Experiment 4
Op-Amp Circuits

Purpose: In this experiment, you will learn about operational amplifiers (or op-amps). Simple circuits containing operational amplifiers can be used to perform mathematical operations, such as addition, subtraction, and multiplication, on signals. They can also be used to take derivatives and integrals. Another important application of an op-amp circuit is the voltage follower, which serves as an isolator between two parts of a circuit.
*Several op-amp chips have the same pin configuration, two examples are the 741 op-amp and the LF351 op-amp. The demo version of Capture doesn't have the LF351 version available, so the modeling will be done using the 741 op-amp (a.k.a. µ741 op-amp.) The LF351 is somewhat better for our application and will extend the battery lifetime. **Model the experiment in PSpice using the 741 but build it using the LF351 if you experience problems with the 741 in the circuit.***

Equipment Required:

- **DMM** (HP 34401A 6-1/2 Digit Multimeter)
- **Rensselaer IOBoard Rev D** (with Mobile Studio Desktop)
- **DV Voltage Source** (Rensselaer IOBoard, and two 9V batteries) Students in JEC 4107/4104 should use the HP power supply, and avoid battery issues.
- **Analog I/O** (Rensselaer IOBoard)
- Protoboard
- Some Resistors (50, 1k, 10k and 100kΩ)
- uA741 or LF351 op-amp (LF351 is used if uA741 in kit is bad)

Helpful links for this experiment can be found on the links page for this course:
http://hibp.ecse.rpi.edu/~connor/education/EILinks.html#Exp4

Part A – Introduction to Op-Amp Circuits

Background

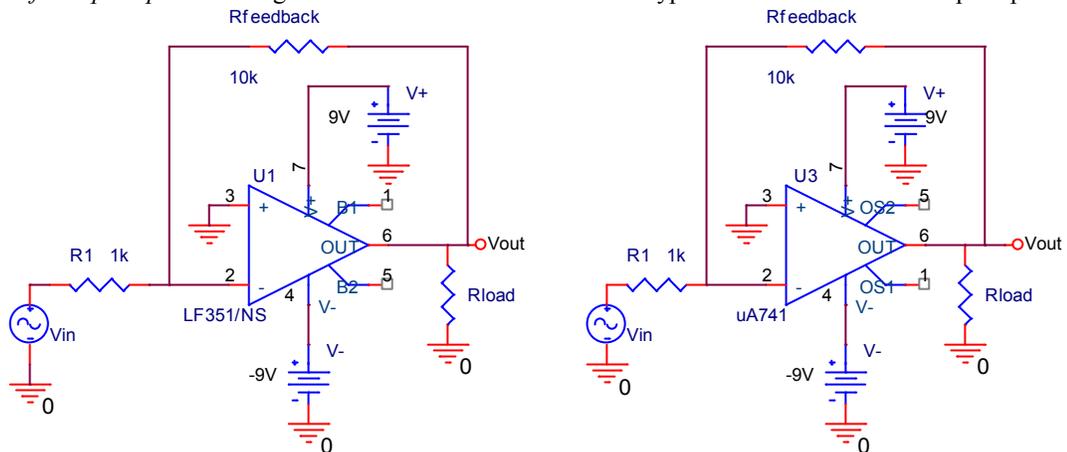*Elements of an op-amp circuit*: Figure A-1 below is a schematic of a typical circuit built with an op-amp.

Figure A-1. Drawn with both 741 and a LF351 op-amps (Rload ≈ 1kΩ).

The circuit performs a mathematical operation on an input signal. This particular op-amp circuit will invert the input signal, Vin, and make the amplitude 10 times larger. This is equivalent to multiplying the input by -10. Note that there are two DC voltage sources in addition to the input. These two DC voltages power the op-amp. The circuit needs additional power because the output is bigger than the input. Op-amps always need this additional pair

of power sources. The two resistors R$_{feedback}$ and R1 determine how much the op-amp will amplify the output. If we change the magnitude of these resistors, we do not change the fact that the circuit multiplies by a negative constant; we only change the factor that it multiplies by. The load resistor R$_{load}$ is not part of the amplifier. It represents the resistance of the load on the amplifier.

Powering the op-amp: The two DC sources, (labeled as V+ and V-, but also often labeled as ± V$_{CC}$), that provide power to the op-amp are typically set to have an equal magnitude but opposite sign with respect to the ground of the circuit. This enables the circuit to handle an input signal which oscillates around 0V, like most of the signals we use in this course. (Note the signs on the sources in the circuit above.) The schematic in Figure A-2 shows a standard ± V$_{CC}$ configuration for op-amps. The schematic symbols for a battery are used in this schematic to remind us that these supplies need to be a constant DC voltage. They are not signal sources.
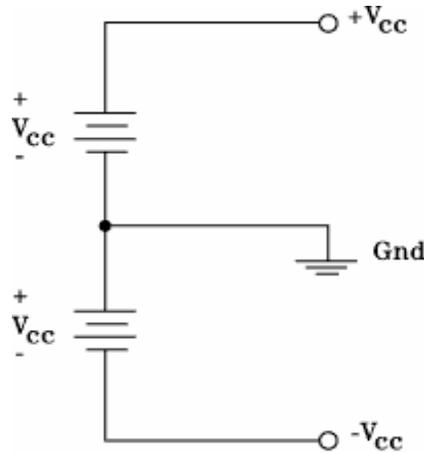

Figure A-2.

Students in JEC 4201 use two 9V batteries for power. Batteries are self explanatory.
Students in JEC 4107/4104 should use the HP E3631A supplies for power. The HP E3631A power supply provides two variable supplies with a common ground (for ±V$_{CC}$) plus a variable low voltage supply (not used in this lab). *The power supply jack labeled "COM" between the V$_{CC}$ supplies should be connected to circuit ground.* When you supply power to the op-amp, adjust the two voltage levels so that +V$_{CC}$ and -V$_{CC}$ are equal, but opposite in sign, at 9V. These are independent and adjusted separately. Note that in PSpice, there are two ways to represent a source with a negative sign. Figure A-3 shows the two options: you can either set the voltage source to a negative value, or you can reverse the polarity of the source.
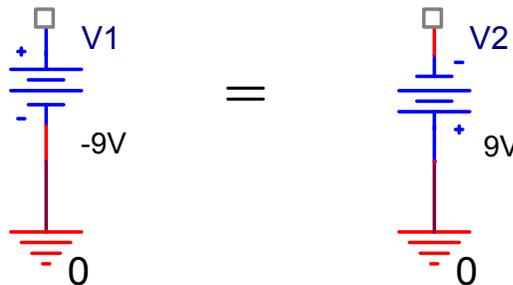

Figure A-3.

The op-amp chip: Study the chip layout of the LF351 and 741 op-amps is shown in Figure A-4. Both have the same pin numbers and names. Either op-amp can be used without changing the circuit. The standard procedure on DIP (dual in-line package) "chips" is to identify pin 1 with a notch in the end of the chip package. The notch always separates pin 1 from the last pin on the chip. In the case of the LF351, the notch is between pins 1 and 8. Pin 2 is the inverting input. Pin 3 is the non-inverting input, and the amplifier output, V$_O$, is at pin 6. These three pins are the three terminals that normally appear in an op-amp circuit schematic diagram. The +V$_{CC}$ and -V$_{CC}$ connections (7

and 4) MUST be completed for the op-amp to work, although they usually are omitted from simple circuit schematics to improve clarity.
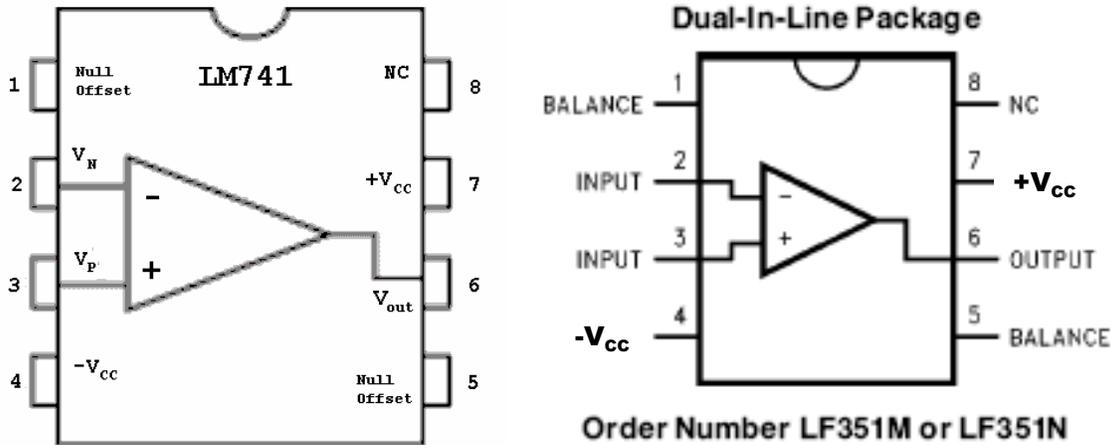

Figure A-4.

The balance (or null offset) pins (1 and 5) provide a way to eliminate any offset in the output voltage of the amplifier. The offset voltage (usually denoted by Vos) is an artifact of the integrated circuit. The offset voltage is additive with V_O (pin 6 in this case). It can be either positive or negative and is normally less than 10mV. Because the offset voltage is so small, in most cases we can ignore the contribution V_{OS} makes to V_O and we leave the null offset pins open. Pin 8, labeled "NC", has no connection to the internal circuitry of the 741 or LF351, and is not used.

Op-amp limitations: Just like all real circuit elements, op-amps have certain limitations which prevent them from performing optimally under all conditions. The one you are most likely to encounter in this class is called *saturation*. An op-amp becomes saturated if it tries to put out a voltage level beyond the range of the power source voltages, $\pm V_{CC}$. For example, if the gain tries to drive the output above 9V, the op-amp is not supplied with enough voltage to get it that high and the output will cut off at the most it can produce. This is never quite as high as 9V because of the losses inside the op-amp. Another common limitation is amount of current an op-amp can supply. Large demands for current by a low resistance load can interfere with the amount of current available for feedback, and result in less than ideal behavior. Also, because of the demands of the internal circuitry of the device, there is only so much current that can pass through the op-amp before it starts to overheat. A third limitation is called the *slew rate* and is the result of limit internal currents in the op-amp. Delays caused by the slew rate can prevent the op-amp circuit from displaying the expected output instantaneously after the input changes. The final caution we have about op-amps is that the equations for op-amps are derived using the assumption that an op-amp has infinite intrinsic (internal) gain, infinite input impedance, zero current at the inputs, and zero output impedance. Naturally these assumptions cannot be true, however, the design of real op-amps is close enough to the assumptions that circuit behavior is close to ideal over a large range.

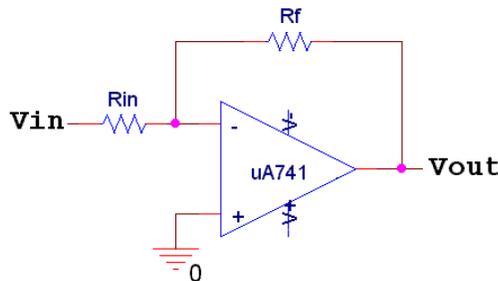*The inverting amplifier:* Figure A-5 shows an *inverting amplifier*.

Figure A-5.

Its behavior is governed by the following equation: $V_{out} = -\dfrac{Rf}{Rin}V_{in}$. The negative sign indicates that the circuit will invert the signal. (When you invert a signal, you switch its sign. This is equivalent to an180° phase shift of a sinusoidal signal.) The circuit will also amplify the input by Rf/Rin. Therefore, the total gain for this circuit is – (Rf/Rin). Note that most op-amp circuits invert the input signal because op-amps stabilize when the feedback is negative. Also note that even though the connections to V+ and V- (±V$_{CC}$) are not shown, they must be made in order for the circuit to function in both PSpice and on your protoboard.

The non-inverting amplifier: Figure A-6 shows a *non-inverting amplifier*. Its behavior is governed by the following equation: $V_{out} = \left(1 + \dfrac{Rf}{Rin}\right)V_{in}$.
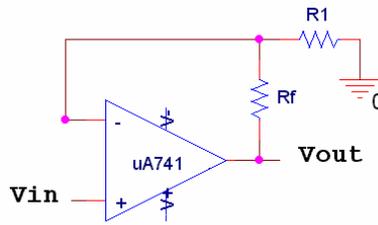

Figure A-6.

This circuit multiplies the input by 1+(Rf/Rin) and, unlike the previous op-amp circuit, the output is not an inversion of the input. The overall gain for this circuit is, therefore, 1+(Rf/Rin). The inverting amplifier is more commonly used than the non-inverting amplifier. That is why the somewhat odd term "non-inverting" is used to describe an amplifier that does not invert the input. If you look at the circuits, you will see that in the inverting op-amp, the chip is connected to ground, while in the non-inverting amplifier it is not. This generally makes the inverting amplifier behave better. When used as a DC amplifier, the inverting amp can be a poor choice, since its output voltage will be negative. However, for AC applications, inversion does not matter since sines and cosines are positive half the time and negative half the time anyway.

Experiment

The Inverting Amplifier
In this part of the experiment, we will wire a very simple op-amp circuit using PSpice and look at its behavior.

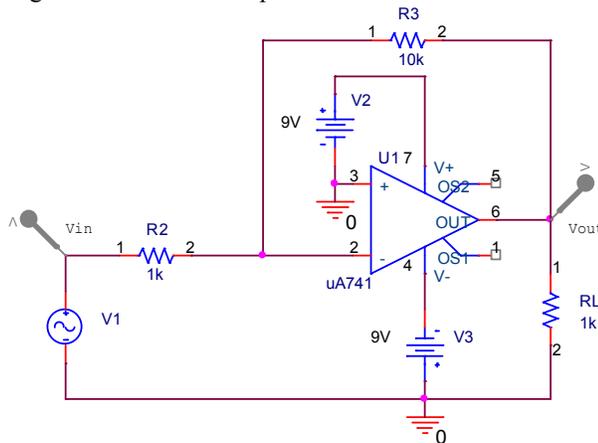- Wire the circuit shown in Figure A-7 below in PSpice.

Figure A-7.

- o The input should have 200mV amplitude, 1kHz and no DC offset.
- o The op-amp is called uA741 and is located in the "EVAL" library.

- o Be careful to make sure that the + and – inputs are not switched and that the two DC voltage supplies have opposite signs.
  - o Note the location of the input voltage, Vin. In this case, R2 is the input resistor, Rin, so the marker goes to its left.

- Run a transient simulation of this circuit that displays three cycles.
  - o What does the equation for this type of circuit predict for its behavior?
  - o Use the cursors to mark the amplitudes of the input and output of the circuit.
  - o Calculate the actual gain on the circuit. Is this close to the gain predicted by the equation?
  - o <u>Print out this plot and include it with your report.</u>

- Run a transient of the circuit with a much higher input amplitude.
  - o Change the amplitude of the source to 5V and rerun the simulation.
  - o What does the equation predict for the behavior this time? Does the circuit display the output as expected? What happened?
  - o Use the cursors to mark the maximum value of the input and output of the circuit.
  - o What is the magnitude of the output of the circuit at saturation?
  - o <u>Print out this plot and include it with your report.</u>

Build an Inverting Amplifier
In this part of the circuit, you will build an inverting amplifier. Build the circuit using the 741 or LF351 op-amp. Students in JEC 4201 use batteries to provide the +9V and –9V power sources.

- Build the inverting op-amp circuit in Figure A-7 on your protoboard.
  - o Don't neglect to wire the DC power voltages at pins 4 and 7. Do not connect pin 4 and 7 to ground. They go through the power supply to ground.
  - o For students in JEC 4107/4104, do not forget to set both the positive and negative values on the DC power supply. One does not automatically set when you set the other. Do not forget to attach the common ground for the power supply voltages to the ground for the circuit as a whole.

- Examine the behavior of your circuit.
  - o <u>Take a picture with the IOBoard software of the input and output of the circuit at 1kHz and 200mV amplitude and include it in your report.</u>
  - o What was the gain of your circuit at this amplitude and frequency? [Use the signals to calculate the gain, not the values of the resistors.]
  - o Saturation: Change $R_{feedback}$ to a 22kΩ resistor. Vary the amplitude of the function generator until the op-amp output starts to saturate. At about what input amplitude does this happen? What is the magnitude of the output of the circuit at saturation? How does this compare with the saturation voltage found using PSpice?

Summary

As long as one remains aware of some of their limitations, op-amp circuits can be used to perform many different mathematical operations. That is why collections of op-amp circuits have been used in the past to represent dynamic systems in what is called an analog computer. There are some very good pictures of analog computers and other computers through the ages at H.A. Layer's Mind Machine Web Museum. A link is located on the course links page.

Part B – Voltage Followers

Background

The voltage follower: The op-amp configuration in Figure B-1 is called a *voltage follower* or *buffer*. Note that the circuit above has no resistance in the feedback path. Its behavior is governed by the equation: $V_{out} = V_{in}$.
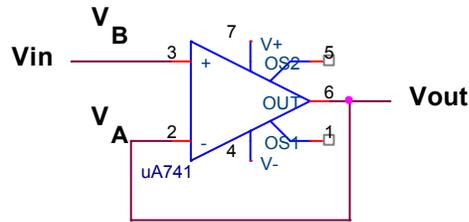
Figure B-1.

If one considers only the equation $V_{out} = V_{in}$, this circuit would appear to do nothing at all. In circuit design, however, voltage followers are very important and extremely useful. What they allow you to do is completely separate the influence of one part of a circuit from another part. The circuit supplying Vin will see the buffer as a very high impedance, and (as long as the impedance of the input circuit is not very, very high), the buffer will not load down the input. (This is similar to the minimal effect that measuring with the scope has on a circuit.) On the output side, the circuit sees the buffer as an ideal source with no internal resistance. The magnitude and frequency of this source is equal to Vin, but the power is supplied by $\pm V_{CC}$. The voltage follower is a configuration that can serve as an impedance matching device. For an ideal op-amp, the voltages at the two input terminals must be the same and no current can enter or leave either terminal. Thus, the input and output voltages are the same and $Z_{in} = V_{in}/I_{in} \rightarrow \infty$. In practice Z_{in} is very large which means that the voltage follower does not load down the source.

Experiment

A Voltage Follower Application
In this part, we will investigate the usefulness of a voltage follower using PSpice.

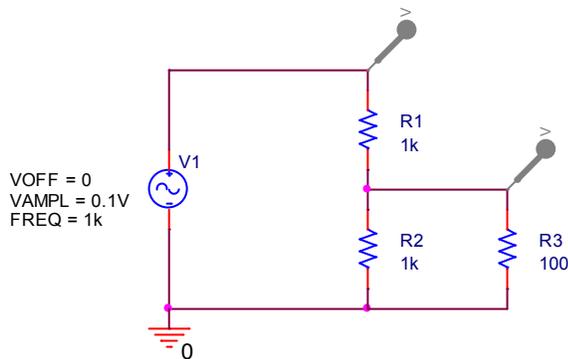- Begin by creating the circuit pictured in Figure B-2 below in PSpice.

Figure B-2.

  o The source has amplitude of 100mV and a frequency of 1kHz.
  o The impedance of the function generator is assumed to be negligible and has been left out.
  o R1 and R2 are a voltage divider and R3 is the load on the voltage divider.

- Run a simulation that displays three cycles of the input.
  o Run the simulation, mark the amplitude of the voltages shown, and print the plot for your report.
  o If we combine R3 and R4 in parallel, we can demonstrate that the amplitude of the output is correct for this circuit.
  o What if our intention when we built this circuit was to have the input to the 100Ω resistor be the output of the voltage divider? i.e. We want the voltage across the load (R4) to be ½ of the input voltage. Clearly the relationship between the magnitudes of the 100Ω resistor and the 1kΩ resistor in the voltage divider will not let this occur. A voltage follower is needed.

- Modify the circuit you created by adding an op-amp voltage follower between R3 and R4, as shown in Figure B-3 on the next page:
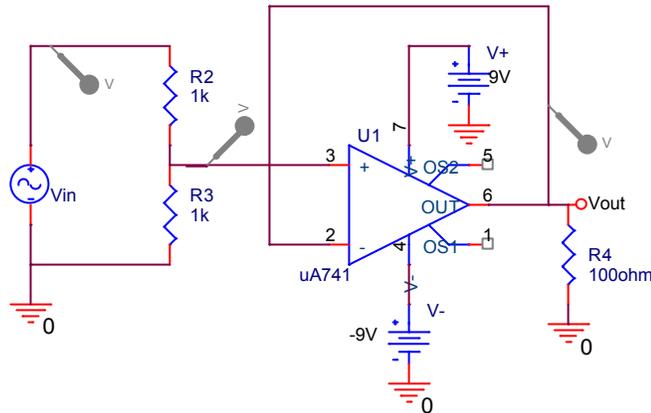

Figure B-3.

- o The op-amp is called uA741 and is located in the "EVAL" library.
- o Be careful to make sure that the + and – inputs are not switched and that the two DC voltage supplies have opposite signs.

- Rerun the simulation
  - o Place voltage markers at the three locations shown.
  - o Rerun the simulation, mark the amplitude of the voltages shown, and print the plot for your report.
  - o What is the voltage across the 100Ω load now? Have we solved our problem?
  - o The voltage follower has isolated the voltage divider electrically from the load, while transferring the voltage at the center of the voltage divider to the load. Because every piece of a real circuit tends to influence every other piece, voltage followers can be very handy for eliminating these interactions when they adversely affect the intended behavior of our circuits.
  - o It is said that the voltage follower is used to isolate a signal source from a load. From your results, can you explain what that means?

- Voltage followers are not perfect. They are not able to work properly under all conditions.
  - o To see this, change R4 to 1Ω.
  - o Rerun the simulation, mark the amplitude of the voltages shown, and print the plot for your report.
  - o What do you observe now? Can you explain it? Refer to the spec sheet for the 741 op-amp on the links page. How have we changed the current through the chip by adding a smaller load resistance?

- Finally, it was noted above that the input impedance of the voltage follower should be very large. Determine the input impedance by finding the ratio of the input voltage to the input current for the follower.
  - o Return the value of R4 back to the original 100Ω.
  - o Recall that R=V/I. We can obtain the voltage we need by placing a voltage marker at the non-inverting input (U1:+) of the op-amp.
  - o PSpice will not allow us it place a current marker at the positive op-amp input. We can find the current anyway by finding the difference between the current through R2 and R3. Place a current marker on R2 and another on R3.
  - o Set up an AC sweep for the circuit from 1 to 100kHz.
  - o From your AC sweep results, add a trace of V(U1:+)/(I(R2)-I(R3)). (Note that your voltage divider resistors might have different names if you placed them on the schematic in a different order.) Include this plot in your report.
  - o What is the input impedance of the op-amp in the voltage follower at low frequencies? (Since PSpice tries to be as realistic as possible, you should get a large but not infinite number.)
  - o Run the sweep again from 100kHz to 100MegHz. Is the input impedance still high at very high frequencies? (Note M is mega and m is milli in PSPice voltage displays.)

Summary

The voltage follower is one of the most useful applications of an op-amp. It allows us to isolate a part of a circuit from the rest of the circuit. Circuits are typically designed as a series of blocks, each with a different function. The output of one block becomes the input to the next block. Sometimes the influence of other blocks in a circuit prevents one block from operating in the way we intended. Adding a buffer can alleviate this problem.

Part C – Integrators and Differentiators

Background

Ideal differentiator: Figure C-1 shows an *ideal differentiator*. Its behavior is governed by the following equation:

$$V_{out} = -RfCin\left(\frac{dVin}{dt}\right).$$

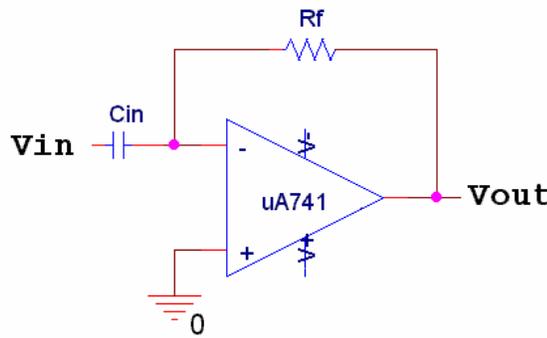**Figure C-1.**

The output of this circuit is the derivative of the input INVERTED and amplified by Rf×Cin. For a sinusoidal input, the magnitude of the gain for this circuit depends on the values of the components and also the input frequency. It is equal to (ω×Rf×Cin). The circuit will also cause a phase shift of -90°. It is important to remember that there is an inversion in this circuit. For instance, if the input is sin(t), then you would expect the output of a differentiator to be +cos(t) (a +90° phase shift). However, because of the inversion, the output phase of this circuit is -90° (+90 - 180). Also note that, because one cannot build a circuit with no input resistance, there is no such thing as an ideal differentiator. A real differentiator differentiates only at certain frequencies. This distinction is discussed in the power point notes for the course.

Ideal integrator: The circuit shown below in Figure C-2 is an *ideal integrating amplifier*. Its behavior is governed by the following equation: $V_{out} = -\dfrac{1}{Rin\,Cf} \int Vin\,dt$.
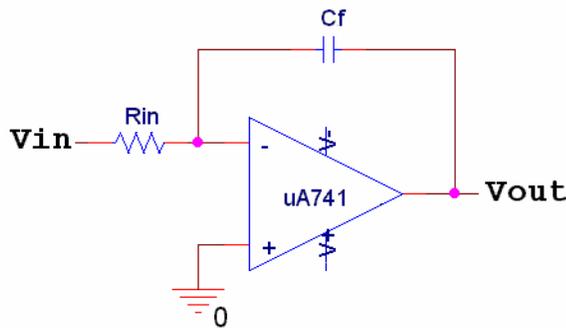

Figure C-2.

The output of this circuit is the integral of the input INVERTED and amplified by 1/(Rin×Cf). For a sinusoidal input, the magnitude of the gain for this circuit depends on the values of the components and also the input frequency. It is equal to 1/(ω×Rin×Cf). The circuit will also cause a phase shift of +90°. It is important to remember that there is an inversion in this circuit. For instance, if the input is sin(t), then you would expect the output of an integrator to be -cos(t), a -90° phase shift. However, because of the inversion, the output phase shift of this circuit is +90° (-90 + 180). Also, because the integration of a constant DC offset is a ramp signal and there is no such thing as a real circuit with no DC offset (no matter how small), wiring an ideal integrator will result in an essentially useless circuit. A Miller integrator is an ideal integrator with an additional resistor added in parallel with Cf. It will integrate only at certain frequencies. This distinction is discussed in the power point notes for the course.

Experiment

Using an Op-amp Circuit to Integrate an AC Signal in PSpice
In this section, we will observe the operation of a Miller integrator on a sinusoid. You will examine the way in which the properties of the integrator change both the amplitude and the phase of the input.

- Build the integration circuit shown below in Figure C-3. V1 should have a 200mV amplitude and 1kHz frequency.
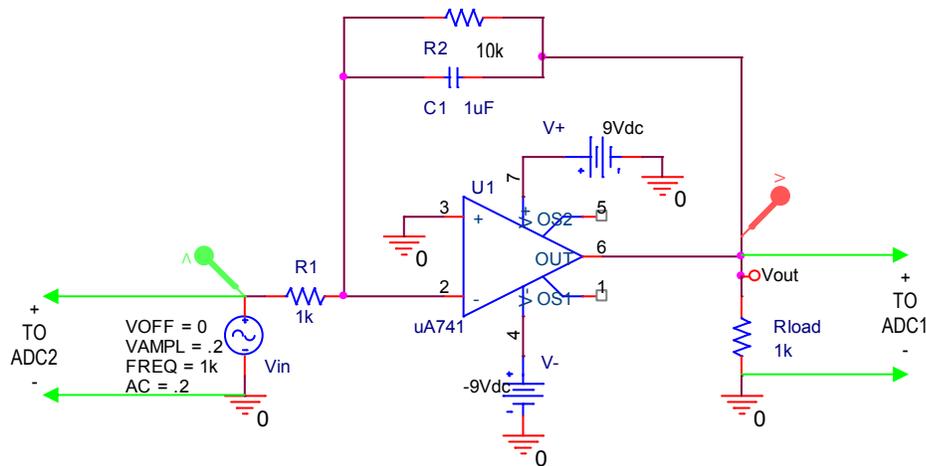

Figure C-3.

- Run a transient analysis.
  - We want to set up the transient to show five cycles, but we also want to display the output starting after the circuit has reached its steady state. Set the run time to 15ms, the start time to 10ms, and the step size to 5us.
  - Obtain a plot of your results. Just like in mathematical integration, integrators can add a DC offset to the result. Adjust your output so that it is centered around zero by adding a trace that adds or subtracts the appropriate DC value. After you have done this, mark the amplitude of your input and output with the cursors.
  - <u>Print this plot and include it in your report.</u>

- Use the equations for the ideal integrator to verify that the circuit is behaving correctly.
  - The equation that governs the behavior of this integrator at high frequencies is given by:

$$if \quad \omega_c >> \frac{1}{R_2 C_1} \quad then \quad v_{out}(t) \approx -\frac{1}{R_1 C_1}\int v_{in}(t)dt$$

  - Recall that the integration of sin(ωt) = (-1/ω)cos(ωt). Therefore, the circuit attenuates the integration of the input by a constant equal to -1/ωR1C2. The negative sign means that the output should also be inverted.
  - What is there about the transient response that tells you that the circuit is working correctly? Is the phase as expected? The amplitude? Above what frequencies should we expect this kind of behavior?

- Now we can look at the behavior of the circuit for all frequencies.

- o   Do an AC sweep from 100m to 100kHz.
- o   Add a second plot and plot the phase of the voltage at Vout. (Either add a p(Vout) trace or add a phase marker at the output of the circuit.) What should the value of the phase be (approximately) if the circuit is working more-or-less like an integrator? Mark the region on the plot where the phase is within ±2° of the expected value.
- o   Print this plot
- o   You will mark this sweep with the data from the circuit that you build.

- We can also use PSpice to check the magnitude to see when this circuit acts best as an integrator.
  - o   Rerun the sweep. Do not add the phase this time.
  - o   Using the equation above, we know that at frequencies above fc, $V_{out} = -V_{in}/(\omega RC)$, where R = R1, C = C1, and $\omega = 2\pi f$. [We plot the negation of the input because the equation for the transfer function of the circuit has an inversion. In a sweep, only the amplitude matters, so the sign is not important.]
  - o   Change the plot for the AC sweep of the voltage to show just V_{out} and $-V_{in}/(\omega R1C1)$. Note that you need to input the frequency ω as 2*pi*Frequency in your PSpice plot. (Pspice recognizes the word "pi" as the value of π and the word "Frequency" as the current input frequency to the circuit. Also note that you must enter numbers for R1 and C1.)
  - o   When are these two signals approximately equal? It is at these frequencies that the circuit is acting like an integrator. Mark the point at which the two traces are within 100mV of each other.
  - o   Calculate fc=1/(2πR2C1). How close are the amplitudes of the two signals at that frequency? At a frequency much greater than fc, the circuit should start behaving like an integrator. Mark the corner frequency on your plot.
  - o   Print this plot.

Using an Op-amp Integrator to Integrate a DC Signal
Another way to demonstrate that integration can be accomplished with this circuit is to replace the AC source with a DC source and a switch.
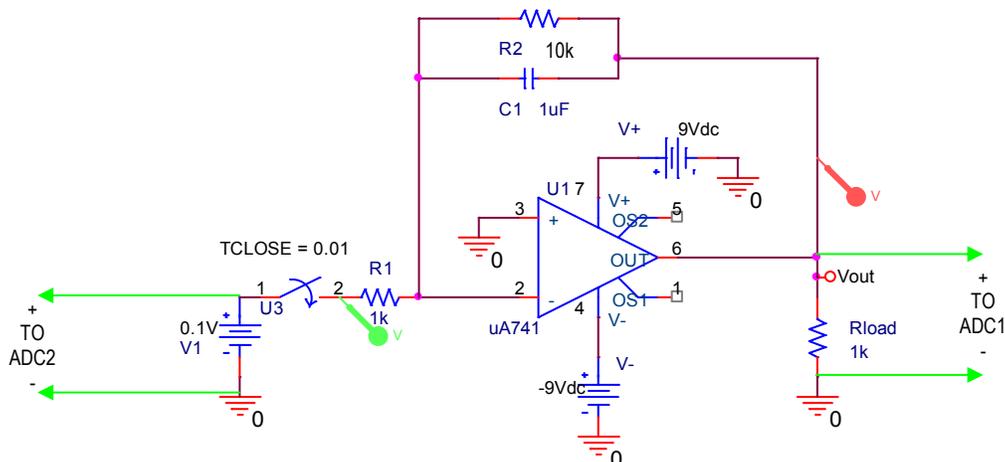

Figure C-4.

- Modify your circuit by replacing the AC source with a DC source and a switch as shown in Figure C-4.
  - o   Note that the switch is set to close at time t=0.01sec. Use a voltage of 0.1V to avoid saturation problems.
  - o   The switch is called Sw_tClose and is in the EVAL library.

- Analyze the circuit with PSpice.
  - o   Do a transient analysis for times from 0 to 150ms with a step of 10us.
  - o   Rather than plotting the output voltage (voltage at Vout), plot the negative of the output voltage. You should see that this circuit does seem to integrate reasonably well.
  - o   Print this plot.

- o How close is the output of your circuit to an integration of the input? The integration of a constant should be a ramp signal of slope equal to the constant. The output of an integrating op-amp circuit should be the inversion of the ramp signal multiplied by a constant equal to (1/(R1C1)).
- o Calculate the approximate slope of the output. Write it on your output plot. Also write the theoretical slope on the plot. For what range of times does it integrate reasonably well? (This is somewhat subjective.)

- • Modify the feedback capacitor
  - o Decrease C2 to 0.01μF and repeat the simulation. Only run it from 0 to 14ms this time. Don't forget to plot the negative of the output voltage.
  - o <u>Print your output.</u>
  - o Mark the theoretical slope on the plot. Calculate the theoretical slope of the output. Don't forget that the constant, 1/(R1C2), is different because C2 has changed.
  - o Does the circuit integrate -- even approximately -- for any period of time? Can you think of any reason why we might prefer to use a smaller capacitor in the feedback loop, even though the circuit does not integrate as well over as long a period of time?

- • Create an ideal integrator
  - o The circuit we have been looking at is a Miller integrator. An ideal integrator does not have an extra resistor in the feedback path. What would happen if we changed our circuit to an ideal integrator?
  - o Set the feedback capacitor back to its initial value of 1uF. Remove the resistor from the feedback loop and run your transient analysis again.
  - o You should see that the circuit no longer works. Negate the output voltage again.
  - o <u>Print your output.</u>
  - o What is wrong with the output? The ideal integrator circuit will operate on both the AC and DC inputs. In any real circuit -- no matter how good your equipment is -- noise will create a small variable DC offset voltage at the inputs. The problem with this circuit is that there is no DC feedback to keep the DC offset at the input from being integrated. Therefore, the output voltage will continuously increase and, in addition, it will be amplified by the full intrinsic gain of the op-amp. This immediately saturates the op-amp.

Building an Op-amp Integrator and an Op-amp Differentiator
In this part of the experiment, we will build an op-amp integrator and an op-amp differentiator on the protoboard and look at the output for a variety of inputs.

- • Build the op-amp integrator circuit as shown in Figure C-3.

- • Observe the behavior of the circuit at three representative frequencies.
  - o Use the sine wave from the function generator for the voltage source, set the amplitude to 0.2V ($0.4V_{P-P}$).
  - o Obtain measurements of the input and output voltages at frequencies of 500Hz, 1kHz, and 5kHz. <u>Add your experimental points</u> for both the amplitude and phase to your PSpice AC sweep plot for the above circuit.
  - o <u>Obtain a picture of each of these signals with the Mobile Studio software.</u>

- • Observe the output of the integrator for different types of inputs
  - o Set the function generator to a frequency that gives reasonable signal amplitude and integrates fairly well. This is somewhat subjective; we just want you to see the shapes of the outputs for different input wave shapes.
  - o Set the function generator to the following types of inputs:
    - • sine wave
    - • triangular wave
    - • square wave
  - o What should the integration of each of these types of inputs be?
  - o <u>Take a picture of the output for each input with the Mobile Studio software.</u>
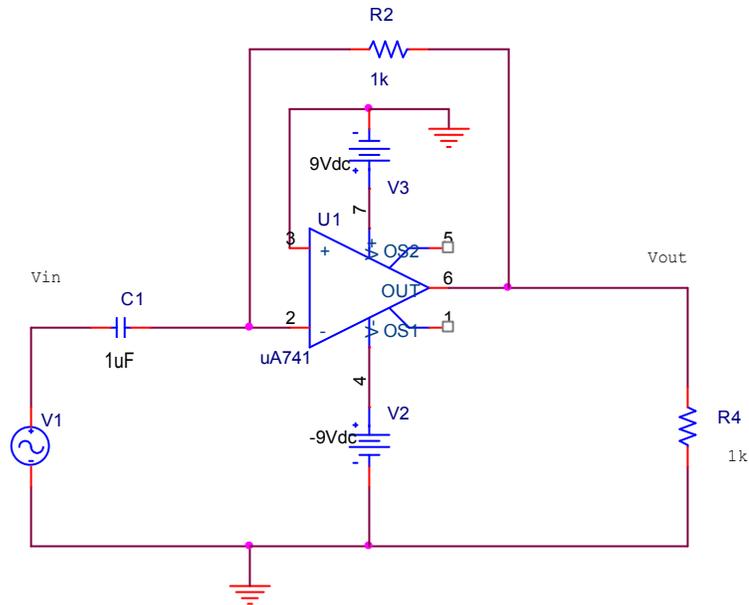
- • Create a differentiator.

Figure C-5.

- o   Remove the feedback capacitor, C2. Replace R1 with an input capacitor, C1=1µF. Replace the 10k feedback resistor with a 1k resistor, R2. Your circuit should now look like Figure C-5.

- •   Set the function generator to a frequency that gives a reasonable signal amplitude and differentiates fairly well. This is somewhat subjective; we just want you to see the shapes of the outputs for different input wave shapes.

- •   Observe the output of the differentiator for different types of inputs.
  - o   Set the function generator to the following types of inputs:
    - •   sine wave
    - •   triangular wave
    - •   square wave
  - o   What should the differentiation of each of these types of inputs be?
  - o   Take a picture of each situation with the Mobile Studio software.

Summary

Op-amp circuits can be used to do both integration and differentiation. The ideal versions of both circuits are not realizable. Therefore, the real versions of these circuits do not work well at all frequencies. Also, as both types of circuits approach optimal mathematical performance, the amplitude of the output decreases. This makes designing an integrator or a differentiator a trade-off between the desired mathematical operation and signal strength.

Part D – Using Op-Amps to Add and Subtract Signals

Background

Op-amp adders: Figure D-1 below shows an *adder*. Its behavior is governed by the following equation:

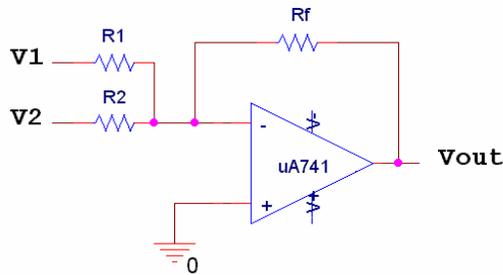$$V_{out} = -Rf\left(\frac{V1}{R1} + \frac{V2}{R2}\right).$$

Figure D-1.

The gain for each input to the adder depends upon the ratio of the feedback resistance of the circuit to the value of the resistor at that input. The adder is sometimes called a weighted adder because it provides a means of multiplying each of the inputs by a separate constant before adding them all together. It can be used to add any number of inputs and multiply each input by a different constant. This makes it useful in applications like audio mixers.

The differential amplifier: The circuit in Figure D-2 is a *differential amplifier,* also called a *difference amplifier.* Its behavior is governed by the following equation: $V_{out} = \dfrac{Rf}{Rin}\,(V1 - V2)$.
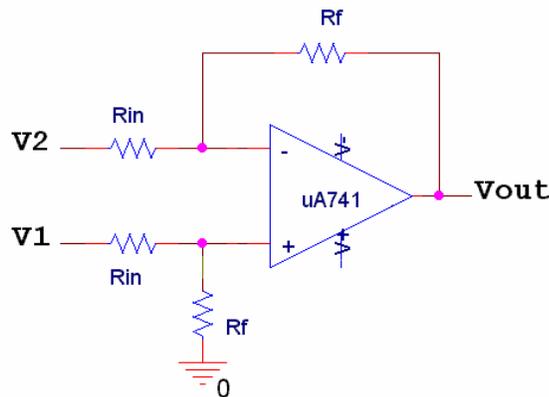

Figure D-2.

It amplifies the difference between the two input voltages by Rf/Rin, which is the overall gain for the circuit. Note that the ability of this amplifier to effectively take the difference between two signals depends on the fact that it uses two pairs of identical resistances. Also note that the signal that is subtracted goes into the negative input to the op-amp. Be careful with the term "differential". In spite of its similarity to the term "differentiation", the differential amplifier does not differentiate its input.

Amplifying the output of a bridge circuit: You may recall from Experiment 4, that it was difficult to measure the AC voltage across the output of the bridge circuit because both of the output connections had a finite DC voltage. Without a special probe, the black leads of the scope are always attached to ground. That meant that one could not just connect one of the scope channels across the output, since the scope would short one of the voltages to ground. The differential amplifier allows us to get by this problem, since neither input is grounded. A very large fraction of measurement circuits use some kind of a bridge configuration or are based on some kind of comparison between two voltages. Thus, the operation of the differential amplifier is very important to understand for project 2.

Experiment

PSpice Simulation of an Adder

- Set up the circuit shown in Figure D-3 in PSpice.
    - Note the polarity of the voltage sources providing ±Vcc to the op-amp.
    - Note that the amplitude of the source, V3, is 2V and that the amplitude of the source, V4, is 1V.

- Run a simulation that shows 3 cycles of the input.
  - What is the gain of the adder? What should it do to the two input signals? Is the adder working correctly?
  - Include this simulation with your report.

- Adders are often used as mixers that give different emphasis to each input signal and then combine the inputs together into one signal. What would we have to set R2 to, if we wanted twice as much of the signal from V3 to pass through the adder as the signal from V4. [Note: This does not mean changing nothing because V3 already has twice the amplitude as V4. It means mixing in twice as much of the amplitude of V3 as the amplitude of V4 into the final output signal.]
  - Modify resistor R2, rerun the simulation, and verify that the output of the signal is as expected.
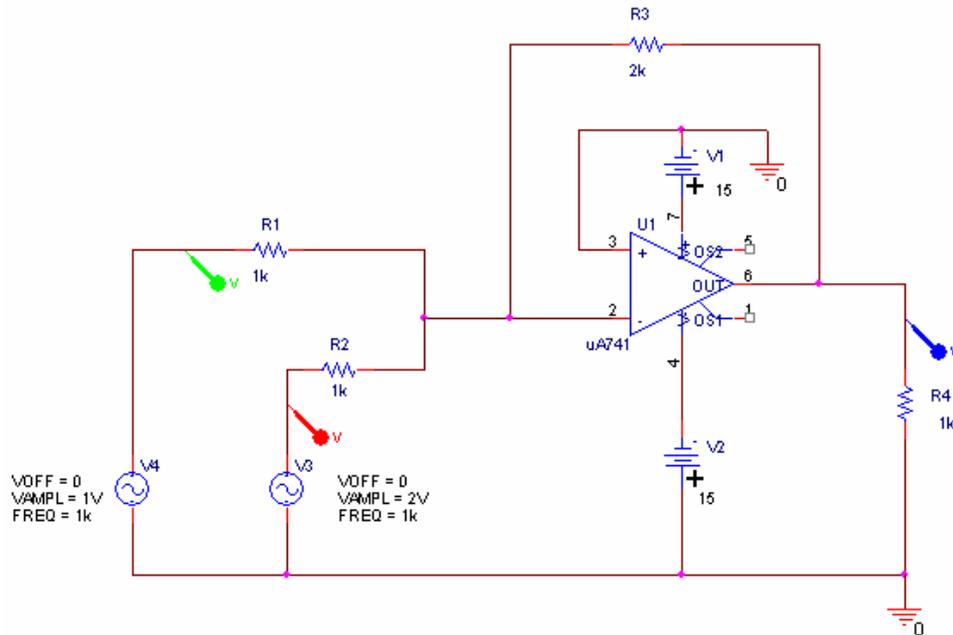 - Include the output of the simulation in your report.

Figure D-3.

Summary

In this experiment we used an adder to add two signals. Then, we modified it so that it would combine the signals with different emphasis, as in an audio mixer.

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 (14 points)

Include the following plots:
1. PSpice transient of inverting amplifier with input amplitude of 200mV and both traces marked. (2 pt)
2. PSpice transient of inverting amplifier with input amplitude of 5V and both traces marked. (2 pt)
3. IOBoard picture of inverting amplifier circuit. (2 pt)

Answer the following questions:
1. What is the theoretical gain of your inverting amplifier? What gain did you find with PSpice when the input amplitude was 200mV? How close are these? (2 pt)
2. What was the actual gain you got for the inverting amplifier you built? How did this compare to the theoretical gain? How did this compare to the PSpice gain? (2 pt)
3. What value did you get for the saturation voltage of the 741 op-amp in PSpice? What value did you get for the saturation voltage of the real op-amp in your circuit? How do they compare? (2 pt)
4. At what input voltage did the op-amp in the amplifier you built on the protoboard begin to saturate? (2 pt)

Part B (10 points)

Include the following plots:
1. PSpice transient of the voltage divider with 100Ω load and no voltage follower. (1 pt)
2. PSpice transient of the voltage divider with 100Ω load and a voltage follower. (1 pt)
3. PSpice transient of the voltage divider with 1Ω load and a voltage follower. (1 pt)
4. PSpice AC sweep of the input impedance for the voltage follower. (2 pt)

Answer the following questions:
1. Compare the transients of the output with and without the buffer circuit in place. What is the function of the buffer circuit? (2 pt)
2. Why is the follower unable to work properly with a small load resistor? (1 pt)
3. What is the typical value of the input impedance of the voltage follower when it is working properly at low frequencies? (1 pt)
4. Is the magnitude of the input impedance of the voltage follower high enough at high frequencies for it to work effectively? (1 pt)

Part C (38 points)

Include the following plots:
1. PSpice transient plot of the integrator. (1 pt)
2. AC sweep of amplitude (with three experimental points marked) and phase (with three experimental points marked.) The frequency at which the phase gets close to ideal should also be marked. (3 pt)
2. AC sweep plot of the integrator voltage and -Vin/ωRC with the location of fc and the place where the voltage gets close to ideal indicated. (2 pt)
3. PSpice plots of the integrator with DC source with slope and theoretical slope (if any) indicated on plot. One should be when C2=1uF and the other for C2=0.01uF (2 plots) (2 pt)
4. PSpice plot of the ideal integrator (without feedback resistor) (1 pt)
5. IOBoard pictures of your circuit trace (input vs. output) at 500Hz, 1kHz and 5kHz. (3 plots) (3 pt)
6. IOBoard pictures of your integrator output with sine wave, triangular wave and square wave inputs (input vs. output) (3 plots) (3 pt)
7. IOBoard picture of your differentiator output with sine wave, triangular wave and square wave inputs (input vs. output) (3 plots) (3 pt)

Answer the following questions:
1. Using the rules for analyzing circuits with op-amps, derive the relationship between V_{out} and V_{in} for the integrator circuit. (3 pt)
2. Why is the integrator also called a low-pass filter? Take the limits of the transfer function at high and low frequencies to demonstrate this. (3 pt)
3. What are the features of the AC sweep and transient analysis of an integrator that show it is working more-or-less as expected according to the transfer function? For about what range of frequencies does it act like an inverting amplifier? For about what range of frequencies does it act like an integrator? (3 pt)
4. Consider the phase shift and the change in amplitude of the output in relation to the input when the circuit is behaving like an integrator. Use the expected change in phase and amplitude (from the ideal equation) to demonstrate that the circuit is actually integrating. (3 pt)
5. Why would we prefer to use the 0.01uF capacitor in the feedback loop even though the circuit does not integrate quite as well over as large a range? (1 pt)

6.   What happens when we try to use an ideal integrator? (1 pt)
7.   In the hardware implementation, you used a square-wave input to demonstrate that the integrator was working approximately correctly. If it were a perfect integrator, what would the output waveform look like? Is it close? (3 pt)
8.   When we built the differentiator, what did the output waveform look like for the square-wave input? What did the differentiator circuit output look like for a triangular wave input? If it were a perfect differentiator, what would the output waveform look like? Is it close? (3 pt)

Part D (10 points)

Include the following plots:
1.   Transient simulation of the output of the adder with both input resistors set to 1k. (1 pt)
2.   Transient simulation from PSpice with R2 modified. (1 pt)

Answer the following questions:
1.   Demonstrate that the original adder circuit (plot D-1) works as expected. (3 pt)
2.   Demonstrate that the modified adder circuit (plot D-2) works as expected. (3 pt).
3.   Give an example of a system (electrical, mechanical, chemical or some combination) with negative feedback and an example of a system with positive feedback. (2 pt)

Overall (8 points)

1.   Material should be in logical order, easy to follow and complete. (6pt)
2.   List member responsibilities. (2 pt)

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.