

Lab #2 {150 pts. Total} Op Amp Circuits

OBJECTIVES:

- Become familiar with operational amplifier circuits.
- Design circuits containing operational amplifier(s) to meet given requirements.
- Construct, test, and debug operational amplifier circuits.

PARTS LIST:

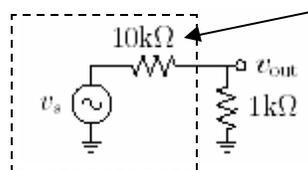
IMPORTANT: Some parts below are determined by your PRE-LAB.

- | | |
|---|---|
| <ul style="list-style-type: none"> ❑ At least: 2 - 0.1μF, 0.01μF ceramic capacitors (values not critical) ❑ 2 - 100μF electrolytic or tantalum capacitors ❑ (2) 100Ω, (2) 100kΩ, 10kΩ, 1kΩ, 47kΩ, and other R's determined by PRE-LAB ❑ condenser microphone (may be available for check-out, otherwise purchase or use your own) ❑ 2N3904 and 2N3906 transistors | <ul style="list-style-type: none"> ❑ 10kΩ potentiometer, single turn (trim pot) (any size from 5kΩ to 100kΩ will do) ❑ speaker (may be available for check-out, purchase or you may use your own) ❑ some operational amplifier(s): LM741, TL084, LF353, or similar {LF353 has better performance than 741) |
|---|---|

PRE-LAB: (30 points)

Procedure:

1. Assume the following circuit:



This circuit cannot be altered. It represents a “large circuit” lumped into a Thevenin equivalent. The 1k load cannot be changed.

Fig. 1

The v_s and 10k resistance are the Thevenin equivalent of a larger circuit. This circuit is needed to drive another circuit with an input resistance of 1k. This 1k resistance is considered the “load” resistance for the larger circuit. By analyzing this circuit, this circuit will produce an output:

$$v_{\text{out}} = \frac{R_L}{R_s + R_L} v_s = 0.09v_s$$

The situation depicted in **Fig. 1** is not a good situation. Think of the signal v_s as a voice signal connected to a microphone that has an equivalent resistance of 10k and the 1k resistance as a speaker. By reducing the amplitude by one-tenth, v_s would not be heard at all through the speaker. Therefore, try to keep the signal approximately the same.

(10 pts)

- (1a) Design a circuit that will keep the signal approximately equal at the output - $v_{out} \sim v_s$. You cannot alter the input v_s , the 10k Ω or the 1k Ω resistors. They all have to be in the circuit.

IMPORTANT: *Because you will be connecting and testing this circuit during the lab time, it is recommended that you connect the circuit before coming to lab. Hint: You will be connecting a circuit between the 10k resistor and the 1k output resistor in **1d**.*

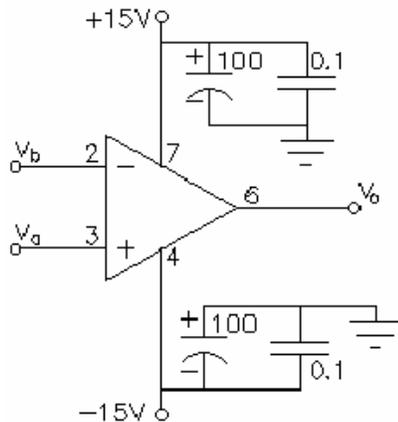
2. (10 pts)
- Design a non-inverting amplifier to produce a gain of 101 V/V. This designed amplifier will be used in Experiment 3(below)
 - Build the non-inverting amplifier on your breadboard.
- (2a) (10 pts)
- Describe in your own words what *slew rate* means.
 - Describe in your own words what *clipping* means.

BACKGROUND INFORMATION:

Connecting power supplies, compensating a scope probe, potentiometers, and noise on the probes.

Power Supply Connections:

Whenever you connect power supplies to the operational amplifier, connect them as follows:



CAUTION

It is very easy to leave the power supply set so that a small accidental touch of the round knob can suddenly change your supply voltage by as much as 10 V– that’s not good for your circuit.

To protect against this, don’t leave the knob set to adjust the most significant digit, or simply hit the +6 V button and the knob will be set to adjust a voltage that we are not using.

Note: The bypass capacitors attached from the power supply pin to ground. This protects your circuit from power supply noise and any noise or glitches that your power leads might pick up. Sometimes they will help keep the circuit from spuriously oscillating. If you look at commercially produced circuit boards, you will often see bypass caps by every IC. Your bypass capacitors should be right on your breadboard and as close to the IC as possible.

For each voltage from the supply, use a parallel combination of a fair size electrolytic or tantalum capacitor (100 μF shown here) and a smaller low-inductance ceramic disk or monolithic ceramic capacitor (0.1 μF shown here). Pay close attention to the electrolytic polarities. They can blow up if you hook one up backwards. See the **Capacitors** box from **Lab #1**.

Scope Probes:

You might want to begin using the 10x scope probes rather than the BNC –to-clip cables(1x). It is your choice, just be aware of the following information:

The 10x refers to the fact that the input impedance to the scope is 10 times higher when using a 10x scope probe. When you hook the scope directly to your circuit, the circuit “sees” a load of 1 $\text{M}\Omega$ in parallel with about 13 pF. When using a 10x probe, the circuit will instead “see” a load of 10 $\text{M}\Omega$ in parallel with about 1.3 pF. This lessens the effect of the scope on the circuit, making your measurement more accurate. The price of this higher impedance is that the scope will only get one-tenth the signal that it would with a 1x probe. Be aware of these tradeoffs in scope performance to get the most accurate measurements from the scope.

In this lab neither the input impedance nor the signal strength will be a problem, so the only reason to use the 10x probes is that they’re a little nicer to use. There is, however, one little “gotcha” with 10x probes. Notice that I said that the scope input capacitance was “about” 13 pF. Well, each scope is a little different and the scope probe will need to be adjusted or “compensated” each time it is used with a different scope. This is especially true in our lab, where the same probes are used with the Tektronics and HP scopes. The frequency response of an uncompensated probe can seriously mess-up your measurements. You should always compensate a 10x probe before using it.

To compensate a probe:

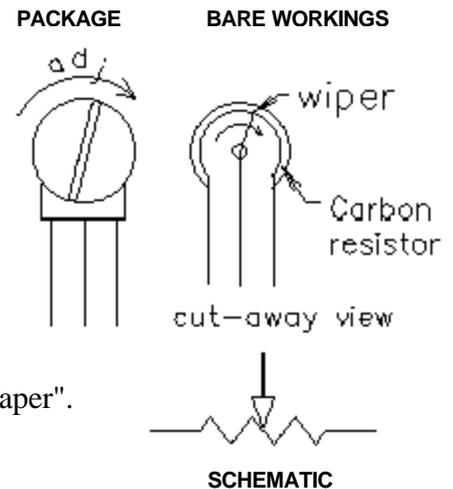
- 1) Connect the probe to the scope as normal.
- 2) Switch the probe to 10x if it has a switch.
- 3) Determine how to adjust your probe. Most probes have a small adjustment screw somewhere on the probe. Some probes adjust by twisting some part of the probe. (In rare cases the adjustment may be located at the scope end of the probe lead.)
- 4) Find a small metal contact near the scope screen marked with a square wave: "Probe Adjust", "Cal", or something similar. This is a square wave output provided by the scope specifically for probe adjustment. Connect the probe tip to this contact.
- 5) Adjust the scope controls to get a trace. You should see a good square wave if the probe is properly compensated.
- 6) If the leading edge of the square wave is distorted (rounded or peaked), adjust the probe for the best possible square wave (the most square).

Potentiometers:

An adjustable resistor is called a potentiometer. (The name comes from its old-time use in voltage measurements.) All potentiometers (pots) work in a similar manner: with a moveable slider or "wiper" that makes contact with a resistor somewhere between its two ends. A rotary potentiometer is shown here: in its package, as bare workings, and as a schematic symbol.

The resistance of a pot is the full resistance between the two outside terminals and is usually written on the part somewhere. Almost every time

you turn a knob on a stereo, TV, or instrument in the lab, you are turning a potentiometer. The resistor in a potentiometer can have a linear or an audio "taper". Audio taper potentiometers are used as volume controls in audio circuits.



Noise on the probes:

Noise can be a problem with these scopes, especially with all the computers in close proximity. The noise will usually manifest itself as a high frequency fuzz that "rides" your signal waveform. If you're having trouble with noise, see the following box:

Noise Problems:

Turn the **BW limit** "on" for each of the analog channels (**A1** and **A2**). This limits the bandwidth of the scope somewhat, cleans up the signal, and won't seriously affect our measurements.

Also, trigger the scope off v_o rather than v_i and do whatever is needed to make the signal stable (**Reject HF**, **Noise Reject**, etc. under **Mode / Coupling**).

EXPERIMENT 1: (30 points)

Procedure:

1. (5 pts) Using the circuit you designed in PRE-LAB **step (1a)**, build and measure the circuit to obtain the following information:

(1a) Fix $v_s = 5V$ DC. Measure v_{out} .

- If it is not close ($0.8 v_s < v_{out} < 1.2 v_s$) modify your design until it is.

- (1b) Measure the current through the 1k “load” resistor and compare that to the value calculated without the added amplifier.
- (1c) Describe in detail where the additional current comes from. Describe what situation you would want to use this circuit. (In a later project, this type of circuit becomes very important)

2. (15 pts.)

- (2a) Apply a sinusoidal signal as v_s . Select amplitude that is easy to observe on the scope. Vary the input frequency and take measurements to create the magnitude Bode plot for this circuit similar to how you did it in Lab 1.
- (2b) Make a rough sketch of the magnitude Bode plot with all relevant points marked. These points should include but are not limited to:
- low-frequency value in the flat section
 - "corner" frequency (f_C)
 - downward slope

IMPORTANT: *A circuit such as the voltage follower does not treat all input waveforms equally. In particular, the circuit will not have a gain of 1 for all input frequencies. The op-amp cannot act like an ideal op-amp at all frequencies— it has limitations.*

The frequency response of a circuit is a measurement of how some characteristic of the circuit varies as a function of the input frequency, in our case, we'll see how the gain changes.

3. (5 pts) How does your measured f_C (the corner frequency or -3dB point) compare to your expected f_C ?

IMPORTANT: *For unity gain, $f_C = f_T$.*

Look up f_T in the data sheet for your op-amp (shown Bandwidth or as BW @ AV=1, read: bandwidth at gain = 1) on a data sheet. Notice that the data sheet shows minimum and typical values for the bandwidth. Your f_C may well be higher. There is nothing wrong with a part exceeding specifications.

4. (5 pts.) From the Bode plot created in 2a, pick an AC input signal v_s with reasonable frequency and amplitude to verify $v_{out} \sim v_s$.

IMPORTANT: *Remember...*

- *The voltage you actually get out of the HP/Agilent function generator is twice the voltage it shows. So, if you want 100 mVpp, set the HP/Agilent to 50 mVpp. Be careful, this little flaw could be a big headache someday.*
- *That gain is V_{out}/V_{in} , and refers to how much bigger the output is than the input-- in this case it's no bigger at all. Incidentally, if you want the scope voltage measurements to make sense, be sure to let it know that your probes are both 1x (Hit A1 and use the right-most button under the screen, repeat for A2). It won't detect the probe automatically.*

- *Be on the lookout for bad probes—I found several, but not before wasting quite a bit of time wondering why all I could get on the scope was noise.*

EXPERIMENT 2: (50 pts)

Procedure:

1. Test the non-inverting amplifier you built in PRE-LAB **step (1b)** and obtain the following information for both amplifiers:

- (1a) Start testing by applying a 100 mVpp sine wave at 1kHz (shown as 50 mVpp on the HP/Agilent). Note that the input signal looks awfully noisy, but most of that is actually scope noise. To prove that, simply hook the scope probe to its own ground and notice how much of the noise is still there.
- (1b) (15 pts) Verify the gain at a relatively low frequency. If not, make modifications to your circuit to get a gain of 101V/V for the non-inverting amplifier.

IMPORTANT: *You may also see “distortion” for the non-inverting amplifier because the gain is very high. If you do, turn down the amplitude and/or frequency so there is no distortion.*

(1c) (5 pts) Find the -3dB point.

(1d) How does this compare to your expected f_c ?

IMPORTANT: *The expected -3dB point can be calculated from the bandwidth value given on the datasheet for your operational amplifier using **Formula 1.0***

Formula 1.0 $GBP = \text{gain} * \text{bandwidth}$ where GBP is the Gain Bandwidth Product on the datasheet

Rearranging **Formula 1.0** and using the -3dB point from (1c), the bandwidth of your circuit can be found by **Formula 1.1**

Formula 1.1 $\text{bandwidth} = GBP/\text{gain}$ where GBP is the Gain Bandwidth Product on the datasheet

IMPORTANT: *If the datasheet does not have a GBP, it will have either a f_T or bandwidth listed. This will be the bandwidth at unity gain and can be treated as the GBP. Again, watch for any distortion of the output waveform and turn down the input amplitude if you see any.*

(1e) (5 pts) Take two measurements beyond f_c and determine the slope at this part of the frequency response curve.

- Sketch a simple frequency response curve showing what you’ve measured.

FOR THE NON-INVERTING CIRCUIT:

- (1f) (5 pts) Change the input coupling of both analog channels (A1 and A2) to AC to eliminate any DC offset in your signals and make phase measurements more accurate.
- Measure the phase shift at f_c . Clever use of the scope's time measurements would help here.
 - Use the Time Cursors to measure the difference between the sine wave zero crossings (Δt) and the Time Period to measure the overall period (T). The phase (degrees) can be found by **Formula 1.3**

Formula 1.3 $\frac{\Delta t}{T} = \frac{\phi}{360^\circ}$ where T is the time period, ϕ is the phase (degrees)

IMPORTANT: A shift to the left is leading (+ angle) and a shift to the right is lagging (- angle).

- The math of a single-pole filter says that the phase angle of the output should be -45° at f_c . How does your measured angle compare? (If you can't measure the right phase, make sure that the scope is triggering on one channel only.)
- (1g) (5 pts) Measure the gain again at $5f_c$
- Measure both the gain and phase shift again at $10f_c$.
 - Did the gain decrease by a factor of 2 between $5f_c$ and $10f_c$ (-20dB per decade roll-off)?
 - Is the phase shift nearly -90° at $10f_c$? These are the theoretically expected results.

2. Measuring the Slew Rate for the non-inverting amplifier.

- (2a) (5 pts) Change the frequency to about 100 kHz and the input voltage to about 0.8 Vpp (shown as 400 mVpp on the HP). Notice that the output signal looks triangular. The op amp simply can't change its output voltage fast enough to keep up with the input.
- The straight ramp in the output indicates that the op amp is changing its output as fast as it can (slewing). The slope of that ramp is its slew rate.
 - Sketch this waveform in your notebook and indicate the slewing on the drawing.
 - Measure this slope. (Measure only the straight part of the line--don't include the rounded tops and bottoms where the op amp isn't slewing.). Use the time cursors and the volts / div marks on the scope to help you. (Make certain that the scope "knows" that you're using 10x probes).
 - Calculate your slew rate as volts/ μ s.
 - Find the slew rate on a data sheet and compare it to your measured slew rate.

IMPORTANT: Your measured value may be higher than that shown in the data sheet, and the part would still be within spec.

(2b) (5 pts) Turn down the frequency until you no longer see the slew rate distortion.

- Find the highest frequency with no distortion (f_{max}).
- Measure the output voltage (V_{pp}).
- Measure the frequency and compare to the theoretical maximum distortion-less frequency. Compute the theoretical maximum distortion-less frequency by using **Formula 1.4**

Formula 1.4 $f_{max} = SR/pV_{pp}$, where SR is slew-rate, V_{pp} is Voltage Peak To Peak

IMPORTANT: *Because the op amp has a limited slew rate, there will be a trade-off between maximum output voltage and maximum frequency.*

For a larger output voltage swing without the slew rate distortion (output triangular): limit the input frequency.

To handle higher frequencies: limit the output voltage swing. The slew rate makes you choose.

3. Measuring the Clipping Levels (saturation)

(3a) (5 pts) Change the input coupling of both analog channels (A1 and A2) back to DC. Turn down the input frequency to about 1 kHz.

- Turn up the input voltage until the output clips both top and bottom (tops and bottoms of the output waveform are cut off)
- Sketch this waveform in your notebook, and indicate the clipping on the drawing.
- Measure both clipping levels as L^- and L^+ (see textbook, p. 18).
- Find the output limits on a data sheet and compare them to your measurements. (Look for “Output Voltage Swing” for $V_S = +15V$ and $R_L > 10 k\Omega$.)

(EXPERIMENT 3 on following page)

EXPERIMENT 3: (40 points)

Procedure:

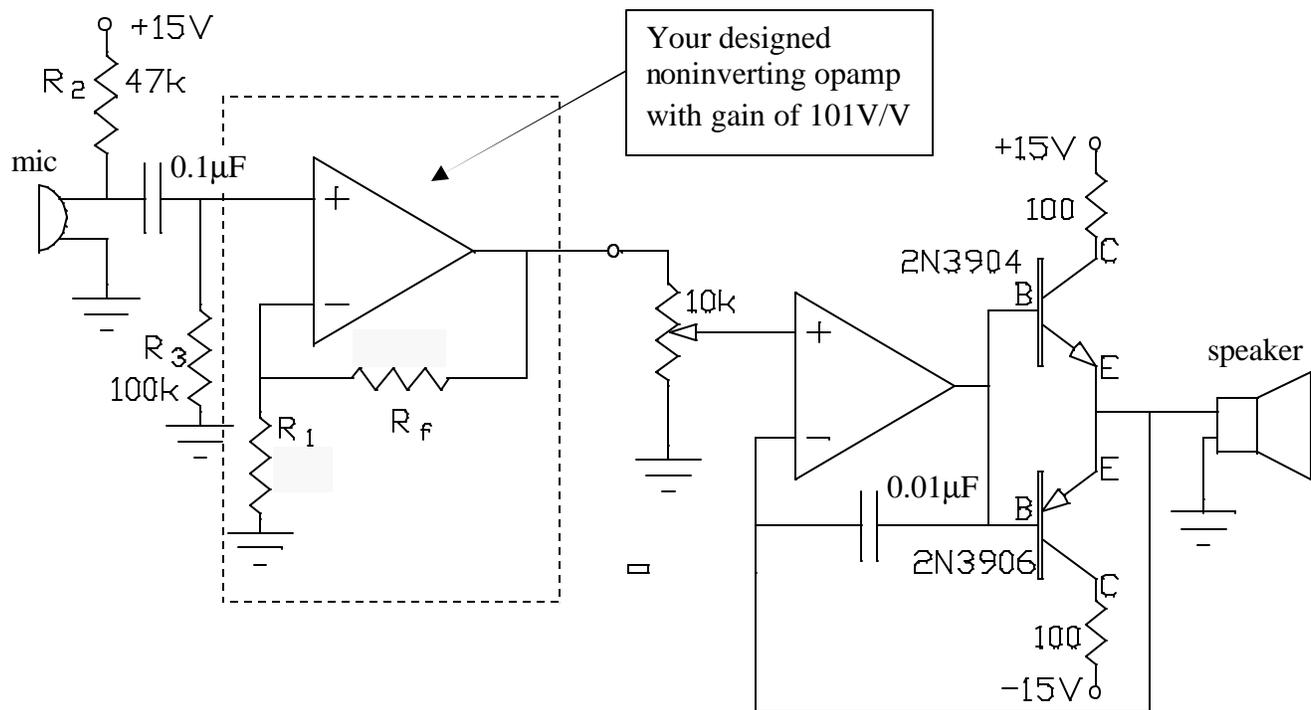
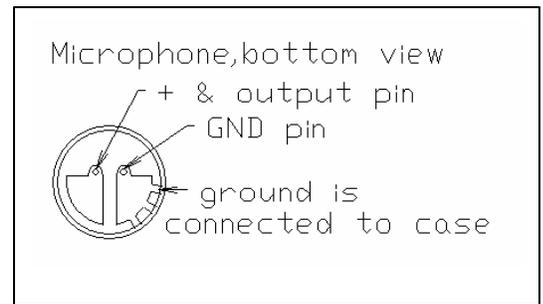


Figure 2. Microphone and Speaker circuit

1. (25 pts) Build and test the circuit in **Fig. 2**.

IMPORTANT: You can break the circuit into parts if it is not working by using the function generator in place of the microphone and testing that the first op amp is indeed amplifying the signal correctly. The function generator can also be connected as the input to the second op amp in place of the first op amp. You should hear an output at the speaker when you speak into the microphone.

Note: Fig. 3 on the following page shows a possible breadboard layout.

Note: The 2N3904 and 2N3906 front is considered the flat part. With the flat part facing you, count from left to right 1,2,3. 1 is considered the Emitter (E), 2 the Base (B), and 3 the Collector (C). E, B, and C are shown on the schematic for each transistor. Verify your connections with your Lab TA before testing it.

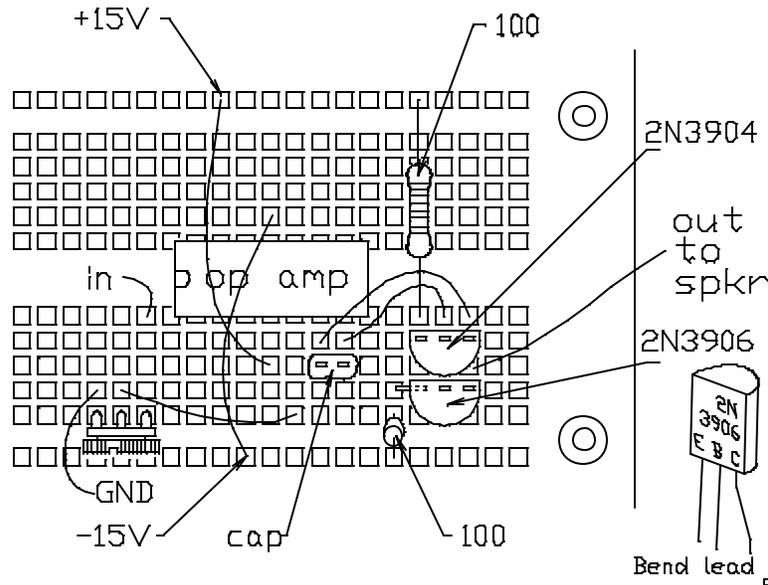


Fig. 3

2. (8 pts) The part shown between the output of the first op amp and the input to second op amp is a potentiometer, used as a volume control in this case.

- Describe in your notebook how the volume control works.

3. (7 pts) Determine how much current is being pulled from the power supply.

IMPORTANT: You ***WILL NEED*** this information to build the power supply project (***PROJECT #1***). ***DO NOT DISCONNECT THIS CIRCUIT. YOU WILL USE IT FOR THE REST OF THE PROJECT'S.***