Linear Circuit Experiment
MAE 171a - Winter 2009

Prof. R.A. de Callafon, email: callafon@ucsd.edu, phone (858) 5343166
**TA: Younghee Han, email: y3han@ucsd.edu , phone: tel. (858) 8221763/8223457**

Overview and Aim of Experiment

In this experiment you will learn how to build and analyze active linear circuits that use to perform basic signal conditioning: **amplification of signals, adding signals and filtering of signals**. The circuits are active (instead of passive) due to the fact that we will use Operational Amplifiers (op-amp's) to create the desired functionality for signal conditioning. By building different interfaces around an op-amps and cascading different op-amp circuits we will eventually make a circuitry that is able to amplify and filter signals coming from a microphone and output to a speaker for audible confirmation of your circuit.

Background on Op-Amps

An operational amplifier (op-amp) is a direct current (DC) coupled high-gain electronic voltage amplifier with differential inputs and typically a single output. In most situations the output of the op-amp is controlled either by negative feedback, which largely determines the magnitude of its output voltage gain. In addition, positive feedback can be used to allow regenerative gain and oscillation circuits. High input impedance at the input terminals and low output impedance are important typical characteristics of an op-amp.

The figure on the right is the schematic of an ideal op-amp. The main part in an amplifier is the dependent voltage source $G*V_i$ that amplifies with a factor G in relation to the voltage drop $V_i = V_+ - V_-$ across the input impedance R_{in}. Many uses have been found for operational amplifiers and an ideal op-amp seeks to characterize the physical phenomena that make op-amps useful.
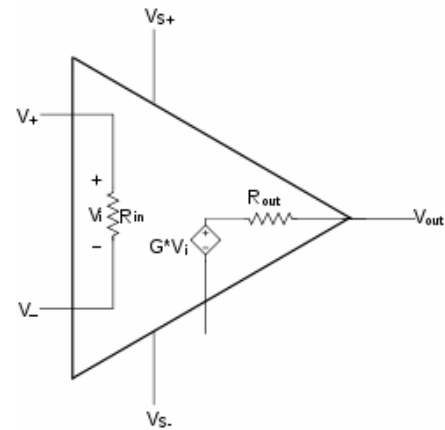
Supply voltages V_{s+} and V_{s-} are used internally to implement the dependent voltage sources. The positive source V_{s+} acts as an upper bound on the output, and the negative source V_{s-} acts as a lower bound on the output. The *internal* V_{s+} and V_{s-} connections are not shown here and will vary by implementation of the operational amplifier.

An *ideal* op-amp has the following properties:

- Infinite (open-loop) gain G. When doing theoretical analysis this means that a limit should be taken as open loop gain G goes to infinity.
- Zero output impedance or $R_{out} = 0$. This means that any output load connected to the op-amp will not influence the output voltage.
- No dynamics or infinite bandwidth and slewrate. This means that the transfer function or frequency response is simply G, without any delay or phase shift, implying infinitely fast settling time.
- Infinite input impedance, making $R_{in} = \infty$. This means that no current (zero input current) would flow from V_+ terminal to the V_-

- No leakage or bias currents. This means that together with the infinite input impedance absolute no input currents are present.
- Zero offset voltage. This means when $V_i = V_+ - V_- = 0$ (by connecting the input terminals), the output $V_{out} = 0$ too. With $R_{out} = 0$ this implies that the output would virtually connected to ground.

Obviously, a real op-amp will approximate these ideal conditions. In this Linear Circuit Experiment we will be using an LM324 (low power quad operational amplifier) and we will test the limits of op-amps and test various circuits for signal conditioning based on realistic op-amp's.

Background on Audio Application
In audio applications, signal conditioning by means of amplification, mixing (adding) and filtering mainly focuses on low-power signals composed primarily of frequencies between 20 Hertz to 20k Hertz (the human range of hearing). These low-power signals typically need to be amplified to a level suitable for driving loudspeakers and is the final stage in a typical audio playback chain. The power of an input signal to an audio signal conditioning circuit may measure only a few hundred microwatts, whiles its output may be hundreds of watts.

In our experiment we will generate low-power signals for your signal conditioning circuitry by a DC-biased condenser (capacitor) microphone. The electronic symbol for a microphone has been depicted on the right. Such a DC-biased microphone uses a thin membrane to respond to changes in sound pressure. This movement is subsequently translated into an electrical signal by having the thin membrane acts as a one of the plates of a capacitor. For that purpose, the plates are DC-biased with a fixed charge Q. Since the Voltage V in Volts across a capacitor is given by $V = Q / C$, where Q = charge in Couloms and C = capacitance in Farads, fluctuations in capacitance C will produce fluctuations in the Voltage V across the capacitor. In the condenser (capacitor) microphone where one of the plates is a thin membrane, the capacitance C varies due to variations in the distance between the plates induced by the sound pressure variations.
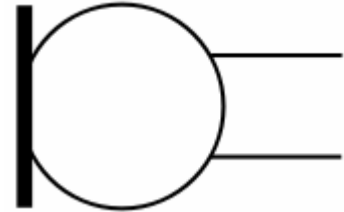
To create (amplified) sound (air pressure variations) of a level audible by the human ear, the signals from the DC-biased condenser (capacitor) microphone need to be amplified and send to a loudspeaker. The loudspeaker is an electroacoustical transducer that converts an electrical signal back into to air pressure variations. Typically, in a loudspeaker an electrical signal is applied to a low impedance voice coil to create a magnetic field. The voice coil and the driver's magnetic system interact to generate the mechanical force to move a lightweight cone back and forth to reproduce the longitudinal pressure waves for sound.

Since loudspeaker have a low impedance (typically 8 Ohm) voice coil and typically require large currents, special speaker driver circuits based on power transistors are required. In our experiment we will create such a power transistor to drive a (low fidelity) speaker to verify our designed signal conditioning circuitry.

WEEK 1

The task of week 1 for the Linear Circuit Experiment is to build a simple circuitry to amplify a low-power microphone signal. During the lab you will experimentally verify the amplification and make sure intermediate and amplified signal do not get `clipped' against the limits of the Voltage supply. To accomplish this task, follow the steps below:

Task1-1: measure and display a signal from a microphone (RC circuit)
Using the MIKE-74 electret microphone and the concept of DC-biasing, first build a DC-bias circuit for the microphone to measure (sound) pressure variations. A basic electrec microphone powering circuit is depicted below and **keep in mind the polarity of the microphone** when building the circuitry: **red wire = positive, black wire = negative.**
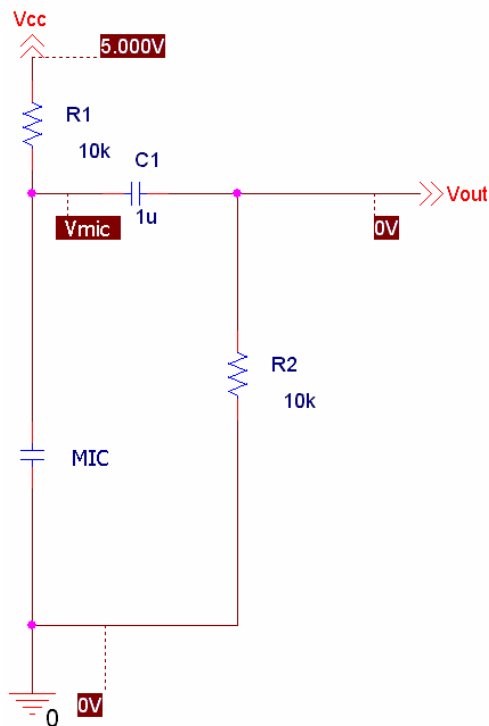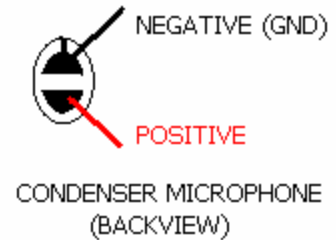

Explanation of circuitry: the microphone circuit must be powered from a single level power source with a **voltage Vcc = +5V** and a **ground GND**, found on your proto-board. The resistor R1 and R2 can be chosen as **R1 = 10kΩ, R2 = 10kΩ**. The resistor R1 is used to create the DC bias voltage/charge for the microphone and R2 is the load resistor. The load resistor together with the decoupling capacitor **C1 = 1μF** creates a high pass filter that eliminates the bias DC bias Voltage on your output signal Vout.

All voltage measurements you will be taken (for example at points Vmic or Vout) are always with respect to the ground.

- Build the circuitry above. Keep in mind the polarity of the microphone.
- Measure the DC-bias (offset) voltage at Vmic. Based on your measurements, compute the (internal) resistance R value of the microphone MIC.
- Measure the DC-bias (offset) voltage at Vout. Explain why it is (close to) 0.
- Display and save time plots of the signal Vout on the scope and estimate its variance by:
  - Shortcut the microphone (Vmic = 0 by connecting Vmic to ground). Theoretically Vout = 0, but due to noise this will not be the case. Hence, this measurement gives the variance of the noise of your circuit design.
  - Remove shortcut and without talking into the microphone. This will give you variance due to the background noise and sounds picked up by the microphone.
  - With talking into the microphone. This will give you estimate of the variance of signal that will eventually will have to be amplified.

Task 2-1: analyze and build op-amp amplifier
This task can be done independently from Task 1-1. Using the LM324 (low power quad operational amplifier) we are going to create a basic (non-inverting) amplifier that we could use to amplify the microphone signal. For that purpose, build the circuitry below/left using the connection diagram of the quad op-amp LM324 depicted on the right.
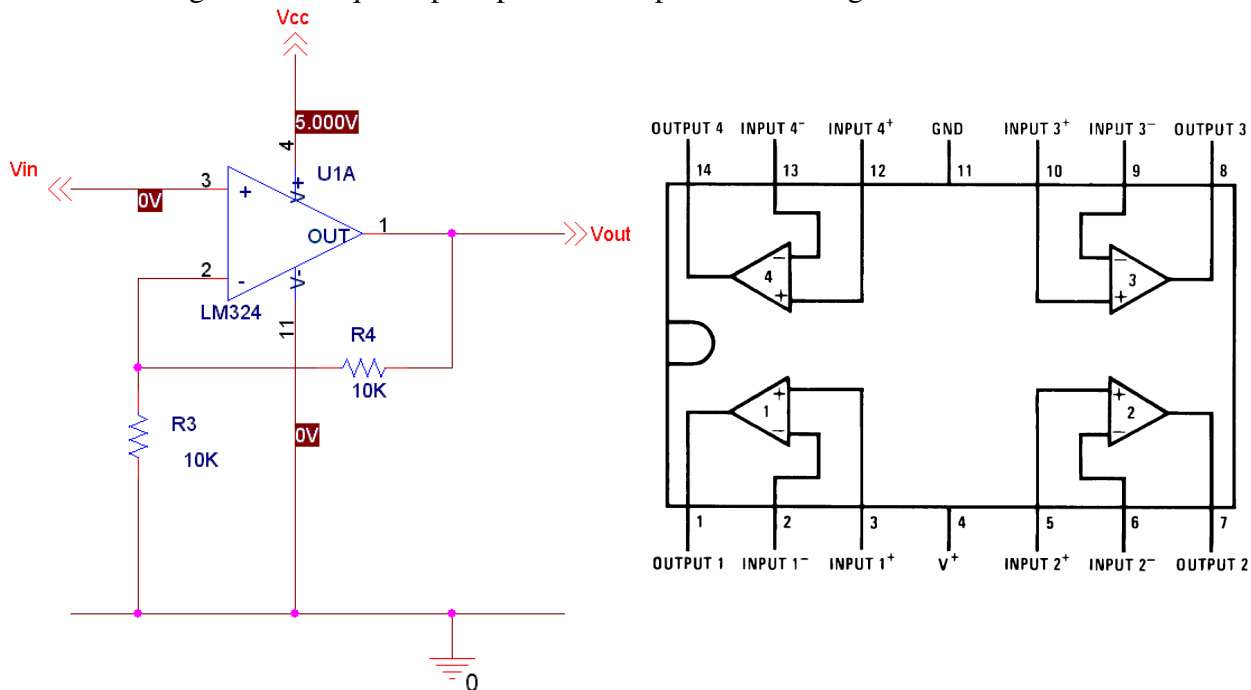
- Build the circuitry above, using **R3 = R4 = 10 kΩ**. With these resistor values, what is the gain/amplification of the (non-inverting) amplifier?
- Shortcut Vin (Vin = 0 by connecting Vin to ground) and measure the DC-bias (offset) of Vout. Explain the difference between the theoretical value of 0 and your measured value.
- Experimentally verify the gain of your op-amp circuitry as follows:
  o Use the function generator to create a **100mVolt peak-peak 1kHz** (sinewave) input signal with a **0 Volt offset** and connect to Vin.
  o Display and save time plots of the signal Vout on the scope. Explain why the output signal Vout gets `clipped' for the negative values of the input signal.
  o Use the function generator to create a **100mVolt peak-peak** (sinewave) input signal and choose an offset to **avoid 'clipping'of the amplified output signal**.
- Now replace resistor **R3 = 1kΩ** resistor and **R4 = 100 kΩ** resistor. With these resistor values, what is the gain/amplification of the (non-inverting) amplifier?
- Experimentally verify the gain of your op-amp circuitry as follows:
  o Use the function generator to create now a **10mVolt peak-peak 1kHz** (sinewave) input signal Vin. The 10mVolt represents the typical signal size from a microphone.
  o Display and save time plots of the signal Vin and Vout on the scope and **make sure Vout does not get `clipped' for both positive and negative values** of the input signal by and choosing an appropriate offset for Vin.
  o Change the frequency of the sinusoid input Vin and observe gain and phase shift between input Vin and output Vout signal. Are you able to see changes in gain and phase shift with different frequency? Save time plots for 100Hz, 1kHz, 10kHz and 20kHz and try to explain the observed results.

Task 3-1: combining microphone and amplifier circuitry

This task combines the results from Task 1 & Task 2. A direct combination of the previous two circuits leads to the schematics right. Unfortunately, due to the fact that the signal from the microphone can be both positive and negative with respect to the ground, the amplified microphone signal will be clipped in the op-amp circuitry for negative values and only amplified for the positive values. To overcome this problem, we modify the combined circuitry as below, where it can be seen that an extra resistor **R5 = 10kΩ** (from the Vcc to the + of the op-amp) and a capacitor **C2=1μF** (from R3 to ground) have been added.
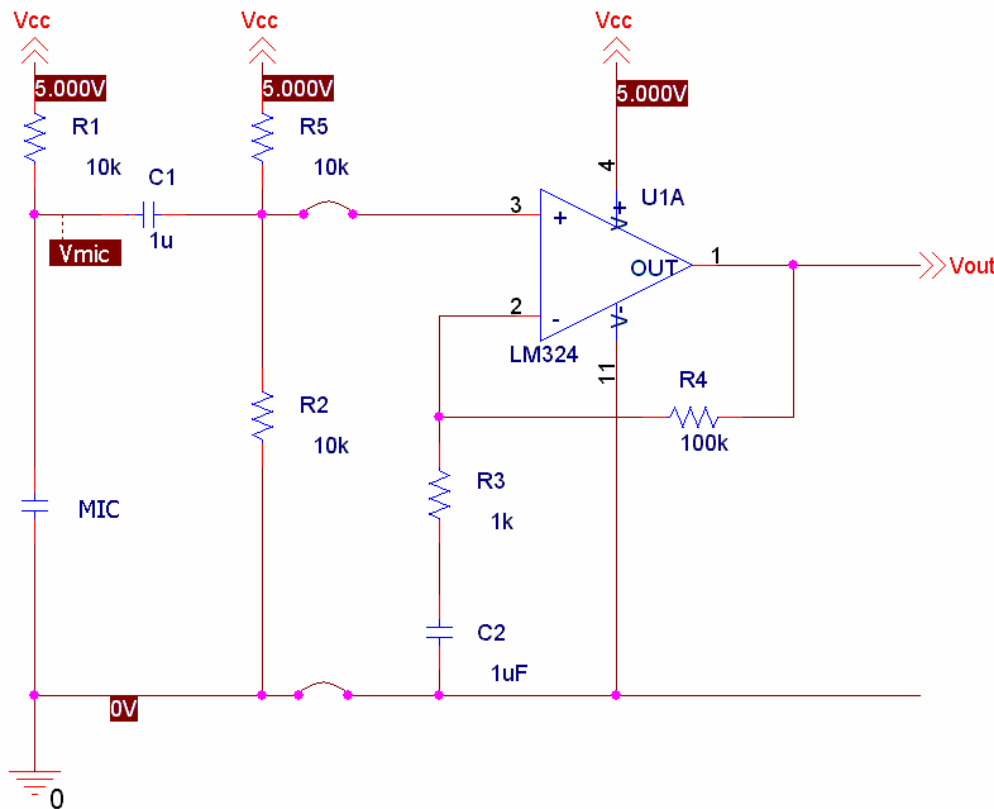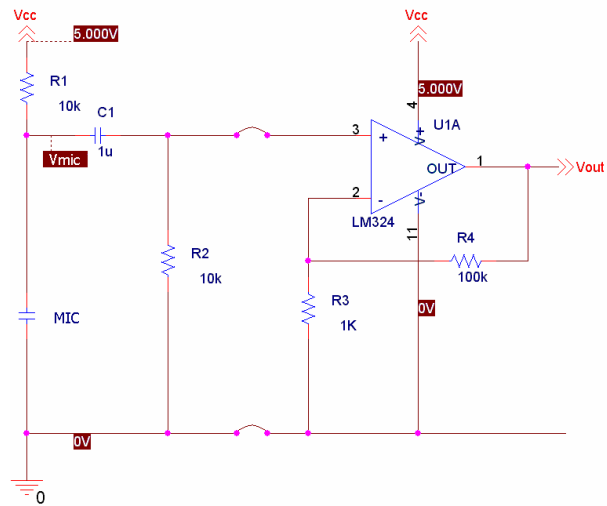

- Combine both microphone and op-amp circuitry by building the circuitry above.
- Measure the DC-bias (offset) at the + input of the op-amp and DC-bias (offset) at Vout. Explain why both of them should (theoretically) be at 2.5V. Explain any differences between this theoretical values and your measured value.
- Display and save time plots of the signal Vmic and Vout on the scope and conclude on the performance of your microphone pre-amplifier (you can create interesting sounds yourself) Are you able to amplify the signal about 100 times without `clipping' the signal?

WEEK 2
The task of week 2 for the Linear Circuit Experiment is to build a simple **adding circuitry and differential amplifier** to build a mixer for your microphone signal by adding a signal. During the lab you will experimentally verify the gain of a differential amplifier, and verify the subtraction or adding of two signals and make sure intermediate and amplified signal do not get `clipped' against the limits of the Voltage supply.

Task 2-1: non-inverting summing circuit
This task can be done independently from Task 2-2. We are going to create a basic *(non-inverting) signal summing circuit* that allows **adding of (multiple) signals**. We use a *non-inverting* signal summing to allow summing of two signals within the single level power supply of 0-5V. The summing will not be perfect, as the summing junction of signals is not connected to a virtual ground as in an inverting summing circuit (causing voltage offset due to small input currents). Using one of the other LM324 op-amps present in your quad op-amp, the circuitry below demonstrates summing of two signals.
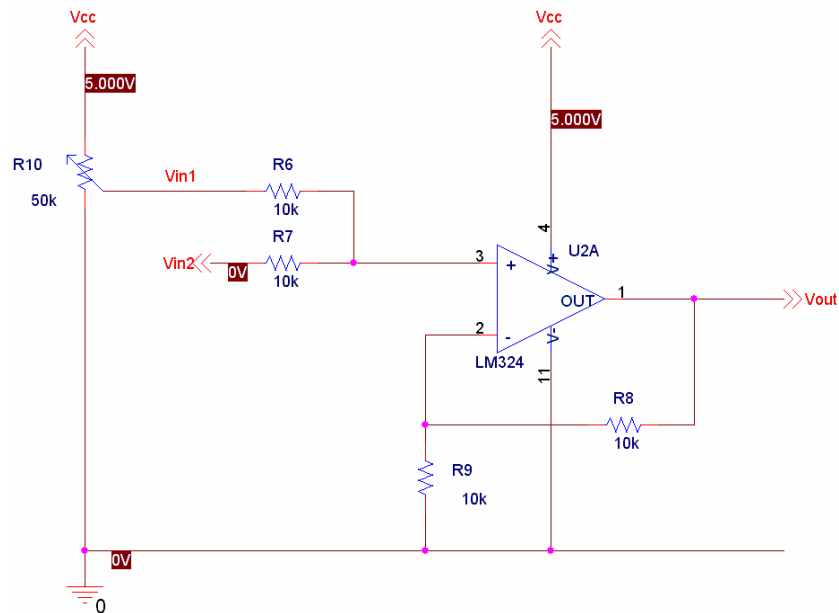
- Build the circuitry above using **R6 = R7 = R8 = R9 = 10kΩ** and **R10 = 50 kΩ potentiometer** (variable resistor) using the connection diagram of the quad op-amp LM324 depicted above.
- Explain (in your report) that Vout = Vin1 + Vin2 in case **R6 = R7 = R8 = R9.**
- Connect Vin2 to ground, measure Vin1 and adjust the potentiometer R10 so that Vin1=2.5Volt. Obviously, this will provide the right DC-bias voltage as Vout = 2.5Volt in case Vin2=0 Volt.
- Experimentally verify the functioning of your summing circuitry as follows:
  - o Use the function generator to create a **1Volt peak-peak 1kHz** (sinewave) input signal with a **0 Volt offset** and connect to Vin2.
  - o Display and save time plots of the signal Vin2 and Vout on the scope.
  - o Verify that indeed Vout = Vin1 + Vin2. Explain why you can use the potentiometer R10 to adjust the DC-offset (bias) of the output signal Vout.
- Explain (in your report) why small input currents in/out of the op-amp + and - inputs cause a non-ideal summing circuit. What will happen if **R6 = R7 = R8 = R9 = 100kΩ**?
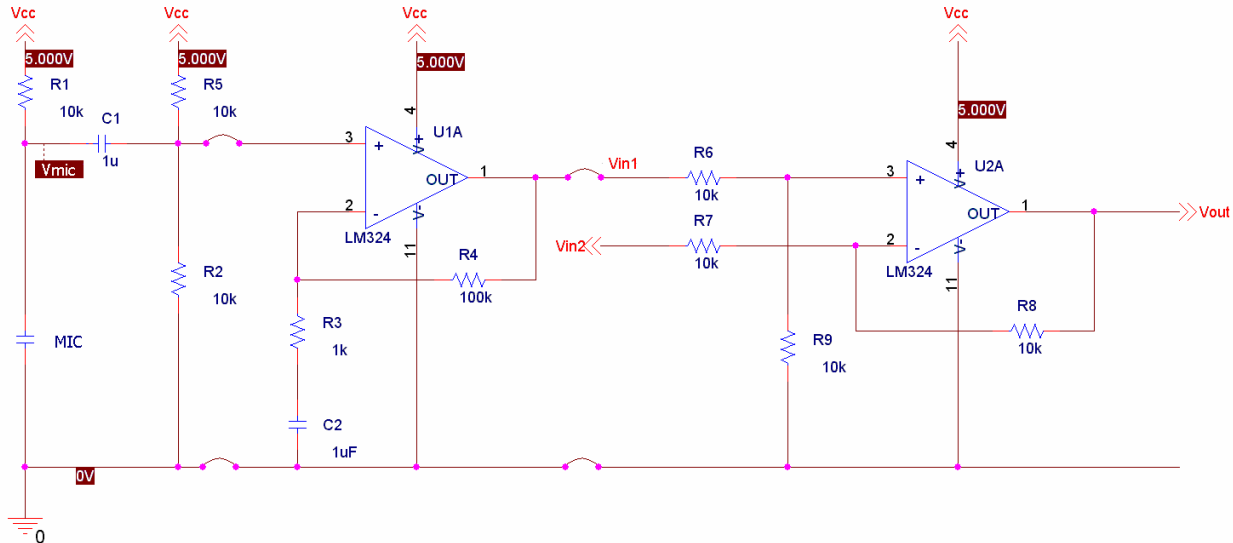
Task 2-2: differential amplifier
This task can be done independently from Task 2-1. We are going to create a basic differential amplifier that we could use to **amplify the difference between two signals**. Using one of the other LM324 op-amps present in your quad op-amp used in your first week, the circuitry below will be used to demonstrate the differential amplifier.
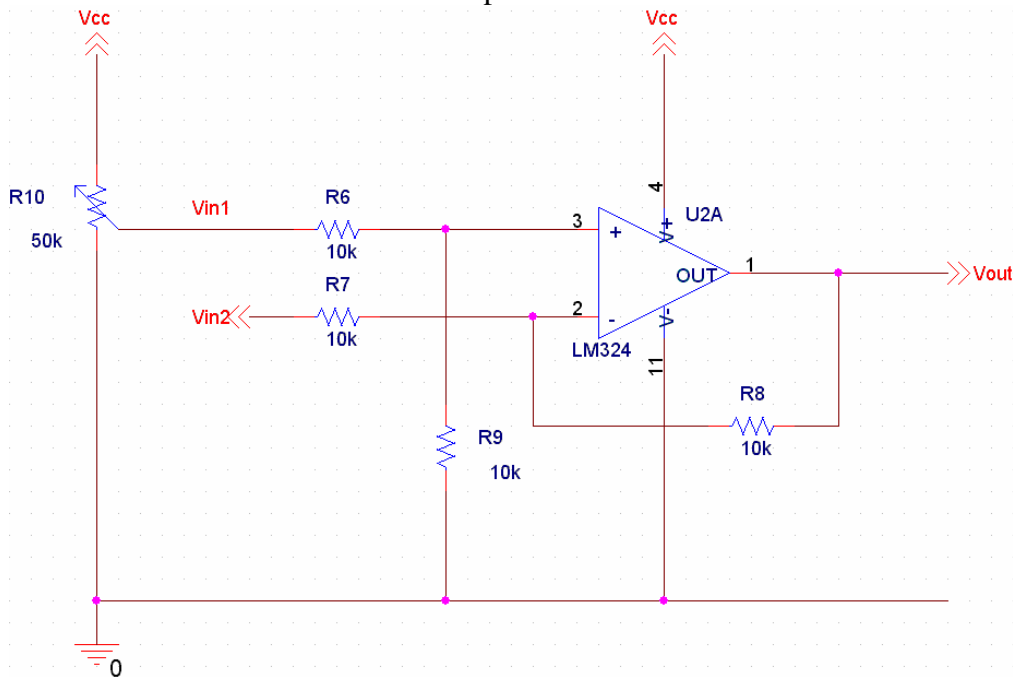
- Build the circuitry above using **R6 = R7 = R8 = R9 = 10kΩ** and **R10 = 50 kΩ potentiometer** (variable resistor) using the connection diagram of the quad op-amp.
- Explain (in your report) that Vout = Vin1 – Vin2 in case **R6 = R7 = R8 = R9.**
- Connect Vin2 to ground, measure Vin1 and adjust the potentiometer R10 so that Vin1=2.5Volt. Obviously, this will provide the right DC-bias voltage as Vout = 2.5Volt in case Vin2=0 Volt.
- Experimentally verify the functioning of your differential amplifier as follows:
  o Use the function generator to create a **100mVolt peak-peak 1kHz** (sinewave) input signal with a **0 Volt offset** and connect to Vin2.
  o Display and save time plots of the signal Vin2 and Vout on the scope.
  o Verify that indeed Vout = Vin1 – Vin2 and explain why you can use the potentiometer R10 to adjust the DC-offset (bias) of the output signal Vout.
  o Adjust R10 so that the DC-offset (bias) of Vout is *as small as possible*, without **'clipping'the output signal** Vout.
- Leave **R8 = R9 = 10kΩ** but make **R6 = R7 = 1kΩ**. Based on these resistor values, compute the differential gain γ in Vout = γ(Vin2-Vin1).
- Experimentally, determined the gain as follows: **create a graph** in which you plot the amplitude of Vin (x-axis) against Vout (y-axis) for different Vin amplitudes. The graph can be used to inspect the linearity of your circuit and the slope of the curve reflects the gain of your circuit.
- Explain (in your report) the difference in gain/amplitude and offset of Vout. Adjust R10 to avoid `clipping' of the output signal Vout if necessary.

Task 2-3: combining microphone amplifier circuitry of week 1 with differential amplifier
This task combines the end results from Task 1-3 (Week 1) and the differential amplifier of Task 2-1 above. A direct combination of the circuits leads to the schematics below. With **R6 = R7 = R8 = R9 = 10kΩ** the extra input signal Vin2 is simply subtracted from the amplified microphone signal circuitry designed in Week 1.

- Combine the Week 1 amplified microphone signal circuitry and the differential amplifier circuitry from Task 2-1 by building the circuitry above.
- Explain why the potentiometer R10 has been eliminated from the circuitry. E.g. why is there no need to introduce a bias Voltage on Vin1 of the differential amplifier?
- Again measure the DC-bias at Vin1 and the DC-bias (offset) at Vout. Explain why both of them should (theoretically) be at 2.5V. Explain any differences between this theoretical values and your measured value.
- Experimentally verify the functioning of your circuitry as follows:
  - Use the function generator to create a **100mVolt peak-peak 1kHz** (sinewave) input signal with a **0 Volt offset** and connect to Vin2.
  - Display and save time plots of the signal Vout on the scope by also creating sound signals for the microphone. Verify that can see the 100m Volt sine wave on your amplified microphone signal without cliping the output signal Vout.
- Since the differential amplifier could be used for additional amplification of the microphone signal, suggest changes to the above circuitry to boost the amplified microphone signal with an additional factor of 2 using the differential amplifier (second op-amp in your circuitry only). Keep in mind DC-offset and possible clipping of your output signal. Test your suggestions and display and save time plots of the signal Vout.

WEEK 3

The task of week 3 for the Linear Circuit Experiment is to build a simple **low pass filter** to filter both the microphone and the mixer signal. During the lab you will experimentally verify the frequency response of your filter (by sending in signals with different frequencies). Again you need to verify that filtered signal do not get `clipped' against the limits of the Voltage supply.

Task 3-1: first order filter with high output impedance
This task can be done independently from task 3-2. Simply combining an RC circuit with an op-amp acting as a voltage follower allows you to build a 1st order low pass filter with a high output impedance:
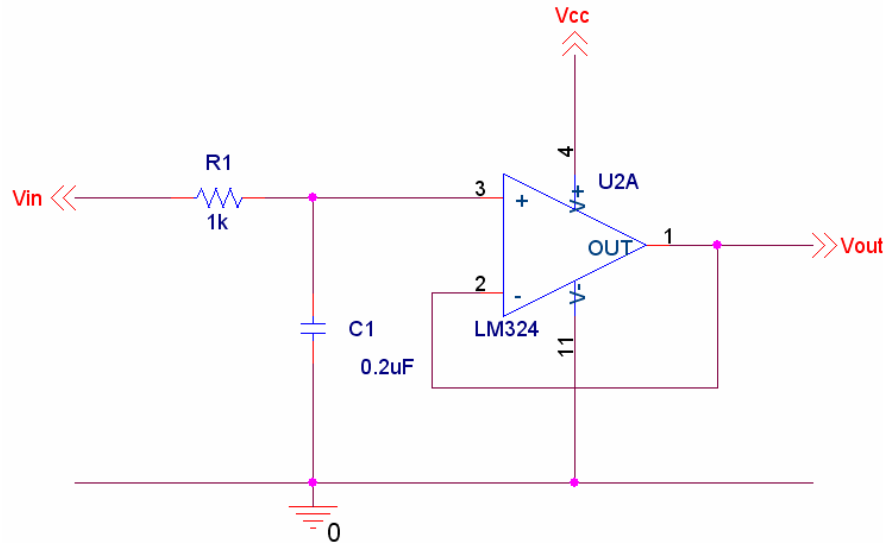

The resistor value R1 and capacitor value C1 are chosen in such a way that a low pass filter with a cut-off frequency of approximately 800Hz is obtained. Consider the following tasks:

- **Build above circuitry** using the connection diagram of the quad op-amp. Combine capacitors (think about having to put them is series or parallel) to get the 0.2μF.
- Given the resistor values above, **compute the exact cut-off frequency** of the 1st order low pass filter.
- For your report, **write down the transfer function of this filter** and compute the Bode response with Matlab (see also lecture notes)
- Experimentally verify the amplitude frequency response of this filter as follows:
    o Use the function generator to create a **1Volt peak-peak** (sinewave) input signal with a **2 Volt offset** and connect to Vin. The offset ensures the output does not get clipped or distorted.
    o **Vary the frequency** from 10Hz – 10kHz of the sinewave input signal and record the amplitude of the output signal Vout for various (at least 10) frequency points. Best is to choose the frequency points with a logarithmic difference, e.g. 10Hz, 25Hz, 50Hz, 100Hz, 250Hz, 500Hz, 1kHz, 2kHz, 5kHz, 10kHz.
    o **Create a loglog graph** in which you plot the applied frequency (log x-axis) against amplitude of Vout (log y-axis) for the different frequencies. Compare the graph (or measured points) with the frequency response created by Matlab. Keep in mind that gain in Db = 20*^{10}log(gain).
    o **Experimentally verify the cut-off frequency** of your filter and verify that this filter (only) gives a 20Db/dec reduction above the cut-off frequency.

Task 3-2: 2nd order low pass Butterworth filter

This task can be done independently from task 3-1. To increase the reduction of the amplitude of the filtered signal at higher frequencies, while still maintaining approximately the same cut-off frequency of the 1st order filter in task 3-1, you are going to build and test a 2nd order filter:
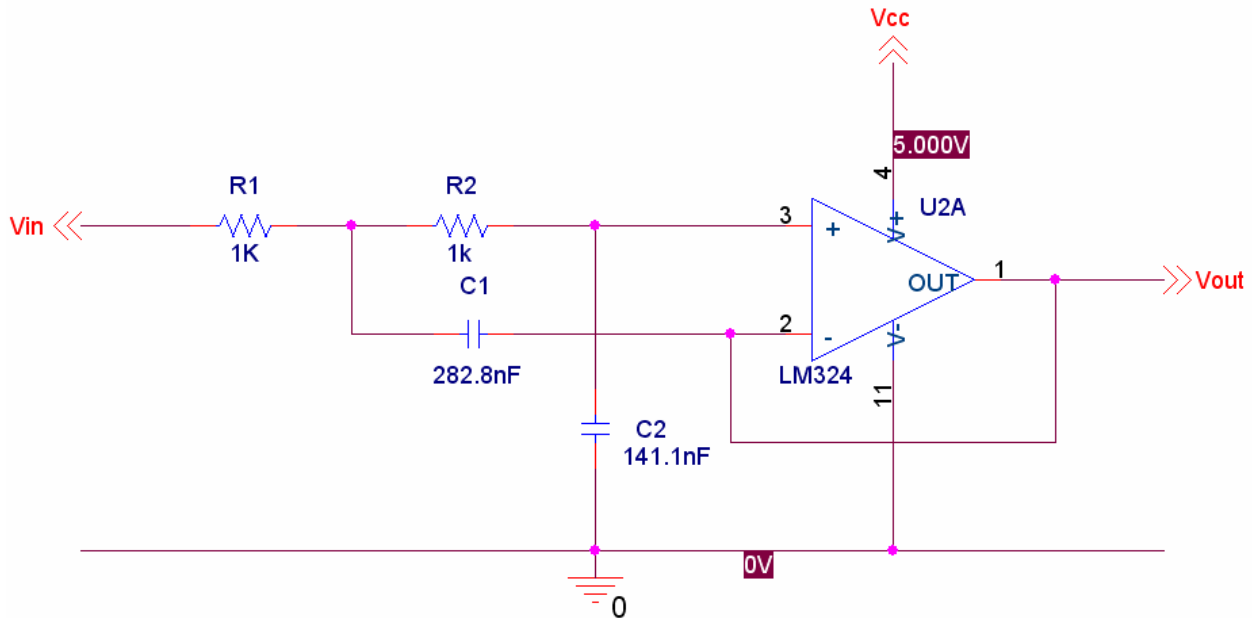

- **Build above circuitry** using the connection diagram of the quad op-amp. The capacitor values are the ideal values to obtain a so-called 2nd order low-pass Butterworth filter, characterized by a resonance frequency of $\omega_n = $ **5000rad/s** (approximately 800Hz) and a damping ratio of $\beta = \sqrt{1/2}$ (approximately 0.707). Combine capacitors (think about having to put them is series or parallel) to approach the values as closely as possible.
- For your report, **compute the Bode response with Matlab** using the numerical value for ω_n above (see also lecture notes)
- Experimentally verify the amplitude frequency response of this filter as follows:
    o Use the function generator to create a **1Volt peak-peak** (sinewave) input signal with a **2 Volt offset** and connect to Vin. The offset ensures the output does not get clipped or distorted.
    o **Vary the frequency** from 10Hz – 10kHz of the sinewave input signal and record the amplitude of the output signal Vout for various (at least 10) frequency points. Best is to choose the frequency points with a logarithmic difference, e.g. 10Hz, 25Hz, 50Hz, 100Hz, 250Hz, 500Hz, 1kHz, 2kHz, 5kHz, 10kHz.
    o **Create a loglog graph** in which you plot the applied frequency (log x-axis) against amplitude of Vout (log y-axis) for the different frequencies. Compare the graph (or measured points) with the frequency response created by Matlab. Keep in mind that gain in Db = $20*^{10}$log(gain).
    o **Experimentally verify the cut-off frequency** of your filter and verify that this filter (only) gives a 40Db/dec reduction above the cut-off frequency.

Task 3-3: combine microphone amplifier and mixing circuitry of week 2 with low pass filter

This last task is to combine of the low pass filters you designed and testing this week (1st order or 2nd order Butterworth) with the completed circuitry of week 2. This is your final circuit that contains all parts you have designed and looks pretty complicated. IF you have made it this far – congratulations! To finish up the final testing of your circuit, we ask you to do a few final tests that should be included in your report:

- Connect Vin2 (mixer/differential amplifier input) to ground and display and save time plots of the signal Vin1 (the amplified microphone signal) and Vout (the amplified and filtered microphone signal) on the scope.
- Connect Vin2 (mixer/differential amplifier input) to the function generator and generate a 100mV 5kHz zero offset sine wave signal. This signal is supposed to represent a disturbance on your microphone signal that was mixed in. Again display Vout (the amplified and filtered microphone signal) on the scope and verify that the 5kHZ disturbance has been reduced.
- As a final last fun test you can listen to the amplified and filtered microphone signal via a (high impedance) ear phone speaker. You do this as follows: **connect the output of the last op-amp (filter design stage) via a 1µF capacitor (to remove the bias voltage) to your ear phone speaker**. Now you can listen to the amplified and filtered microphone signal. An optional power boost can be given to the last op-amp to also power a low impedance speaker (see next page).
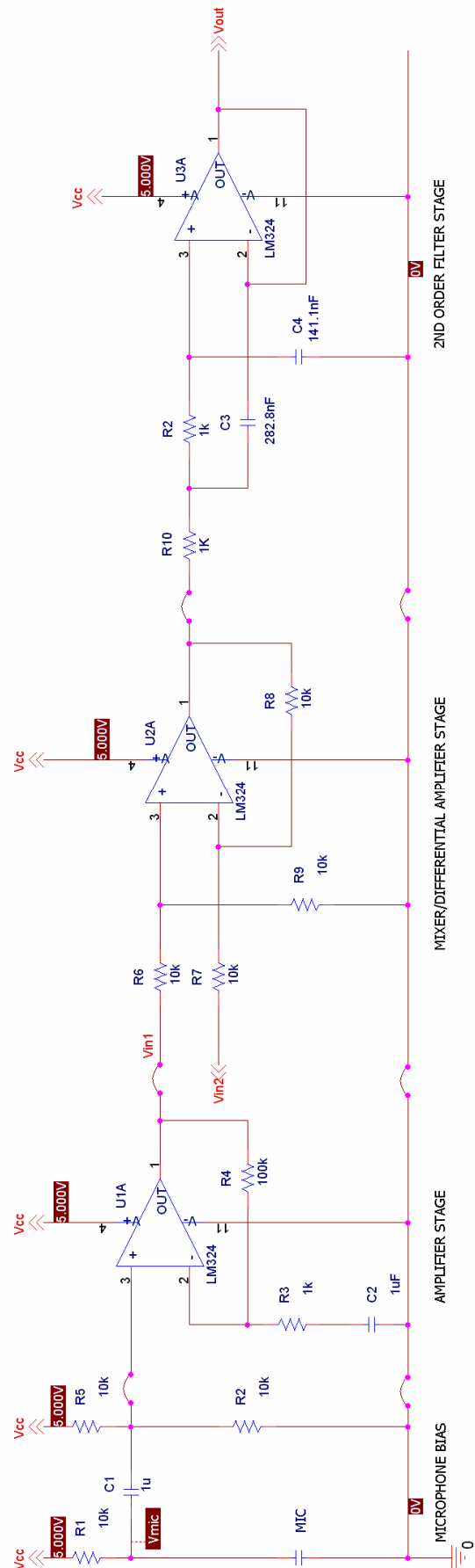
Task 3-4 (optional): power boost stage

The LM324 op-amp we are using in our circuits is limited in its output current and output power it can generate. To boost the power of the op-amp, an additional output stage can be added by means of power transistors that can handle larger currents. Since currents will have to flow in and out of the op-amp due to fluctuating (high frequency) signals we like to send out, the output stage consists of a pair of NPN and PNP transistors that act as voltage followers:
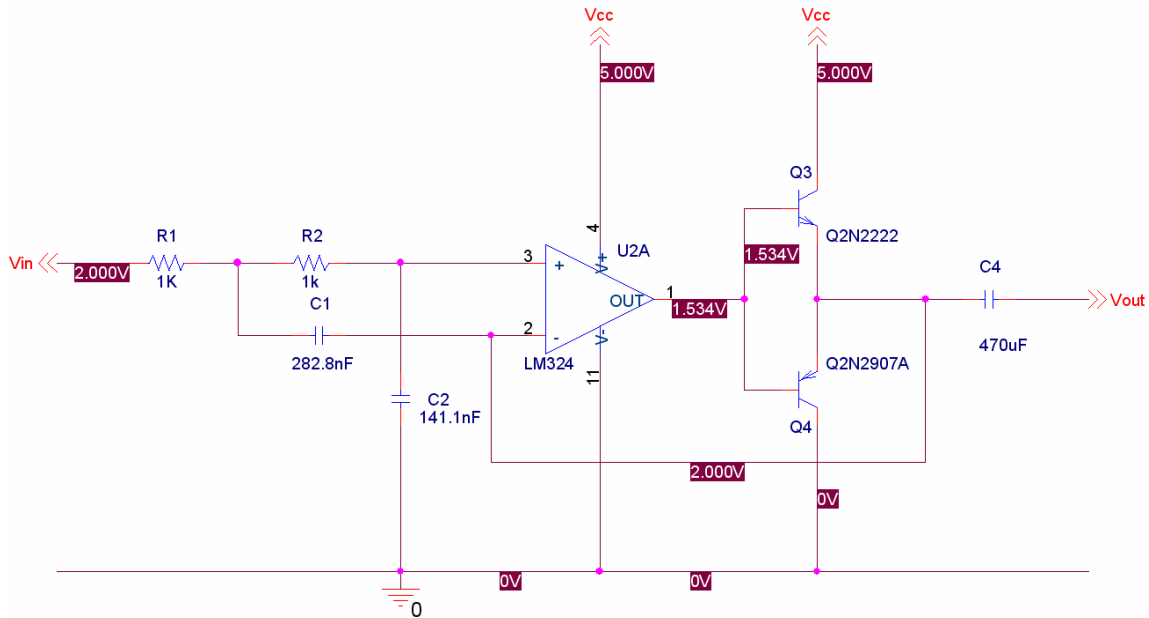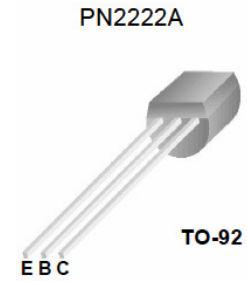
2nd order Butterworth filter with ω_n = 5000rad/s and output power boost

The output stage is quite simple and this particular power boost can deliver quite some power into an 8-Ohm speaker. Notice the following:
- The feedback back into the V- input of the op-amp now comes from the output stage instead of the output of the op-amp. This is to compensate also for the (non-linear) behavior of the NPN/PNP transistor pair.
- The bias voltage at the output the op-amp is approximate 0.5Volt lower than the input bias voltage applied to the V+ of the op-amp. This is due to the fact that the NPN and PNP transistor with their internal diode structure always loose 0.5-0.6V as voltage follower.
- Since a bias voltage is present on Vin (and thus the output of the op-amp), both NPN and PNP transistors always conduct current and no resistor/diode pairs are needed to keep the transistors from conducting current. Otherwise an additional double resistor and diode pair has to be added to each transistor basis.
- A large capacitor C4 is used to filter of the bias voltage. The capacitor C4 along with a low impedance load (such as an 8-Ohm speaker) creates a high pass filter that filter of the bias voltage. A large capacitor is needed to make the cut-off frequency as low as possible handle for a low impedance 8-Ohm load.

You can use this circuitry to power the 8-Ohm speakers provided in the lab (optional).