Breadboard Experiments for the NI myDAQ

Texas Instruments

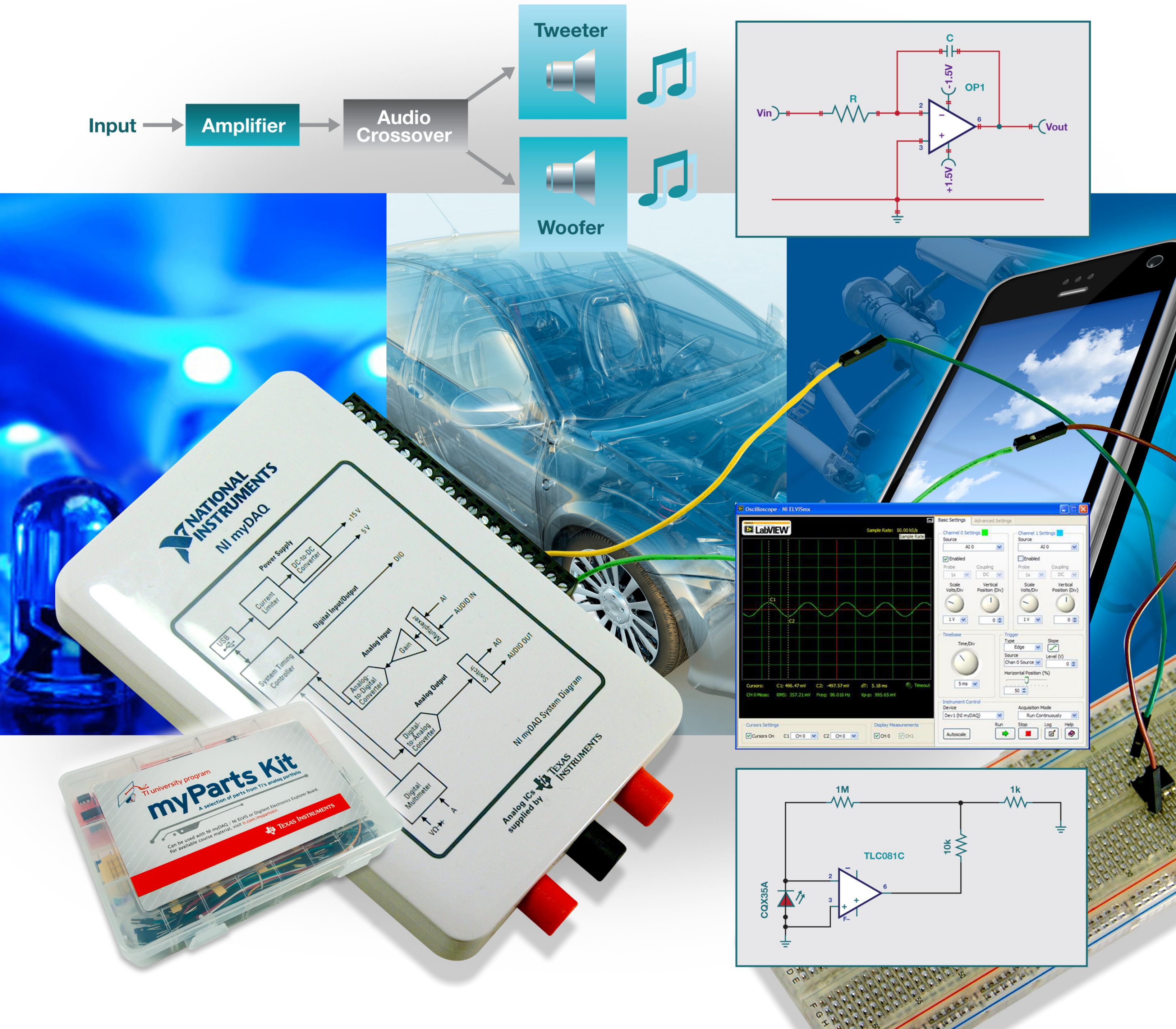


Table of Contents

Introduction:

This supplement to the myDAQ tool was created in an effort to introduce you to electronics theory and circuits as well as provide a practical method of getting to know the capabilities of the myDAQ tool. The experiements include all you need to study, build, and test some preselected circuits that typify some of the basic elements of electronics used in many applications from cell phones to MP3 players to automotive control systems.

About TI

Texas Instruments Incorporated (TI) is a global semiconductor design and manufacturing company that develops analog ICs and embedded processors. By employing the world's brightest minds, TI creates innovations that shape the future of technology. TI is helping more than 100,000 customers transform the future, today. Learn more at www.ti.com.

Getting Started

Parts List

Operational Amplifier	TL072
Comparator	LP311
Transistor	2N3904
LEDs	
Capacitors	
Buzzer	
Wire Kit	
Potentiometers	

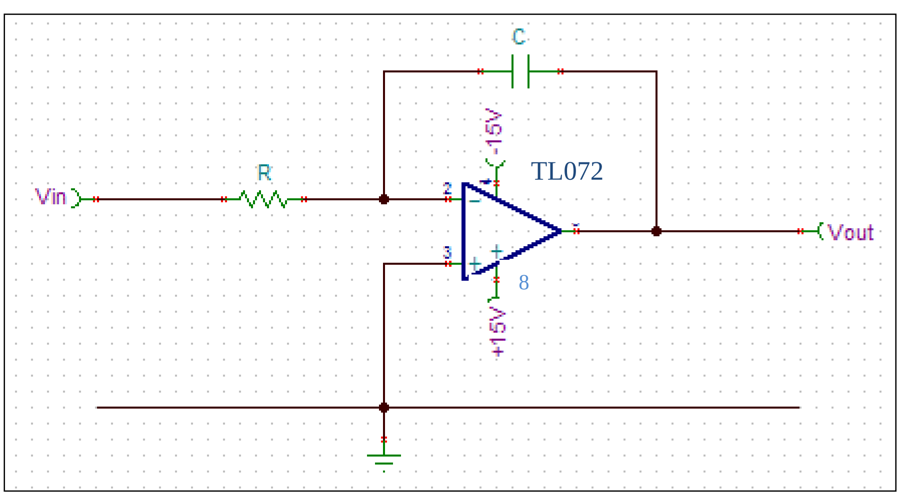
Breadboard Tutorial

This kit comes with two breadboards (part number BB830T) from BPS. They will be used to connect the circuitry together as well connect to the myDAQ that will supply power for the components and also the interconnections required to study the waveforms and test the circuits. The reasoning behind having two breadboards is that some experiments feed into one another to create a more complex circuit. By having two breadboards you can keep a tested circuit intact, build and test the second circuit on the second breadboard, and then connect the two together for the final experiments.

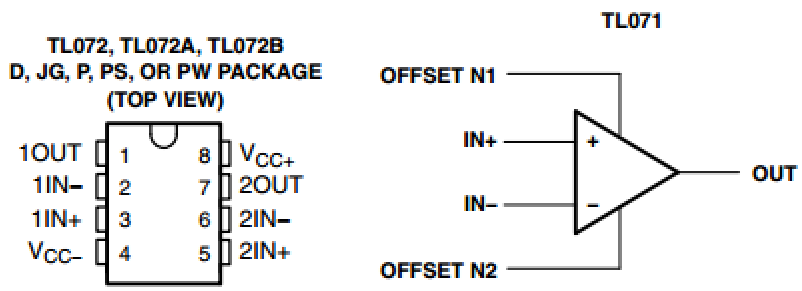
Internal Connections

The BB830T breadboard has 63 vertical columns on top and 63 columns below. Each column has 5 connected holes each (the green lines). This is the circuit area. There are also 4 “rails” (or distribution strips) for power and ground running horizontally (the red and blue lines). A distribution strip can be used to carry a signal if it is not needed for power or ground. Below is an example of how to assemble and connect a circuit to the breadboard.

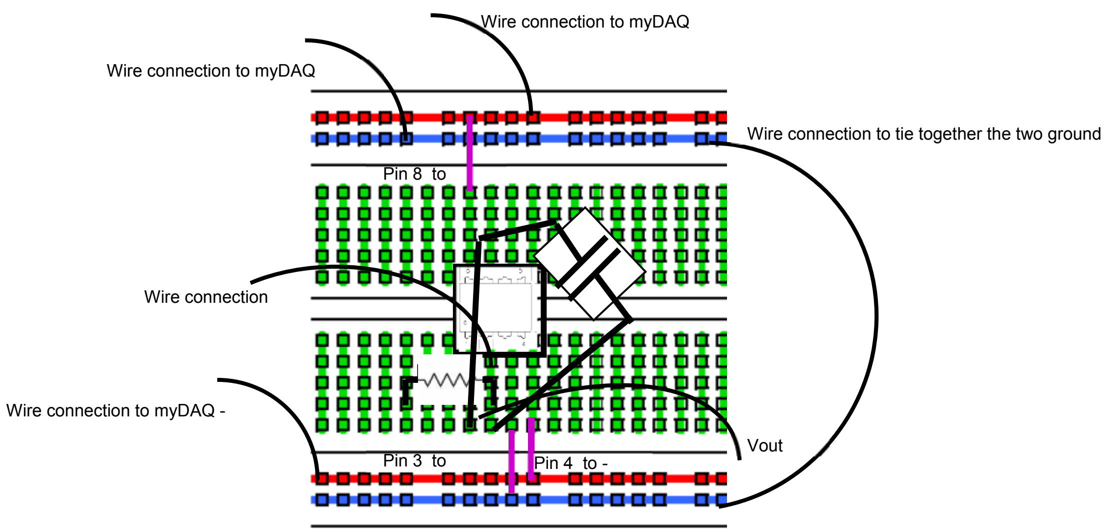
We start with the schematic, in this case an integrator circuit used later.



The operational amplifier used in this circuit is a Texas Instruments component, part number TL072 (http://www.ti.com/product/tl072). It has a pinout and equivalent circuit shown below. Please ignore the offset pins as this function is not used throughout the experiments.



This is how it should be assembled on the breadboard.



Connecting the Breadboard to the myDAQ

The connections are made using the screw terminal connector that comes with the myDAQ tool. The table below shows the pinout for the device. Each experiment will walk through what connection points have to be made and where they will connect on the breadboard.

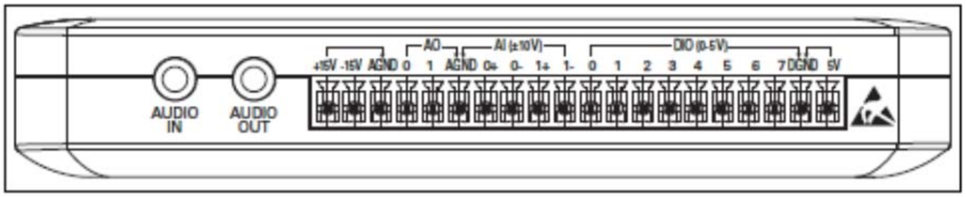


Figure 4. NI myDAQ 20-Position Screw Terminal I/O Connector

Table 1. Screw Terminal Signal Descriptions

Signal Name	Reference	Direction	Description
AUDIO IN	—	Input	**Audio Input**—Left and right audio inputs on a stereo connector
AUDIO OUT	—	Output	**Audio Output**—Left and right audio outputs on a stereo connector
+15V/–15V	AGND	Output	+15 V/–15 V power supplies
AGND	—	—	**Analog Ground**—Reference terminal for AI, AO, +15 V, and –15 V
AO 0/AO 1	AGND	Output	Analog Output Channels 0 and 1
AI 0+/AI 0–; AI 1+/AI 1–	AGND	Input	Analog Input Channels 0 and 1
DIO <0..7>	DGND	Input or Output	**Digital I/O Signals**—General-purpose digital lines or counter signals
DGND	—	—	**Digital Ground**—Reference for the DIO lines and the +5 V supply
5V	DGND	Output	5 V power supply

Experiment 1: Filters

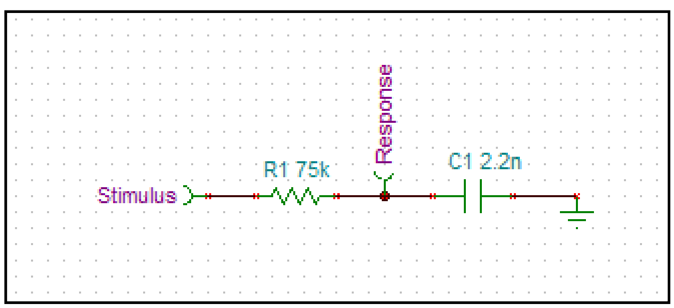
Introduction: Electronic filters are used to modify, emphasize, or reject certain ranges of the frequency content of a signal by altering the gain and phase response of a specific frequency band. Filters are a fundamental component of many electronic circuits in a variety of applications, including power electronics, communications, and audio systems.

This lab will demonstrate the theory and operation of several types of first-order filters, including low-pass, high-pass, and band-pass filters. Passive and active filters will also be discussed, as well as a brief introduction to higher order filtering.

Task 1.1: Passive Low-Pass Filter

This section will address the theory and operation of a simple first-order, passive, low-pass filter. Low-pass filters preserve the low frequency content of a signal while rejecting the high frequency content. The operation of the filter will be demonstrated in both the time and frequency domain.

Schematic:



Schematic 1: Passive Low Pass Filter

Equations: The transfer function of Schematic 1 can be derived from a simple voltage division between the stimulus and ground.

$$V_{out} = V_{in}\left(\frac{\frac{1}{j\omega C}}{R+\frac{1}{j\omega C}}\right) = V_{in}\left(\frac{1}{1+j\omega RC}\right)$$

$$\frac{V_{out}}{V_{in}} = \left(\frac{1}{1+j\omega RC}\right)$$

Equation 1: Transfer Function of a Passive Low Pass Filter

Notice that when the frequency ω is small, the gain of the circuit is approximately one. As the frequency becomes high, the gain of the circuit goes to zero. Alternatively, consider that when the frequency of the input signal is low, the capacitor operates like an open circuit, and $V_{out} = V_{in}$. When the frequency is high, the capacitor operates like a short circuit, and V_{out} approaches zero.

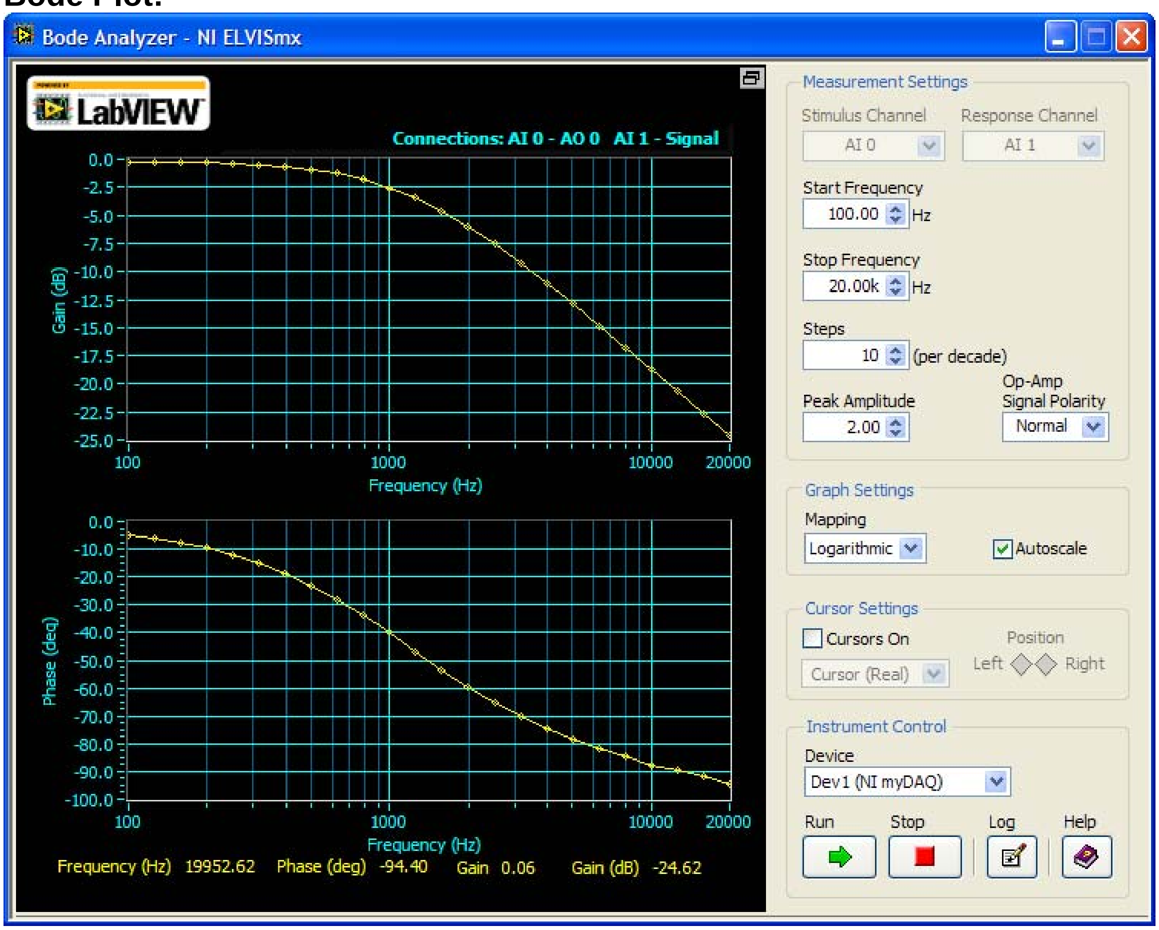
At the cutoff frequency, f_c, the gain of the signal is down 3dB, equivalent to a gain of 0.708. The cutoff frequency is given by:

$$f_c = \frac{1}{2\pi RC}$$

Equation 2: Cutoff frequency of a first order filter

Therefore the theoretical cutoff frequency of the circuit show in Schematic 1 is 964Hz. The actual -3dB point observed in the following Bode plots may fall above or below this value, depending on the accuracy of the components used.

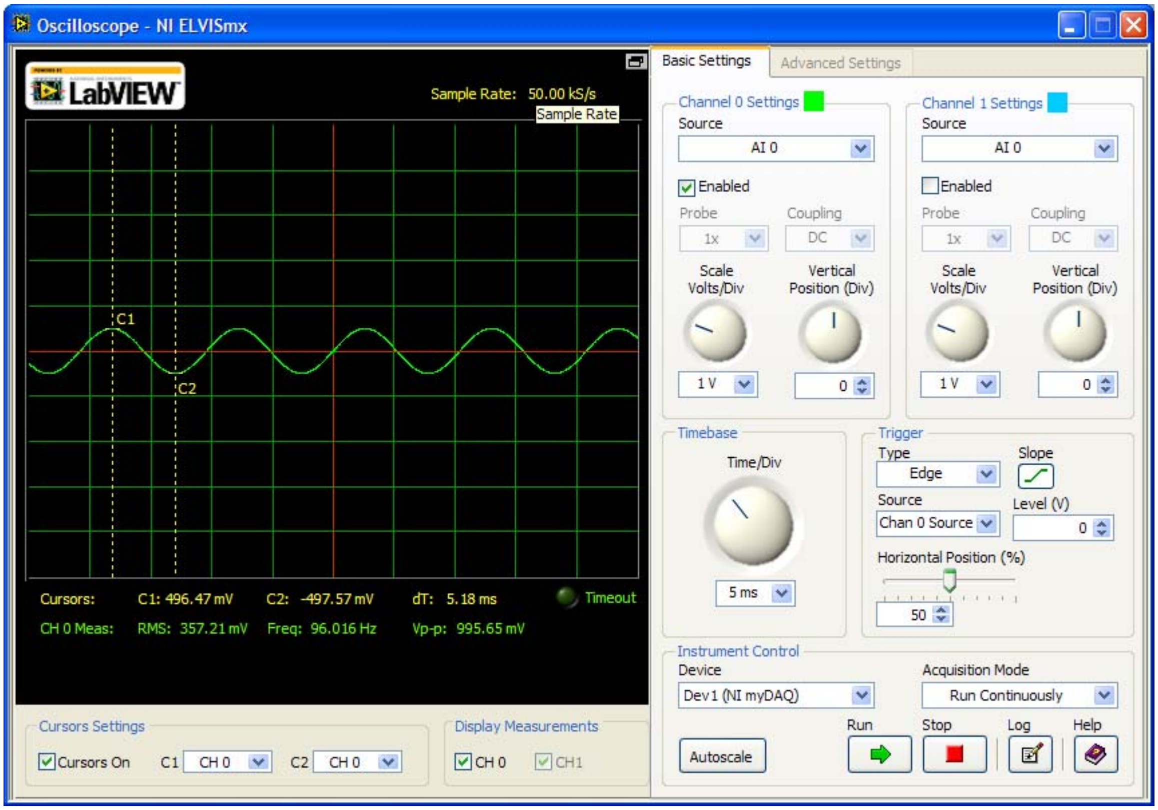
Bode Plot:



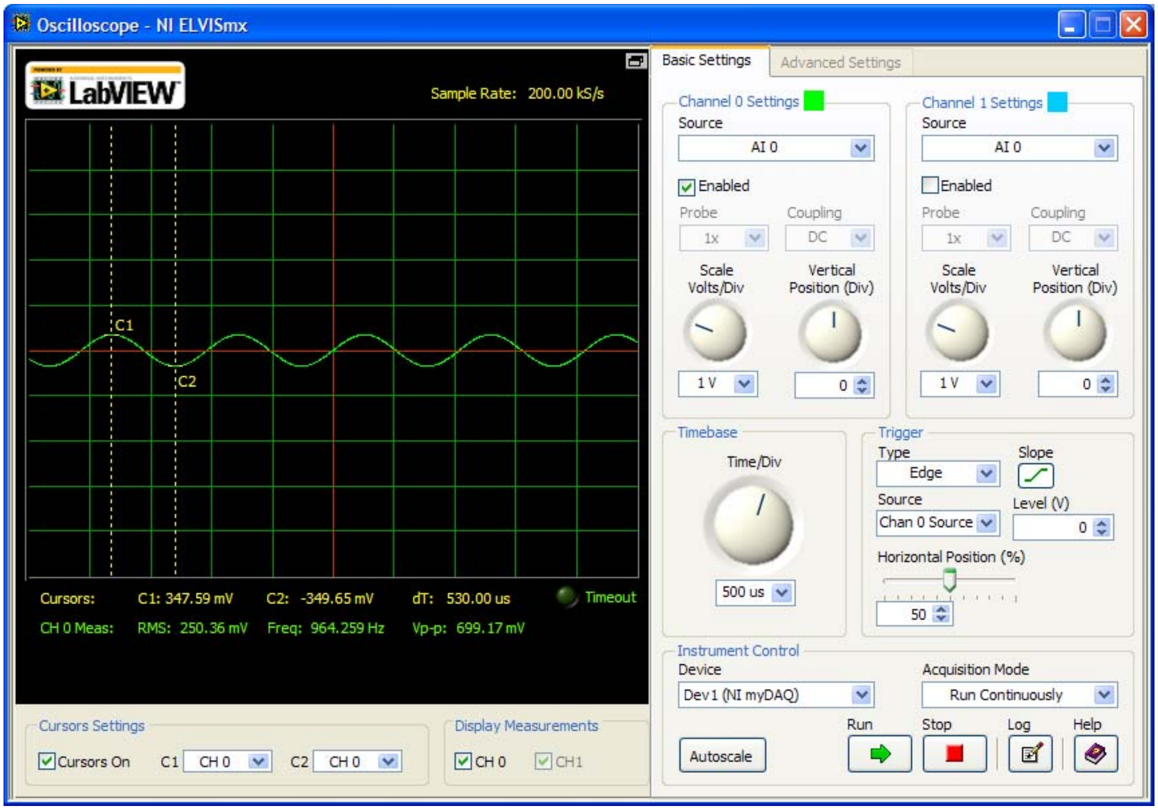
Notice that at the cutoff frequency, the signal gain is down by approximately 3dB.

To show the effect of the filter in the time domain, apply a 96-Hz, 1-Vpp sine wave to the input of the circuit, and view the response on the oscilloscope.

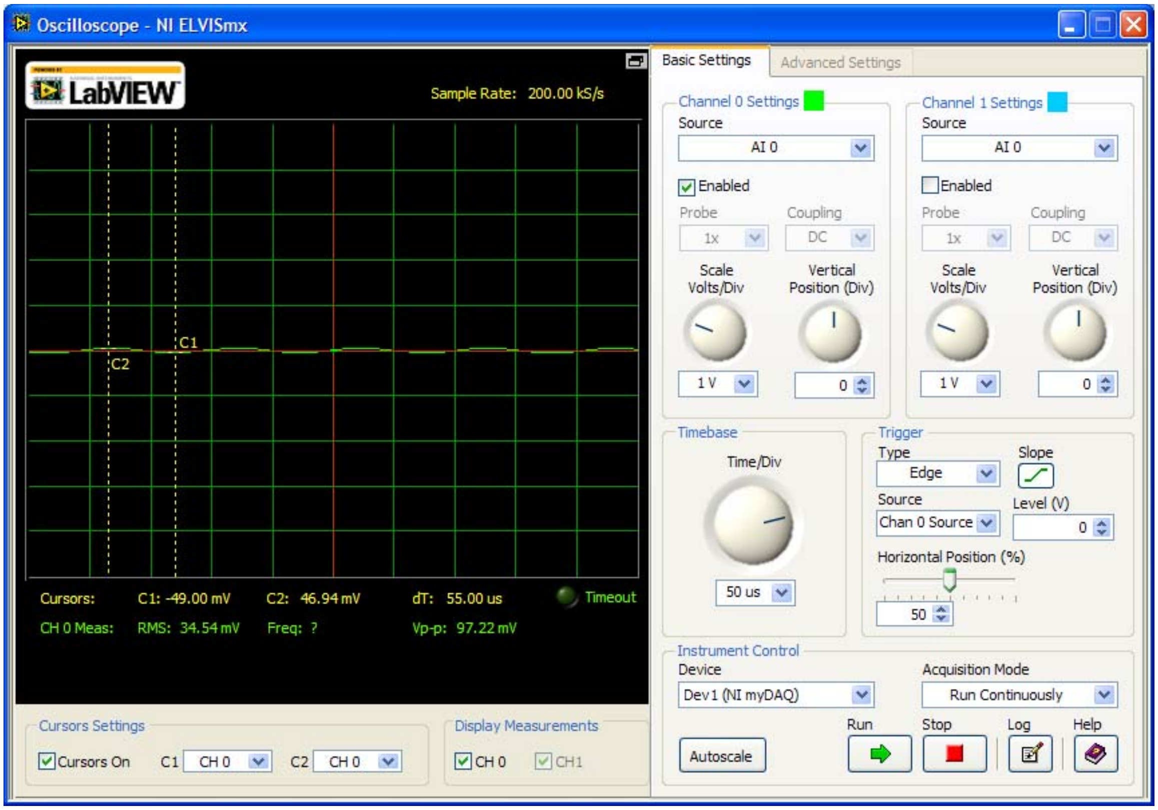
Note: Component values for this lab have been selected arbitrarily to accommodate the limitations of the myDAQ bode analyzer. Other component values and cutoff frequencies can be accommodated by performing the following exercises at one decade below the cutoff frequency, at the cutoff frequency, and one decade above the cutoff frequency.



Notice that the output waveform is approximately 994mVpp, close to unity gain. Now apply a 964-Hz, 1-Vpp sine wave to the input.

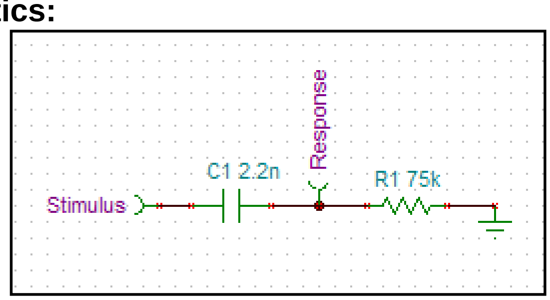


Notice that the output signal is approximately 697mVpp, or approximately 3dB down from the input. Finally, apply a 9640-Hz, 1-Vpp sine wave to the input.



Notice that the output signal has been reduced to approximately 10% of the magnitude of the input signal.

Additional Schematics:



Schematic 2: Passive High Pass Filter

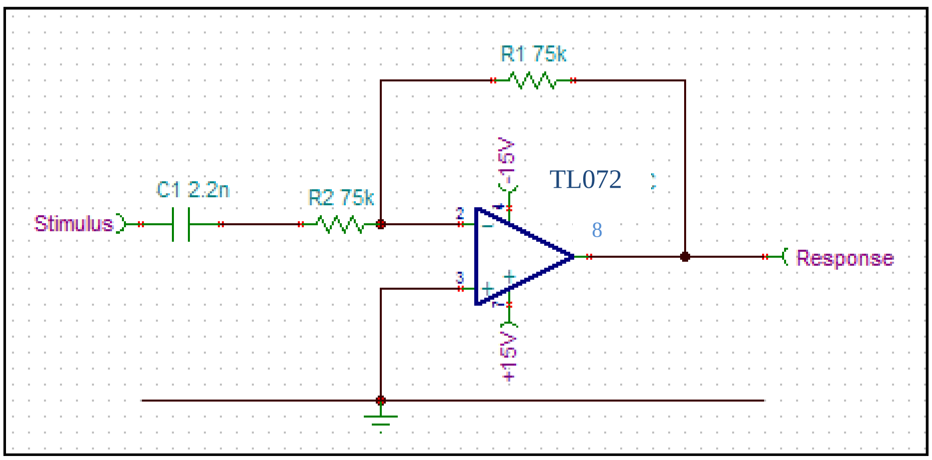
Task 1.2: Inverting Active High Pass Filter

Introduction: The passive filters demonstrated in Task X.1 are valued for their ease of design and implementation. Passive filters can be used to easily introduce additional poles into a circuit with a simple calculation, but passive filters also have disadvantages. First, the precision of the filter is inherently related to the tolerance of the components. Additionally, passive filters often require large resistor and capacitor values, which can result in high output impedance.

Active filters can reduce the output impedance of a filter, as well as apply gain to a signal and buffer the filter from the rest of the circuit. Most important, active filters are the fundamental building block of higher order filters. This section will demonstrate the operation of an inverting, first-order, high-pass filter.

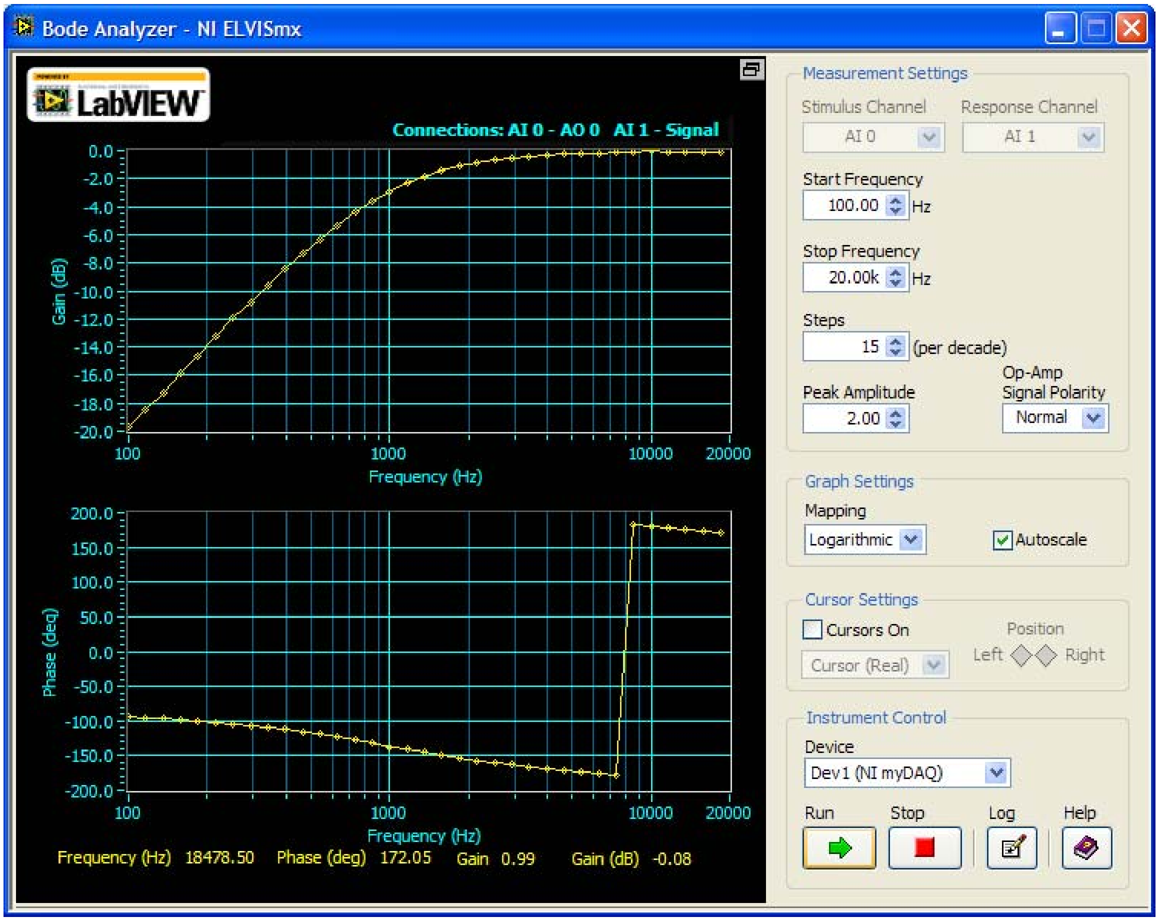
For first-order, active filters, the cutoff frequency is given by Equation 2, as before, and the gain is given by the ratio (R_1 / R_2).

Schematic:



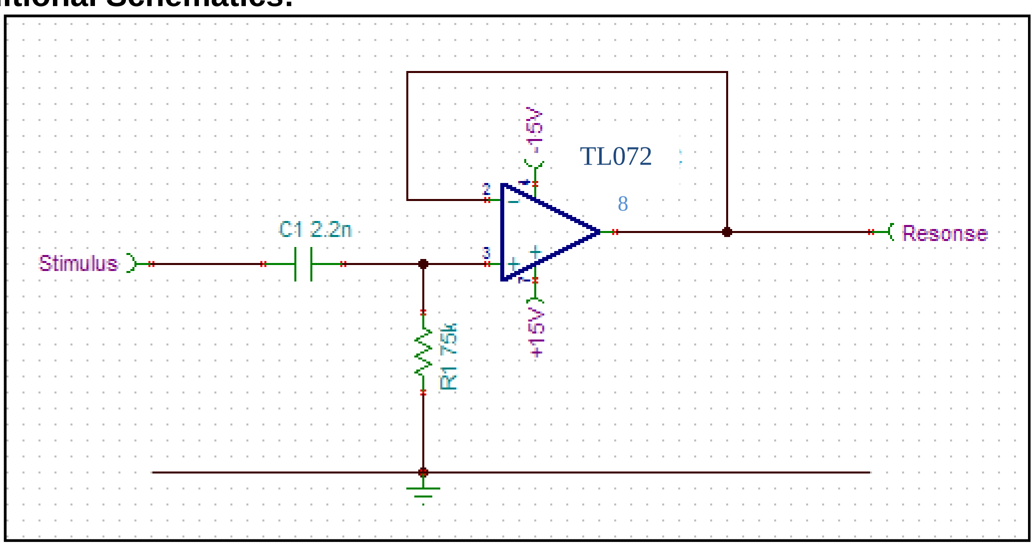
Schematic 3: Inverting Active High Pass Filter

Bode Plot:

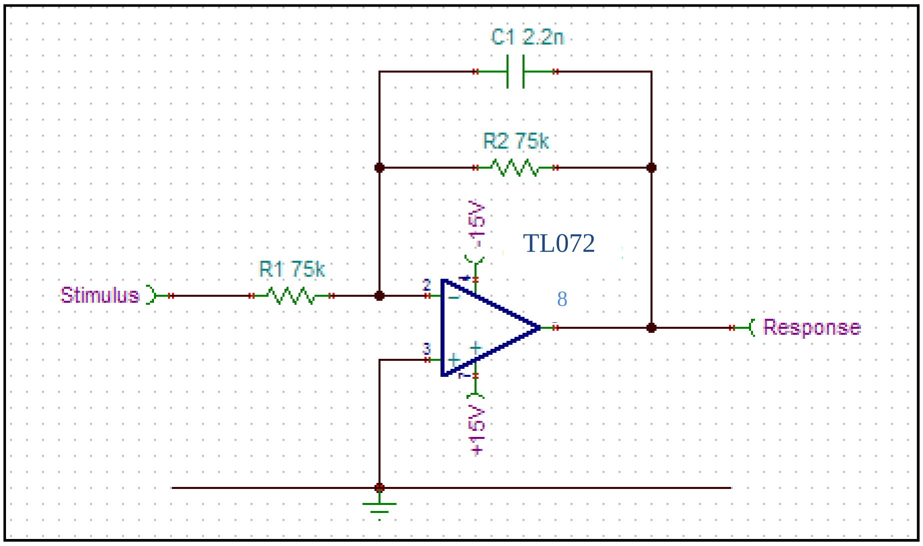


Now apply input signals of 96Hz, 964Hz, and 9.64kHz and view the output on the oscilloscope, as in Task X.1. What happens?

Additional Schematics:



Schematic 4: Non-Inverting Active High Pass Filter

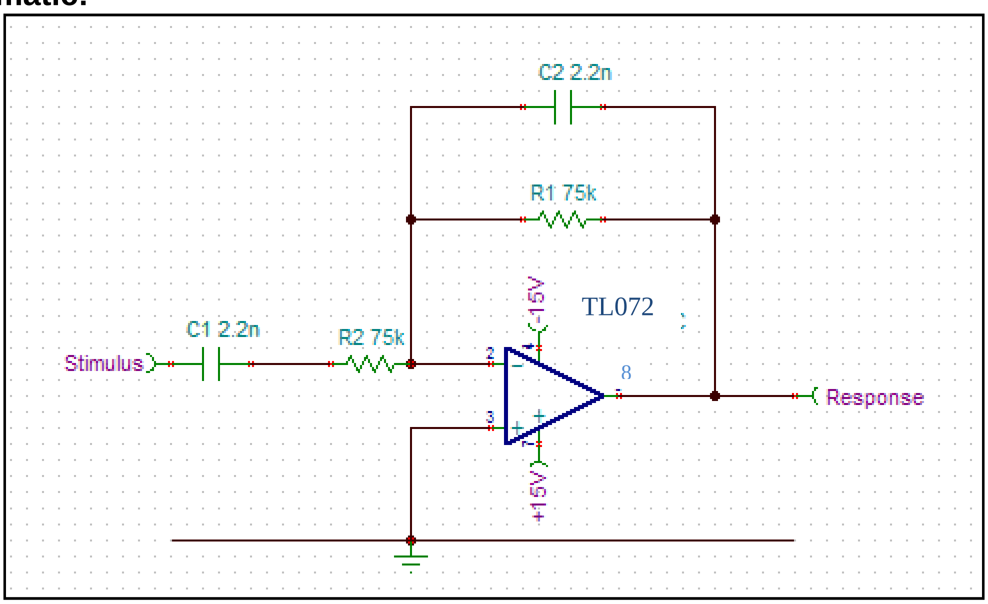


Schematic 5: Inverting Active Low Pass Filter

Task 1.3: Inverting Band Pass Filter

Introduction: First-order bandpass and band-stop filters can be designed easily by implementing a high-pass and low-pass filter simultaneously. This section demonstrates the operation of an inverting, active bandpass filter.

Schematic:



Schematic 6: Inverting Active Bandpass Filter

Notice that this is simply a high-pass filter and a low-pass filter, both with cutoff frequencies of 964Hz. The gain of the circuit is again given by the ratio (R_1/R_2).

Bode Plot:

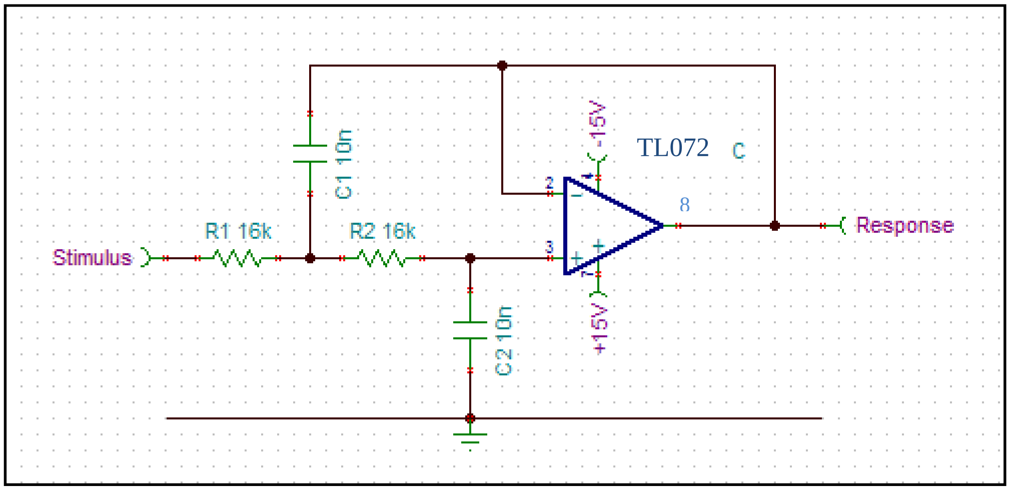
How can Schematic 6 be modified to implement a band-stop filter?

Task 1.4: Second-Order, Low-Pass Filter

Introduction: This section will introduce second order filters and demonstrate the operation of a Sallen-Key second-order, low-pass filter.

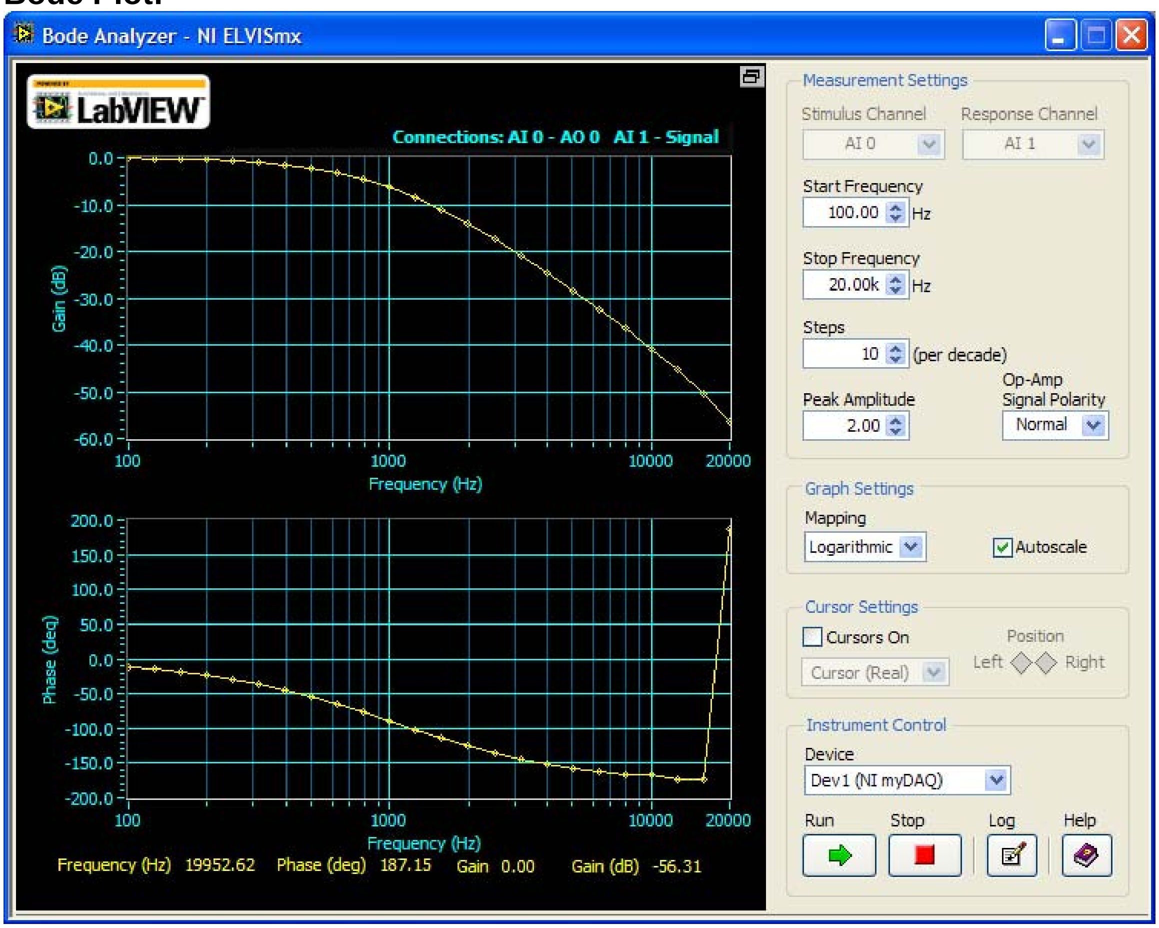
There are many common filter topologies used to realize high-order filtering, such as Butterworth, Bessel, and Chebyschev. The Sallen-Key circuit is valued for its simplicity and gain accuracy in realizing various second-order transfer functions. In designing higher order filters, various tradeoffs determine the ideal filter topology for achieving optimum performance in a given application. A more complete discussion of second-order filtering and the performance characteristics of various filter topologies can be found at http://focus.ti.com/lit/an/sloa049b/sloa049b.pdf.

Schematic:



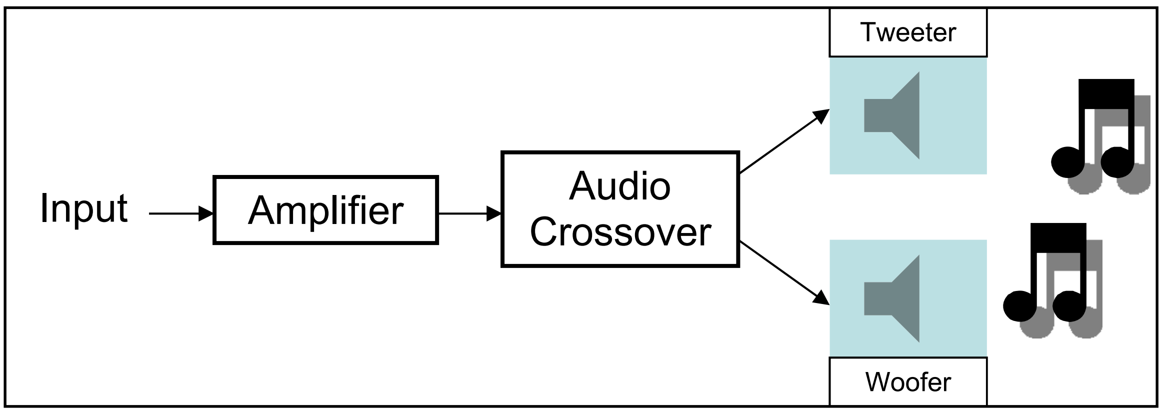
Schematic 7: Sallen-Key, Second-Order Low-Pass Filter

Bode Plot:



The filter shown in Schematic 7 has a cutoff frequency of 997Hz. Notice that at the cutoff frequency, the signal gain is down 6dB rather than only 3dB, as in the first order case. Also note that the second order filter results in a 180˚ phase shift, compared to 90˚ in the first order filters.

Preview: Audio crossovers are an important class of filters used to direct different frequency components of a signal to different speakers in an audio system. High frequency content is routed to speakers optimized for high frequency content (tweeters), while low frequency content is routed to speakers optimized for low frequency content (woofers).



Subsequent experiments will characterize the frequency content of common audio signals and demonstrate the effect of filters on audio signals.

Experiment 2: R-2R Resistive DAC

This lab will create an 8-bit, digital-to-analog converter (DAC). To do this, select the "DigOut" virtual instrument (VI) on the "NI ELVISmx Instrument Launcher." The eight digital I/O pins will be set as outputs and used to bias an R-2R resistive network. This will in turn create an analog voltage that will be fed through a voltage follower op amp topology to supply current to a load. This is commonly done in practice to remove loading effects on a signal or power supply rail.

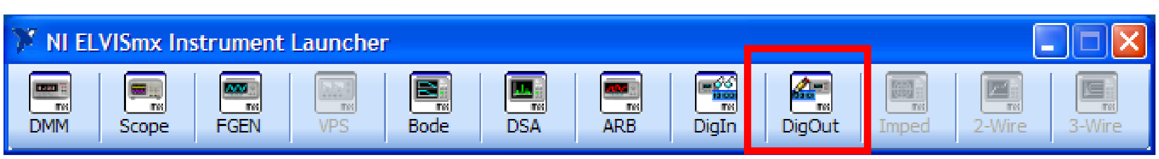


Figure 2.1: NI Instrument Launcher

Background:
The R-2R resistive DAC will work with any two sets of resistors with as close to an R-2R ratio as possible. First Nodal Analysis will be used on simpler 2-bit DAC to show the math behind how a resistive DAC works.

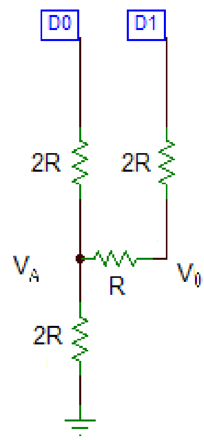


Figure 2.2: The 2-Bit R-2R Resistive DAC

To perform nodal analysis, the sum of the current going into and out of each node will be set equal to each other. Mathematically this can be done by labeling each node with a voltage reference label; then the current terms work out to be the change in voltage divided by resistance between each node. The output voltage will be solved for in terms of D_0 and D_1, which is shown below.

<u>Node V_A:</u>

$$\frac{D_0 - V_A}{2R} + \frac{V_O - V_A}{R} = \frac{V_A}{2R}$$
$$D_0 - V_A + 2V_0 - 2V_A = V_A$$
$$D_0 + 2V_O - 4V_A = 0$$

<u>Node V_O:</u>

$$\frac{D_1 - V_O}{2R} = \frac{V_O - V_A}{R}$$
$$D_1 - V_0 = 2V_0 - 2V_A$$
$$2V_A = 3V_O - D_1$$

$$\textit{Substitute } 2V_A \textit{ in equation on left}$$
$$D_0 + 2V_O - 2(3V_O - D_1) = 0$$
$$D_0 + 2V_0 - 6V_0 + 2D_1 = 0$$
$$4V_0 = D_0 + 2D_1$$
$$V_O = \frac{1}{4}D_0 + \frac{1}{2}D_1$$

The process would repeat for however many bits used. If you used a 4-bit DAC with bits D0 to D3 left to right, their corresponding weights would be 1/16, 1/8, 1/4 and 1/2. If the digital level for the I/O bits were set to 3.3 volts, the four possible output values for the four possible inputs would be as follows.

Table 2.1: 2-Bit DAC Output Values

Decimal	Binary	Output Voltage
0	00	0.000
1	01	0.825
2	10	1.650
3	11	2.475

This can also be shown graphically by the following.

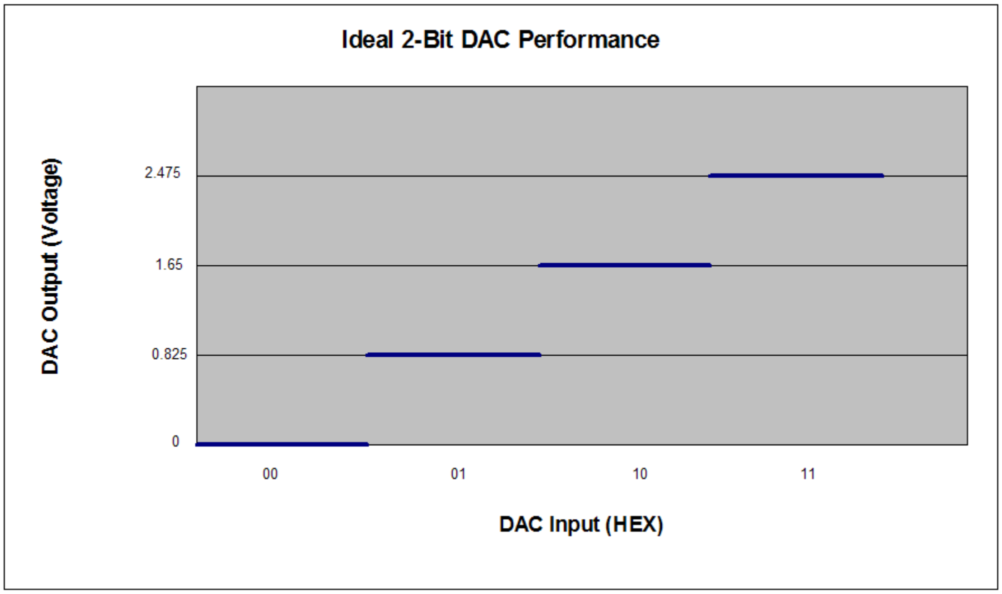


Figure 2.3: Ideal 2-Bit DAC Performance

You may have noticed that the DAC cannot output the full scale range of voltages up to the level of the digital I/Os. The maximum output voltages of these types of resistive DACs are as follows.

$$V_{MAX} = (V_{DIG})\frac{2^N - 1}{2^N}$$

Besides full-scale voltage, another very important characteristic with dealing with DACs is resolution. The resolution is the smallest increment in voltage the DAC is capable of outputting. Another way to say this is the resolution is the change in voltage due to asserting the least significant bit (LSB). This can be defined by:

$$RES = \frac{V_{DIG}}{2^N}$$

where VDIG is the supply level of the digital I/Os and N is the number of bits the DAC has.

As mentioned before the converter that will be built in this lab will be an 8-bit converter; the resolution will be 12.9 mV. Below is the schematic of the 8-bit DAC that will be used for this lab. The resistors chosen are 27k-ohms and 56k-ohms.

<u>Note</u>: The closer the resistor values are to being an R-2R ratio the more accurate the DAC will be.

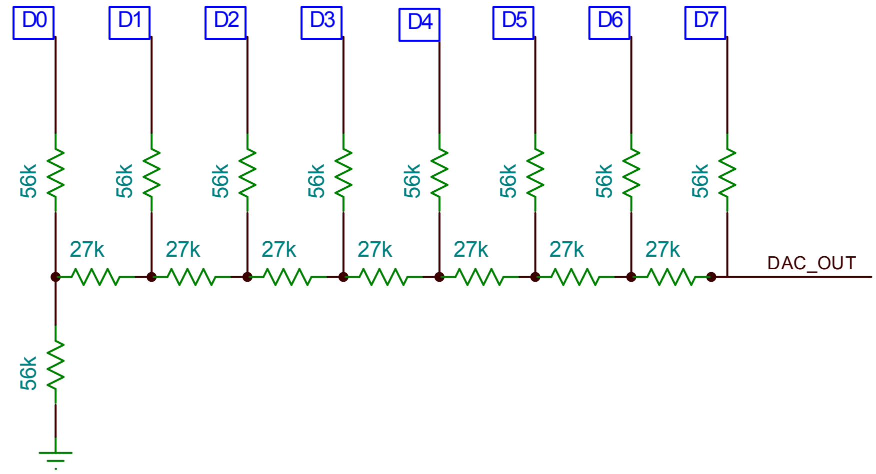


Figure 2.4: Schematic for 8-Bit DAC

The ground symbol on the lower left corner of the schematic will need to be connected to AGND and DGND pins on the myDAQ. The eight digital I/Os on the myDAQ will each be connected to their corresponding D0 through D7 pins on the schematic.

With 8 bits there are 2^8 or 256 possible output voltage settings. Once the resistive network DAC is set up, the following buffer op amp will be built at the output to reduce the effects of output loading. This means that when a load is attached to the DAC, the output will not vary as much.

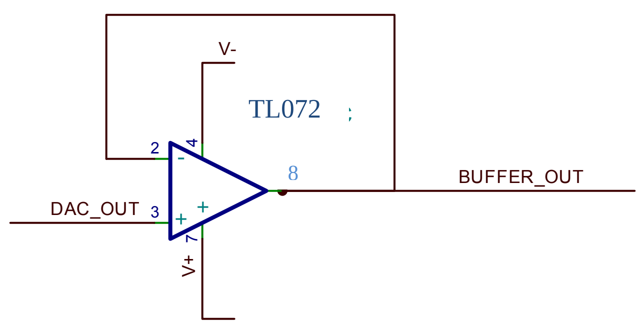


Figure 2.5: Buffer Operational Amplifier

With this op amp topology the output voltage will remain the same as the input voltage.

The V+ and V- pins on the TL072 chip will be connected to the +15 and -15 on the myDAQ, respectively. To test this circuit, first set the DigOut VI with “Lines to Write” to “0-7.” This enables all eight digital I/O pins instead of the default four, which is shown below.

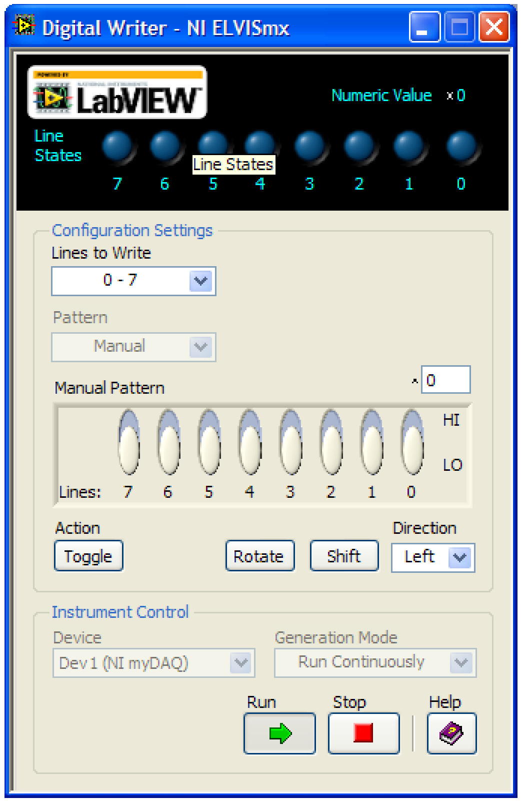


Figure 2.5.1: Setting the Digital I/O Pins

To check the output voltage, open the “DMM” VI on the NI Instrument Launcher.

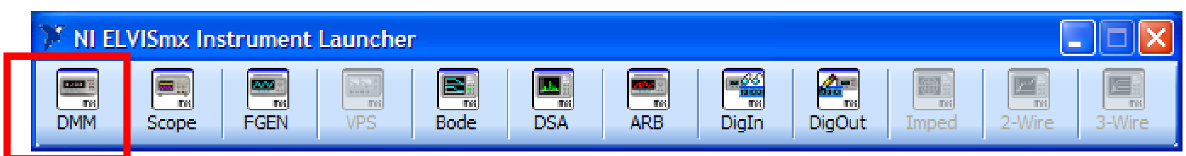


Figure 2.6: NI Instrument Launcher

The following window will open. For the “Mode” setting, select “Auto.”

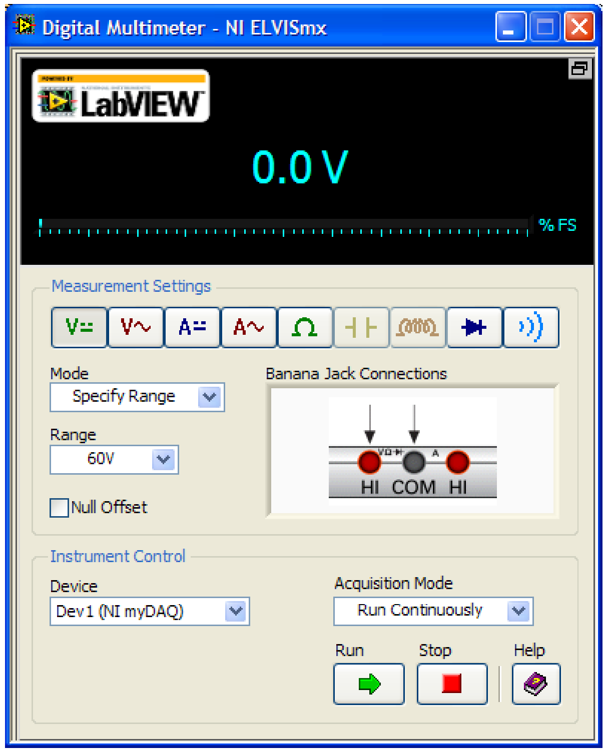


Figure 2.7: Digital Multimeter VI

To measure voltage, connect the two banana plugs to the HI and COM connectors on the bottom of the myDAQ. Attach the red plug to the HI port used for voltage, and attach the black banana plug to the center COM port. Next, connect the black probe to a ground reference and the red probe to the output of the op amp.

To test the functionality of the circuitry, start from 0x00 hex on the Digital Writer window and increment the output pins. Check the corresponding output voltage using the formula below.

$$V_{OUT} = \frac{1}{256}D0 + \frac{1}{128}D1 + \frac{1}{64}D2 + \frac{1}{32}D3 + \frac{1}{16}D4 + \frac{1}{8}D5 + \frac{1}{4}D6 + \frac{1}{2}D7$$

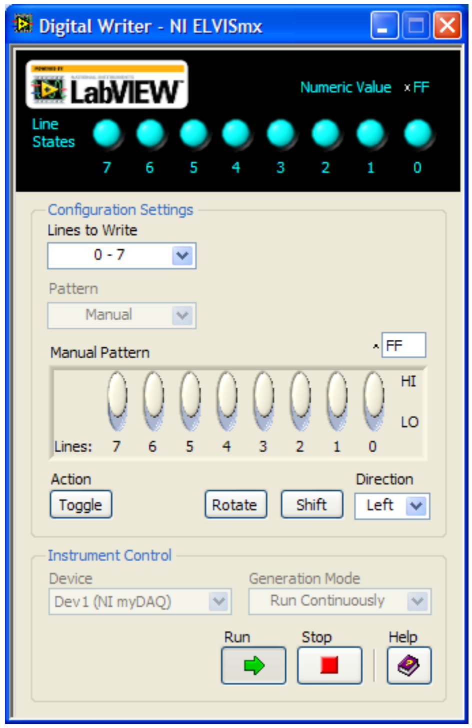


Figure 2.8: Setting the Digital I/O Pins

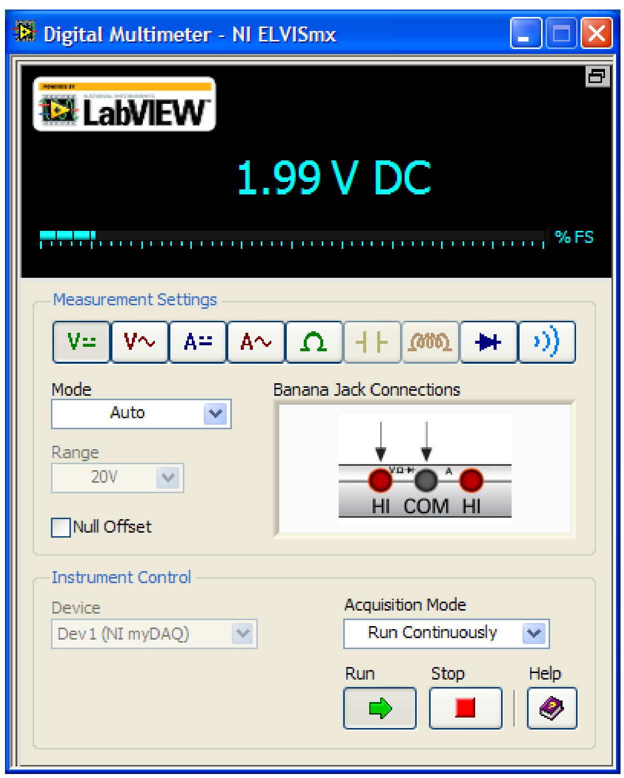


Figure 2.9: Digital Multimeter VI Sensing DAC Output Voltage

NOTE: -The first few increments may not register on the Multimeter.
-Keep this circuitry on your breadboard for future labs.

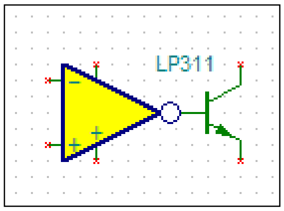
Experiment 3: Triangle Wave Generator

This lab will introduce a triangle wave generator that can be used to create pulse width modulated (PWM) signals.

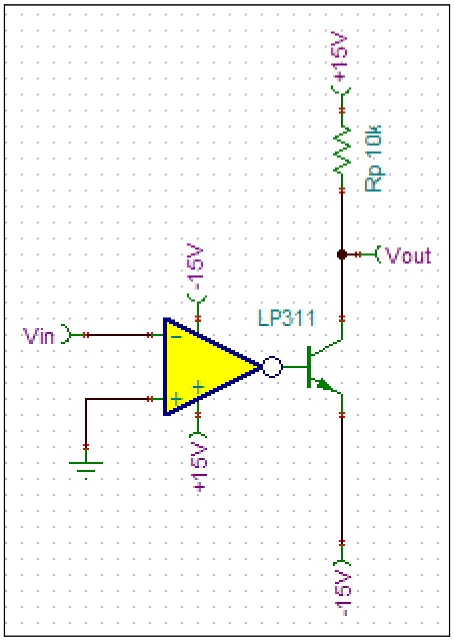
Task 3.1: Basic Comparator Circuit

First we will discuss a basic comparator circuit.

Schematic:



Schematic 3: LP311 Block Diagram



Schematic 2: Basic Comparator

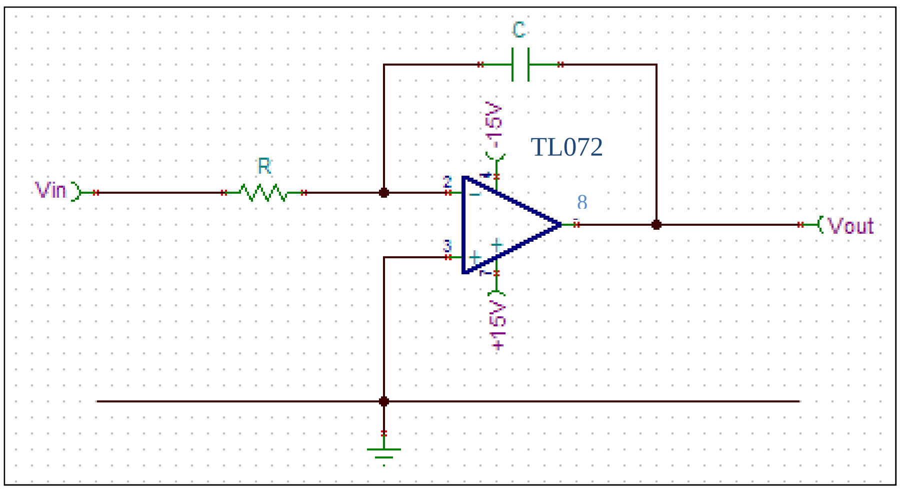
Equations: A comparator essentially compares two voltages. If the voltage at the positive input is greater than the voltage at the negative input, the output will be high. If the voltage at the positive input is lower than the voltage at the negative input, the output will be low. The LP311 is considered an open collector comparator, meaning that the output is the collector of a BJT. When the positive input is greater than the negative, the output looks like an open circuit. Thus, you'll need a pull-up resistor on the output to get a high level out. When the negative input is greater than the positive, the output looks like a short circuit to –Vcc. The output of a comparator as a function of the inputs is given as:

$$V_{out} = \begin{cases} +Vcc & if \;\; (V_{IN+} - V_{IN-}) > 0 \\ -Vcc & if \;\; (V_{IN+} - V_{IN-}) < 0 \end{cases}$$

Task 3.2: Basic Integrator Circuit

Introduction: An inverting op-amp can be set up as an integrator by simply replacing the feedback resistor with a capacitor. This will create a circuit that integrates the input voltage. Build the circuit shown in Schematic 3.

Schematic:



Schematic 3: Basic Integrator Circuit

Equations: From ideal op-amp theory, the inputs of the op-amp will remain at the same potential. Since the positive input is tied to ground, the negative input will also be at ground potential. Thus, the current into R is:

$$i_R(t) = \frac{V_{IN}(t)}{R}$$

The current into a capacitor is (note that the current is considered negative due to the passive sign convention, where Vcap is taken from Vout to the positive input of the op amp):

$$i_C(t) = -C\frac{dV_C(t)}{dt}$$

Rearranging, taking the integral, and accounting for the initial voltage gives:

$$V_C = V_0 - \frac{1}{C}\int_{t_1}^{t_2} i_C(t)dt$$

Since the current into the input of an ideal op-amp is assumed to be zero, $i_C = -i_R$ and the output voltage is equal to the voltage across the capacitor:

$$V_{OUT} = V_0 - \frac{1}{RC}\int_{t_1}^{t_2} V_{IN}(t)dt$$

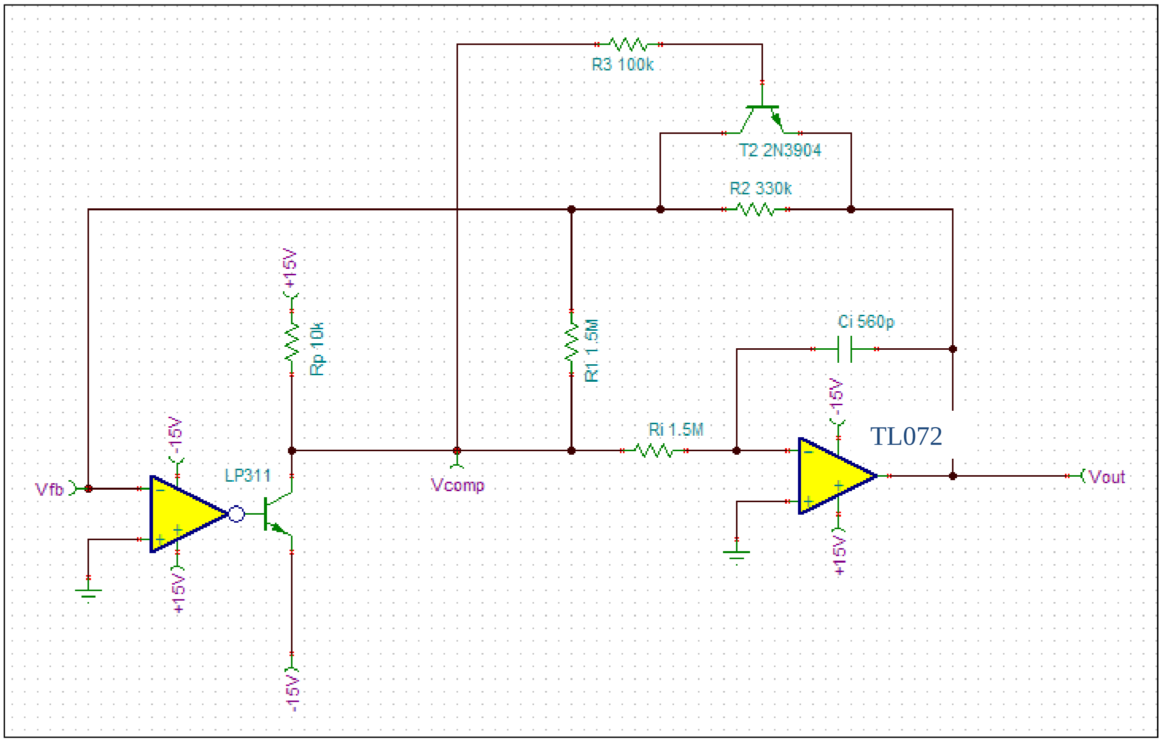
This equation shows that the circuit integrates the input voltage at a rate of 1/RC. If V_{IN} is a DC voltage than the output will change with a constant slope of –V_{IN}/(RC) versus time. Therefore, if the input to the integrator is a square wave, the output will rise when the input is a constant low level and will fall when the input is a constant high level. This is the basis of the triangle wave generator.

Task 3.3: Triangle Wave Generator

Introduction: By combining the integrator with a comparator, we can make a simple triangle wave generator. Since the comparator only has two output levels, it can be used as a square wave generator. As the output of the triangle wave climbs, feedback can be used to trip the comparator as the output passes through a desired value. The integrator will then begin to fall and again, the comparator will be tripped when the integrator passes through a certain value.

Build the circuit shown in Schematic 4. Keep in mind that the transistor shown at the output of the comparator is actually internal to the LP311 and thus, no transistor should be installed there. You can find the datasheet and pin diagram of the LP311 and TL072 by going to www.ti.com and using the “Search by Part Number” box.

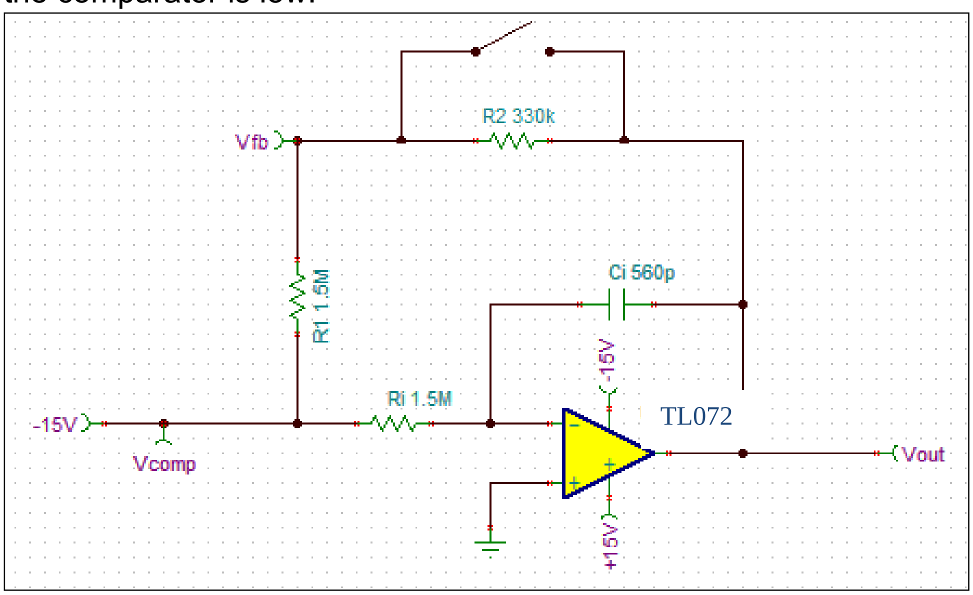
Schematic:



Schematic 4: Triangle Wave Generator

Equations:
The best way to analyze this circuit is to look at what happens when the comparator is high and what happens when the comparator is low.

When the comparator is low:



Schematic 5: Comparator Low

Schematic 5 shows the triangle generator after being redrawn with some simplifications made. For the sake of simplifying the discussion, the saturation voltage of the LP311 has been ignored. The 2N3904 that is being used as a switch is off when the comparator is low. Also, assume that Vout is sitting at 0V.

First, the comparator trip point will be determined such that the comparator output will transition from low to high. Since the negative input is tied to ground, when the output of the comparator is low, the positive input must be negative. Since the input to the integrator is negative, the output of the integrator will climb, during which V_{fb} becomes less negative. When V_{fb} passes through zero, the output of the comparator will switch. The output voltage that causes V_{fb} to reach 0V can be calculated as follows:

$$I_{R2} = \frac{V_{out} - V_{comp}}{R_1 + R_2}$$

$$V_{fb} = Vout - R_2 \cdot I_{R2}$$

$$Solve\ for\ Vout\ when\ V_{fb} = 0V$$

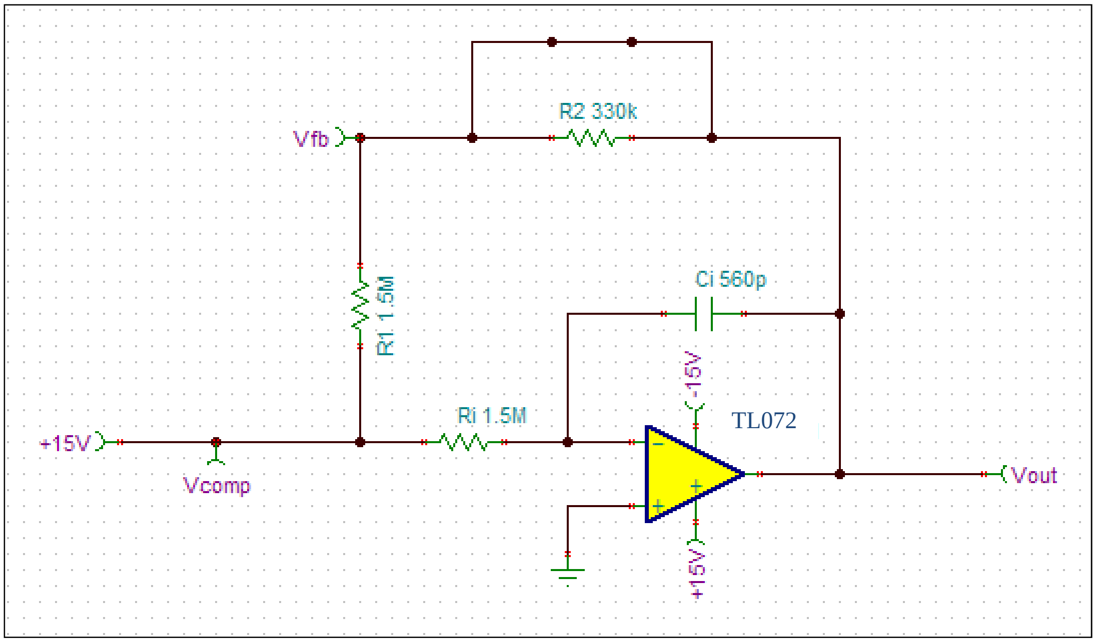
$$Vout = R_2 \cdot I_{R2} = 330k\Omega\left(\frac{V_{out} - (-15V)}{1.5M\Omega + 330k\Omega}\right)$$

$$Vout = 0.1803(Vout + 15V)$$

$$Vout = \frac{0.1803 \cdot 15V}{0.8197} = 3.3V$$

Therefore, the integrator will ramp up the output voltage until it hits 3.3V, at which point the comparator output will transition from low to high.

Now we will find the lower trip point. The schematic redrawn for a high comparator output is shown below:



Schematic 6: Comparator High

Notice that the BJT now looks like a short circuit because the transistor has become saturated, which shorts out R2. Therefore, it is easy to find the lower trip point because V_{fb} is equal to V_{out}. Therefore the lower trip point is 0V. This means that we will have a triangle wave that starts at 0V, climbs to 3.3V, and falls back to 0V. This process will repeat over and over.

Next, we will determine the climbing and falling times of the triangle wave. As shown before, the output voltage of the comparator can be calculated by:

$$V_{OUT} = V_0 - \frac{1}{RC}\int_{t_1}^{t_2} V_{IN}(t)dt$$

Looking at the rising output case first:

$$V_{OUT} = -\frac{1}{(1.5M\Omega)(560\,pF)}\int_0^{t_r}(-15V)dt$$

$$V_{OUT} = \frac{15}{(1.5M\Omega)(560\,pF)}\left(t_r - 0\right)$$

$$t_r = V_{OUT}\left(\frac{(1.5M\Omega)(560\,pF)}{15}\right)$$

Since the comparator has been calculated to switch when V_{OUT} reaches 3.3V:

$$t_r = (3.3V)\left(\frac{(1.5M\Omega)(560\,pF)}{15}\right) = 184.8\mu s$$

By similar logic, for the case when the comparator is high and the output is falling, where V_0 = 3.3V:

$$V_{OUT} = (3.3V) - \frac{1}{(1.5M\Omega)(560\,pF)} \int_0^{t_f} (15V)\,dt$$

$$V_{OUT} = (3.3V) - \frac{15}{(1.5M\Omega)(560\,pF)} \left(t_f - 0\right)$$

$$t_f = \left(3.3V - Vout\right) \frac{(1.5M\Omega)(560\,pF)}{15}$$

Since the comparator will switch when V_{OUT} is 0V:

$$t_f = \left(3.3V\right) \frac{(1.5M\Omega)(560\,pF)}{15} = 184.8\mu s$$

Notice that $t_r = t_f$. This tells us that the output triangle wave will be symmetrical. The frequency of the triangle wave can then be calculated by:

$$f_\Delta = \frac{1}{t_r + t_f} = 2.7kHz$$

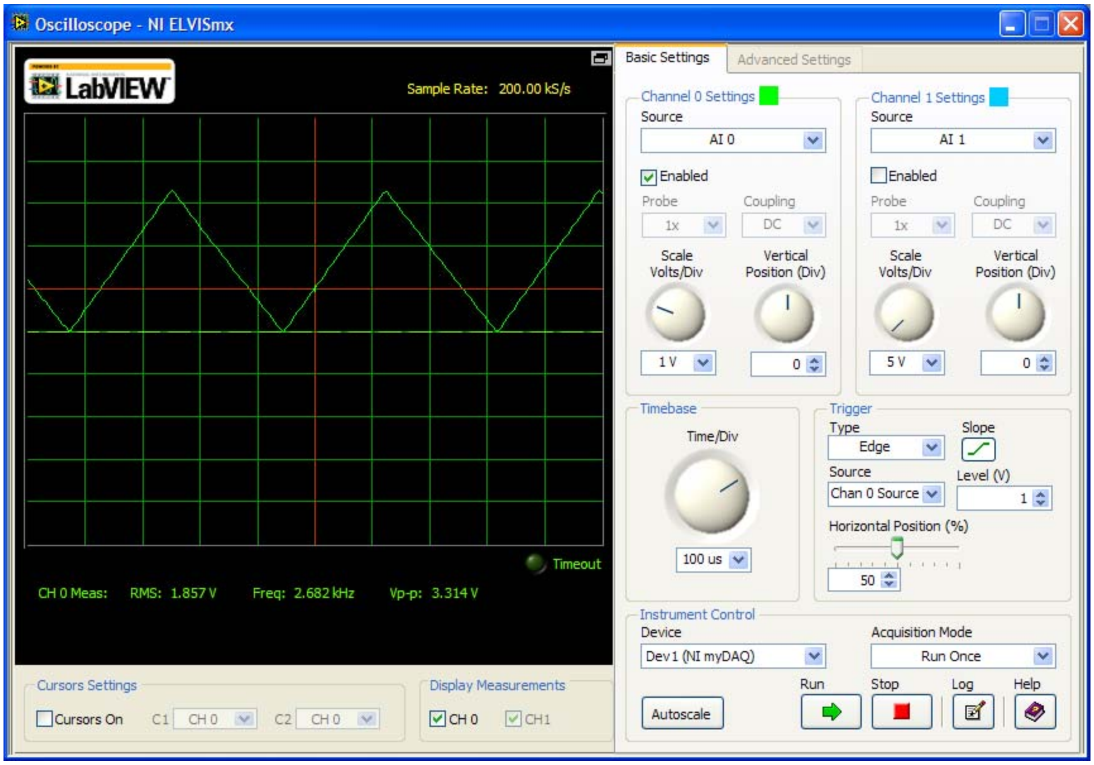
The screen shot below shows the output of the triangle wave generator. Notice that the triangle wave is not perfectly symmetric and the frequency is slightly off from our calculated value. This is due to the assumptions that were made in the beginning of the calculations. The second screen shot shows the triangle wave and comparator output together for a more intuitive sense of the circuit's operation (The frequency and voltages are slightly off due to loading effects at the comparator output).

Questions:

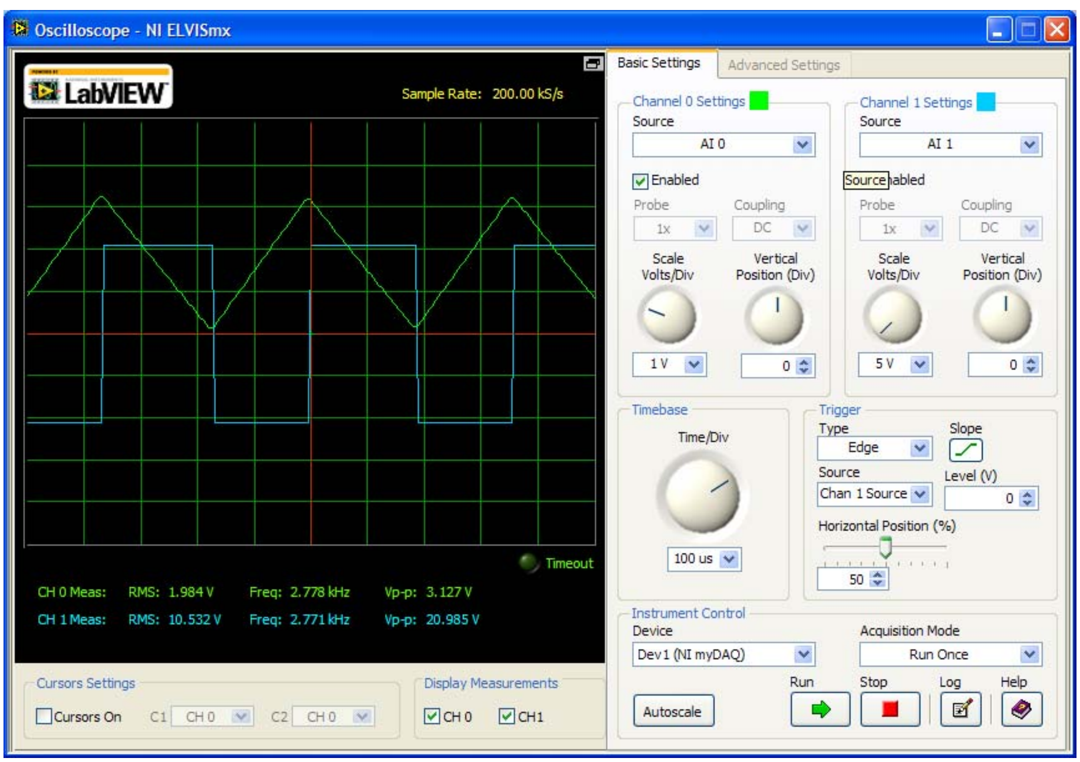
Try changing the integration capacitor from a 560pF to a 1000pF. How does the output change?

What would happen if R3 and T2 were removed from the circuit? Pull out the components and see how the output of the generator changes.

Screenshots:



Screenshot 1: Triangle Wave Output



Screenshot 2: Triangle Wave and Square Wave

Experiment 4: PWM Generator

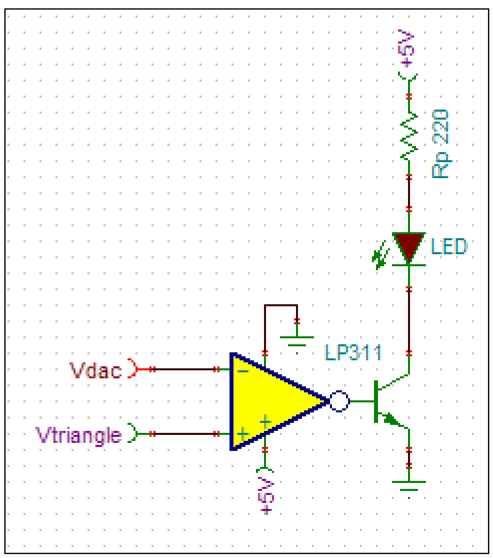
Introduction: This lab will put together the triangle wave generator and DAC for use as an LED dimmer.

Task 4.1: Pulse-Width Modulator

Introduction: Using a triangle wave and a comparator, a pulse-width modulated signal can be created. We will use the DAC as our input signal and the resulting output signal will have a duty cycle that is proportional to the input voltage. We will then use the PWM signal to dim an LED.

Build the circuit shown in Schematic 1 below. Hook up the triangle wave generator to the node labeled “Vtriangle” and the output of the DAC to the node labeled “Vdac.” Play around with the bits of the DAC and observe how the brightness of the LED changes.

Schematic:



Schematic 1: Pulse-Width Modulator

Equations:

Just like the comparator circuit from the triangle wave generator, when the positive input is lower than the negative, the output will be low. Therefore, when $V_{triangle}$ is greater than V_{dac}, the output is low. When the output is low, the output transistor looks like a short circuit. The forward diode voltage for a red LED is approximately 2V. Thus, the current through the LED can be calculated as:

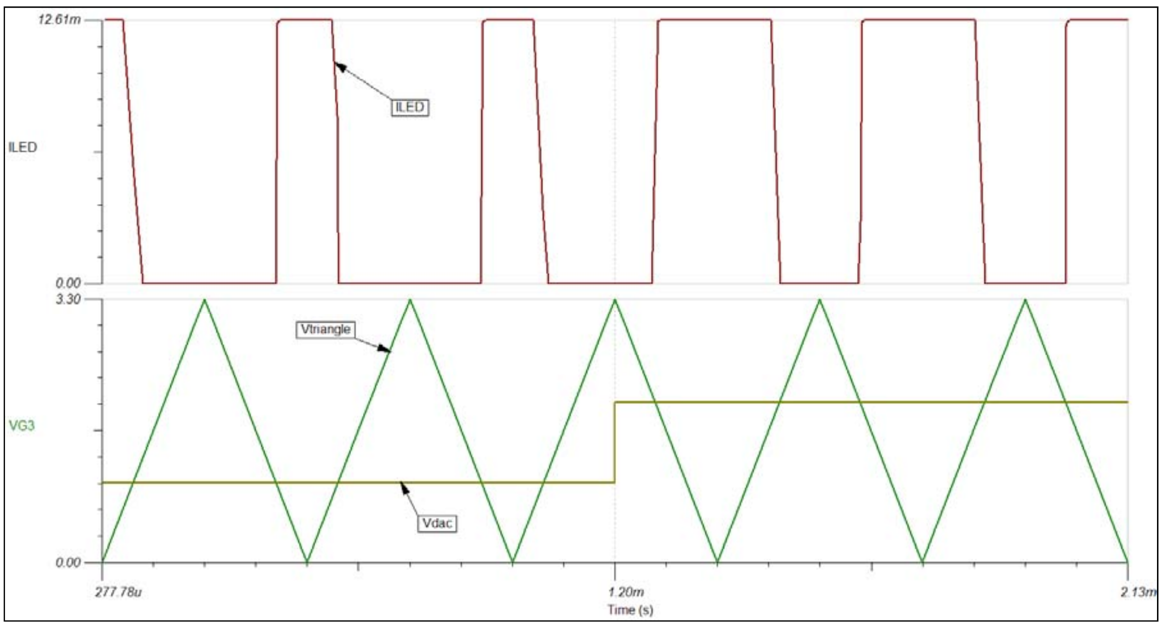
$$i_{LED} = \frac{5V - 2V}{R_P}$$

$$i_{LED} = \frac{5V - 2V}{220\Omega} = 13.6mA$$

When $V_{triangle}$ is greater than V_{dac}, the output is high. In the case of the LP311, when the output is high, this translates to the output transistor being off, which looks like an open circuit. If the output transistor looks like an open circuit, then no current can flow through the LED, and therefore the LED is off.

Since the triangle wave is periodic and the DAC output voltage is fixed at a DC level, the current through the LED will be a square wave with a duty cycle that is proportional to the DC level. Higher DAC output voltage will result in a higher current duty cycle, which will make the LED brighter. Lower DAC output voltage will result in a lower current duty cycle, which will make the LED dimmer. The SPICE simulation results are shown below. Two values of V_{dac} are shown in the simulation for better understanding of the operation of the circuit.

Screenshots:

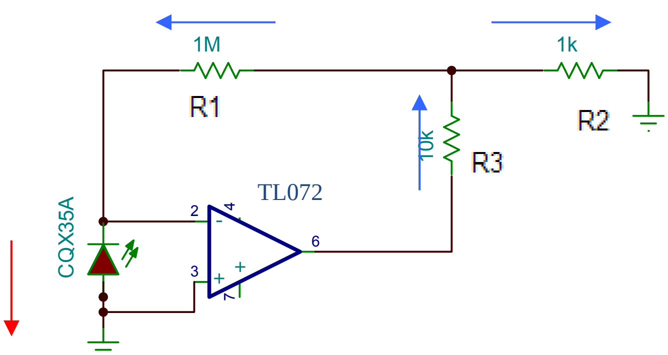


Screenshot 1: Simulation Results of PWM

Experiment 5: Light Detector Circuit

With the comparator output driving an LED, the next step will be to sense the intensity of the LED. This lab will build a light detector circuit using an op amp and an everyday light emitting diode (LED). It's common knowledge that when a current flows through an LED it emits light, but you may not have known that they also work in reverse. When an LED is exposed to light, it will create a small current from cathode to anode due to the photovoltaic effect. Using this property of LEDs, we can use them not only to produce light, but to detect it as well. This small current then can be amplified into a large voltage through the use of a transimpedance amplifier. However there are some limitations to this approach. First, LEDs best detect light at wavelengths shorter than the light they normally emit. This circuit will first be preformed using the same type of blue LED that is used in the "PWM Generator" lab.

This circuit will output a voltage that is proportional to the intensity of ambient light. The higher intensity the incident light the LED is exposed to, the higher the output voltage of the ambient light sensor circuit will be. The following schematic will be used to sense ambient light.



<u>Figure 5.1</u> Schematic of Light Detector Circuit

This circuit can be described very simply. When light shines on a LED there is a corresponding current that flows backwards through the LED (direction shown by red arrow in Figure Z.1). Also, by characteristics of ideal operational amplifiers, both input terminals equal 0 volts. The blue arrows represent the direction of current that will be used to solve for output voltage in terms of resistance.

$$\frac{V_{OUT}-V_1}{R3}=\frac{V_1-0}{R_1}+\frac{V_1-0}{R_2} \qquad \text{and} \qquad V_1=R_1 I_D$$

$$V_{OUT}-V_1=\frac{V_1 R_3}{R_1}+\frac{V_1 R_3}{R_2}$$

Substitute V_1 into the equation on the left:

$$V_{OUT} - (R_1 I_D) = \frac{(R_1 I_D) R_3}{R_1} + \frac{(R_1 I_D) R_3}{R_2}$$

$$V_{OUT} = I_D R_3 + \frac{R_1 I_D R_3}{R_2} + R_1 I_D$$

$$V_{OUT} = I_D \left(R_1 + R_3 + \frac{R_1 R_3}{R_2} \right)$$

For this lab the following component values will be used:

- R1 = 1M-ohm
- R2 = 1k-ohm
- R3 = 10k-ohm

Therefore the circuit ideally would have an output voltage equal to:

$$V_{OUT} = 11{,}010{,}000 * I_D$$

This means that a very small current can correspond to a measurable output voltage. To test this circuit shine the light from the LED on the output of the comparator directly at the LED of the light detector (point the tops of the LEDs towards each other). By increasing the output of the DAC the LED will get brighter and the measured voltage on the output of the light detector circuit will increase. By setting the output of the digital I/O pins to 0111 1111, or half of full value the following capture was made.

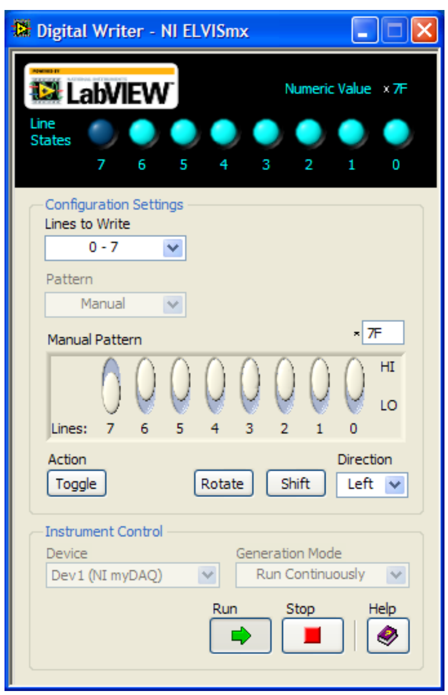


Figure 5.2 Setting the Digital I/O Pins

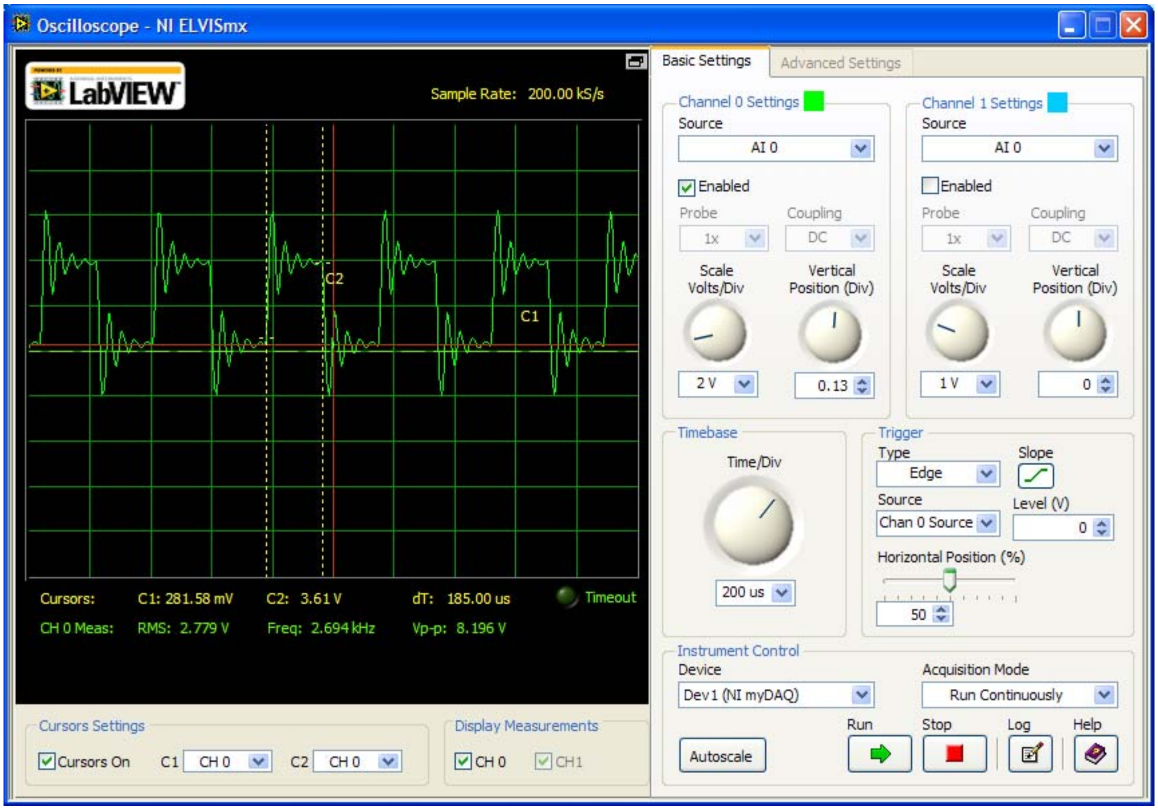


Figure 5.3 Capture of Light Detector Circuit

Notice that the Frequency of the captured waveform is at 2.694 kHz which is what the frequency of the triangle wave generator was set at. Also note that the duty cycle of the captured is at 49.8% due to the fact the DAC was set at 50% of full scale.

Infrared and remote controls

Most remote controls for electronic appliances use a near infrared diode to emit a beam of light that reaches the device. A 940 nm wavelength LED is typical. This infrared light is invisible to the human eye, but picked up by sensors on the receiving device.

By turning on and off the diode at a specific frequency and modulating (adding or mixing) data (keystrokes from the remote), a device can communicate wirelessly to the intended target (tv, radio, etc.). Looking back at your experiment, you can imagine an infrared detector in the appliance that will convert the pulses of light received to digital data.

Different manufacturers of infrared remote controls use different protocols to transmit the infrared commands. The RC-5 protocol that has its origins within Philips, uses, for instance, a total of 14 bits for each button press. The bit pattern is modulated onto a carrier frequency that, again, can be different for different manufacturers and standards, in the case of RC-5, a 36 kHz carrier is being used.

Other consumer infrared protocols are, for instance, the different SIRCS versions used by Sony, the RC-6 from Philips, the Ruwido R-Step, or the NEC TC101 protocol.

By changing the LED in the previous section from Blue to Red the IR light can be sensed in an everyday remote control. Below is a capture taken from pointing a remote control at the light detector circuit with and red LED in place of the original blue LED. The circuit was able to pick up a click of the remote control.

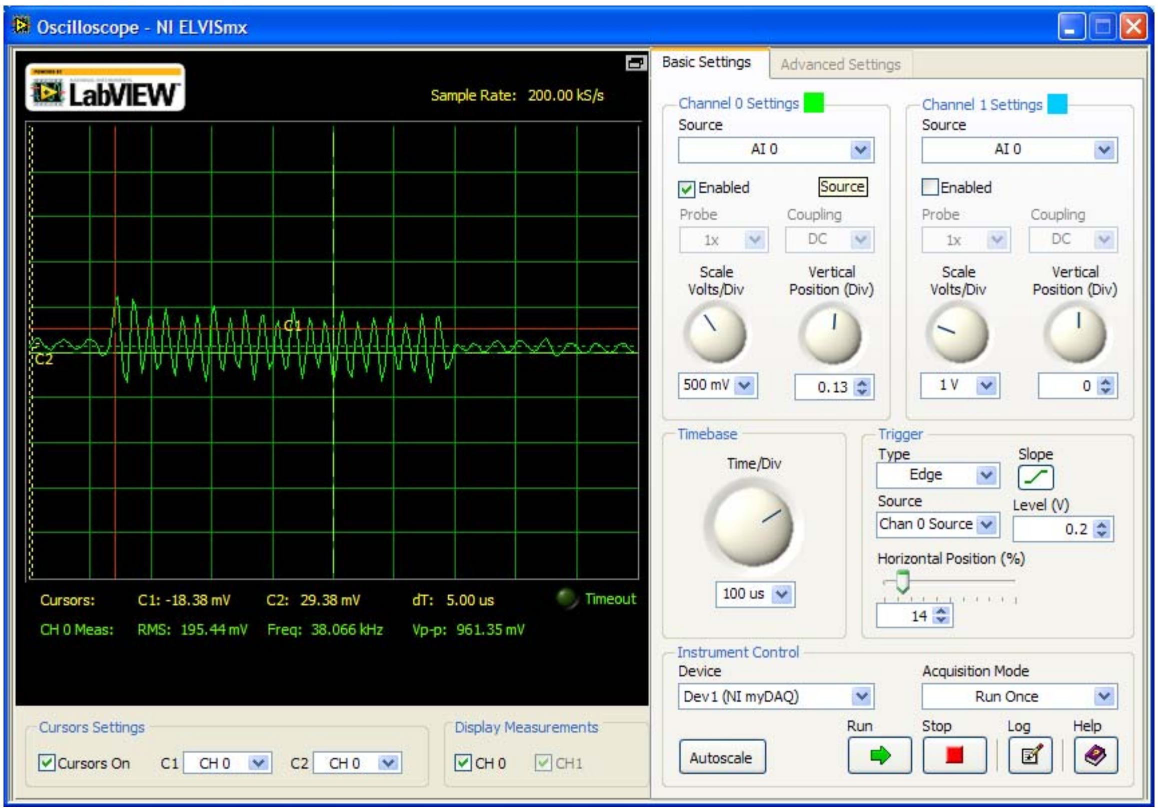


Figure 5.34 Capture of a click of a Remote Control

Try replacing the Red LED with the original Blue LED. Can you still detect a press of a button on the Remote Control?

Experiment 6: Audio

Introduction: This lab will demonstrate the theory and operation of non-inverting amplifiers as well as their practical application to amplify the signal of a microphone. We will construct a non-inverting amplifier to increase the output signal of a microphone, and use this circuit to measure the performance of a loudspeaker

Task 6.1: Microphone Pre-Amplifier

Electret capsule microphones are very popular due to their decent performance and low price. The simplest way to imagine the microphone's operation is to view it as a resistor whose value varies when a sound wave hits it. If a bias current flows through this varying resistance, it will create a small voltage that must be amplified for useable purposes.

Schematic:

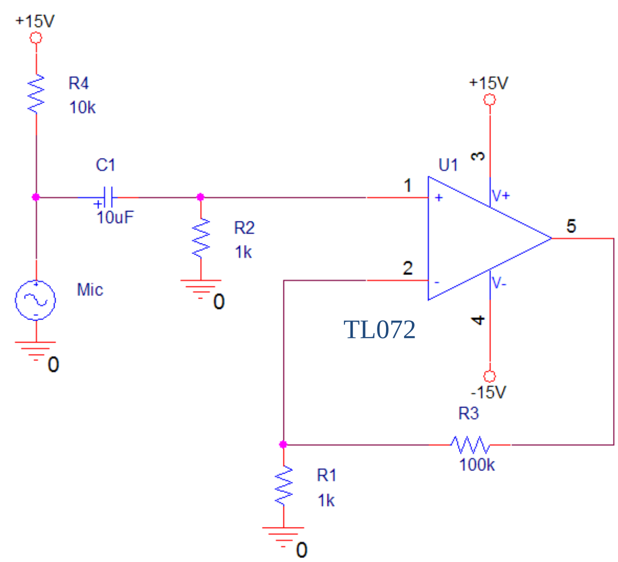


Figure 4: Microphone and amplifier circuit

In the above circuit, R4 provides the bias current for the microphone. Capacitor C1 prevents the DC voltage across the microphone from being amplified, while allowing the AC signal to pass to the amplifier. Resistor R2 provides a DC bias current path for the TL071 but also creates a high-pass filter with C1, whose cut off frequency can be determined as:

$$f_c = \frac{1}{2\pi R_2 C_1} = \frac{1}{2\pi(1k\Omega)(10uF)} = 15.92Hz$$

The TL071, R1 and R3 form the non-inverting amplifier whose gain can be described with the equation:

$$A_V = \frac{V_{OUT}}{V_{IN}} = 1 + \frac{R_3}{R_1} = 1 + \frac{100k\Omega}{1k\Omega} = 101$$

PSpice was used to verify the performance of the circuit using an AC Sweep.

Simulated Performance:

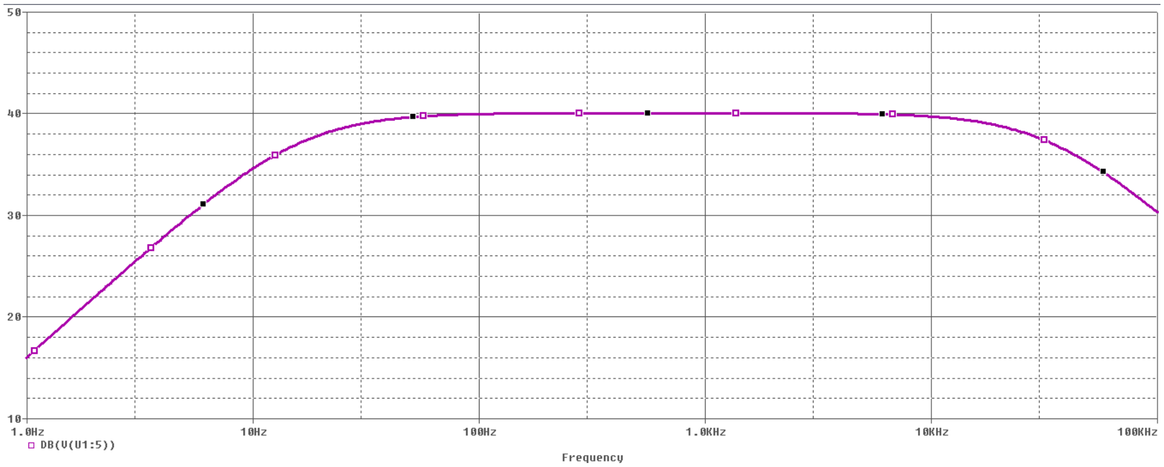


Figure 5: Frequency response of the microphone amplifier circuit.

The simulation shows that the circuit has a gain of 40 dB as predicted and it's low frequency roll-off occurs around 15.9 Hz. The upper frequency roll-off is created by the bandwidth of the opamp itself. For small signals, the easiest method to predict the bandwidth of the amplifier is to use the gain/bandwidth product (GBW) specified by the manufacturer. The TL072 has a GBW of 3 MHZ, by dividing this number by our gain of 101, we can predict the 30 kHz bandwidth of the amplifier seen in the simulation.

The Bode application included with the myDAQ can be used to verify the operation of the circuit once it's built. Be sure to install the capacitor with polarity in the correct orientation and care should be taken when inserting the microphone into the breadboard.

Figure 6: Real-world performance of the microphone amplifier circuit

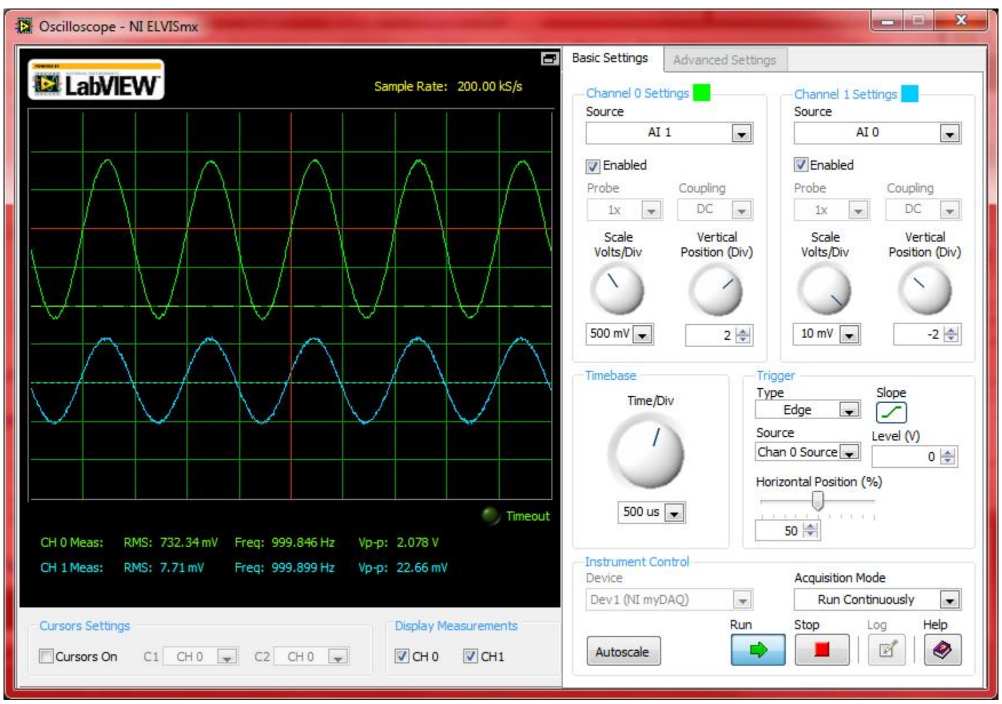


Figure 7: Screenshot using "scope" to verify the operation of the circuit.

Task 6.2: Measuring the performance of a loudspeaker

The non-inverting amplifier constructed in the previous section makes it much easier to interface the microphone to the myDAQ. Now the microphone can be used for real-world measurements and recording.

Schematic:

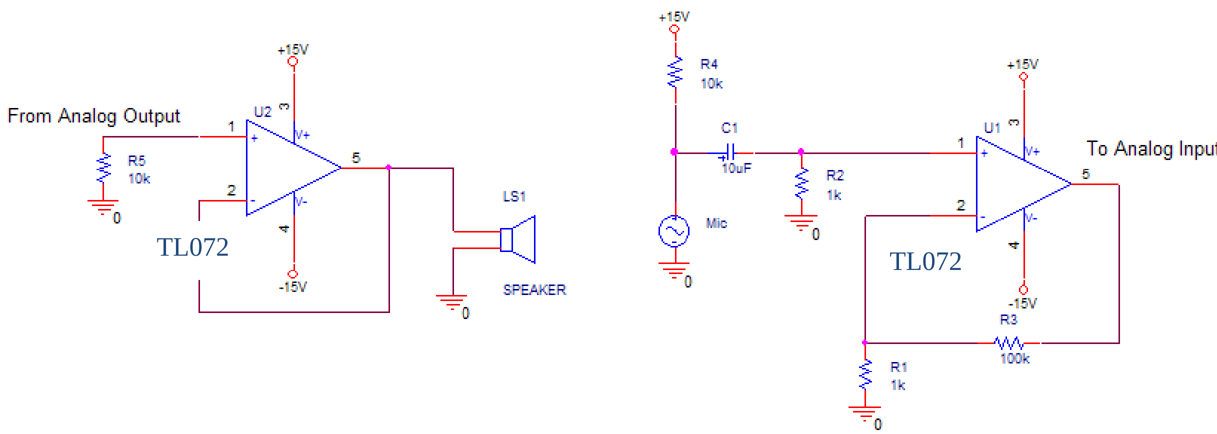


Figure 8: Circuit configuration for measuring the performance of a loudspeaker.

In order to drive the included speaker with the myDAQ, a buffer circuit needs to be constructed. By removing the feedback resistors from the non-inverting amplifier circuit and connecting the output of the opamp directly to the inverting input, the circuit's gain is now reduced to 1. This means that the voltage at the output of the opamp will match the voltage at the input. You may be wondering what the benefit of the circuit is if it doesn't provide any gain to the signal? The advantage of a buffer is that it provides the necessary current to drive a low impedance load while still presenting a high-impedance load to the source. Notice in the above schematic that the analog output of the myDAQ does provide current to the 150 Ohm speaker, the opamp does. The impedance "seen" by the analog output is the value of R5, 10k Ohms in this case.

By connecting the buffer input to the analog output of the myDAQ and the microphone amplifier to an analog input, the frequency response of the speaker can be measured. The DSA application can be used to view the harmonic content of the speaker's output.

Figure 9: Frequency response of the included loudspeaker, measured by the microphone circuit.

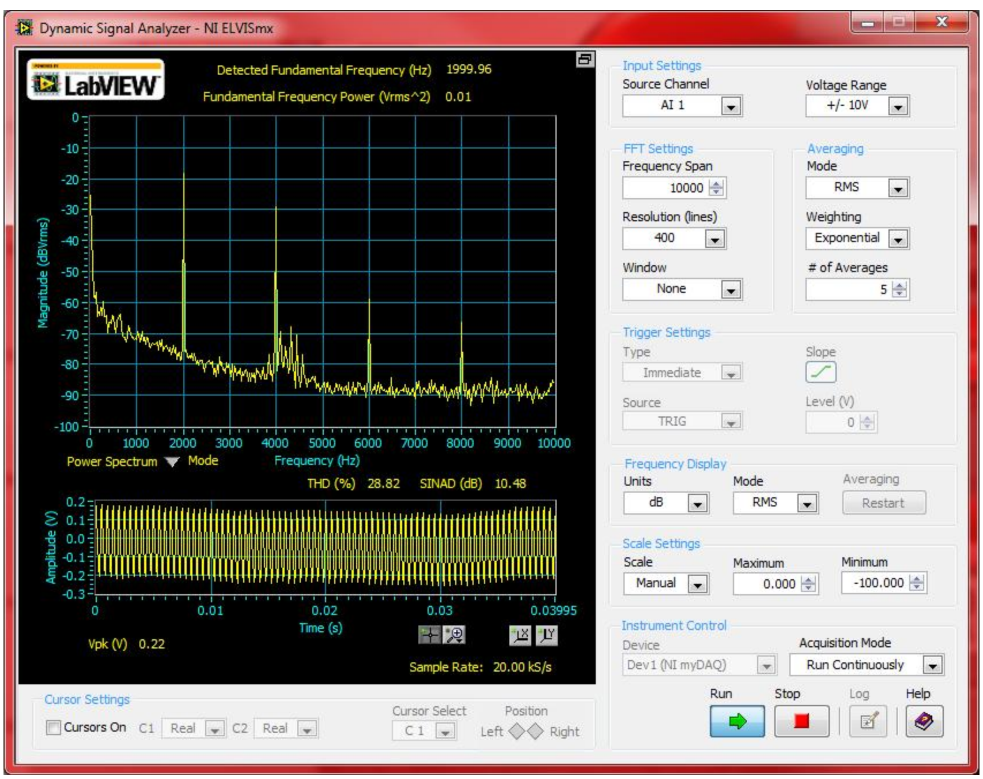


Figure 10: Harmonic content of the included loudspeaker when playing a 2kHz tone.