Project 4
Optical Communications Link

In this project you will build a transmitter and a receiver circuit. The transmitter circuit uses pulse frequency modulation to create a series of light pulses that encode an audio signal. The receiver takes in the light pulses, demodulates them, amplifies the signal back to the level the audio amplifier needs to see, and sends it through an audio amplifier to a speaker where it can be heard. Your initial design will reconstruct the input to create an audible signal of poor quality. In the final design, you will modify the circuit to create a much better quality demodulated audio signal.

Figure 1 shows an input signal (top) and its pulse frequency modulated equivalent. Simply averaging this signal will demodulate it to some extent. The initial design for the receiver circuit provides this functionality more-or-less by accident. If you enhance the time-averaging ability of the circuit with the addition of an integrator, you will be able to improve the reconstruction of the audio signal. The integration, however, will attenuate the signal level somewhat. To make up for this, you will have to adjust the gain of the circuit after the integrator is added. You will also be asked to add a smoothing capacitor to the circuit to further improve your output signal. The modified circuit is to be tested to be sure that it is an improvement over the initial design.

Another part of this project is to determine what each block of the circuit you are building does. Figure 2 shows a block diagram for the circuit. The piece of the circuit between each of the marked points is a sub-circuit that performs some function. You have encountered most of these smaller circuits sometime during the semester. (The one key exception is the audio amplifier, for which information is contained in this handout.) You should be able to identify the function of each block shown in Figure 2 and show that the block is functioning as expected by comparing the signal before the block to the signal after it.

Please note: The Analog Discovery Board is very likely up to the task, but it may not sample fast enough to measure all the details of the output signal of the 555 timer. This is a typical engineering challenge – you don’t have all the information that you desire. The traces will be what they are and you will have to use your understanding of the circuits and components to be able to debug the project.
Part A - Initial Design

In the initial design of this circuit you will build a transmitter and a receiver. You will also use a PSpice model of the circuit and the optical transmission and reception process to better understand how this all works together.

Transmitter Circuit
The initial transmitter circuit is pictured in Figure 3.

Receiver Circuit
The initial receiver circuit is shown in Figure 4. The 100Ω resistor protects the speaker from too much power.
In this circuit, the input is the light pulses detected by the phototransistor. The output is a demodulated and amplified signal heard on a speaker. Note the locations of the points D, E, F, G and H. You will be measuring the voltage signals at these points and defining the circuit blocks between them. The 10K pot is marked 103.

The audio amplifier
Your circuit contains an audio amplifier. Details on this block of the circuit are contained in the spec sheet for the 386 amplifier. The image in Figure 5 is taken from that source.

PSpice Model
You are provided with a PSpice circuit that models the function of the initial transmitter and receiver circuit combination. It is available on the links page under project 4. It is not necessary to enter it by hand. You should generate PSpice output showing the function of each circuit block and also the function of the circuit as a whole. Use the circuit diagrams above to build the hardware.

Figure 5.

Figure 6.
The model: Figure 6 shows the different pieces of the model. An audio source is approximated using a small sine wave at 2kHz and a square wave at 500Hz. This helps demonstrate how the circuit works. You may want to run the simulation with only one source on at a time. The transmitter circuit is separated from the receiver circuit with a buffer. (In the circuit that you build in hardware, this separation will be the air through which the light pulses are transmitted. In the simulation, we don’t have light or air, but we can at least isolate the two circuits electronically with a buffer.) There is also a pot that creates the difference in amplitude between the pulses generated by the transmitter and the pulses picked up by the phototransistor. The audio amplifier in the receiver is modeled with a different amplifier (E2) than the 386 used in the real circuit. (PSpice does not have a 386 model). In order to keep the amplitude of the output at the same order of magnitude as the input, the gain on this amplifier (1) is 1/10 of the gain on the 386 without the optional capacitor. The picture also shows the locations you will use to add the circuit elements you need for your final design.

Run the simulation: Start by running the simulation provided. Remember that the initial design should give you a poor quality reproduction of your input signal. Therefore, you should expect the model of the initial design to give you a poor quality reconstruction of the signal at the output. The input (point A) and output (point H) of the simulation should look something like Figure 7. The figure below shows 2ms of the traces. The input trace has one cycle of the lower frequency square wave (at 500Hz) with four cycles of the higher frequency wave (at 2kHz) superimposed on it. As you can see, the output shows some of the features of the input, but it is dominated by the high frequency samples taken by the 555 timer oscillating between about minus 5V and plus 5V. Note that the average of the signal when the input is positive is less than the average when the input is negative. It is the average that carries most of the sampled information from the input.

Determine your sampling frequency: The sampling frequency in the PSpice model is representative of a good sampling frequency for your circuit. It is hard to determine exactly what this frequency is by looking at the pulses because the frequency of the pulses changes with the input signal. You can get a reasonable average by averaging over several samples in Figure 7 or by calculating the frequency of the 555 timer circuit using the astable mode equations and the values of the components in the R-R-C combination (10kΩ-27kΩ-0.001µF). Be sure that the PSpice frequency and the actual sampling frequency of your transmitter circuit match reasonably well when you take your data and design your integrator. In the actual circuit, you can match the frequencies either by turning the 100K pot, replacing the 100K pot in the circuit with a 10k resistor, or by changing resistor R3 in the PSpice model.

Examine the different blocks: Once you have the simulation running, look at the output of each block separately. Also use the magnifying glass and look more closely at the wave shapes. (For example, look at the signals at point A and point B only. Which one has the DC offset and why?) This will give you an idea of what the block is doing and also what to expect from the circuit you build in hardware. Also, it is generally easier to identify the function of each block by observing what happens to the signals from each of the two sources separately. You can do this by
setting one of the voltages to zero, but leaving the source in the circuit for future use. The square wave input is especially helpful in demonstrating that it is the average of the sampled output that matters.

Note: The next task is to build the transmitter and receiver in hardware. Do not use the PSpice diagrams to build your hardware circuits. Use your hand-drawn diagrams.

Building and Testing the Circuits
Build the two circuits as shown in the circuit diagrams. Leave out the optional 10µF capacitor to start. If the signal sounds very soft, you might try adding the 10µF capacitor, but it should work without it. Turn up the volume. If the audio output sounds distorted, you should remove the capacitor. Test your circuit using a square wave signal from the function generator representative of an audio wave. You may want to also try a sine waveform, and try changing the signal frequency. When the circuit works, have a staff member sign off on it and then take your data.

Test input signal: The transmitter is a 555 timer circuit very similar to an astable multivibrator. The only difference is that instead of generating a regular string of pulses, it generates a string of pulses which vary in frequency in response to the input signal at pin 5. It is best to use a square wave to debug this circuit. A typical audio signal is pictured in Figure 8. Note that the signal below has several frequencies. There are about 10 cycles of a typical wave in a 5ms division. This corresponds to a frequency of about 2kHz. Also note that the amplitude reaches a maximum of about 800mVp-p. Our circuit works best with a slightly larger amplitude, so set the function generator to 0.6V, sine or square wave at 1.5kHz.

![Figure 8.](image)

No common ground: Build the transmitter and receiver on separate boards. The two circuits should generally NOT have a common ground. They should be completely separate and the transmission of the light pulses should be the only interaction. When recording signals using the Analog Discovery Board, you will on occasion, need to tie both grounds together.

Orientation of the phototransistor: The phototransistor has the flat side near the collector lead. The collector goes toward the resistor; the emitter is connected to ground.

Verify that the transmitter is working: You should make sure that the pulses from your circuit are modulated by observing the voltage at the input (point A) and the voltage at the output (point C). The output should look like a pulse modulated signal. (If the input is low, the pulses should get closer together and if the input is high, the pulses should get farther apart.) As you turn the pot, the frequency of your output pulse modulated signal should change.

Set your sampling frequency: CD quality sound uses 44.1k samples per second to recreate an audio signal. We will use a somewhat lower frequency so the sampling effects will be clearer. Each pulse in your modulated signal is like a digital sample. A high sampling rate is desirable if you want to get high quality sound from your circuit. The sampling rate should also be higher than the range of audible frequencies so that you cannot hear any artifacts from the sampling. To vary your sampling rate, turn the 100K pot. To determine your sampling frequency, find the frequency of the pulses using the scope. You will not be able to get an exact measure of the sampling frequency of your circuit because it changes. However, you can get an estimate by averaging over several cycles. Since you will
be comparing your data to the PSpice model (and using it to design your integrator), it is important that the sampling frequencies for your actual circuit and your PSpice circuit match fairly well. Either try to match your circuit to the PSpice (by using the pot or replacing it with a 10k resistor) or change the sampling rate of the PSpice circuit by altering the value of R3.

A working receiver: The receiver circuit is more complicated than the transmitter circuit. If yours does not work, try debugging it in pieces. Check to see that the phototransistor is generating a set of pulses that correspond to the pulses from the LED. Check to see that the inverting amplifier is making the pulses bigger. Make sure the volume pot alters the amplitude of the signal. Finally, check to see if the audio amplifier amplifies the signal again. If you identify a block that does not work, debug it before you continue on to the next one.

Your audio signal: You will need to demonstrate that your circuit works using the function generator and an audio signal. Your laptops have audio output jacks. You can bring your own music or find some on the internet. If you prefer, you can use your portable audio device. Probably the most flexible option is to use the Analog Discovery Wavegen in Player mode as your source, because it allows you to easily set the amplitude level. If you choose this option, read the instruction file on the course website (http://www.ecse.rpi.edu/courses/S14/ENGR-2300/Wave%20Files%20for%20Analog%20Discovery.pdf). Make sure that you use the scope to check the amplitude of your input signal. Set the volume of your signal so that it corresponds roughly to the 400mV amplitude of the test input. If you cannot turn the volume up that high, you may need to use another input device. Do not let your input amplitude exceed 2Vp-p. This will interfere with the 555 timer’s ability to sample the input effectively. There are 1/8” mini stereo plugs for this use in the studio. The bare uninsulated wire is the ground connection and the red (right) or white (left) wires are the stereo output voltage signals.

Signature: When your circuit works, have a staff member listen to the circuit with the sine wave input and with the audio input and sign the cover/signature page at the end of this handout. If available, replace your LED with a bright white LED and observe the differences in system performance.

Taking your data
Take data showing the input to and output from each block of the circuit and also the overall function of the circuit. All signals should be taken relative to ground. It is easiest to see how the circuit works if you use a square wave. Therefore, we ask you to take most of your data using this signal. YOU MAY TAKE MORE THAN ONE PICTURE OF EACH BLOCK. In some cases, it helps to take a “close-up” of the individual pulses and a “wide-angle” of the overall signal shape. Whatever data you take, make sure that the PSpice picture and the Analog Discovery picture are at about the same time scale so that they can be compared. No signatures are required.

Simulation—Obtain PSpice data: Before you take plots of the PSpice simulation, verify that the sampling frequency is about the same as your circuit. If not, you can alter the sampling frequency by changing R3 or turning the pot in your circuit. For these plots you should reduce the amplitude of the sine wave voltage source, or remove it completely. Plot the following pairs of points for both a sine wave and square wave 1kHz input:

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Experiment—Obtain function generator data: The following is a list of the oscilloscope plots you should generate using both a sine wave and square wave 1kHz signal from the function generator as input. For each Pair put one signal on channel 1 and the other on channel 2. Be careful to keep track of which signal is which. If you want, in System (A-H), you can invert the output at H with the scope to get a better visual comparison with the input at A. Some of the speakers we have (with low impedance) distort the output at H. If you have this problem, you can remove the speaker when you take the data at point H.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Take audio data: Take the following additional data using your chosen audio signal as input. If you want, in System (A-H), you can invert the output at H with the scope to get a better visual comparison with the input at A.
Some of the speakers we have (with low impedance) distort the output at H. If you have this problem, you can remove the speaker when you take the data at point H.

Pair(E-F): ______ System(A-H): ______

Comparison
The final step in the initial design is to compare the output of the PSpice model to the output of the actual circuit. Are they similar? Also examine the function of each circuit signal Pair. Does each Pair work as expected?

Part B - Final Design
The initial design for the receiver of this project should have reconstructed the signal well enough to be audible, but likely does a very poor job. You can improve the output of your circuit by adding an integrator and a smoothing capacitor. Your final signal (at point H) should look and sound much more like the original signal coming from your audio device (at point A).

Adding an integrator
The first change you will make to your circuit is to add an integrator.

*What the integrator should do:* When you integrate the modulated signal, you take advantage of the fact that the pulses vary in frequency. Your pulses (at point E) are square waves centered around zero. For the positive part of the pulse, the signal (the integration of a positive constant) ramps up. For the negative part of the pulse, the signal (the integration of a negative constant) ramps down. Since the pulses vary in frequency (and width) with the signal, adding this integration will bring out the variation in the amplitude of the original signal. Figure 9 shows the input (point A) and the output (point H) of the PSpice model with an appropriate integrator added. Note that you can still see the sampling pulses, but the output (which is inverted) captures the overall shape of the wave much better. You can see both the shape of the lower frequency and the higher frequency of the input. (These plots are for a pair of sine wave inputs, one at 500Hz and one at 4kHz. You can choose to do these frequencies or use the separate 1kHz sine wave and square wave signals.

![Figure 9.](image)

There are a few things you should note in this output. The first is that the signal is inverted because there is an inverting amplifier in the receiver circuit. Another is that there is a time shift between the input and the output. It takes a finite amount of time for the signal to travel through the circuit. The DC offset also wavers about a bit in the output. This will not affect the sound quality at the speaker.
Types of integrators: You can either add a passive or an active integrator to your circuit. Both work well, so it is a matter of preference. Both are discussed in this section:

- Passive Integration
  Figure 10 shows a passive (approximate) integrator. It only works at higher frequencies. When $\omega \gg \omega_c = 1/RC$, then $V_{out} = \frac{1}{RC} \int V_{in} \, dt$. If you add this integrator, you will add it at point E. When you add this integrator, you may notice a marked change in the amplitude of your signal. You can change the amplitude by a factor of 10 by adding or removing the 10\(\mu\)F capacitor in the audio amplifier. You can make finer adjustments by changing the gain of the inverting amplifier.

- Active Integration
  Figure 11 shows an active integrator. To add this type of integrator, you can modify the inverting amplifier in your circuit by adding a capacitor to the feedback path. This integrator also does not work well at very low frequencies. Note – this is a modification of the existing 741-based amplifier, not a new amplifier. When $\omega \gg \omega_c = 1/(R_fC)$, then $V_{out} = -\frac{1}{R_fC} \int V_{in} \, dt$.

Integrator design: You want your integrator to not affect a useful audio signal but to integrate (average out) higher frequencies. The higher frequencies will be part of any signal delivered to the speaker because of the pulses produced by the 555 timer in the transmitter. All unwanted signals (those that are not in the audio range of interest) are noise. In the PSpice model, the example input frequency varies between 500Hz and 4kHz. This range covers much of the range we hear in common speech and music and should adequately cover the frequencies that will not be significantly affected by the integrator. If you want to, you can look closely at the audio signal you are using and determine its lowest and highest frequencies. If they fall outside this range, change the frequencies of the two sine wave sources. With either integrator, you need to choose R and C such that your chosen corner frequency is much less than the frequencies you want to integrate, $f_c = 1/(2\pi RC) << f_{noise}$ and possibly even lower than some of the useful audio range. In the passive integrator, you can choose R arbitrarily, in the active integrator, R is the value of your feedback resistor. There is more information on integrators in the Project 4 slides and videos.
**PSpice test:** Once you have chosen an integrator and your target component values, try adding the integrator to the PSpice model. Run the simulation and see if it works. Note that you will probably have to increase the gain of E2 by a factor of 10. If you have to do this in PSpice, do not forget to also add the 10 μF capacitor between pins 1 and 8 of the 386 amplifier in your actual protoboard circuit. Try adjusting the value of the integrator capacitor until you get a good integration with adequate final amplitude. Remember, you don’t need to worry about the pulses from the 555 timer sampling. You just want to get a good reconstruction of the input with the sampling pulses superimposed on it as shown in Figure 9. [If you need to, you can use the inverting amplifier to change the gain as well.]

**The hardware integrator:** After you have used PSpice to design an integrator, add the integrator to your circuit. Don’t forget to also add any other changes you made in the PSpice model (like amplifying the signal). Does your integrator work? Many times the best design predicted by PSpice will not be the one that works optimally in your circuit. If you find a design that works better in the actual circuit, go back and adjust the PSpice simulation.

**Signature:** When your circuit works as expected with the integrator, have a staff member listen to the circuit with the sine wave input and sign the cover/signature page at the end of this handout.

**Take your data:** When you have a working integrator, take an oscilloscope picture of the integrator functioning and the overall performance of the circuit and the model. If you want, in System(A-H), you can invert the output at H with the scope and with PSpice to get a better visual comparison to the input at A. *Again, some of the speakers we have (with low impedance) distort the output at H. If you have this problem, you can remove the speaker and when you take the data at point H.*

**PSpice output:** Pair(E-F): ______  System(A-H): ______

**Circuit with sine wave input:** Pair(E-F): ______  System(A-H): ______

**Smoothing out the sampling frequency**

The final addition you will make to your circuit is to add a smoothing capacitor to eliminate the pulses created by the sampling.

![Graph showing the effect of a smoothing capacitor](image)

**Figure 12.**

**What the smoothing capacitor should do:** Recall that when you added a smoothing capacitor to the output of a rectifier, the bumps from the rectified signal were smoothed over and the output was much more even. You can apply this concept to the signal in this circuit as well. If we add a smoothing capacitor of the correct size between point F and ground, we should be able to smooth out the pulses caused by the sampling and come up with an even closer reconstruction of the original audio wave. Figure 12 shows the result of adding a smoothing capacitor to the integrated signal. It shows the original input (at point A), the output of the circuit (at point H), and the inversion of the output (at point H). Note that the original input frequencies have been preserved, but the pulses (that are shown
in Figure 9) have been largely eliminated. Aside from a change in amplitude, a slight time delay, and some inconsistencies in DC offset (caused by the integration), the input and the output are now very much the same. You can see that this is a substantial improvement over the initial data shown in Figure 7.

**What capacitance should I use?** The exact value of your smoothing capacitor will depend on your integrator and your sampling frequency. A capacitor that is too large will smooth over the high frequencies of your input signal. A capacitor that is too small will not smooth over the sampling pulses. Try experimenting with the PSpice model by adding a 1µF capacitor between point F and ground. Look at the output. Should the capacitance be increased or decreased? Increase or decrease the capacitance in increments of an order of magnitude (0.001µF, 0.01µF, 0.1µF, 1µF, 10µF, 100µF, 1000µF) until you find one that gives you the best reconstruction. You can then fine tune the capacitance using the values you have in your kit. If you find you need a capacitance that you do not have, consult the staff to find out what we have available.

**Circuit Instabilities:** As you try larger values of capacitance, it is likely that an instability will occur manifested by the appearance of additional wiggles in the signal. This is similar to the instability we observed in the differentiator circuit and do to the same cause (additional phase shift in the feedback). If you observe such an artifact in your data, you have two options to deal with it. First, you can reduce the value of your smoothing capacitor until the instability disappears. If the signal still looks good (very little residue from the square sampling pulses), then your smaller capacitance is fine. If, however, you need more smoothing, you can add a small resistor (10-50Ω) between point F and where you connect the capacitor. That is, after point F you should have a small series resistor, then a capacitor to ground (your smoothing capacitor) and then the potentiometer.

**Check the smoothing capacitor in your circuit:** We have found that the smoothing capacitors do not always work well in the actual circuit. Try adding the smoothing capacitor to your circuit and see if it works. You may need to try a slightly different size capacitor than the one predicted by PSpice. If you can find a design that both integrates and smoothes the signal, take a scope picture of the output for a sine wave input signal and have it signed by a staff member. Include it for extra credit. Make sure you indicate what components you used to get the circuit working.

**Take your PSpice data:** Once you have decided on a smoothing capacitor that works well in PSpice, take your data. If you want, in System(A-H), you can invert the output at H with the scope and with PSpice to get a better visual comparison to the input at A. Once again, some of the speakers we have (with low impedance) distort the output at H. If you have this problem, you can remove the speaker and when you take the data at point H.

PSpice output: **Pair(E-F): _____** **System(A-H): _____**

**Take your circuit data:**
If you were able to get the smoothing capacitor to work, take this last picture with both the integrator and the smoothing capacitor. If you did not, take it with only the integrator.

Final circuit with audio source input: **Pair(E-F): _____** **System(A-H): _____**

**Signature:** When your circuit works as expected with the integrator (and possibly the smoothing capacitor), have a staff member listen to the circuit with the audio input and sign the cover/signature page at the end of this handout.
Part C - The Report

Instead of a formal report this semester, we are asking you to provide the following packet. Basically, we want you to show a comparison between the PSpice output and the output from the circuit. You will be graded out of 70 points for the packet plus an additional 10 point general assessment. If you want you can include more than one pair of plots for any Pair. Also include the cover/signature page attached to the end of this handout. ALL PLOTS SHOULD INDICATE WHICH TRACE CORRESPONDS TO THE SIGNAL AT WHICH POINT. Based on your simulation and experiment, identify and discuss any performance limitations of your system.

A. Plots for Initial Design with Function Generator Input (40 points)
   Two cases – 1kHz sine and square wave.
   Part A1: Pair A-B
   • Output from PSpice  (1 pt)
   • Output from circuit  (1 pt)
   • Pair description and brief comparison  (3 pt)
   Part A2: Pair A-C
   • Output from PSpice  (1 pt)
   • Output from circuit  (1 pt)
   • Pair description and brief comparison  (3 pt)
   Part A3: Pair A-D
   • Output from PSpice  (1 pt)
   • Output from circuit  (1 pt)
   • Pair description and brief comparison  (3 pt)
   Part A4: Pair A-E
   • Output from PSpice  (1 pt)
   • Output from circuit  (1 pt)
   • Pair description and brief comparison  (3 pt)
   Part A5: Pair A-F
   • Output from PSpice  (1 pt)
   • Output from circuit  (1 pt)
   • Pair description and brief comparison  (3 pt)
   Part A6: Pair A-G
   • Output from PSpice  (1 pt)
   • Output from circuit  (1 pt)
   • Pair description and brief comparison  (3 pt)
   Part A7: Pair A-H
   • Output from PSpice  (2 pt)
   • Output from circuit  (2 pt)
   • Pair description and brief comparison  (6 pt)

B. Plots for Initial Design with Audio Input (4 points)
   Part B1: Pair E-F
   • Output from circuit  (2 pt)
   Part B2: Circuit Function A-H
   • Output from circuit  (2 pt)

C. Final Design with Integrator only (12 points)
   Part C1: Pair E-F – Function Generator Input (both sine and square wave)
   • Output from PSpice  (2 pt)
   • Output from circuit  (2 pt)
   • Brief comparison  (2 pt)
   Part C2: Circuit Function A-H – Function Generator Input (both sine and square wave)
   • Output from PSpice  (2 pt)
   • Output from circuit  (2 pt)
D. Final Design with Integrator and Smoothing Capacitor (14 points)

Part D1: Final Design Schematic
- Print out of final design schematic from PSpice (3 pt)
- Calculation of corner frequency of integrator (2 pt)
- State value of smoothing capacitor (1 pt)

Part D2: Pair E-F – Function Generator Input (both sine and square wave)
- Output from PSpice (2 pt)
- Compare to output without smoothing (1 pt)

Part D3: Circuit Function A-H – Function Generator Input (both sine and square wave)
- Output from PSpice (2 pt)
- Compare to output without smoothing (1 pt)

Part D4: Circuit Function A-H – Audio Input
- Output from circuit (2 pt)

Total: 70 points for project packet
+10 points for general assessment of packet
+20 points for attendance
100 points

Attendance (20 possible points)
- 3 classes (20 points), 2 classes (10 points), 1 class (0 points)
- Minus 5 points for each late.
  No attendance at all = No grade for project.

Here is an example of an appropriate “block (signal pair) description” and “brief comparison”

**Block Description:** Block F-G is a volume control. As you can see, in both the PSpice simulation and the actual circuit output, the output signal (the trace marked G) is smaller than the input signal (the trace marked F). The amplitude of the output at G is some fraction (between 0 and 1) of the input at F. Turning the pot changes the relative amplitude of G. The output and the input are the same basic shape and the frequencies are preserved exactly.

**Brief Comparison:** The PSpice output and the output from the circuit have comparable sampling frequency (around 48kHz). The amplitude of the PSpice output is larger, primarily due to the fact that the output signal from the real photodiode is smaller than the one from the simulation. The overall shape of both (triangular samples superimposed over a signal which marginally represents the input) are comparable.
Project 4
Electronic Instrumentation

Group: ______

Report Grade: ______

__________________________ Attendance: ________________________
__________________________ Attendance: ________________________
__________________________ Attendance: ________________________
__________________________ Attendance: ________________________

Signatures

1. Hand-Drawn Circuit Diagrams ___________________________

2. Initial Design
   Function Generator Input ________________________________
   Audio Input __________________________________________

3. Final Design with Integrator Only
   Function Generator Input ________________________________

4. Final Design with Integrator (and Smoothing Capacitor)
   Audio Input __________________________________________