EECE 2413
Electronics Laboratory

Lab #1: Operational Amplifiers (Op Amps)

Goals

The goals of this lab are to review the use of DC power supplies, function generators and oscilloscopes. Afterwards, you will build and study circuits that use operational amplifiers. Op amps are very useful and versatile circuit elements, but they also have some flaws. Therefore, some of the limitations of op amps will also be investigated.

Once you are more familiar with the way that op amps work, you will design and test a simple audio amplifier for a microphone.

Prelab

Prelabs will be collected for grading at the *beginning* of the lab. Keep a photocopy for your own use during the lab.

1. Read the review section and familiarize yourself with the test equipment used in this lab.
2. A circuit for a RS-232 serial port requires a ±12 volt DC power supply and a +5 volt power supply. First sketch two Agilent E3647A power supplies and their output terminals. Then show on your sketch how you would connect the terminals to create these three voltages (see Fig. 1).
3. An Agilent 33220A function generator is set-up to produce an output of $v(t)=5+10\sin(2\pi*100t)$ at the Output connector.
   - 3.1. Describe how you would increase the frequency to obtain an (open-circuit) output of $5+10\sin(2\pi*2000t)$ volts? Give step-by-step instructions explaining which buttons you would press.
   - 3.2. If a 50-ohm resistor is then placed across the Output of the Function Generator, what would be the new output voltage?
   - 3.3. Describe how you would eliminate the DC offset voltage? Give step-by-step instructions explaining which buttons you would press.
4. Consider the op amp circuit shown in Figure 5.
   What is the voltage gain of this amplifier if R1 = 50 Ω and R2 = 5000 Ω?

What is the input resistance of this amplifier?
How much would the output voltage of the 33220A function generator decrease when it is connected to R1? (Hint: What is the voltage at the inverting op amp input? Remember to take into account the resistance of the function generator.)

Part 1: Review of power supplies, function generators, and oscilloscopes.

The Power Supply

You will use a DC power supply for almost every electronic circuit, so let's start by reviewing a few of the features of a typical DC supply. In Figure 1, the front panel of a DC power supply is shown. This DC supply contains three independent voltage sources. The output voltage of each source is controlled by the big black knob at the upper right corner of the power supply. The second knob at the upper right is used to control the *maximum allowed current. However, you need to use the "Select" button from the panel to hover into voltage and current from the display for a change.* The voltage and current value can be seen in the digital display.



Figure 1a. The KEYSIGHT EDU36311A Dual DC Power Supply

Setting the voltage current mode limit:

You can set the voltage and current limit values from the front panel using the following method.

1. Turn on the power supply.
2. Click on "*Unlock*" reflected on the display to start.



Figure 1b. The KEYSIGHT EDU36311A Dual DC Power Supply

3. There are three supply channels, and the limit values are displayed for each corresponding channel. *The first channel can provide up to 6V, however, the second and third channel can supply up to 30V.*
4. Set the knob to current control mode by pressing the key.
5. Move the blinking digit to the appropriate position using the resolution selection keys, and change the blinking digit value to the desired current limit by turning the control knob.
6. Press the "*On*" key to enable the corresponding output. After about 5 seconds, the display will go to output monitoring mode automatically to display the voltage and current at the output.

For this lab you should set the current limit to **50 mA.**

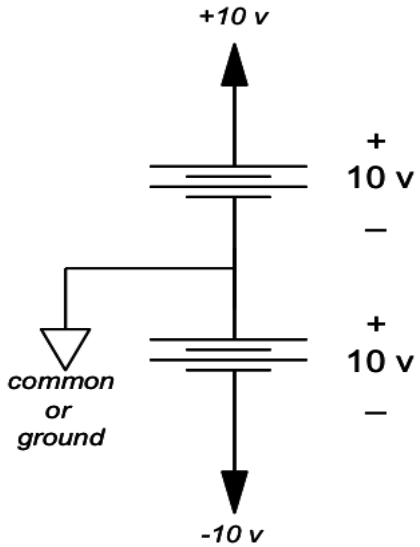


Figure 2. Schematic for ±10 volt Configuring a dual power supply for plus and minus voltages.

Often a circuit will require both a positive voltage source and a negative voltage source. Figure 2 shows the schematic for a ±10 volt source. Often, only the up- and down-arrows will be shown, and the remainder of the circuit in Figure 2 is implied. This voltage supply configuration is usually used with op amp circuits as you will see later in this lab.

In Figure 1b, notice the four terminals on the lower right-hand side. Output 2 has DC+ and DC- terminals, and Output 3 has DC+ and DC- terminals. To configure your power supply as a ±10 volt source, follow the connections shown in Figure 1: Output 2's DC+ has a red wire connected to it. This is +10 volts. Output 2's DC- is connected to Output 3's DC+ with a green wire. This green wire is the common node in Figure 2. If it is important to have an *earth ground* in your experiment, you should connect another wire from Output 2's DC- to the ground of your experimental board. Finally, the -10 volt source is connected using another green wire at Output 3's DC- terminal of the power supply.

The Function Generator

The *function generator* lets you inject well-controlled voltage signals into your circuit. Typically, you can select sine waves, triangle waves and square waves and then control the frequency and voltage amplitude of the wave. Most circuits are meant to receive "real world" signals, but these signals are often noisy and irregular. Later, you will use a microphone and you will see how difficult it is to make a good measurement of circuit performance due to noise and interference. The function generator produces a clean, regular signal so that you can test your circuit.

The Keysight 33212A function generator is shown in Figure 3 below. After turning on the signal generator by using the white square button at the bottom left corner, click on the "*System*" button to start selecting the waveform parameters. The waveform is selected by pushing one of the buttons beside the screen. For this example, let us assume that the sine wave is selected. This will allow us to generate a signal in the form of

$$\mathbf{v(}t\mathbf{) = V_{DC} + A}\ sin\ \mathbf{(2\pi \cdot} f \cdot t\mathbf{)}$$

where V_{DC} is a DC OFFSET voltage that is added to the sine wave. The AMPLITUDE is the peak value of the sine wave, A, and *f* is the amplitude and FREQUENCY of the sine wave.

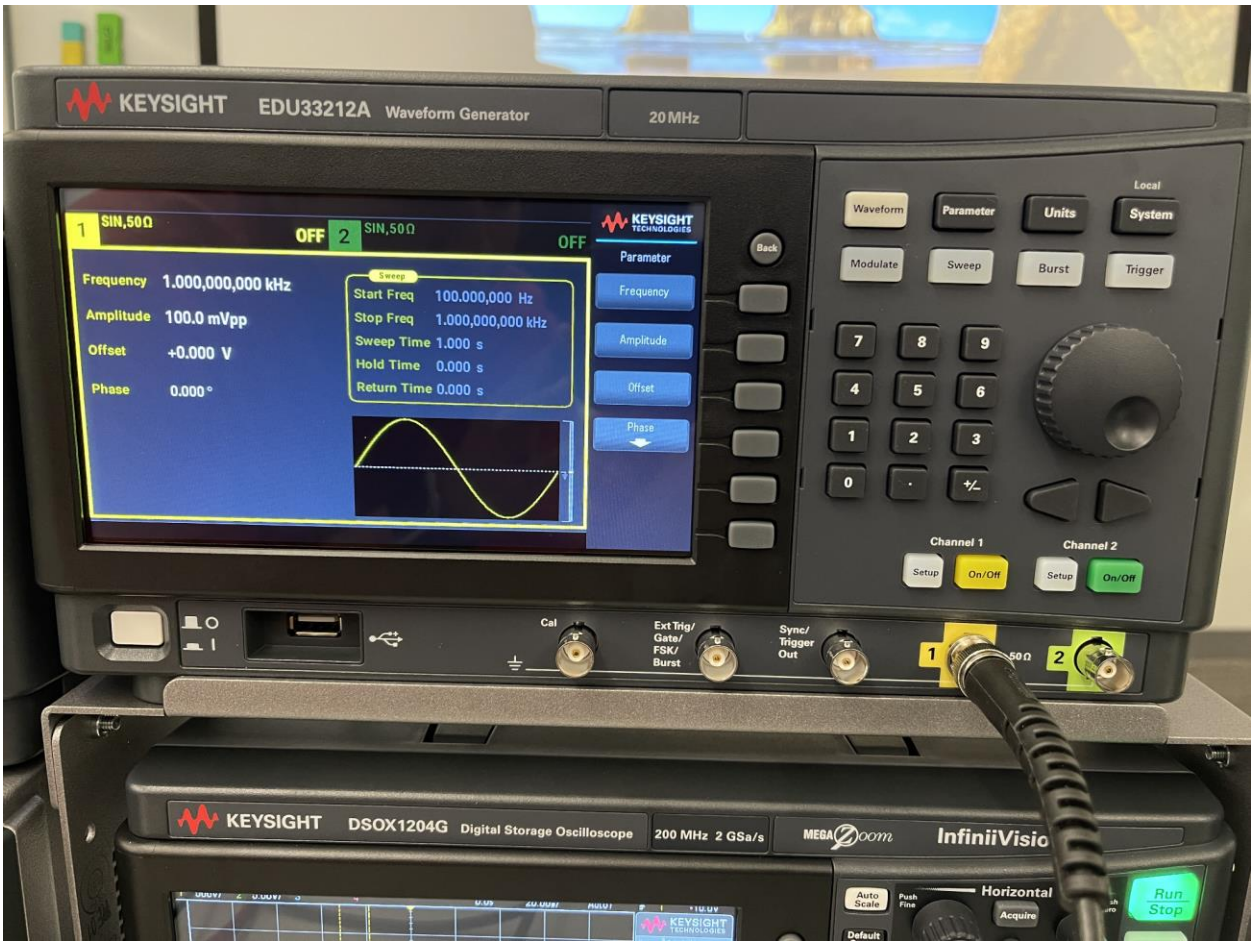
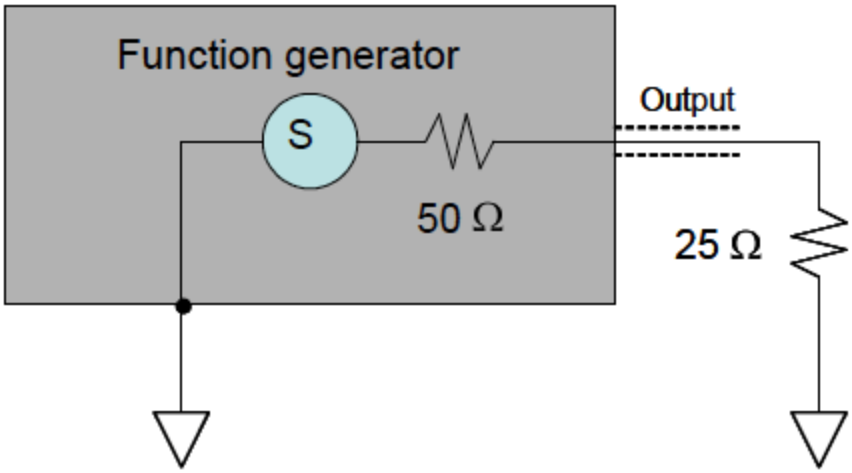


Figure 3. KEYSIGHT 33212A Function Generator

OUTPUT:

The output voltage from the function generator is produced at the BNC connector Channel 1 and Channel 2. Press the “on/off” key above the selected channel to enable the Output connector. Similar to the oscilloscope, the outer conductor of the BNC is grounded, so make sure that you always connect this to the ground node of your circuit. [Note: This output has a Thevenin equivalent resistance of 50 Ω, which means that when you connect a 25 Ω resistor to this output, the *output voltage* will be divided by three as shown by the voltage divider below: 25 Ω /(25 Ω + 50 Ω)]



AMPLITUDE:

The output voltage level of the sine wave can be set by the numeric keypad on the right. To set the amplitude, you need to press the softkey beside "Amplitude" first. After that, use the numeric keypad to enter the desired amplitude value and select the desired units using the softkeys. [Note: this voltage range is correct if nothing is connected to Output. If you connect a circuit to Output, the actual output voltage drops due to the internal 50 Ω Thevenin equivalent resistance.]

FREQUENCY:

The output frequency of the sine wave can be set by the numeric keypad on the right. To set the frequency, you need to press the softkey beside "Frequency" first. You can press the softkey again to change the period of the output signal. After that, using the numeric keypad enter the desired frequency value and select the desired units using the softkeys.

DC OFFSET:

This knob controls how much DC voltage is added to the signal. To set the offset voltage, you need to press the softkey beside "Offset" first. After that, using the numeric keypad enter the desired DC offset value and select the desired units using the softkeys. Usually we will leave the DC offset at 0.

PHASE:

The SWEEP features of the function generator allow you to change the phase of the selected waveform. We might not need to use this feature in EECE 2413, however.

The Oscilloscope

Along with the voltmeter, ammeter, and ohmmeter, the oscilloscope is one of the most important measurement tools. A voltmeter allows us to measure a DC or average voltage, but an oscilloscope lets us "see" a time-varying voltage signal, v(*t*). In this lab you will use the 4-channel digital scope shown in Figure 4.

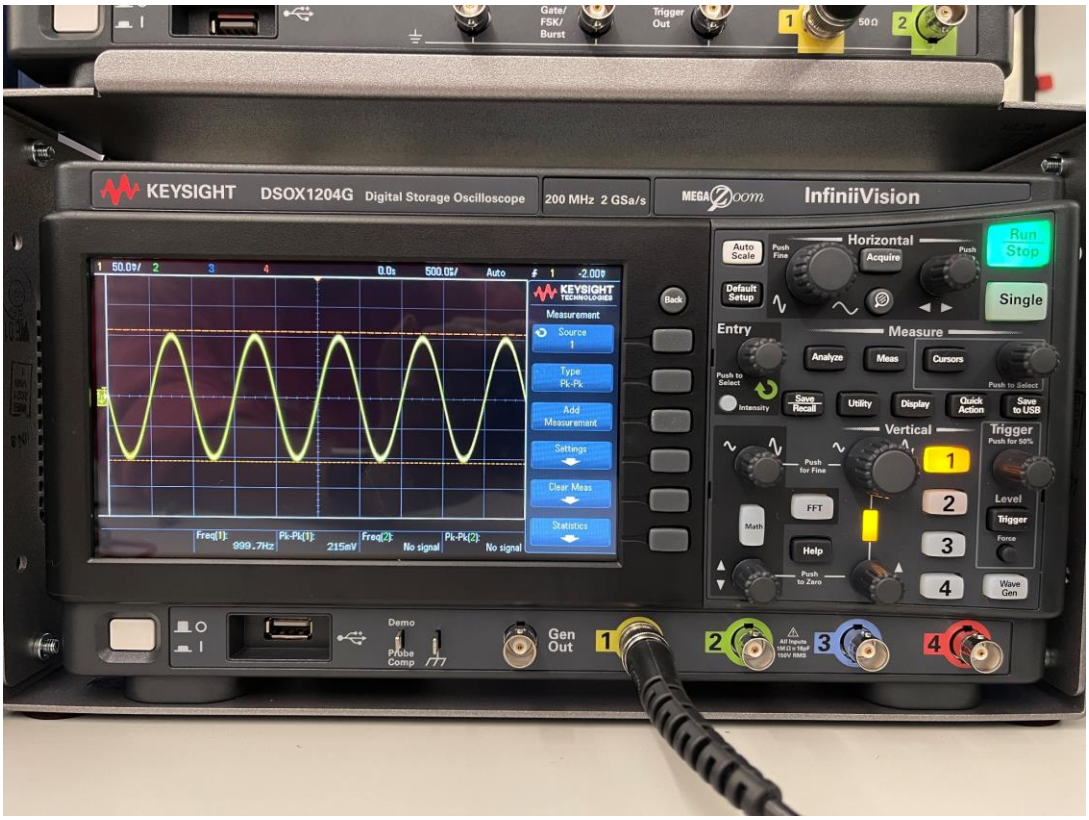


Figure 4. KEYSIGHT DSOX 1204G oscilloscope

Most people think that it is best to learn how to use a scope by simply playing around with it. Here are a few basics to get you started.

Inputs: CH1, CH2, CH3 and CH4

Each "channel" of an oscilloscope accepts a separate voltage input. This scope has four channels (CH1, CH2, CH3 and CH4). The input is a shielded BNC connector. **The outside conductor of the BNC connector is ground!** A common mistake is to

connect this grounded lead to a node in your circuit that is *not* ground. This almost always causes your circuit to fail. Remember, always connect the grounded lead of the scope to the ground of your circuit. Then you may probe your circuit using the other lead –*i.e.,* the lead that is connected to the inner pin of the BNC input of the scope. Each channel may be turned on or off by pressing the 1, 2, 3 and 4 button.

VOLTS/DIV:

The screen is divided by a grid, and each grid line is called a "division" or DIV. The screen displays the input voltage vs. time. The voltage axis is controlled by the medium knob right beside the display. In Figure 3, a sine wave is applied to channel 1. Where's zero volts? On the left side of the screen a small arrow and the number 1 are displayed (1). This is zero volts for channel 1. Likewise, the minimum value of the sine wave is -750mV. From the Measurement at the bottom of the display will show the peak to peak amplitude of the signal.

CH1, CH2, CH3, CH4:

Pressing channel 1,2,3,4 on/off buttons will display a large number of options on the bottom of the screen. Most important is the COUPLING type. Press the channels' on/off key, and then press the Coupling softkey to select the input channel coupling. The options are AC or DC. DC COUPLING shows you the entire signal including any DC voltage component it contains. For example, if the input is $v(t) = 10.0 + 0.2*\sin(2\pi \cdot 1000t)$ volts, the display will show a 0.2 V sine wave located 10 volts above the zero level. It can be hard to see such a small signal (0.2 V) when a large DC voltage is present (10 V). The AC COUPLING lets you eliminate the DC part of the signal and examine just the AC part. In our example, AC COUPLING would cause the scope to display $v(t) = 0.2*\sin(2\pi \cdot 1000t)$ volts even though the actual signal is $v(t) = 10.0 + 0.2*\sin(2\pi \cdot 1000t)$ volts. Many circuits produce very small signals that are superimposed on DC voltages, so the AC COUPLING feature can be quite useful. You will learn more on this topic when we study transistors.

Vertical:

The zero voltage level displayed on the screen can be adjusted up or down using the big knobs in the vertical section in the front panel. This is useful if you are looking at two channels with overlapping waveforms. Simply move channel 1 up and move channel 2 down to get a clearer view of each. However, the "Autoscale" button is very useful to check signals from multiple channels rather than moving the signals manually.

SEC/DIV:

The scope gives you a view of voltage vs. time. The time axis is controlled by the knob beside the display. This tells us how many seconds each horizontal division on the screen represents. Use this control to spread or compress the horizontal axis so that you

can see the signal clearly. The SEC/DIV setting is indicated at the top of the screen. As an example, if the scope is set to 500ms per division. The *period* of the sine wave is approximately 2 divisions * 500ms per division = 1000ms = 1 ms. Therefore, the frequency of the sine wave is $f = 1 / T = 1$ kHz.

Horizontal:

The big knob under the horizontal section allows you to shift the signal horizontally on the screen, and functions just like the vertical position knob.

TRIGGER LEVEL:

On the far bottom right side of the scope are the trigger controls. The scope can be thought of as a camera that takes a picture of the signal. The TRIGGER tells the scope when to take the picture. More precisely, the trigger tells the scope to begin taking and displaying the input signal when a certain *voltage level* is reached. Notice that there is a small triangle marker (T) on the left side of the screen. This marker shows the voltage level that will trigger the scope. This arrow will move up or down as you rotate the TRIGGER LEVEL knob. It is critical that this marker (T) be positioned between the maximum and minimum voltage on the screen, otherwise the "pictures" will be taken at random times, and the voltage trace will appear to jump around on the screen.

CURSORS:

Another helpful feature is the Cursors button. This will activate *guidelines* on the screen that let you measure voltage and time using the vertical and horizontal POSITION knobs. Press the Cursors button. The Cursors menu will be displayed on the screen. Select the waveform or the channel (1/2/3/4) to be measured by pressing the Source softkey and selecting the channel. To measure time select the X cursor, and to measure voltage select the Y cursor. Use the Entry knob to move the cursors. To turn the cursors off, press the Cursors button again.

SPECIAL Features:

If you have applied a signal to the scope input and can't display a signal on the screen, you can use the Autoscale button located at the left. Pushing this button will cause the scope to examine the input signal and automatically choose the settings outlined above. This usually works, but not always. Also, don't rely on this feature too much – not every scope can perform an Autoscale, and you should know how to use all scopes.

The "Meas" button will cause the scope to determine the peak-to-peak/peak and several other entities of a signal, the mean of the signal, the frequency of a signal, etc.

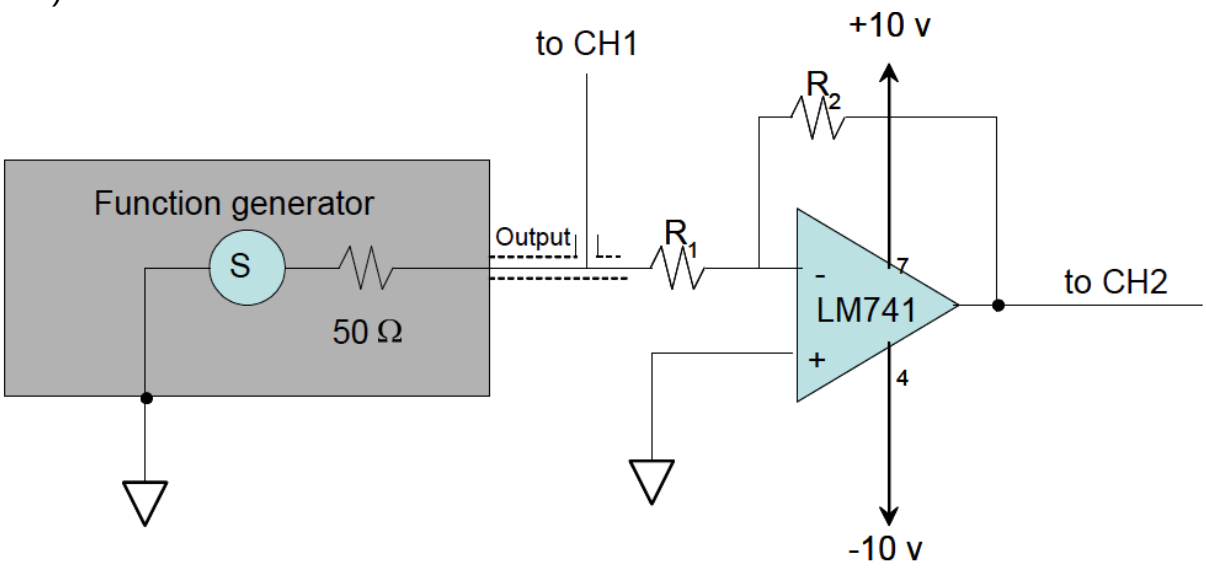
Use this feature with some caution! First, make certain that the screen displays a stable, noise-free signal that is complete (not chopped off). Second, if a question mark is displayed after the data, the scope is telling you that the result is probably not correct: "FREQ 1.937 kHz?" should not be trusted. Either adjust the scope for a clearer display or manually calculate the frequency using the SEC/DIV information.

Part 2: Operation Amplifiers

An operational amplifier, or op amp, is an integrated circuit that contains at least 20 transistors. You can see the schematic for the LM741CN op amp on page 4 of the National Semiconductor spec sheet attached to this lab. As you progress through Electronics, you will begin to understand this schematic, but for now we will just worry about the op amp's overall performance. The pin-out diagrams are shown on the first page of the spec sheet – we will be using the *dual-in-line package* op amp. Remember that the U-shaped indent on the package shows you where pin 1 is located.

The *ideal* op amp has infinite voltage gain. You can see from the spec sheet (p. 3) that the LM741C op amp typically has a voltage gain of 200 V/mV or $A_V \sim 200{,}000$ V/V. For some chips, however, the gain may be as low as 20,000 V/V (see the **Min** spec!). To make the op amp performance more repeatable, we add *negative feedback*. The feedback reduces the voltage gain to a much smaller value, but that value is controlled by external resistors.

Figure 5 shows the op amp used in the *inverting amplifier configuration.* R_2 provides feedback of the signal from the OUTPUT (pin 6) back to the INVERTING INPUT (pin 2).



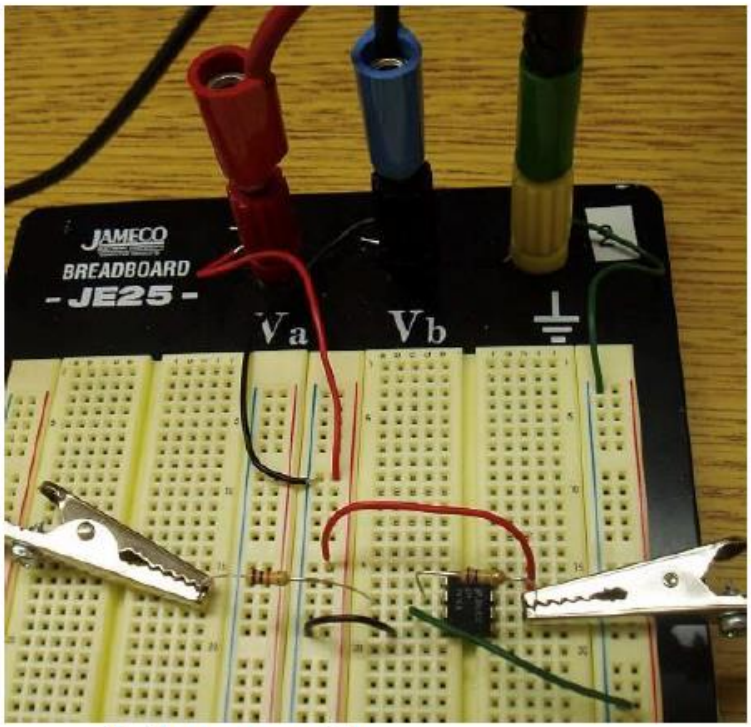


Figure 5. The LM741CN op amp used in the inverting amplifier configuration. The input signal is applied through the clip on the left, and the scope is connected through the clip on the right. All ground connections are stacked on the rightmost banana plug. Keep the layout neat!

Voltage Gain:

Using R_1 = 10kΩ and R_2 = 100kΩ, construct the circuit shown in Figure 5. Use a BNC Tee at the output of the function generator so that the signal can be sent to the oscilloscope (CH1) and to your op amp circuit. Connect the output of the op amp to CH2 of the oscilloscope. Adjust the AMPLITUDE of the function generator to be approximately 0.75 V (peak-to-peak) and set the FREQUENCY to approximately 8 kHz. You should see oscilloscope traces similar to Figure 6 below. Notice that the gain is -10 (15.2V/1.52V) and the output decreases as the input increases (an inverting amplifier). Notice that the output may be slightly shifted from 180^0 relative to the input. **Determine the phase shift of this op amp circuit by (1) finding the time lag (DT) between channel 1 (input) and channel 2 (output). (2) The phase shift is calculated using $360^0 \times (DT/T)$ where T is the period of the sine wave (T=1/*f*).**

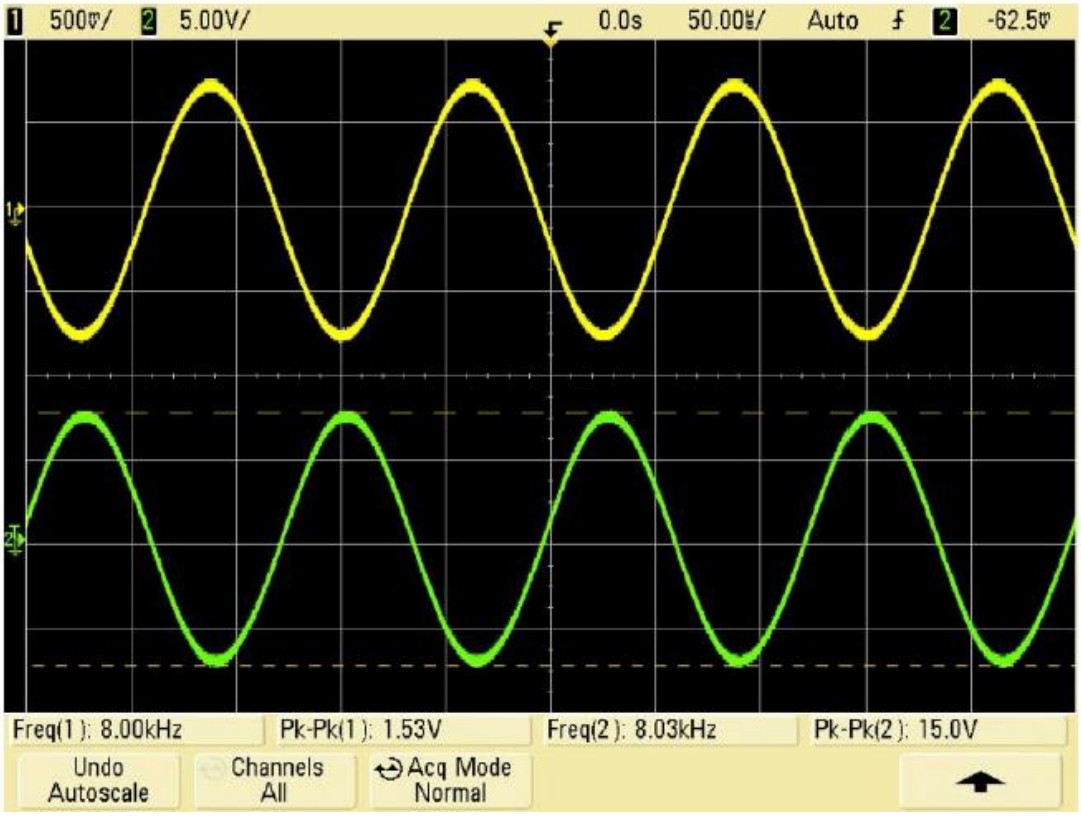


Figure 6. Oscilloscope traces for the inverting amplifier of Fig. 5.

Clipping:

The output voltage of the op amp cannot exceed the power supply voltage. When the output gets close to the power supply voltage the top and/or bottom of the waveform is clipped off. Increase the amplitude of the function generator until you see the output waveform start clipping on the oscilloscope. You will probably need to change the VOLTS/DIV setting in order to keep the output trace displayed on the screen. When clipping, **use the CURSOR function to determine the maximum (most positive) and minimum (most negative) possible output voltages produced by the op amp. How do these voltages compare with the power supply voltages?**
Read the spec sheet to determine the TYPICAL output voltage swing of the LM741C op amp for a ±15v DC supply, and determine the typical "*difference*" between them in volts for $R_L \geq 10k\Omega$. Is your op amp within the typical "*difference*" specification for the ±10v supplies used in this experiment? Explain. Look at the schematic diagram for the LM741 and explain why the output voltage is always less than the power supply voltage.

Figure 7 shows a typical waveform where the output voltage is clipped as it approaches the power supply voltage.

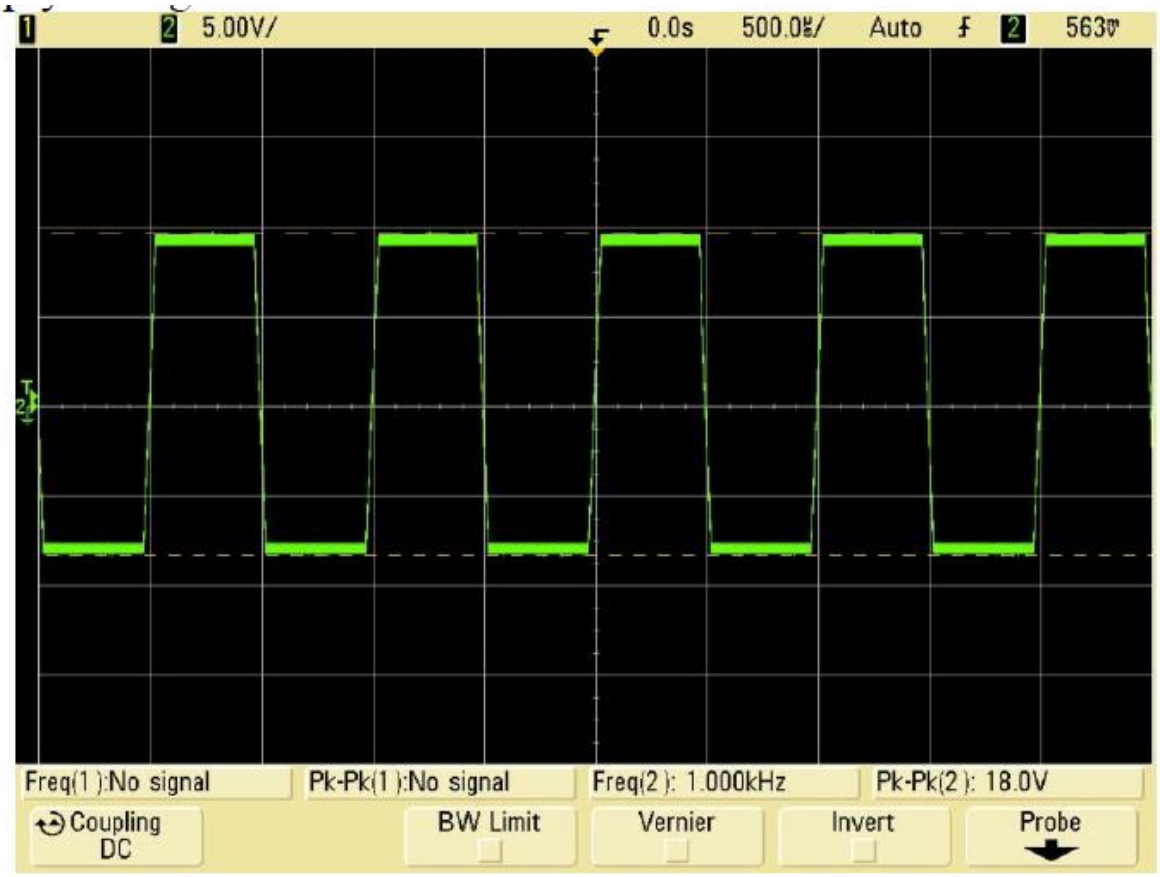


Figure 7. Clipped output signal from a 741 op amp

Slew Rate:

Return the function generator amplitude to approximately 0.75 V volts. Next, slowly increase the frequency while observing the output waveform. Notice that the output waveform gradually transitions from being a sine wave to a triangular wave! This is because the output of the op amp is *slew rate limited*. The fastest a 741 op amp's output voltage can increase or decrease is by about 0.5 volts every microsecond. So if the input signal demands a faster change, the output just ramps to that value as fast as it can. Ultimately, this may limit the usefulness of an op amp. Certainly this circuit would be useless in a 1.8 GHz cell phone application!

Switch the input signal to a square wave. What happens? Explain.

Typical slew rate limited output is shown below:

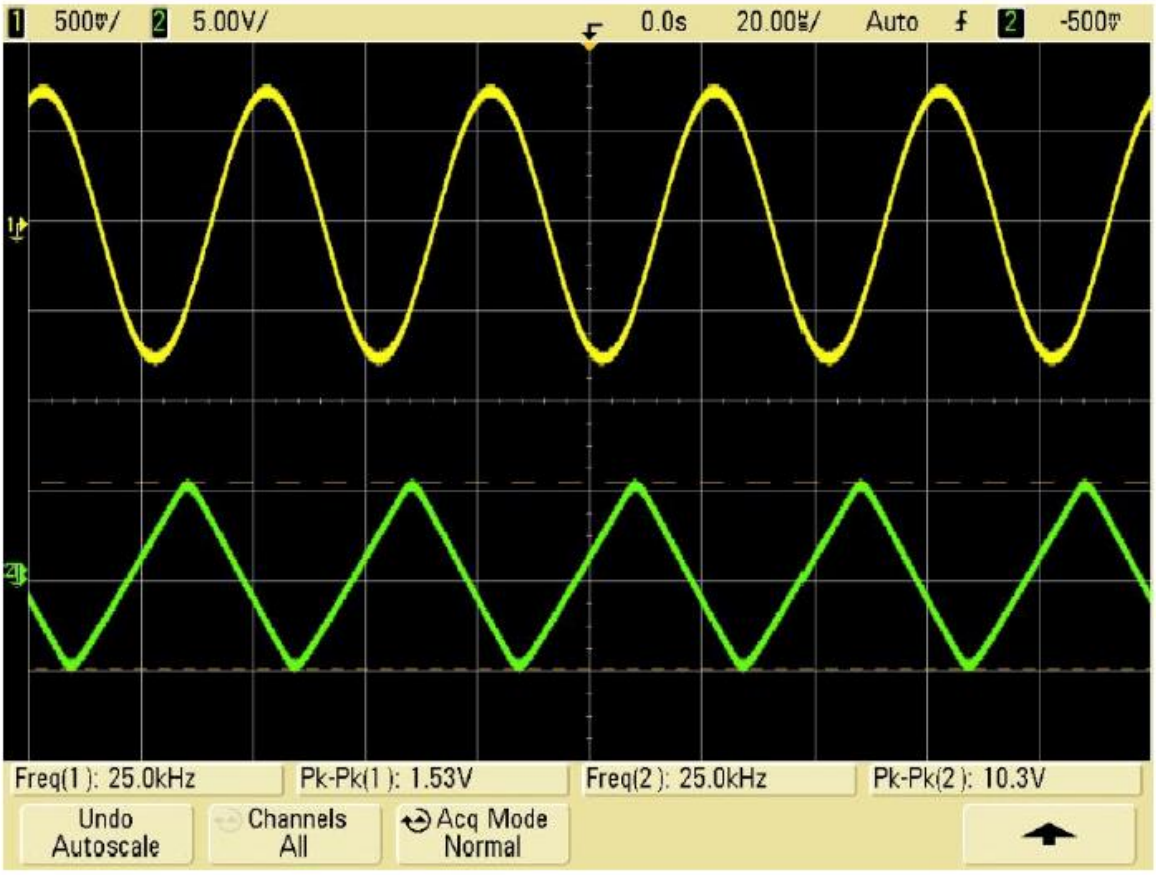


Figure 8. Slew rate limited output (CH2) from a sine wave input (CH1). Notice that the amplitude of the output voltage also decreases if the op amp is slew rate limited.

Extra Info: Why does an op amp have a slew rate? Here is a simplified explanation: If you look at the schematic diagram in the spec sheet, you will notice a 30 pF capacitor (C1). This capacitor prevents the op amp from oscillating due to feedback by reducing the voltage gain at high frequency. Unfortunately, C1 must be charged and discharged through transistor Q13, which acts as a constant current source. Remember, for a capacitor $I = C\, dV/dt$. For large output signals, the current I is constant from Q13, so dV/dt is also constant and the output voltage is a ramp as shown in Figure 8.

Now we are going to use the USB-Key to store the data from the oscilloscope.

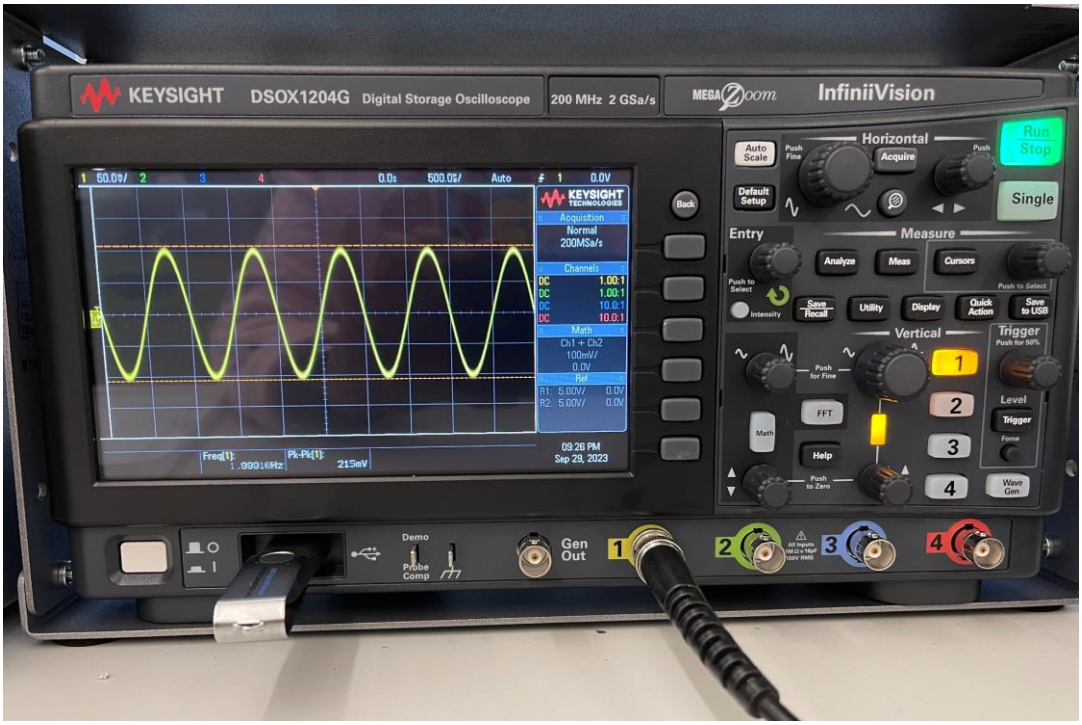


Figure 9. Plugging the USB to the Oscilloscope

Step 1: Plug the USB to the Oscilloscope at the bottom left as shown in Fig 9. *We have used a sinusoidal signal for reference. You can save the image from your corresponding lab experiment.*

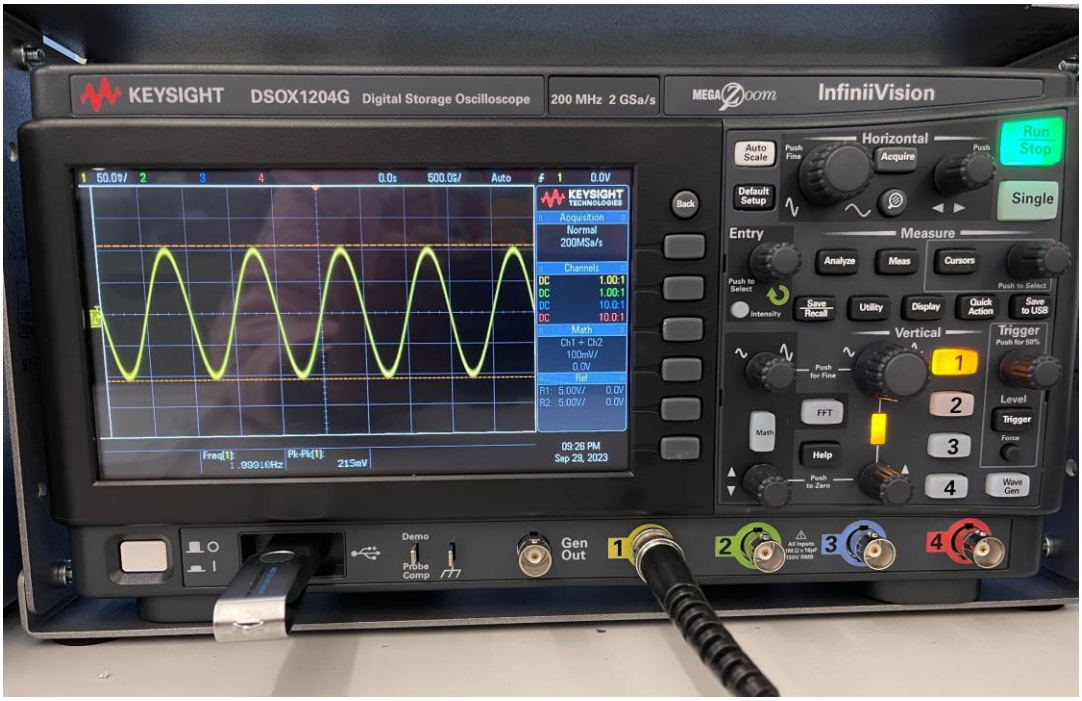


Figure 10. Click on “Save to USB”

Step 2: Click on “Save to USB” from the front panel. The format option will pop-up in the display

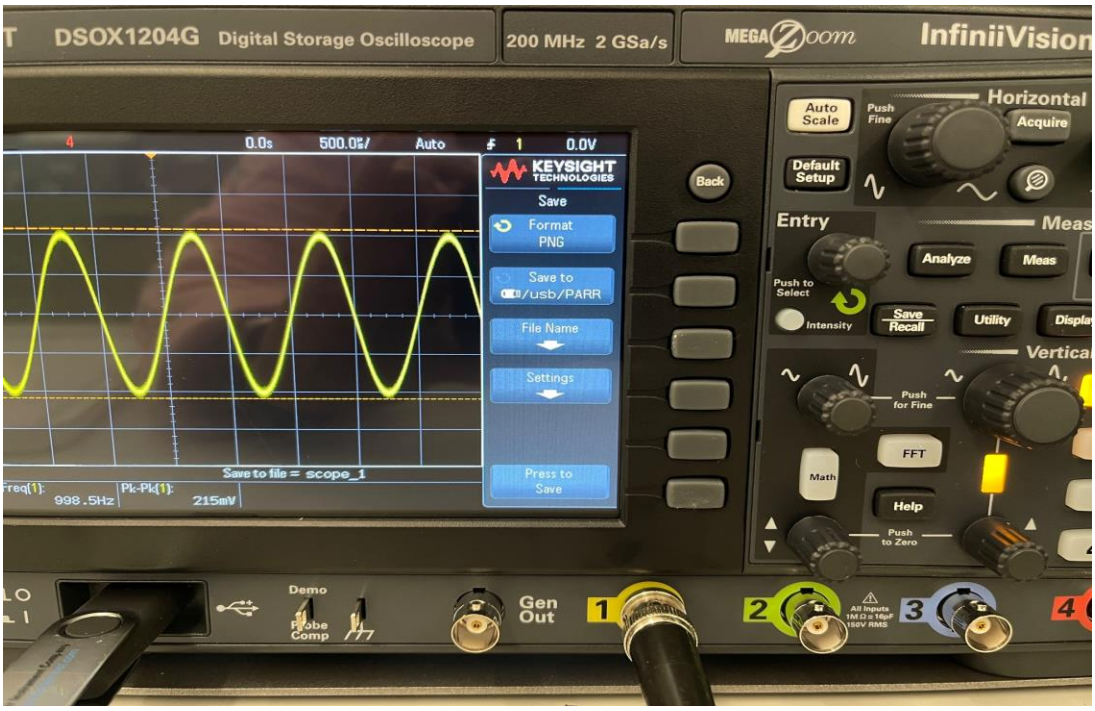


Figure 11. Click on “Format”

Step 3: Now click on “Format” to select the type of file you want to save. Finally click on “Press to Save” at the bottom of the display. For this lab we will select the PNG type for the lab report. In your pendrive the files will be stored in the name as Scope_0, Scope_1 …. Etc.

Part 3: Microphone amplifier design

For the last segment of this lab, you will design and construct an amplified microphone. The microphone requires a *special circuit* which is shown in Figure 9. The DC voltage provides power for the microphone through the upper 4.7k resistor, and the 1.5 uF capacitor blocks the DC supply voltage from the output (because capacitors are open circuits at DC). The 1.5 uF capacitor, however, allows the signal to pass through to the output. The 4.7k resistor at the output of the mic provides are DC current path to bias the first transistor inside the op amp. You will learn more about transistor bias in later labs. Note: Look on the backside of the microphone. The metal case of the microphone is connected to one of the two microphone wires. **The mic case and this wire should be grounded**.

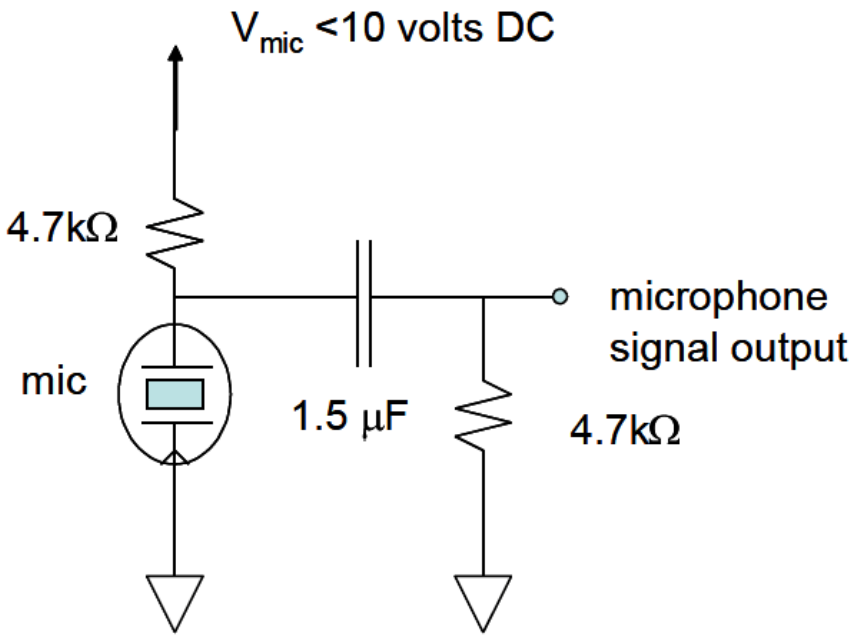


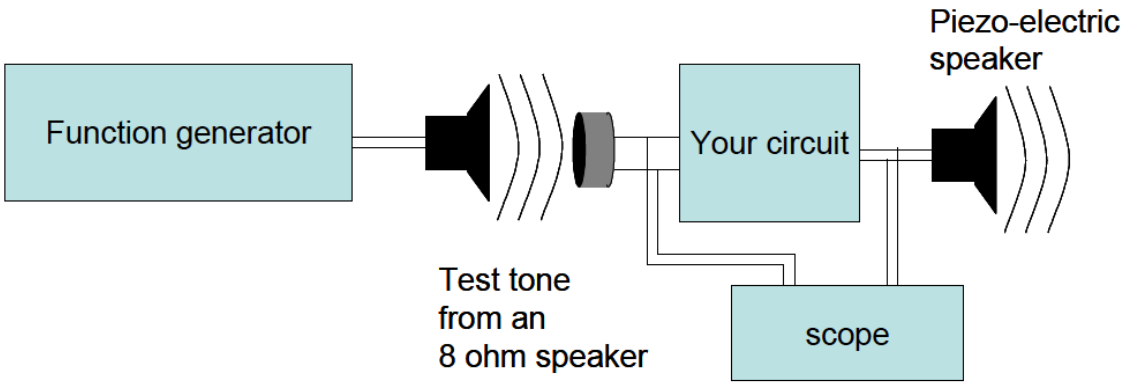
Figure 9. Microphone circuit

DESIGN SPECIFICATION:

Using an op amp, design a circuit to amplify the microphone and drive a small piezo-electric speaker (provided). The amplifier should be non-inverting and have the largest gain possible so that the speaker easily causes feedback with the microphone. **Use the function generator and oscilloscope to measure the voltage gain and record the results in your lab notebook. What is the maximum frequency that you can amplify before the output drops by 3 dB (that is, 1/Ö2)? Try connecting the 8 ohm speaker to your amplifier – what happens? Why?** (Hint: measure the impedance of the piezoelectric speaker using the impedance meter in the lab!)

Include the circuit diagram and the answers to the above questions in your lab notebook. Then have the TA sign-off on your design before leaving the lab.

Block diagram for testing your circuit:



Equipment List -- Lab #1

Keysight EDU33212A Waveform Generator
Keysight DSOX1204G Digital Storage Oscilloscope
Keysight EDU36311A Tripple Output Programmable DC Power Supply
Keysight EDU34450A Digital Multimeter
Proto-Board model PB-103 (or equivalent)
Desktop PC

4 banana plug-terminated test leads
BNC-BNC cable
BNC-banana plug cable (3) with alligator clips
#20 hook up wire
wire strippers

LM741 op amp
Two-lead electret microphone
Piezoelectric speakers
4-8 ohm speaker
1.5 mF capacitor (unpolarized)

Resistors: 1/4 W unless otherwise specified

1.0 kW	±5%	(1)
4.7 kW	±5%	(1)
10 kW	±5%	(1)
100 kW	±5%	(1)

Plus assorted resistors as needed for the design problem

Resistor Color Codes

Black 0
Brown 1
Red 2
Orange 3
Yellow 4
Green 5
Blue 6
Violet 7
Gray 8
White 9

Example: 20 kW =

RED	+	BLACK	+	ORANGE	
2		0	x	10^3	JH8/05

National Semiconductor

August 2000

LM741
Operational Amplifier

General Description

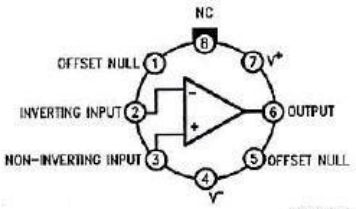
The LM741 series are general purpose operational amplifiers which feature improved performance over industry standards like the LM709. They are direct, plug-in replacements for the 709C, LM201, MC1439 and 748 in most applications.

The amplifiers offer many features which make their application nearly foolproof: overload protection on the input and output, no latch-up when the common mode range is exceeded, as well as freedom from oscillations.

The LM741C is identical to the LM741/LM741A except that the LM741C has their performance guaranteed over a 0°C to +70°C temperature range, instead of −55°C to +125°C.

Connection Diagrams

Metal Can Package

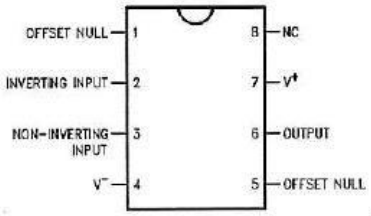


DS009341-2

Note 1: LM741H is available per JM38510/10101

Order Number LM741H, LM741H/883 (Note 1), **LM741AH/883 or LM741CH**
See NS Package Number H08C

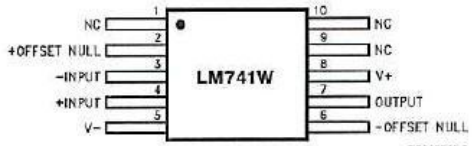
Dual-In-Line or S.O. Package



DS009341-3

Order Number LM741J, LM741J/883, LM741CN
See NS Package Number J08A, M08A or N08E

Ceramic Flatpak

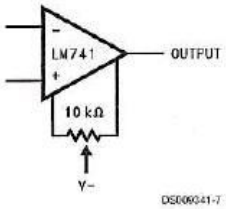


DS009341-6

Order Number LM741W/883
See NS Package Number W10A

Typical Application

Offset Nulling Circuit



DS009341-7

© 2000 National Semiconductor Corporation

Absolute Maximum Ratings (Note 2)

If Military/Aerospace specified devices are required, please contact the National Semiconductor Sales Office/ Distributors for availability and specifications.

(Note 7)

	LM741A	LM741	LM741C
Supply Voltage	±22V	±22V	±18V
Power Dissipation (Note 3)	500 mW	500 mW	500 mW
Differential Input Voltage	±30V	±30V	±30V
Input Voltage (Note 4)	±15V	±15V	±15V
Output Short Circuit Duration	Continuous	Continuous	Continuous
Operating Temperature Range	−55°C to +125°C	−55°C to +125°C	0°C to +70°C
Storage Temperature Range	−65°C to +150°C	−65°C to +150°C	−65°C to +150°C
Junction Temperature	150°C	150°C	100°C
Soldering Information			
N-Package (10 seconds)	260°C	260°C	260°C
J- or H-Package (10 seconds)	300°C	300°C	300°C
M-Package			
Vapor Phase (60 seconds)	215°C	215°C	215°C
Infrared (15 seconds)	215°C	215°C	215°C

See AN-450 "Surface Mounting Methods and Their Effect on Product Reliability" for other methods of soldering surface mount devices.

ESD Tolerance (Note 8)	400V	400V	400V

Electrical Characteristics (Note 5)

Parameter	Conditions	LM741A			LM741			LM741C			Units
		Min	Typ	Max	Min	Typ	Max	Min	Typ	Max	
Input Offset Voltage	T_A = 25°C										
	$R_S \le 10\ k\Omega$					1.0	5.0		2.0	6.0	mV
	$R_S \le 50\Omega$		0.8	3.0							mV
	$T_{AMIN} \le T_A \le T_{AMAX}$										
	$R_S \le 50\Omega$			4.0							mV
	$R_S \le 10\ k\Omega$						6.0			7.5	mV
Average Input Offset Voltage Drift				15							µV/°C
Input Offset Voltage Adjustment Range	T_A = 25°C, V_S = ±20V	±10				±15			±15		mV
Input Offset Current	T_A = 25°C		3.0	30		20	200		20	200	nA
	$T_{AMIN} \le T_A \le T_{AMAX}$			70		85	500			300	nA
Average Input Offset Current Drift				0.5							nA/°C
Input Bias Current	T_A = 25°C		30	80		80	500		80	500	nA
	$T_{AMIN} \le T_A \le T_{AMAX}$			0.210			1.5			0.8	µA
Input Resistance	T_A = 25°C, V_S = ±20V	1.0	6.0		0.3	2.0		0.3	2.0		MΩ
	$T_{AMIN} \le T_A \le T_{AMAX}$, V_S = ±20V	0.5									MΩ
Input Voltage Range	T_A = 25°C							±12	±13		V
	$T_{AMIN} \le T_A \le T_{AMAX}$				±12	±13					V

Electrical Characteristics (Note 5) (Continued)

Parameter	Conditions	LM741A			LM741			LM741C			Units
		Min	Typ	Max	Min	Typ	Max	Min	Typ	Max	
Large Signal Voltage Gain	T_A = 25°C, $R_L \geq 2$ kΩ										
	V_S = ±20V, V_O = ±15V	50									V/mV
	V_S = ±15V, V_O = ±10V				50	200		20	200		V/mV
	$T_{AMIN} \leq T_A \leq T_{AMAX}$, $R_L \geq 2$ kΩ,										
	V_S = ±20V, V_O = ±15V	32									V/mV
	V_S = ±15V, V_O = ±10V				25			15			V/mV
	V_S = ±5V, V_O = ±2V	10									V/mV
Output Voltage Swing	V_S = ±20V										
	$R_L \geq 10$ kΩ	±16									V
	$R_L \geq 2$ kΩ	±15									V
	V_S = ±15V										
	$R_L \geq 10$ kΩ				±12	±14		±12	±14		V
	$R_L \geq 2$ kΩ				±10	±13		±10	±13		V
Output Short Circuit Current	T_A = 25°C	10	25	35		25			25		mA
	$T_{AMIN} \leq T_A \leq T_{AMAX}$	10		40							mA
Common-Mode Rejection Ratio	$T_{AMIN} \leq T_A \leq T_{AMAX}$										
	$R_S \leq 10$ kΩ, V_{CM} = ±12V				70	90		70	90		dB
	$R_S \leq 50$Ω, V_{CM} = ±12V	80	95								dB
Supply Voltage Rejection Ratio	$T_{AMIN} \leq T_A \leq T_{AMAX}$, V_S = ±20V to V_S = ±5V										
	$R_S \leq 50$Ω	86	96								dB
	$R_S \leq 10$ kΩ				77	96		77	96		dB
Transient Response	T_A = 25°C, Unity Gain										
Rise Time			0.25	0.8		0.3			0.3		µs
Overshoot			6.0	20		5			5		%
Bandwidth (Note 6)	T_A = 25°C	0.437	1.5								MHz
Slew Rate	T_A = 25°C, Unity Gain	0.3	0.7			0.5			0.5		V/µs
Supply Current	T_A = 25°C					1.7	2.8		1.7	2.8	mA
Power Consumption	T_A = 25°C										
	V_S = ±20V		80	150							mW
	V_S = ±15V					50	85		50	85	mW
LM741A	V_S = ±20V										
	$T_A = T_{AMIN}$			165							mW
	$T_A = T_{AMAX}$			135							mW
LM741	V_S = ±15V										
	$T_A = T_{AMIN}$					60	100				mW
	$T_A = T_{AMAX}$					45	75				mW

Note 2: "Absolute Maximum Ratings" indicate limits beyond which damage to the device may occur. Operating Ratings indicate conditions for which the device is functional, but do not guarantee specific performance limits.

Electrical Characteristics (Note 5) (Continued)

Note 3: For operation at elevated temperatures, these devices must be derated based on thermal resistance, and T_j max. (listed under "Absolute Maximum Ratings"). $T_j = T_A + (\theta_{jA} P_D)$.

Thermal Resistance	Cerdip (J)	DIP (N)	HO8 (H)	SO-8 (M)
θ_{jA} (Junction to Ambient)	100°C/W	100°C/W	170°C/W	195°C/W
θ_{jC} (Junction to Case)	N/A	N/A	25°C/W	N/A

Note 4: For supply voltages less than ±15V, the absolute maximum input voltage is equal to the supply voltage.

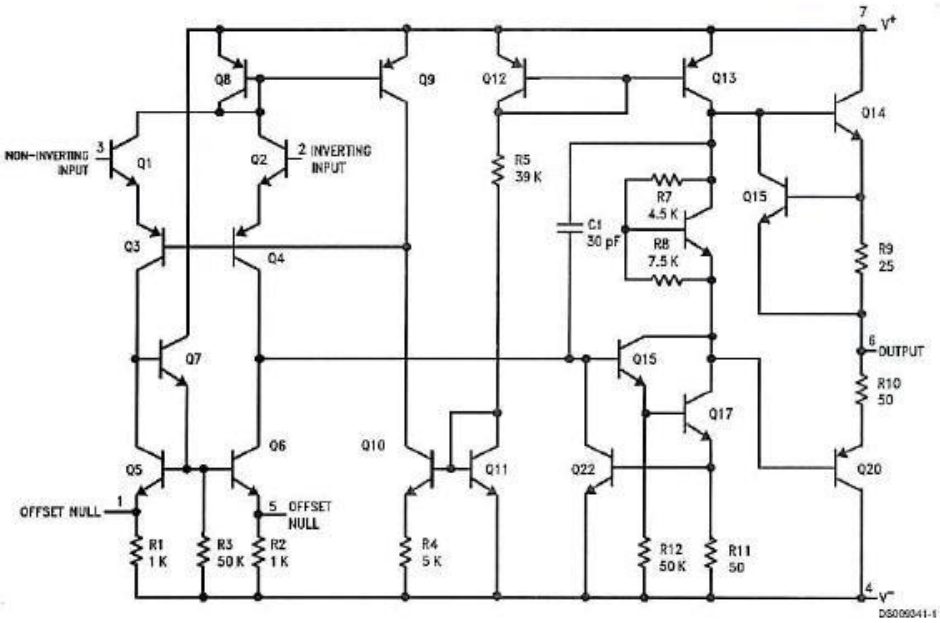
Note 5: Unless otherwise specified, these specifications apply for V_S = ±15V, −55°C ≤ T_A ≤ +125°C (LM741/LM741A). For the LM741C/LM741E, these specifications are limited to 0°C ≤ T_A ≤ +70°C.

Note 6: Calculated value from: BW (MHz) = 0.35/Rise Time(μs).

Note 7: For military specifications see RETS741X for LM741 and RETS741AX for LM741A.

Note 8: Human body model, 1.5 kΩ in series with 100 pF.

Schematic Diagram



PSPICE instruction for Lab 1 (part 1 and 2)

1: Please design your circuit with inverting Op-amp and resistors in PSPICE with the gain as in the measurement lab, and simulate to verify all the three concepts: i) amplification with gain, phase shift, ii) clipping with +/- 10 V DC power supply, iii) slew rate

Repeat all the measurements we did in the lab for these 3 concepts and plot the simulated outputs.

In your lab report you should have (this us the PSPICE part of your lab report)

1. Input signal, amplified output signal by (they should match your measurement data)
2. Phase shift (they should match your measurement data)
3. What input amplitude you can see the clipping occurs. Plot the input and clipped output and mark the V_{min_out} and V_{max_out} (they should match your measurement data)
4. Simulate for slew rate calculation, and plot the input and triangular shaped output. Use markers to show the slope. Write down the input frequency from where you noticed the output signal becomes triangular. (They should match your measurement data.)

Lab 1, Part 3:

1: Plot the input and amplified output of your non-inverting amplifier used for the design in the lab. Verify your gain is in condenses with the measurement results.

2: Plot the 3 dB frequency (Gain vs. frequency) of your design.

You do not have to use any mic, speaker or audio amplifier for this part of your PSPICE simulation.