used to measure phase shift in an electronic circuit, as shown in Fig. 3. An oscillator is connected to the input of the circuit under test. The output of the circuit is connected to the 'scope vertical input, whereas the oscillator signal is connected directly to the horizontal input. The phase-shift angle ϕ may be determined from the relation,

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

where B and A are measured as shown in the figure. 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.

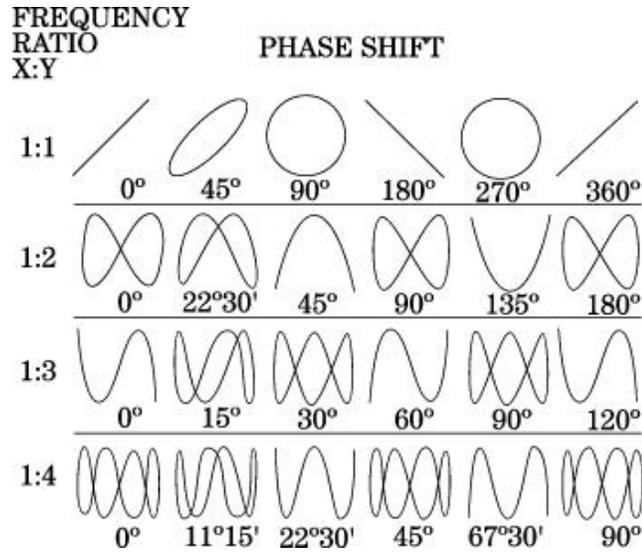
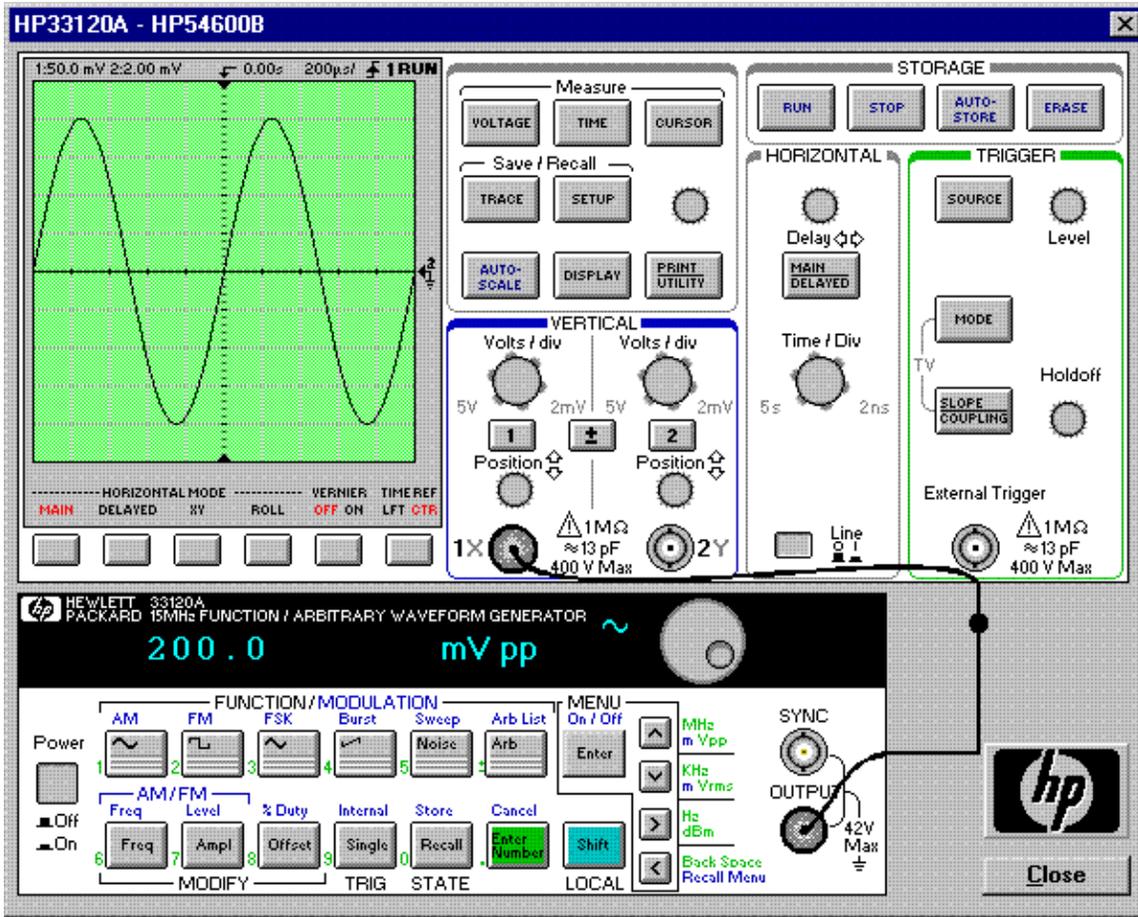


Figure 4 – Lissajous Patterns

The '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. 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 diagrams. Some typical shapes for the Lissajous diagrams are shown in Fig. 4. It may be noted that these shapes can vary somewhat depending on the phase relation between the input signals.

Part A Comparison of Two Sine Wave Signals

In the first part of this exercise, we will generate some Lissajous patterns. 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 in the manner you learned in Experiment 1 to be sure that both function generators are producing the same signal. 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. The setup for the HP function generator and the 'scope is shown in the figure below. This is taken from a software emulator of these devices that is available from Agilent (and now also on the studio computers).



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. If the amplitudes are not the same, the sine waves will not be the same size (see Figure 5). If the phases are not the same, the sinewaves will not line up horizontally (see Figure 6). 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.

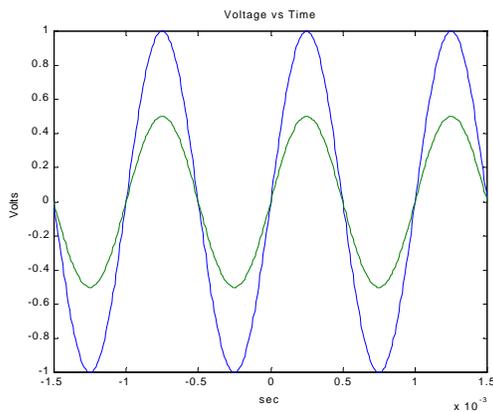


Figure 5 – Different Amplitudes

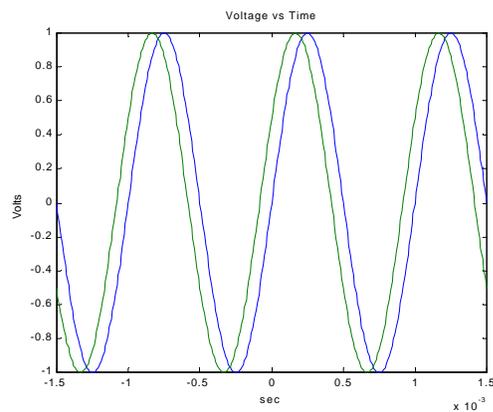


Figure 6 – Different Phases

Once you have both of the signals the same, meaning as close as possible, change the frequency of the HP generator signal up and down and see how the signals compare. You should notice that it is somewhat difficult to

very accurately compare the two signals by displaying them separately. We can now generate a Lissajous pattern by clicking on one of the 'scope buttons. On the 'scope face you will notice that there are several blocked off sections that address different functions. We have used the *VERTICAL* section to adjust the display of the signal amplitudes and the *HORIZONTAL* section to adjust how much time we are displaying. By changing the *Time/Div*, for example, we can change the two cycles shown to many more cycles of the sine wave. 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 that were displayed separately into a single Lissajous pattern. Describe what you see. (You can use the examples shown in Figure 4). Use the benchmark 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 of.]

You should now play with the equipment and produce several other representative patterns. Create three patterns that you find particularly interesting. Which ones have you done?

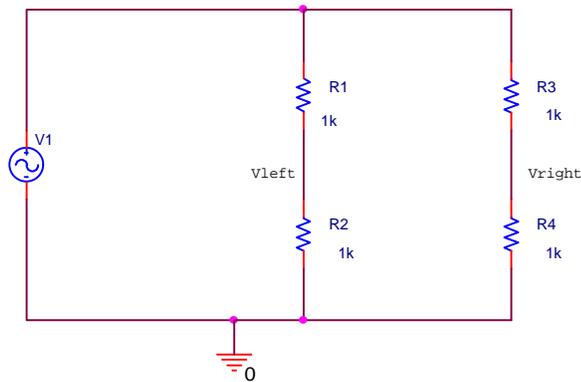
Next, we will do a different kind of comparison of the two sine waves, one that will prove to be very important in the development of measurement techniques. In this measurement, we will compare two signals to see how close they are to one another by subtracting one from the other. Click on the *MAIN/DELAYED* 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.) Once you have done so, click on the channel math button (this is labeled with both a plus and a minus and is half way between the buttons for the two channels in the *VERTICAL* section). 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?

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. We will see another good reason for differential measurements when we address bridge circuits.

Part B Voltage Divider and Bridge Circuits

In Experiment 1, we looked at one of the simplest useful circuits – the voltage divider. In many simple applications of electronics, we have only a small number of standard voltages in whatever circuits we are building. When we use a 9 volt battery as our source we have only one voltage level available, unless we use a voltage divider to get smaller voltages. We can also use a divider to measure resistance, if we have some device with an unknown resistance. For example, if we connect an unknown resistor in series with a known resistor, then the voltage across the unknown resistor can tell us the value of the resistance. An even better measurement can be done by combining two voltage dividers in a configuration like the one shown in the following figure. Note that if $R_1 = R_2 = R_3 = R_4$, the voltages at the two points marked V_{left} and V_{right} will be equal to half of the source voltage. Thus, their difference should be zero. Set up the circuit shown using PSpice, following the procedure given in Experiment 1. Use a 100mV, 1kHz source and setup the transient analysis so that three cycles of the sine wave are shown. Once you finish the analysis and display the voltages at the two points, click on *Trace* on the probe menu and add the trace of the difference between the two voltage. You can use the differential voltage marker, type it in directly as a mathematical expression or click on the symbols in the node list (the latter two choices are found using the Add Traces window). You should find that the difference is indeed equal to zero.

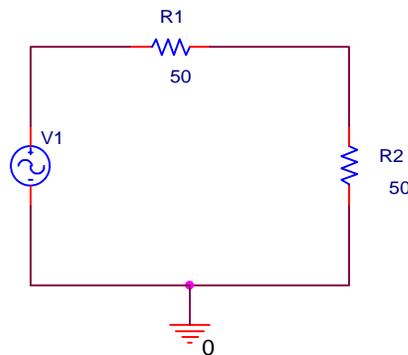


Now, change R4 to be equal to 1.1k ohms. Do the analysis again and add the trace of the difference between the two voltages. What is the amplitude of the difference voltage as a percentage of the source voltage? Print out this plot and include it with your report.

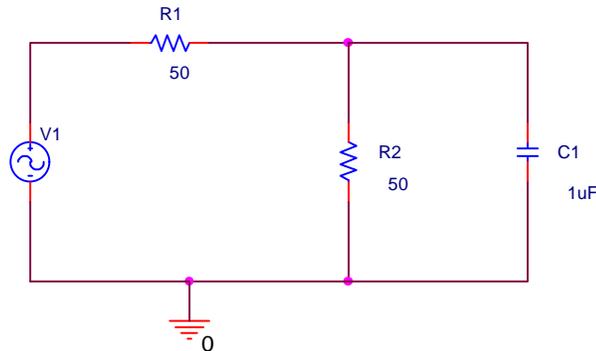
Now analyze this circuit by hand, and find the voltages at the two points and their difference. Make sure that your answer agrees with the PSpice simulation.

Finally, assume that $R1 = R2 = R3$ are known resistors equal to R , and that $R4$ is unknown. Derive a formula for $R4$ in terms of R , the source voltage $V1$, and the voltage difference between the two divider voltages ($V_{left} - V_{right}$).

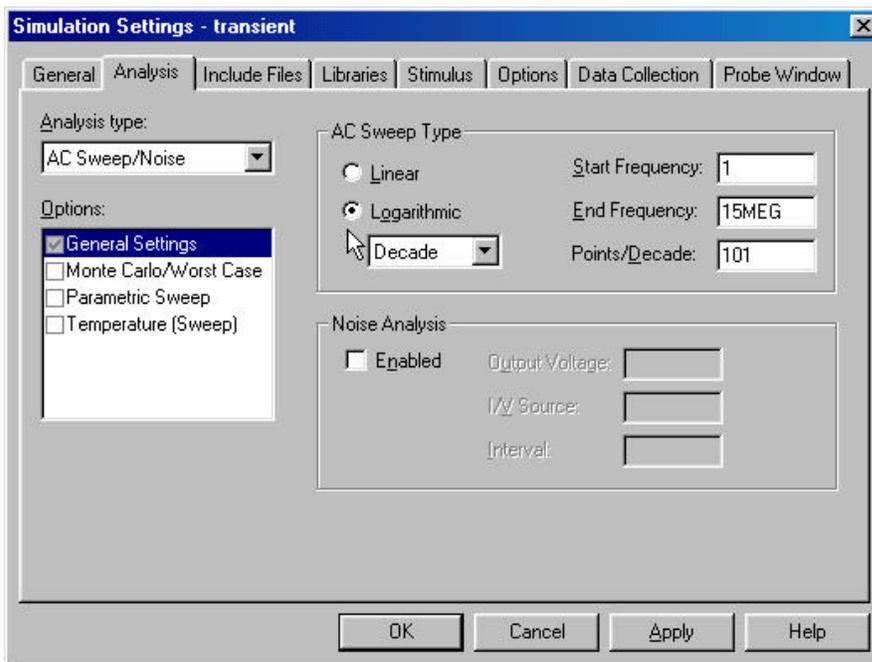
We have been doing only one kind of simulation with PSpice. You may have noticed that there seem to be quite a few other options. Two that we will use quite a bit are DC and AC analysis. For the former, we will need to have a source with a DC (constant with respect to time) voltage. For the latter, we will not learn much unless our circuit also contains some capacitors or inductors. We have not reviewed the properties of these other circuit devices, but we will analyze some simple cases to see how they work. Let us consider the simple voltage divider again to see how a finite DC source value will be reflected in the circuit analysis. Use PSpice to configure the circuit shown below. This is the same source – VSIN – that we have used a couple of times already. Set up VSIN in the same manner as we did in Experiment 1, except now make VOFF = 1V, which will add an offset of 1 volt to the 100mV, 1kHz sine wave. Place a voltage marker on either side of resistor R1. You should find that the sine waves displayed by PROBE will no longer oscillate around zero, but will be positive at all times. Change the value of R2 to 100 ohms. How do the DC values (offset) differ from the case where both resistors are 50 ohms?



Now change R2 back to 50 ohms and add a 1 μ F capacitor in parallel with R2. Run the analysis again. You should see little or no difference between this and the previous results. Capacitors can cause a circuit to behave differently at some frequencies and have no effect at other frequencies.



In order to see this, we will have to do another kind of analysis. Click on the *Edit Simulation Settings* button. Select the *AC Sweep/Noise* analysis and set it up as shown.



The AC Sweep Type has been chosen as *Logarithmic - Decade* since frequency effects usually only become obvious when we change orders of magnitude. This generates a log scale for frequency. The start frequencies and end frequencies are chosen to cover an interesting range. Usually this range is selected from some knowledge of the expected performance of the circuit. However, since we are assuming that we know very little about this circuit, we can set the range to be roughly that covered by the HP function generator. Note that 15MHz had to be written as 15MEG, since PSpice uses both lower case m and upper case M to mean milli. We have to make one more IMPORTANT change before we can do AC analysis. Double click on the voltage source V1 again to bring up its attribute spreadsheet. You will now have to give a value to AC. This can really be anything, but make it equal to the sine wave amplitude 100mV. *In general, it is not a bad idea to do this from the beginning in any analysis involving an AC (or VSIN) source to avoid problems when the analysis type is changed.*

PROBE will produce a plot consisting of a flat line and a line that is flat for lower frequencies but falls to zero for higher frequencies. These lines are the amplitudes of the sine waves at the two nodes where we have placed voltage markers. The DC offset has no effect. For what range of frequencies does the capacitor change the voltage across R2 by less than 5%? Use the cursor to find a reasonably precise answer to this question. Print out your plot with this point clearly labeled. Include this plot with your report.

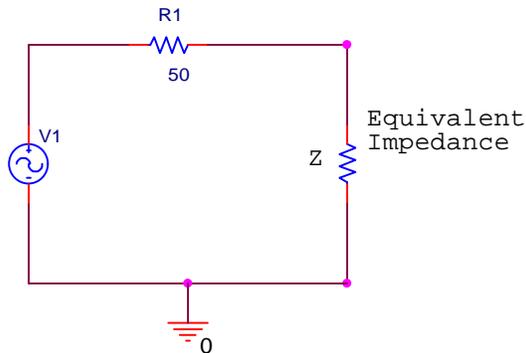
Note that at the higher frequencies the voltage is very small. This means that at high frequencies the combination of the capacitor and R2 looks like a very small resistor indeed. Remember that in a voltage divider, the voltage across the resistors is proportional to the value of the resistance.

Let us see if we can figure out at least the magnitude of the equivalent impedance of the combination of R2 and C1 at the frequency of 1 MHz. Note that here we have used the term *impedance* rather than *resistance*. We will reserve *resistance* for when we are speaking of resistors and use *impedance* for combinations of resistors, capacitors, and inductors. In the circuit you are analyzing, R2 and C1 together have a different impedance at different input frequencies. This means that at any one given frequency, we could replace the combination by a single resistor (which we will call Z).

Go back to your schematic and change the value of the source (V1) frequency to 1MEG. This frequency is only used in the transient analysis, but that is what we want to look at now. You cannot yet run the analysis, since the frequency has been changed so much. Click on the *Edit Simulation Settings* button and set up for transient analysis using smaller times. Since 1 MHz is 1000 times larger than 1 kHz, you will have to make each of the three times you have specified 1000 times smaller to produce three cycles of the oscillating voltage signals. (Since one of the times – the start time – should be zero, making it three times smaller is not necessary.) Once you have made these changes, run the transient analysis. You should see that source oscillates as it did before, although at a much higher rate (you should see the same number of cycles, but the time scale is now much shorter) and that the voltage across R2 and the capacitor seems to not change with time at all. Actually, the latter voltage is still oscillating, but at such a small amplitude that you cannot see it. Delete the trace that shows the source voltage. You can do this by clicking on the node label at the bottom of the PROBE window that corresponds to the side of resistor R1 connected to the source (this should be something like V(R1 :1), but may be something different, depending on the precise location of your voltage marker) and then hitting the Delete button on your computer keyboard. (Make sure that you have deleted the correct label.) Using the cursors, determine the amplitude of the sine wave oscillation across R2 and the capacitor. The voltage will also have a DC level, but we only want to determine the sine wave amplitude. The peak-to-peak amplitude can be determined by subtracting the voltage at a minimum from the voltage at a maximum. The actual amplitude of the sine wave will be half the peak-to-peak value. Write down the amplitude you determined here.

From this amplitude and your knowledge of how voltage dividers work, determine the value of the equivalent impedance of the R2/C1 combination, which we are calling Z. [Hint: $V_z = V_1 * (Z) / (R_1 + Z)$] (Since we have only included one capacitor in the circuit, it should have the label of C1. However, you may have created your circuit in a nonstandard manner and thus ended up with a different label.)

Below is a picture of the circuit with the impedance of R2/C1 represented by a resistor Z. If you want to check your answer, replace the capacitor resistor combination (R2/C1) with a resistor with the value of Z you calculated. Rerun the simulation. Does the output have the same amplitude as the combination?



Part C Potentiometers and Strain Gauges

A very useful type of voltage divider is the *potentiometer*. A potentiometer is a resistor that has a tap that can slide up and down the resistor producing two variable resistors. The symbol for the potentiometer (or *pot* as it is usually called) is very descriptive of its operation. We will first set up a PSpice simulation of a couple of pots with different loads. Remember that the size of the load resistor can be significant in the performance of a circuit.

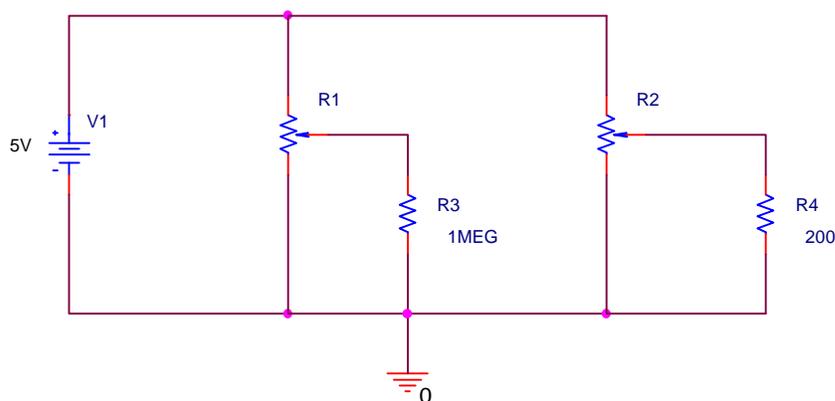


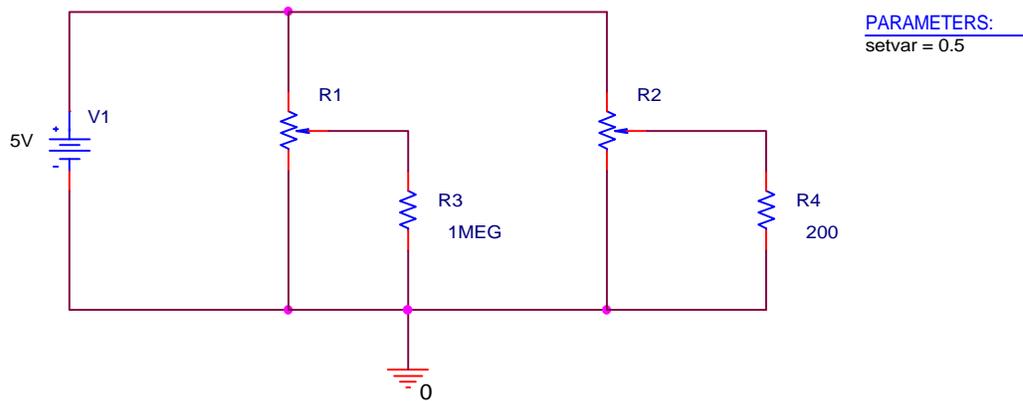
Figure C1

Using PSpice, create the circuit shown. The voltage source is now VDC rather than VSIN, since we only want a DC source this time. The pots are POT in the parts list. However, the library in which POT is found is not loaded by default. You will have to use the *Add Library* button in the *Place Parts* window. The library you will need is called *Breakout*. (This library may already have been loaded, if someone has used this software on your computer previously.) When you do not know the library to look in, you can use the *Browse* feature to find it. Since pots are variable, their values have to be set up before the analysis can be done. Unlike other parts, pots come with a complete set of default attributes. The total resistance of the pot will be 1k ohm and the tap (also called the wiper) will be set at the half way point. Connecting across the entire pot from top to bottom will give a resistance of 1k ohm. Connecting from the top to the tap will give a resistance of 500 ohms, as will a connection from the tap to the bottom. Thus, for the default conditions, we have, in effect, a 500 ohm resistor above the tap and a 500 ohm resistor below the tap. Run a transient analysis at this point and you will see that the voltage at the tap on the left pot is close to half of the source voltage, as it should be. The large 1 M ohm resistor does not load down the pot voltage divider very much. However, the 200 ohm load on the right pot is much smaller than the 500 ohm

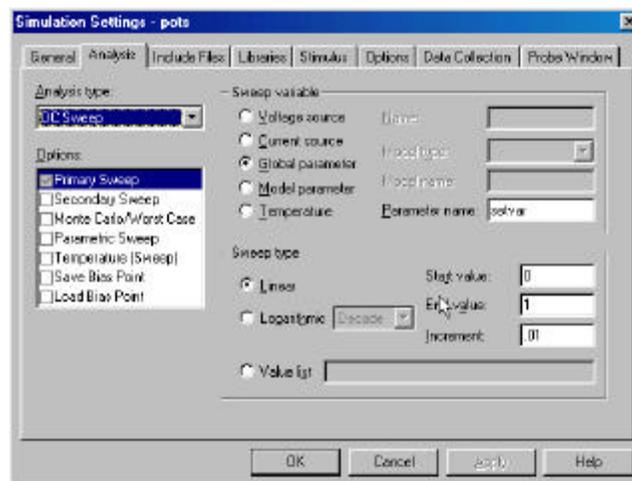
on your schematic. Double click on this word and you will be able to set its attributes. When you have finished, you will see the parameter and its default value listed under Parameters. To declare a global parameter, use the following procedure:

- Place a PARAM part in your design (which you have already done)
- Double click on the PARAM part to display the Parts spreadsheet and click on New Column.
- In the Property name textbox, enter your chosen name (here use *setvar*).
- Enter 0.5 for the default value, since that will leave the wiper half way up in its default position and click ok.
- While this cell is still selected, click *Display*.
- In the *Display Format* window select *Name and Value*, then click *OK*.
- Click *Apply* to update all the changes to the PARAM part.
- Close the Parts spreadsheet.

When you are finished, your circuit look something like this:



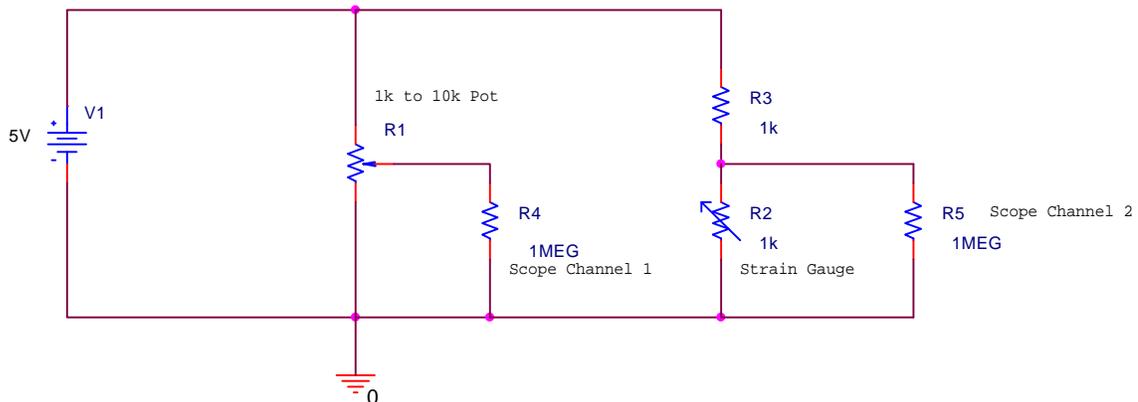
- 3) Next we will set up the analysis. Click on the *Edit Simulation Profile* button or the *New Simulation Profile* button, depending on whether or not you have already run a simulation. Select DC sweep. Then, set up the global parameter which we have called *setvar* to vary linearly from 0 to 1 in steps of 0.01.



Now perform the simulation and you will see the voltages at the taps of the two pots. You should print this particular plot and then discuss why it looks the way it does. Include this in your report.

We will now combine our study of potentiometers and bridge circuits in a practical hardware application. You will need a 1 k ohm resistor, an instrumented beam, and a pot. We used a 1 k ohm pot for this simulation, so try to get one about that value. However, the operation of the bridge is not changed fundamentally as long as a pot with a value from 1 to 10 k ohm is used. The beam comes with two sets of output wires. One, usually covered with black insulation, is connected to a coil located on the support plate near the moveable end of the beam. The other, usually with twisted wires, is connected to a small strain gauge cemented near the fixed end of the beam. We will only be using the strain gauge in this experiment. The strain gauge is relatively fragile, so its wires are usually fixed to the support plate before going to the outside world. It is still a good idea, however, to be very careful with such devices. The ones we are using are quite good (aka not cheap).

Before you build the circuit, use the multimeter to determine the resistance of your strain gauge in its rest position, with the beam deflected down until it touches the support plate, and up an equal amount. *Do not make this measurement when the beam is connected to the circuit.* Do not over extend the beam upwards. Write down the maximum and minimum resistance of your strain gauge. You should see why we indicate this resistance with a variable resistor in the figure below.



The diagram above should have the information you need to build the circuit correctly. Once you get the circuit built, **adjust the pot** so that there is no voltage when the beam is not deflected. The best way to monitor the difference between the two output points is to connect each point to one of the 'scope channels and then use the channel math button to display the difference between the two voltages. When you have everything hooked up, set the beam into free oscillation. You should observe a decaying sinusoidal voltage. Estimate the frequency of the oscillation. You will note that the AUTO SCALE feature will probably not work at this low frequency. This is one of the problems with mechanical systems. The natural frequencies of mechanical devices are usually low. Thus, you will have to learn to set your scales manually if you hope to see anything useful. Once you have obtained a clear signal of a decaying sinusoid, stop the 'scope scan so that you can look at the signal for some time. Then use the Benchlink software to obtain a plot of the signal. Include this plot with your report. Indicate on it how you found the beam frequency. Note that a more accurate estimation of the frequency can be found by averaging over many cycles.

Note that there is a very good reference on strain gauges available in the Helpful Info section of the course webpage. Look for the Interactive Guide to Strain Gauge Technology from Vishay Measurements Group, Inc. To apply any of the materials available there to the experiment we are doing, you need to know that the gauge factor of the strain gauge we are using is 125. (Typical for semiconductor gauges.)

Report and Conclusions

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

Part A

Include the following plots:

1) Benchlink 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. (1 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? (1 pt)
- 2) Discuss which of the Lissajous patterns you were able to generate and what you had to do to create them. (1 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. (1 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 (sec/div)? What value would you need to display three cycles at 5kHz? (1 pt)

Part B

Include the following plots:

- 1) PSpice voltage divider plot with $R_4 = 1.1K$ ohms (1 pt)
- 2) PSpice AC Sweep plot with 5% point marked with cursor (1 pt)

Answer the following questions:

- 1) What is the value of the difference voltage as a fraction of the input voltage for the case where one of the 1k resistors is replaced with a 1.1k resistor? (1 pt) Show that you get the same answer when you analyze the circuit by hand, applying the appropriate formulas to do the analysis. (1 pt)
- 2) Assuming that $R_1 = R_2 = R_3$ are known resistors equal to R, and that R_4 is unknown, DERIVE a formula for R_4 in terms of R, the source voltage V1, and the voltage difference between the two divider voltages ($V_{left} - V_{right}$). (2 pts)
- 3) For the case where VSIN is given a DC offset, what is the DC voltage across resistor R_2 when R_2 is 50 ohms and when it is 100ohms? (This is for the circuit with no capacitor.) (1 pt) When the capacitor is added, for what range of frequencies does the capacitor change the voltage across R_2 by less than 5%? (1 pt)
- 4) What is the value of the equivalent impedance of the resistor-capacitor combination at 1M Hertz? Show how you found it. (1 pt)
- 5) What is the difference between impedance and resistance? (1 pt)

Part C

Include the following plots:

- 1) PSpice setvar sweep plot (1 pt)
- 2) Benchlink plot of beam oscillation with method of calculation of beam frequency on it (1 pt)

Answer the following questions:

- 1) What is the resonant frequency of the beam? (1 pt)

Electronic Instrumentation

ENGR-4300

Fall 2004

Section _____

- 2) What is the resistance of your strain gauge when the beam is in its undeflected position? when it is deflected fully downward? (1 pt) Why must this measurement be made while the strain gauge is not connected to the circuit? (1 pt)
- 3) Draw the equivalent circuit (with regular resistors, not with pots) for figure C1 and then, calculate the voltages you expect to see at the two wipers. (Assume the pots have a set value of 0.5.) (1 pt) Check to be sure that your answer agrees with the PSpice Output File. (1 pt)
- 4) Connect the function generator directly to the 'scope and set its frequency to 10 Hz and peak-to-peak amplitude to 1 volt. Do not press the AUTO SCALE button. Rather, adjust the vertical and horizontal scales (and do whatever else you need to do) to display the voltage on the 'scope. Once you have accomplished this, explain what you had to do and why. (1 pt)

Summarize Key Points (1 pt)

Mistakes and Problems (0.5 pt)

Member responsibilities (0.5 pt)

Total: 25 points for write up

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