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. Then you will build and study circuits that use operational amplifiers. Op amps are very useful and versatile circuit elements, but op amps 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.
    a. Describe how you would increase the frequency to get an (open-circuit) output of $5+10\sin(2\pi*2000t)$ volts? Give step-by-step instructions explaining which buttons you would press.
    b. If a 50 ohm resistor is then placed across the Output of the Function Generator, what would be the new output voltage?
    c. Describe how would you 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 $R_1 = 50\ \Omega$ and $R_2 = 5000\ \Omega$?
   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 R_1? (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 is a dual DC supply, meaning that it contains two independent voltage sources. Each output voltage of each source is controlled by the big white knob at the upper right corner of the power supply. The same knob is used to control the *maximum allowed current.* The voltage and current value can be seen in the digital display.

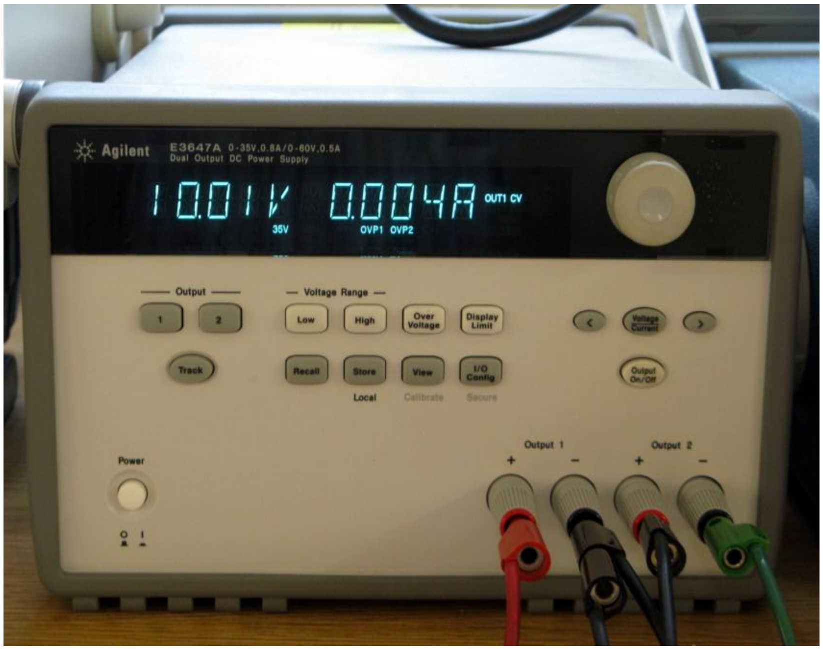


Figure 1. The Agilent E3647A Dual DC Power Supply

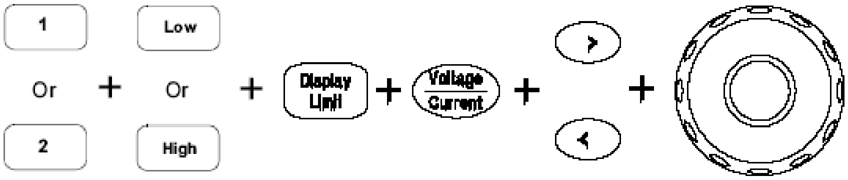
Setting the current mode limit:

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

1. Turn on the power supply.

2. Press Display Limit key to show the limit values on the display.

3. Set the knob to current control mode by pressing Voltage Current key.



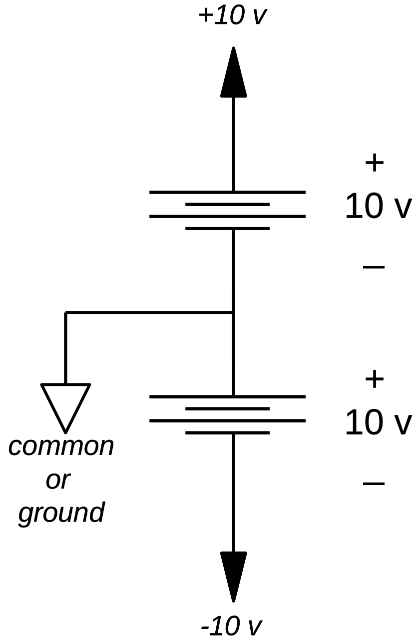
5. Move the blinking digit to the appropriate position using the resolution selection keys and change the blinking digit value to the desired current limit by turning the control knob.

6. Press Output On/Off key to enable the 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.**

Configuring a dual power supply for plus and minus voltages

Figure 2. Schematic for ±10 volt



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 1, notice the four terminals on the lower right-hand side. Output 1 has DC+ and DC- terminals, and Output 2 has DC+ and DC- terminals. To configure your power supply as a ±10 volt source, follow the connections shown in Figure 1: Output 1's DC+ has a red wire connected to it. This is +10 volts. Output 1's DC- is connected to Output 2's DC+ with a black wire. This black wire is the common node in Figure 2. If it is important to have an *earth ground* in your experiment, you should connect another black wire from Output 1's DC- to the ground of your experimental board. Finally, the -10 volt source is connected using a green wire at Output 2'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 Agilent 33220A function generator is shown in Figure 4 below. The waveform is selected by pushing one of the buttons below the screen. For this example, let's assume that the sine wave is selected. This will allow us to generate a signal in the form of

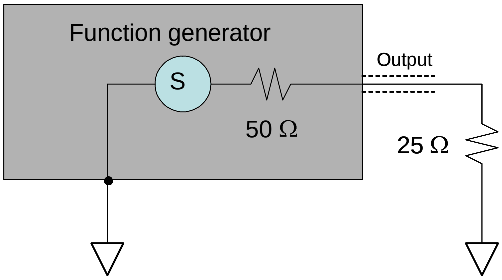
$$\mathbf{v}(t) = \mathbf{V}_{\mathbf{DC}} + \mathbf{A}\, \sin\,(2\pi f t)$$

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 frequency of the sine wave.

Figure 4. The Agilent 33220A Function Generator

OUTPUT:

The output voltage from the function generator is produced at the BNC connector marked “Output”. Press the Output key 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 below “Ampl” 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 below “Freq” 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 below “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.

SWEEP:

The SWEEP features of the function generator allow you to automatically sweep the frequency rather than manually adjusting the frequency using the Sweep button. We will not 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 2-channel digital scope shown in Figure 3.

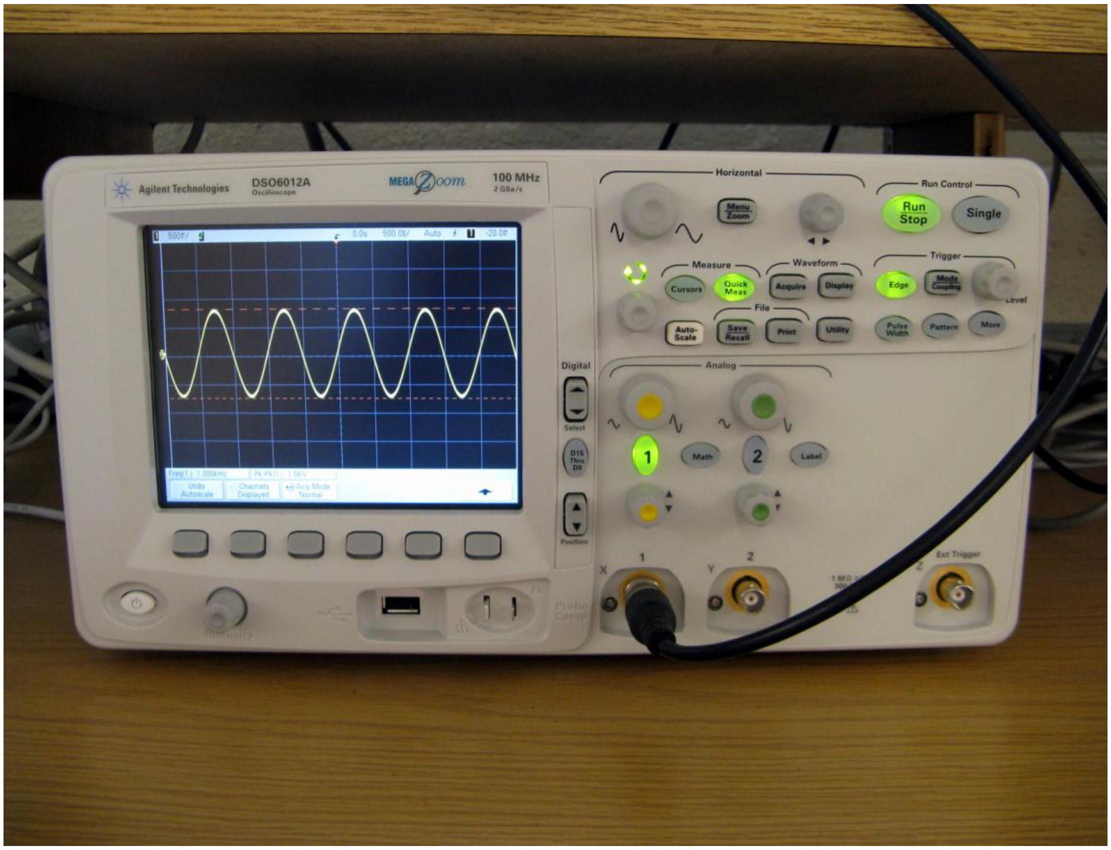


Figure 3. The Agilent MSO6012A oscilloscope

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 and CH2

Each "channel" of an oscilloscope accepts a separate voltage input. This scope has two channels (CH1, CH2), although some scopes have four or more channels. 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 or 2 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 knob above button 1 or 2. In Figure 3, a sine wave is applied to channel 1. The VOLTS/DIV knob has been adjusted to 500mV (as shown in the upper left corner of the scope’s screen). This means each division represents 500 mV. 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. You should be able to see that the peak of the sine wave is approximately 1.5 divisions above the zero marker. This means the amplitude of the sine wave is 1.5 divisions * 5 volts/division = 750mV. Likewise, the minimum value of the sine wave is -750mV.

CH1, CH2:

Pressing channel 1&2 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(1000t) volts, the display will show a 0.2 volt 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 (10v). 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(1000t) volts even though the actual signal is v(t) = 10.0 + 0.2*sin(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.

▲/▼*:*

The zero voltage level displayed on the screen can be adjusted up or down using the knob below button 1 and 2. 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.

SEC/DIV:

As previously mentioned, the scope gives you a view of voltage vs. time. The time axis is controlled by the knob on the top left corner. 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. In Figure 3, the scope is set to 500μs per division. The *period* of the sine wave is approximately 2 divisions * 500μs per division = 1000μs = 1 ms. Therefore, the frequency of the sine wave is *f* = 1 / T = 1 kHz.

◄ ►:

This knob allows you to shift the signal horizontally on the screen, and functions just like the vertical position knob.

TRIGGER LEVEL:

On the far 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 at the bottom of the screen. Press the Mode softkey, and then select Manual. Select the waveform 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.

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.

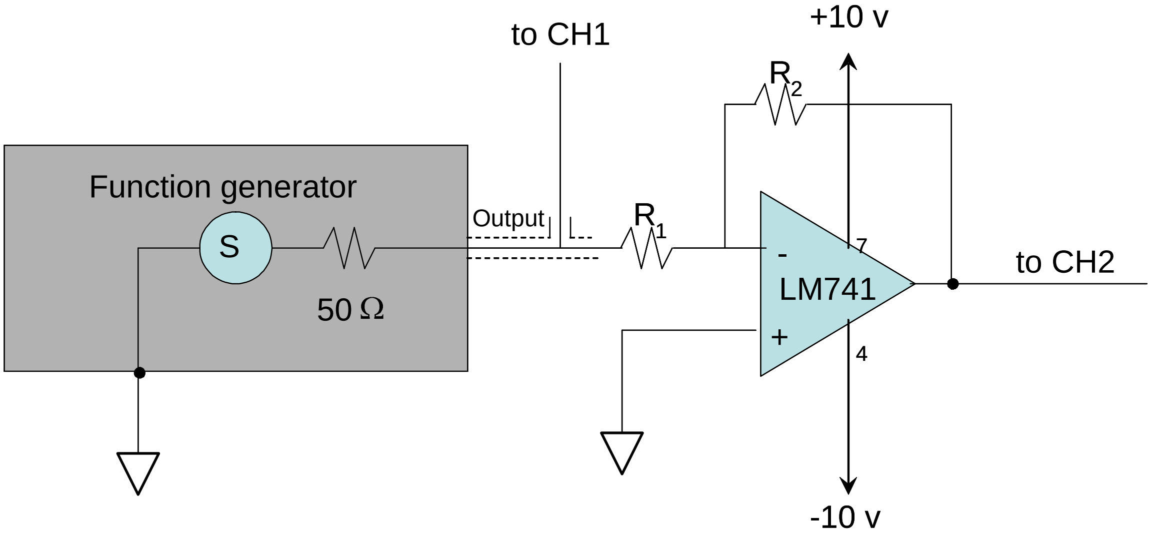
The Quick Meas button will cause the scope to determine the peak-to-peak voltages of a signal, the mean of the signal, the frequency of a signal, etc. Use this feature with some caution! First, make certain that the screen displays a stable, noise-free signal that is complete (not chopped off). Second, 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 ~ 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).



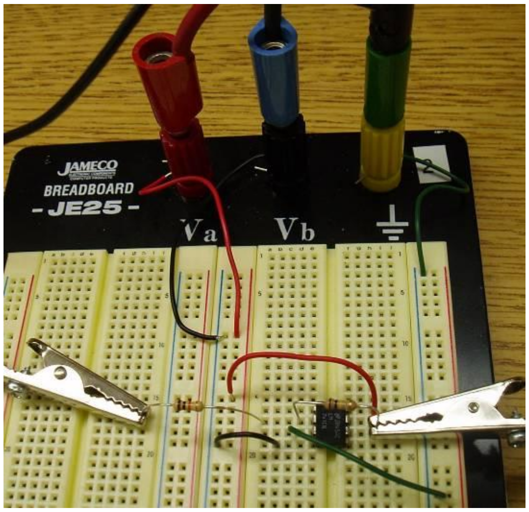
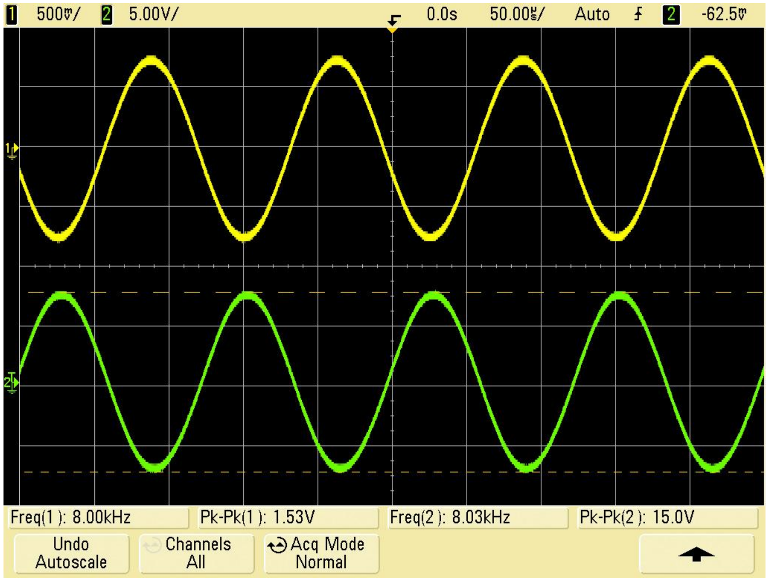


Figure 5. The LM741CN op amp used in the inverting amplifier configuration. The input signal is applied through the clip on the left, and the scope is connected through the clip on the right. All ground connections are stacked on the rightmost banana plug. Keep the layout neat!

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 (ΔT) between channel 1 (input) and channel 2 (output). (2) The phase shift is calculated using 360^0* (ΔT/T) where T is the period of the sine wave (T=1/*f*).**



1 500mV/ 2 5.00V/ 0.0s 50.00µs/ Auto 2 -62.5mV

Freq(1): 8.00kHz
Pk-Pk(1): 1.53V
Freq(2): 8.03kHz
Pk-Pk(2): 15.0V

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.

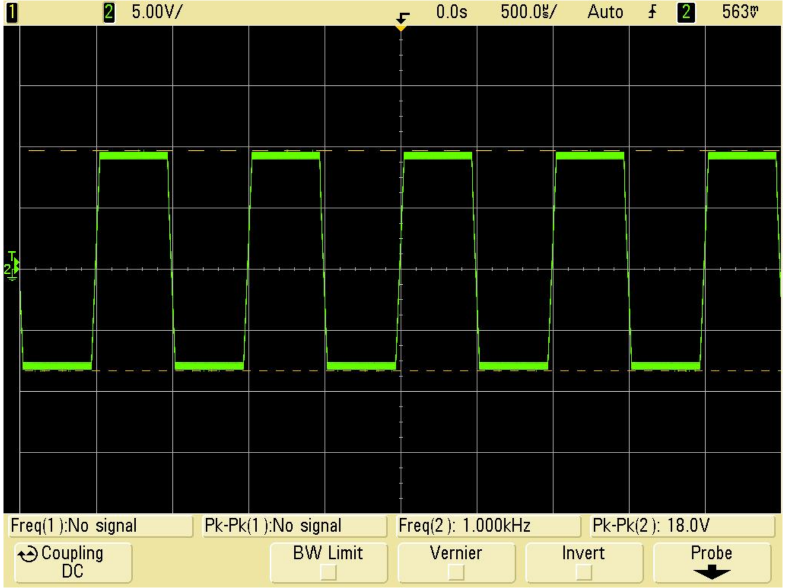


Figure 7. Clipped output signal from a 741 op amp

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.

Now we are going to use the MATLAB Instrument Control Toolbox to download the output waveform so you can calculate the slew rate on a hardcopy plot.

Return the function generator signal to a sine wave.

To open MATLAB on your PC, go to **Start > All Programs > Statistical & Computational > MATLAB**, and select **2011**. Set the Current Directory to "**C:\Temp\Work**". All command source codes are available in this folder.

In order to obtain the waveform from the oscilloscope, a command is needed to activate the General Purpose Interface Bus (GPIB) system and to initialize the oscilloscope setting. To do this, in the MATLAB command window enter: `[scope]=setup_scope`. Now that the oscilloscope has been initialized, the function `slewrateecho` can be used

to download the waveforms from the scope.
** You can find the source code of `setup_scope` and `slewrateecho` in Appendix 1 & 2.

The syntax of calling this function is `output = slewrateecho(n, channel, scope).` This command puts into effect the following steps:

a) Sampling the waveform from the oscilloscope, where the effective sampling rate used is `n` MHz, meaning a rate of $n \cdot 10^6$ samples per second. (Allowed values of `n` are 1, 2, 5, 10, 20, 50, 100 and 200). Parameter `channel` depends on to which channel of the oscilloscope you have connected your output waveform. (Thus, the actual command may be, e.g. `output = slewrateecho(50, 2, scope)`.)
b) The returned value `output` is a two-column array (the name `output` is arbitrary, and you may use any other name you choose): The first column consists of the list of sampling times, and the second column consists of the corresponding samples of the output signal. You may name the two column vectors: “time” and “signal”. They are obtained using the array commands:

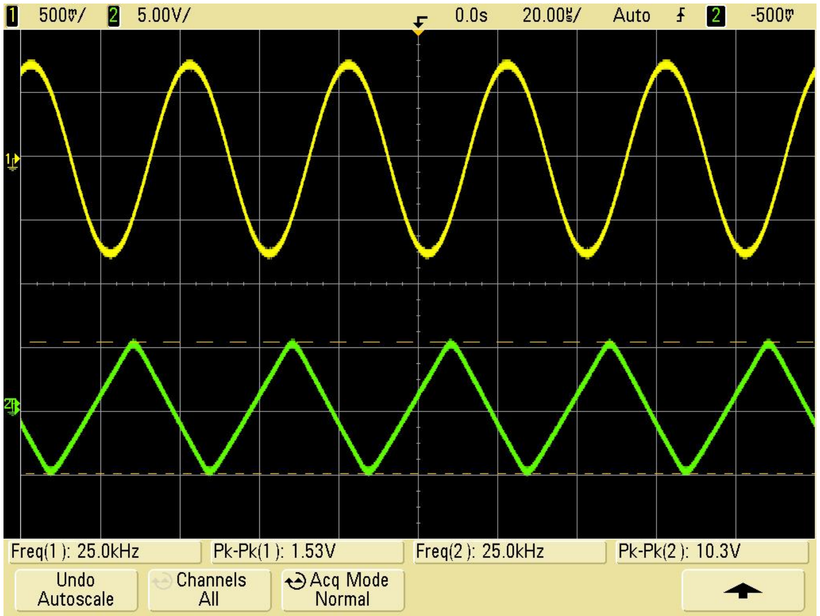
```
time=output(:,1);
signal=output(:,2);
```

Plot `signal` as a function of time using the command `plot(time,signal)`. Provide axis names and title, as well as team #, the names of your team members, and print a copy for each team member to include in their lab report.

Calculate the slew rate of your op amp from the plot. **What is the slew rate in volts/microsecond? Does it meet the specifications for the LM741?**

Note that using MATLAB, you will be able to keep records for your experimental results.

Typical slew rate limited output is shown below:



Freq(1): 25.0kHz
Pk-Pk(1): 1.53V
Freq(2): 25.0kHz
Pk-Pk(2): 10.3V

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.

Prepare your lab report using the guidelines provided in the syllabus section of the lab manual. It is due next week at the beginning of lab.

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**.

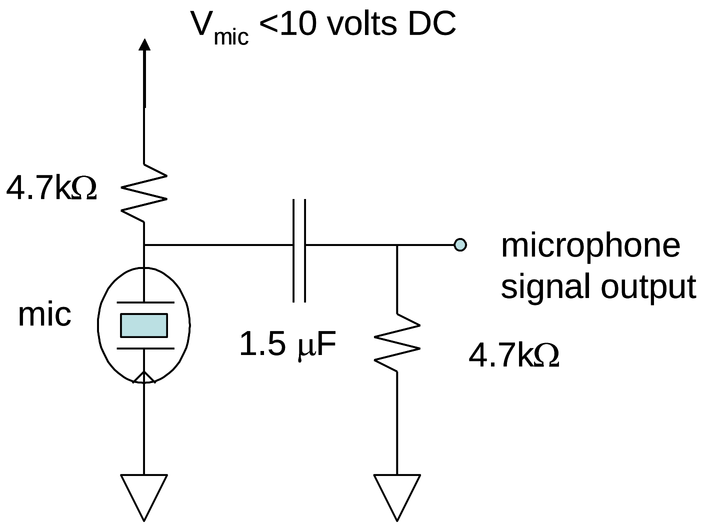


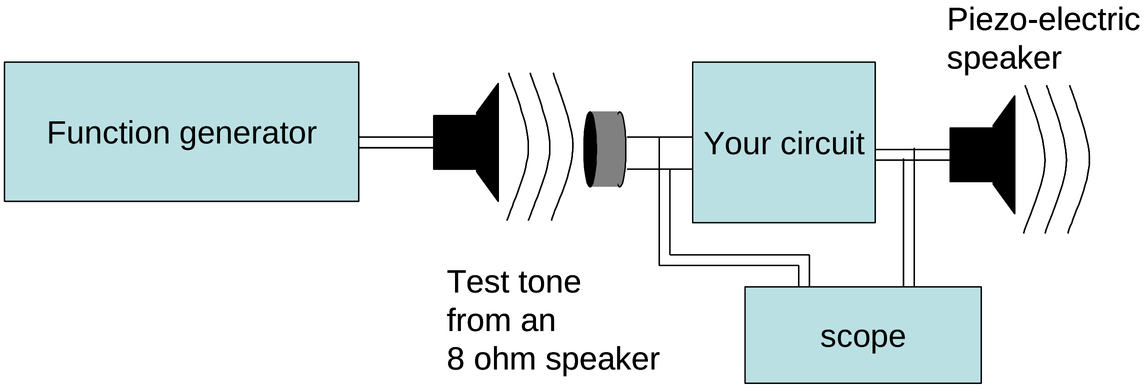
Figure 9. Microphone circuit

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/√2)? Try connecting the 8 ohm speaker to your amplifier – what happens? Why?** (Hint: measure the impedance of the piezoelectric speaker using the impendance 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

Agilent E3647A dual output power supply
Agilent MSO6012A mixed-signal oscilloscope
Agilent 33220A function generator
Proto-Board model PB-103 (or equivalent)
Dell OPTIPLEX 755 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 μF capacitor (unpolarized)

Resistors: 1/4 W unless otherwise specified

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

Plus assorted resistors as needed for the design problem

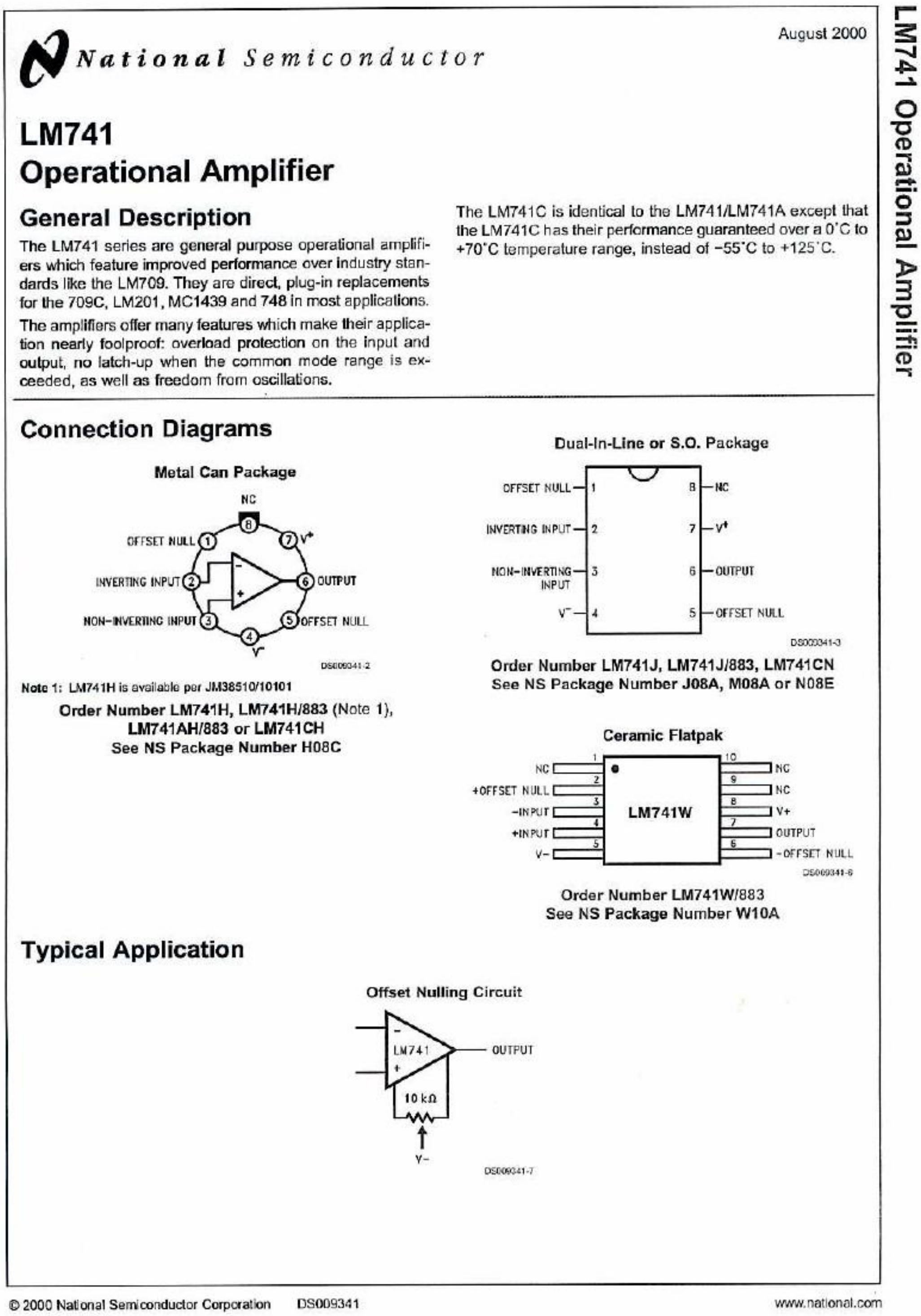
Resistor Color Codes

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

Example: 20 kΩ =

RED	+	BLACK	+	ORANGE
2		0	x	10^3

JH8/05 rev.9/05
rev.7/08
rev.6/10
rev.4/12



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

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

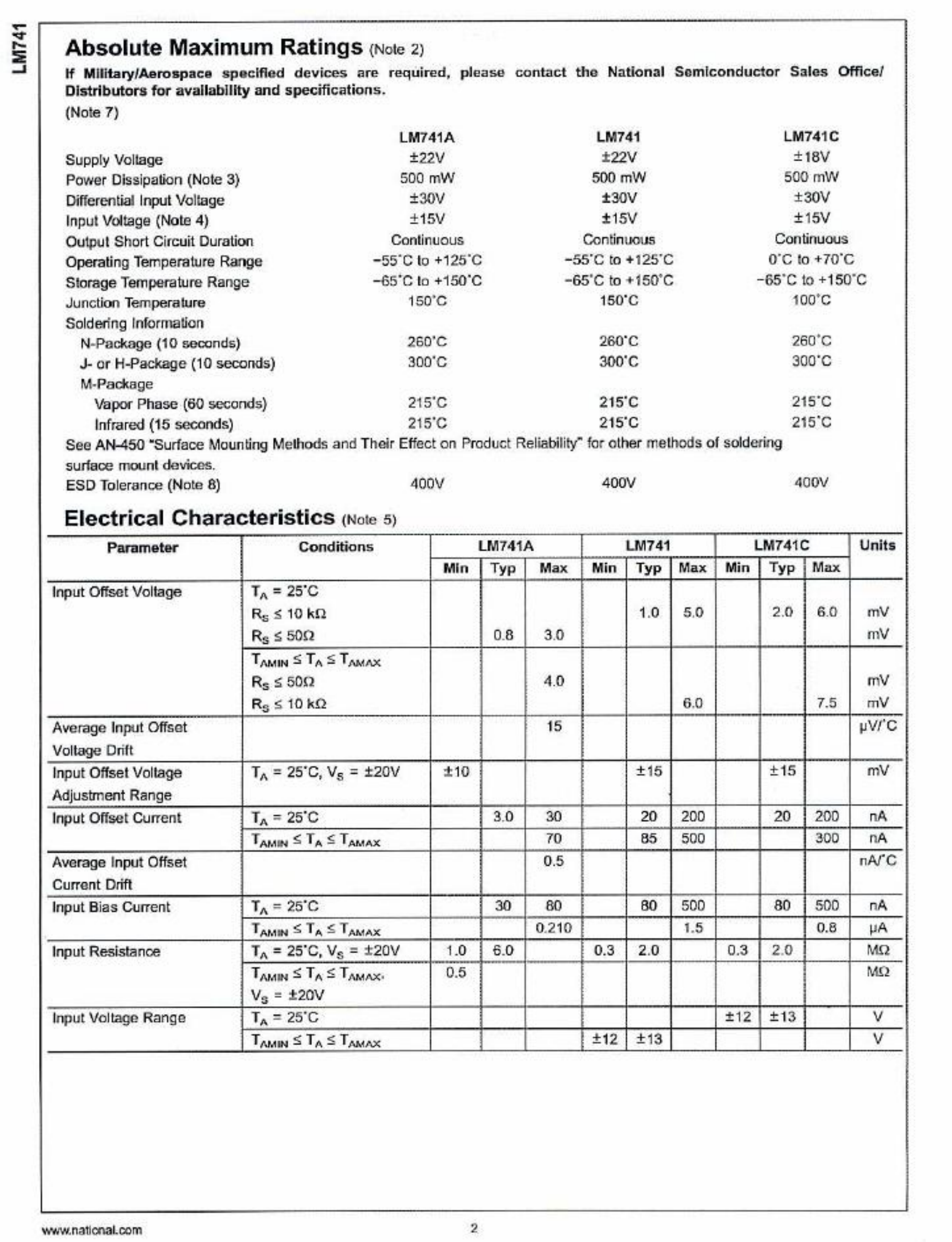
Order Number LM741J, LM741J/883, LM741CN
See NS Package Number J08A, M08A or N08E

Ceramic Flatpak

Order Number LM741W/883
See NS Package Number W10A

Typical Application

Offset Nulling Circuit



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)

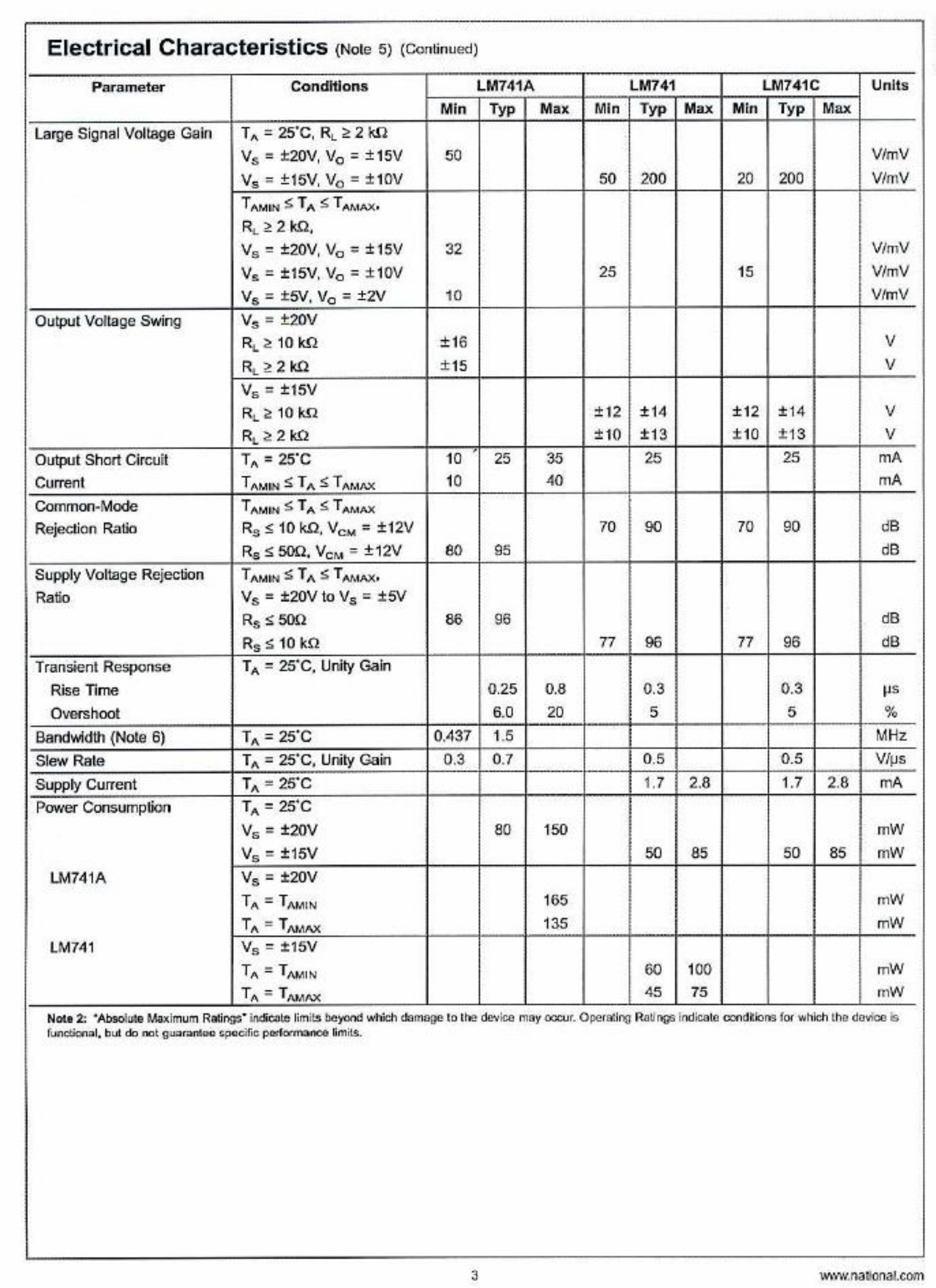
	LM741A	LM741	LM741C
Supply Voltage	±22V	±22V	±18V
Power Dissipation (Note 3)	500 mW	500 mW	500 mW
Differential Input Voltage	±30V	±30V	±30V
Input Voltage (Note 4)	±15V	±15V	±15V
Output Short Circuit Duration	Continuous	Continuous	Continuous
Operating Temperature Range	−55°C to +125°C	−55°C to +125°C	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.

	LM741A	LM741	LM741C
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	$T_A = 25°C$										
	$R_S \leq 10\ k\Omega$					1.0	5.0		2.0	6.0	mV
	$R_S \leq 50\Omega$		0.8	3.0							mV
	$T_{AMIN} \leq T_A \leq T_{AMAX}$										
	$R_S \leq 50\Omega$			4.0							mV
	$R_S \leq 10\ k\Omega$						6.0			7.5	mV
Average Input Offset Voltage Drift				15							μV/°C
Input Offset Voltage Adjustment Range	$T_A = 25°C, V_S = \pm 20V$	±10				±15			±15		mV
Input Offset Current	$T_A = 25°C$		3.0	30		20	200		20	200	nA
	$T_{AMIN} \leq T_A \leq T_{AMAX}$			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
	$T_{AMIN} \leq T_A \leq T_{AMAX}$			0.210			1.5			0.8	μA
Input Resistance	$T_A = 25°C, V_S = \pm 20V$	1.0	6.0		0.3	2.0		0.3	2.0		MΩ
	$T_{AMIN} \leq T_A \leq T_{AMAX}$, $V_S = \pm 20V$	0.5									MΩ
Input Voltage Range	$T_A = 25°C$							±12	±13		V
	$T_{AMIN} \leq T_A \leq T_{AMAX}$				±12	±13					V



LM741

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, $R_L \ge 2$ kΩ										
	V_S = ±20V, V_O = ±15V	50									V/mV
	V_S = ±15V, V_O = ±10V				50	200		20	200		V/mV
	$T_{AMIN} \le T_A \le T_{AMAX}$,										
	$R_L \ge 2$ 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										
	$R_L \ge 10$ kΩ	±16									V
	$R_L \ge 2$ kΩ	±15									V
	V_S = ±15V										
	$R_L \ge 10$ kΩ				±12	±14		±12	±14		V
	$R_L \ge 2$ kΩ				±10	±13		±10	±13		V
Output Short Circuit Current	T_A = 25°C	10	25	35		25			25		mA
	$T_{AMIN} \le T_A \le T_{AMAX}$	10		40							mA
Common-Mode Rejection Ratio	$T_{AMIN} \le T_A \le T_{AMAX}$										
	$R_S \le 10$ kΩ, V_{CM} = ±12V				70	90		70	90		dB
	$R_S \le 50Ω$, V_{CM} = ±12V	80	95								dB
Supply Voltage Rejection Ratio	$T_{AMIN} \le T_A \le T_{AMAX}$,										
	V_S = ±20V to V_S = ±5V										
	$R_S \le 50Ω$	86	96								dB
	$R_S \le 10$ 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.

3 www.national.com

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.

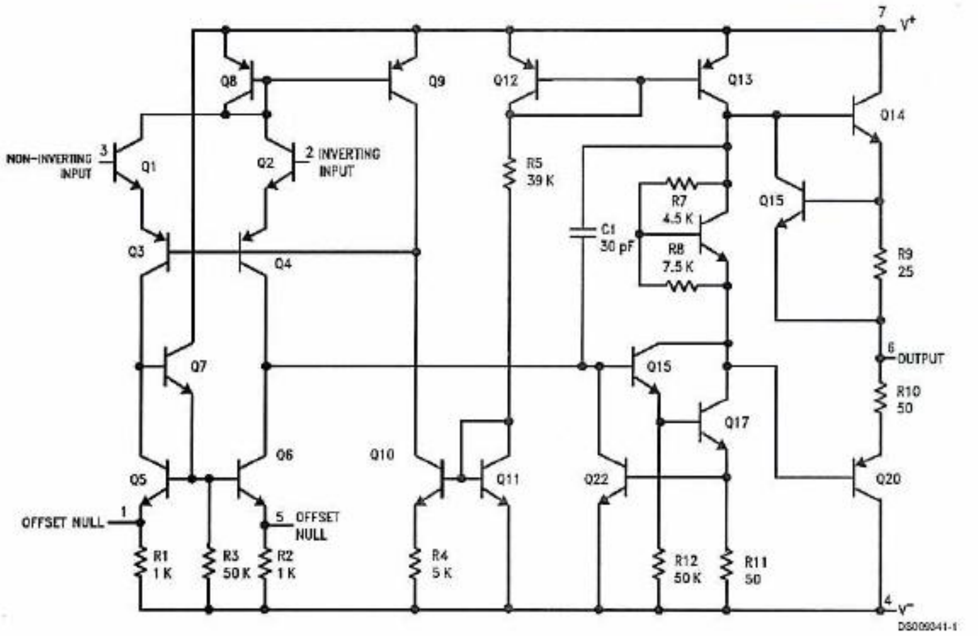
Note 5: Unless otherwise specified, these specifications apply for $V_S = \pm 15V$, $-55°C \le T_A \le +125°C$ (LM741/LM741A). For the LM741C/LM741E, these specifications are limited to $0°C \le T_A \le +70°C$.

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



APPENDIX - 1

Filename: setup_scope.m

```
function [scope]=setup_scope

% initializes the scope function

scope = visa('agilent','GPIB0::7::INSTR'); % open GPIB connection to
scope
set(scope,'InputBufferSize', 1.024E6); % hold 1 meg of data in memory
fopen(scope)
if(scope.Status~='open')
    fprintf('Error opening GPIB connection to oscilloscope\n');
    output = [0,0]; % error flags set
    return;
end

fprintf(scope,':TIMEBASE:MODE MAIN');    % required for deep memory
transfer
fprintf(scope,':TIMEBASE:RANGE 5E-4'); % set scope time window to 5 ms
width
fprintf(scope,':TIMEBASE:REFERENCE LEFT');% put start of window at left

fprintf(scope,':TIMEBASE:DELAY 0'); % move output pulse to left
side %change for delay
fprintf(scope,':CHANNEL1:RANGE 2.0'); % set vertical sensitivity of
channel 1; heidy cambiar amplitud
fprintf(scope,':CHANNEL1:COUPLING DC'); % coupling to DC
fprintf(scope,':TRIG:SOURCE EXT'); % trigger on sync from function
generator
fprintf(scope,':TRIG:SLOPE POSITIVE'); % sync output goes low when
pulse starts
fprintf(scope,':TRIG:LEVEL 1'); % trigger on 1V point

fprintf(scope,':AUT');
fclose(scope)   % disconnect GPIB scope object
```

APPENDIX – 2

Filename: slewrateecho.m

```
function output = slewrateecho(n, channel, scope)
% Agillent MSO6012A Oscilloscope
% slewrateecho(n) - function to digitize the echoes from a 1 MHz
ultrasonic transducer
% n is sample rate (MHz) from choices 200, 100, 50, 20, 10, 5, 2, 1.
% function returns two-column matrix [time, voltage], or [0,0] if
failure
% channel is the channel connected to the output

fopen(scope)
if(scope.Status~='open')
fprintf('Error opening GPIB connection to oscilloscope\n');
output = [0,0]; % error flags set
return;
end
fprintf(scope,':ACQUIRE:TYPE NORMAL'); % required for deep memory
transfer
if (channel == 1)
fprintf(scope,':WAVEFORM:SOURCE CHANNEL1'); % assume input into Channel
1
fprintf(scope,':DIGITIZE CHANNEL1'); % digitize the data
else if (channel == 2)
fprintf(scope,':WAVEFORM:SOURCE CHANNEL2'); % assume input into Channel
2
fprintf(scope,':DIGITIZE CHANNEL2'); % digitize the data
    else fprintf('Error read the channel\n');
    end
end
fprintf(scope,':WAVEFORM:POINTS ALL'); % grab as many points as
possible
fprintf(scope,':WAVEFORM:POINTS?'); % ask for number of points
numpointschar = fscanf(scope); % assume character
%numpoints = str2num(numpointschar(6:length(numpointschar)));% turn
into number
numpoints = str2num(numpointschar); % Heidy... changed way to calculate
number of points
fprintf(scope,':WAVEFORM:FORMAT BYTE'); % deliver one byte per
datapoint
fprintf(scope,':WAVEFORM:PREAMBLE?');% read offsets, etc.
header = fscanf(scope,'%s');
%numheader = parse(header); % turn string into numerical array %heidy
numheader = str2num(header); %modified
xinc = numheader(5); xorig = numheader(6); xref = numheader(7);
yinc = numheader(8); yorig = numheader(9); yref = numheader(10);
fprintf(scope,':WAVEFORM:DATA?'); % request scope to send byte data
bytedata = fread(scope,numpoints+11,'uint8'); % read in bytes from gpib
stream
subdata = bytedata(11:length(bytedata)-1); % trim off number of points
etc.
lastindex = length(subdata); % remember how long data is
index = 1:lastindex; % generate index list
rawtime = ((index - xref) * xinc) + xorig; % scale time appropriately
rawdata = ((subdata - yref) * yinc) + yorig; % scale voltage
```

```
appropriately
startindex = min(find(rawtime>=0)); % ignore pretriggered portion
fprintf(scope,':RUN'); % restore scope free run
switch n
case 200
time = rawtime(startindex:lastindex); % copy positive time portion
data = rawdata(startindex:lastindex); % take every point
case 100
time = rawtime(startindex:2:lastindex); % copy positive time portion
data = rawdata(startindex:2:lastindex); % take every other point
case 50
time = rawtime(startindex:4:lastindex); % copy positive time portion
data = rawdata(startindex:4:lastindex); % take every 4th point
case 20
time = rawtime(startindex:10:lastindex); % copy positive time portion
data = rawdata(startindex:10:lastindex); % take every 10th point
case 10
time = rawtime(startindex:20:lastindex); % copy positive time portion
data = rawdata(startindex:20:lastindex); % take every 20th point
case 5
time = rawtime(startindex:40:lastindex); % copy positive time portion
data = rawdata(startindex:40:lastindex); % take every 40th point
case 2
time = rawtime(startindex:100:lastindex); % copy positive time portion
data = rawdata(startindex:100:lastindex); % take every 100th point
case 1
time = rawtime(startindex:200:lastindex); % copy positive time portion
data = rawdata(startindex:200:lastindex); % take every 200th point
otherwise
time = 0; data = 0;
end

output(:,1) = time'; % assign time data to dummy output column
output(:,2) = data; % assign voltage data to dummy output column
fclose(scope) % disconnect GPIB scope object
```