Linear Circuit Experiment (MAE171a)

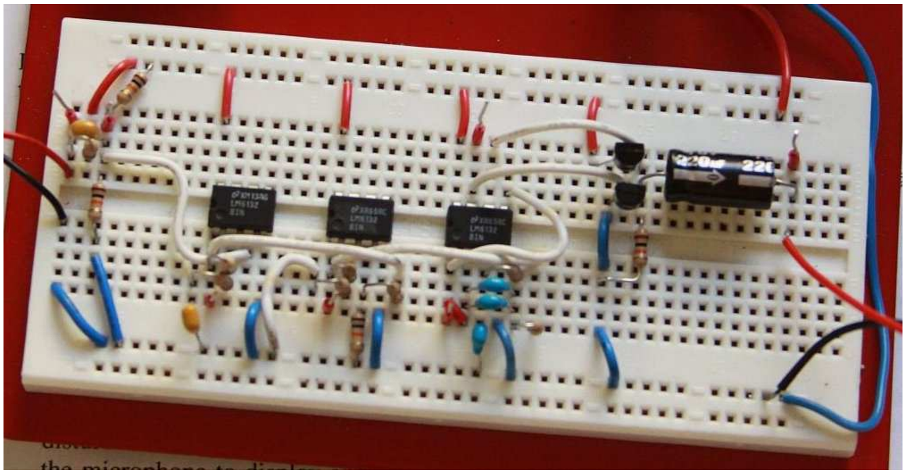
Prof: Raymond de Callafon
email: `callafon@ucsd.edu`

TAs: Sophia Wang, **email**: `s9wang@ucsd.edu`
Yichao Xu, **email**: `yix004@ucsd.edu`

class information and lab handouts will be available on
`http://maecourses.ucsd.edu/labcourse/`

Main Objectives of Laboratory Experiment:

modeling, building and debugging of op-amp based linear circuits for standard signal conditioning



Main Objectives of Laboratory Experiment:

modeling, building and debugging of op-amp based linear circuits for standard signal conditioning

Ingredients:

- modeling of standard op-amp circuits
- signal conditioning with application to audio (condensor microphone as input, speaker as output)
- implementation & verification of op-amp circuits
- sensitivity and error analysis

Background Theory:

- Operational Amplifiers (op-amps)
- Linear circuit theory (resistor, capacitors)
- Ordinary Differential Equations (dynamic analysis)
- Amplification, differential & summing amplifier and filtering

Outline of this lecture

- Linear circuits & purpose of lab experiment
- Background theory
  - op-amp
  - linear amplification
  - single power source
  - differential amplifier
  - summing circuit
  - filtering
  - simulation
- Laboratory work
  - week 1: microphone and amplification
  - week 2: audio amplification and mixing
  - week 3: filtering and complete circuit (with power boost)
- Summary
- What should be in your report

Linear Circuits & Signal Conditioning

Signal conditioning crucial for proper signal processing.
Applications may include:

- Analog to Digital Conversion
  - Resolution determined by number of bits of AD converter
  - Amplify signal to maximum range for full resolution
- Noise reduction
  - Amplify signal to allow processing
  - Filter signal to reduce undesired aspects
- Feedback control
  - Feedback uses reference $r(t)$ and measurement $y(t)$
  - Compute difference $e(t) = r(t) - y(t)$
  - Amplify, Integrate and or Differentiate $e(t)$ (PID control)
- Signal generation
  - Create sinewave of proper frequency as carrier
  - Create blockwave of proper frequency for counter
  - etc. etc.

Purpose of Lab Experiment

In this laboratory experiment we focus on a (relatively simple) signal conditioning algorithms: *amplification*, *adding/difference* and basic (at most 2nd order) *filtering*.

Objective: to model, build and debug op-amp based linear circuits that allow signal conditioning algorithms.

We apply this to an audio application, where the signal of a condenser microphone needs to amplified, mixed and filtered.

Challenge: single source power supply of 5 Volt. Avoid clipping/distortion of amplified, mixed and filtered signal.

Aim of the experiment:

- insight in op-amp based linear circuits (MAE140, MAE170)
- build and debug (frustrating)
- compare theory (ideal op-amp) with practice (build and test)
- verify circuit behavior (experiment and simulation)

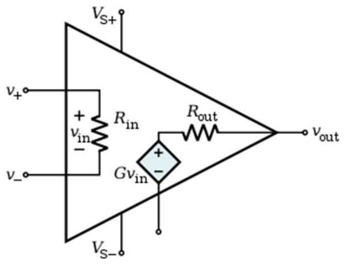
Background Theory - op-amp

op-amp = operational amplifier

more precise definition:
DC-coupled high-gain electronic voltage amplifier with differential inputs and a single output.

- DC-coupled: constant (Direct Current) voltage at inputs results in a constant (DC) voltage at output
- differential inputs: two inputs V_- and V_+ and the difference $V_\delta = V_+ - V_-$ is only relevant and amplified
- high-gain: $V_{out} = G(V_+ - V_-)$ where $G >> 1$.

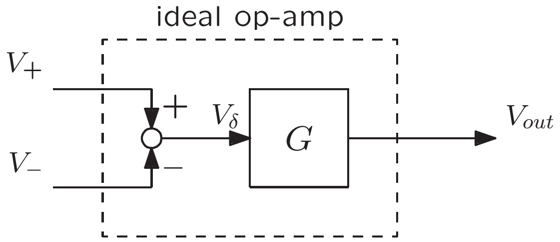
Background Theory - op-amp



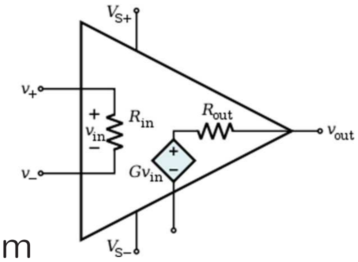
Ideal op-amp (equivalent circuit right):

- input impedance: $R_{in} = \infty \Rightarrow i_{in} = 0$
- output impedance: $R_{out} = 0$
- gain: $V_\delta = (V_+ - V_-)$, $V_{out} = GV_\delta$, $G = \infty$
- rail-to-rail: $V_{S-} \leq V_{out} \leq V_{S+}$

Ideal op-amp (block diagram below)



Background Theory - op-amp



Infinite input impedance ($R_{in} = \infty$) useful to minimize load on sensor/input.

Zero output impedance ($R_{out} = 0$) useful to minimize load dependency and obtain maximum output power.

Rail-to-rail operation to maximize range of output V_{out} between negative source supply V_{S-} and positive source supply V_{S+}.

But why (always) infinite gain G? Obviously:

$$V_{out} = \begin{cases} V_{S+} & \text{if} \quad V_+ > V_- \\ 0 & \text{if} \quad V_+ = V_- \\ V_{S-} & \text{if} \quad V_+ < V_- \end{cases}$$

does not very useful with any (small) noise on V_+ or V_-.

Background Theory - op-amp

Usefulness of op-amp with high gain G only by **feedback**!

Consider open-loop behavior:

$$V_{out} = GV_\delta, \text{ where } V_\delta = V_+ - V_-$$

and create a feedback of V_{out} by choosing

$$V_- = KV_{out}$$

to make

$$V_\delta = V_+ - KV_{out}$$

Then

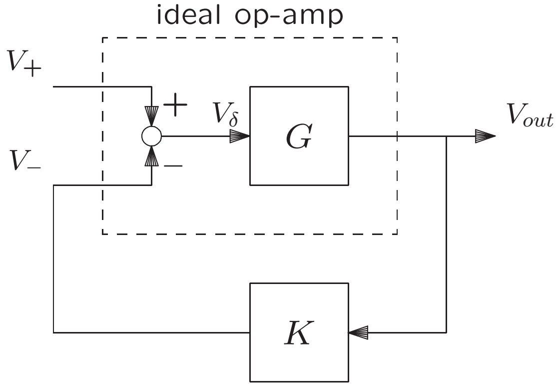
$$V_{out} = GV_\delta = GV_+ - GKV_{out}$$

allowing us to write

$$V_{out} = \frac{G}{1 + GK} V_+$$

Background Theory - op-amp

So, with the feedback $V_- = KV_{out}$ we obtain $V_{out} = \frac{G}{1+GK}V_+$



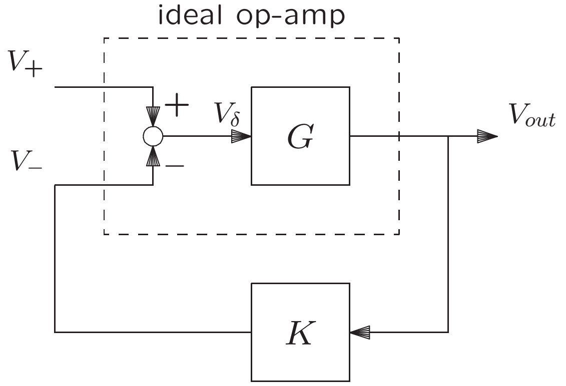
In case $G \to \infty$ we get well-defined relation:

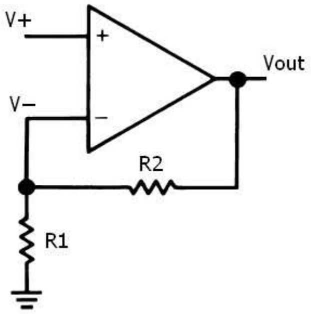
$$V_{out} = \frac{1}{K}V_+$$

- Don't care what gain G is, as long as it is LARGE
- Make sure K is well-defined and accurate
- If $0 < K < 1$ then V_+ is nicely amplified to V_{out} by $1/K$

Background Theory - op-amp

Amplification $1/K$ by feedback K of ideal high gain op-amp:





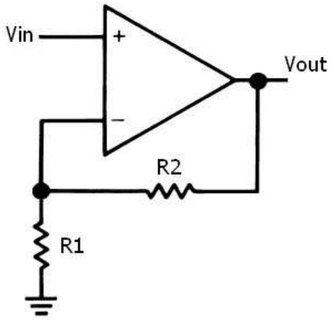
Series of R_1 and R_2 leads to voltage divisor on V_- given by:

$$V_- = \frac{R_1}{R_1+R_2} \cdot V_{out} = K \cdot V_{out}, \;\; 0 < K \leq 1$$

and with ideal high gain op-amp we get

$$\lim_{G\to\infty} V_{out} = \lim_{G\to\infty} \frac{G}{1+GK}V_+ = \frac{1}{K}V_+ = \frac{R_1+R_2}{R_1}V_+ = \left(1+\frac{R_2}{R_1}\right)V_+$$

Background Theory - non-inverting amplifier (voltage follower)



Our first application circuitry:

$$V_{out} = \left(1 + \frac{R_2}{R_1}\right) V_{in}$$

So-called voltage follower in case

$R_1 = \infty$ (not present) and $R_2 = 0$

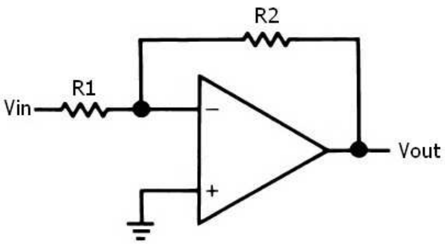
where $V_{out} = V_{in}$ but improved output impedance!

Quick (alternative) analysis based on $V_+ = V_-$ and $i_+ = i_- = 0$:

- Since $i_- = 0$ and series R_1, R_2 we have $V_- = \frac{R_1}{R_1 + R_2} V_{out}$
- Hence

$$V_{in} = V_+ = \frac{R_1}{R_1 + R_2} V_{out} \Rightarrow V_{out} = \frac{R_1 + R_2}{R_1} V_{in} = \left(1 + \frac{R_2}{R_1}\right) V_{in}$$

Background Theory - inverting amplifier



Similar circuit but now negative sign:

$$V_{out} = -\frac{R_2}{R_1} V_{in}$$

Quick (alternative) analysis based on $V_+ = V_-$ and $i_+ = i_- = 0$:

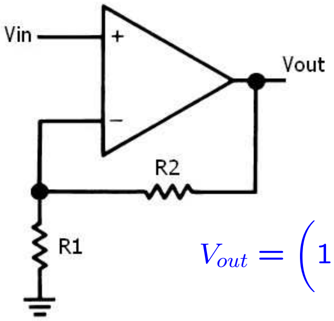
- With $V_- = V_+ = 0$ and $i_- = 0$, Kirchhoff's Current Law indicates

$$\frac{V_{in}}{R_1} + \frac{V_{out}}{R_2} = 0$$

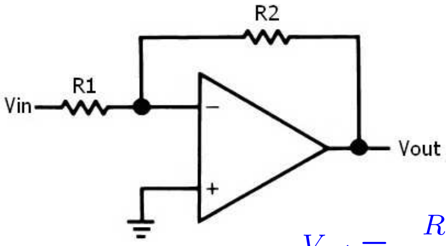
- Hence

$$\frac{V_{out}}{R_2} = -\frac{V_{in}}{R_1} \Rightarrow V_{out} = -\frac{R_2}{R_1} V_{in}$$

Background Theory - effect or rail (source) voltages



$$V_{out} = \left(1 + \frac{R_2}{R_1}\right) V_{in}$$



$$V_{out} = -\frac{R_2}{R_1} V_{in}$$

Formulae are for ideal op-amp with boundaries imposed by negative source supply V_{S-} and positive source supply V_{S+}

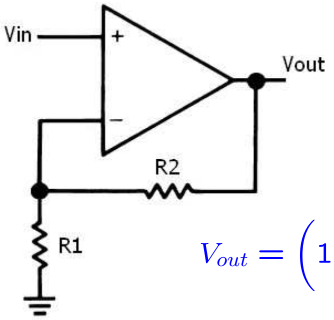
$$V_{S-} \leq V_{out} \leq V_{S+} \quad \text{(rail-to-rail op-amp)}$$

Single voltage power supply with $V_{S+} = V_{cc}$ and $V_{S-} = 0$ (ground):

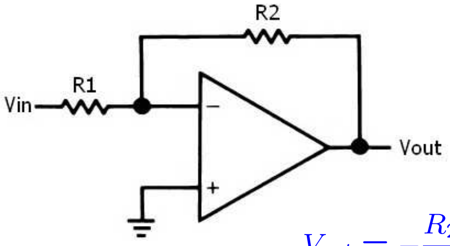
- Limits use of inverting amplifier ($V_{out} < 0$ not possible)
- Limits use of large gain R_2/R_1 ($V_{out} > V_{cc}$ not possible)

Design challenge: $0 < V_{out} < V_{cc}$ to avoid 'clipping' of V_{out}.

Background Theory - effect or rail (source) voltages



$$V_{out} = \left(1 + \frac{R_2}{R_1}\right) V_{in}$$



$$V_{out} = -\frac{R_2}{R_1} V_{in}$$

Single voltage power supply with $V_{S+} = V_{cc}$ and $V_{S-} = 0$ (ground) complicates amplification of

$$V_{in}(t) = a \sin(2\pi f t)$$

as $-a < V_{in}(t) < a$ (both positive and negative w.r.t. ground).

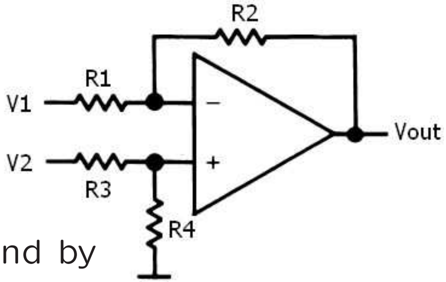
Example: audio application (as in our experiment).

To ensure $0 < V_{out} < V_{cc}$ provide offset compensation

$$V_{in}(t) = a \sin(2\pi f t) + a$$

to ensure $V_{in}(t) > 0$ and use non-inverting amplifier.

Background Theory - differential amplifier



Instead of amplifying one signal, amplify the difference:

$$V_{out} = \frac{R_1 + R_2}{R_3 + R_4} \cdot \frac{R_4}{R_1} V_2 - \frac{R_2}{R_1} V_1$$

Difference or differential amplifier is found by inverting amplifier and adding signal to V_+ via series connection of R_3 and R_4. Analysis:

- With $i_+ = 0$ the series or R_3 and R_4 leads to $V_+ = \frac{R_4}{R_3+R_4} V_2$
- With $V_- = V_+$ and Kirchhoff's Current Law we have

$$\frac{V_1 - \frac{R_4}{R_3+R_4} V_2}{R_1} + \frac{V_{out} - \frac{R_4}{R_3+R_4} V_2}{R_2} = 0$$

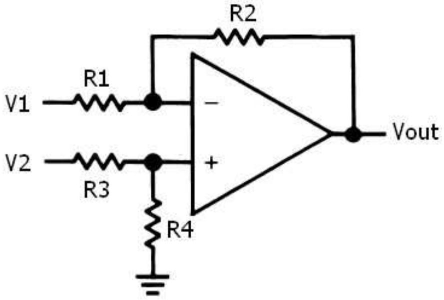
Hence

$$V_{out} = \frac{R_2}{R_1} \cdot \frac{R_4}{R_3 + R_4} V_2 + \frac{R_4}{R_3 + R_4} V_2 - \frac{R_2}{R_1} V_1$$

or

$$V_{out} = \frac{R_1 + R_2}{R_3 + R_4} \cdot \frac{R_4}{R_1} V_2 - \frac{R_2}{R_1} V_1$$

Background Theory - differential amplifier



Choice $R_1 = R_3$ and $R_2 = R_4$ reduces

$$V_{out} = \frac{R_1 + R_2}{R_3 + R_4} \cdot \frac{R_4}{R_1} V_2 - \frac{R_2}{R_1} V_1$$

to

$$V_{out} = \frac{R_2}{R_1}(V_2 - V_1)$$

create a difference/differential amplifier.

Further choice of $R_1 = R_3$, $R_2 = R_4$ *and* $R_2 = R_1$ yields

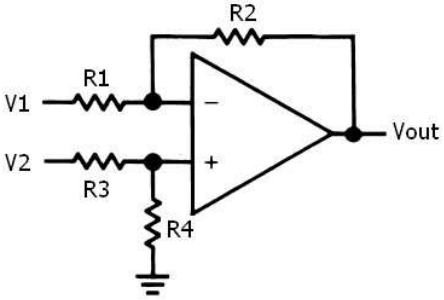
$$V_{out} = V_2 - V_1$$

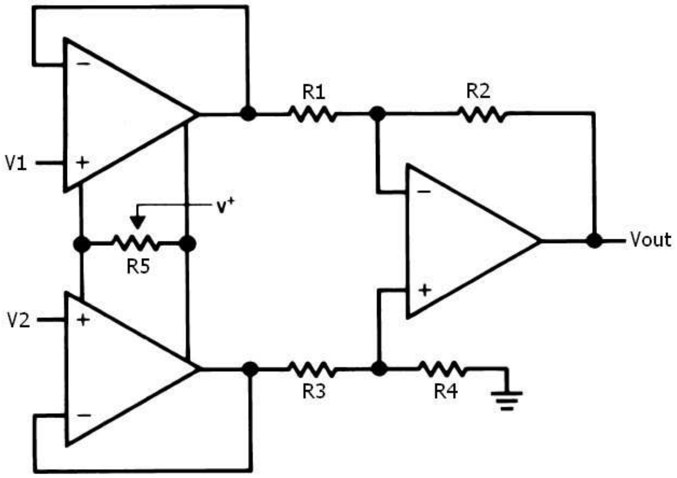
and computes the difference between input voltages V_1 and V_2.

NOTE: $V_2 > V_1$ for a single voltage power supply with $V_{S+} = V_{cc}$ and $V_{S-} = 0$ (ground) to avoid clipping of V_{out} against ground.

Background Theory - more advanced differential amplifiers

Difference amplifier does not have high input impedance (loading of sensors). Better design with voltage followers:





With

$$\frac{R_2}{R_1} = \frac{R_4}{R_3}$$

we have

$$V_{out} = \frac{R_2}{R_1}(V_2 - V_1)$$

R_5 is used to adjust offset (balance)

Background Theory - more advanced differential amplifiers

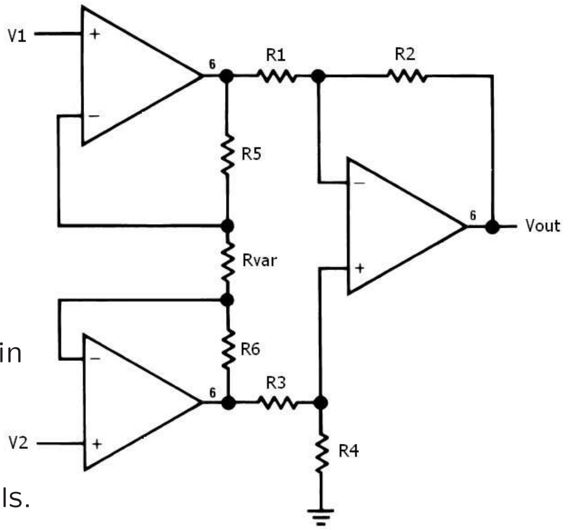
Even better differential amplifier that has a variable gain is a so-called instrumentation amplifier:

Setting all resistors

$$R_i = R, \ i = 1, 2, \ldots 5$$

except R_{var}, makes

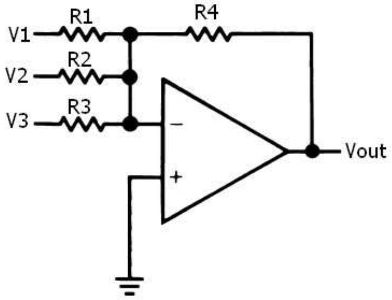
$$V_{out} = \left(1 + \frac{2R}{R_{var}}\right)(V_2 - V_1)$$



High input impedance and variable gain via an (external) resistor R_{var} makes this ideal for the amplification of (non-grounded) instrumentation signals.

Instrumentation amplifiers are made & sold as a single chip.

Background Theory - inverting summing amplifier



Inverting amplifier can also be extended to add signals:

$$V_{out} = -R_4 \left(\frac{V_1}{R_1} + \frac{V_2}{R_2} + \frac{V_3}{R_3} \right)$$

Analysis follows from Kirchhoff's Current Law for the − input of the op-amp:

- With $V_- = V_+$ we have $V_- = 0$
- With $i_- = 0$ we have

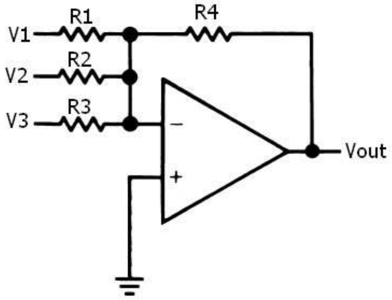
$$\frac{V_{out}}{R_4} + \frac{V_1}{R_1} + \frac{V_2}{R_2} + \frac{V_3}{R_3} = 0$$

Hence

$$V_{out} = -R_4 \cdot \left(\frac{V_1}{R_1} + \frac{V_2}{R_2} + \frac{V_3}{R_3} \right)$$

creating a weighted sum of signals.

Background Theory - inverting summing amplifier



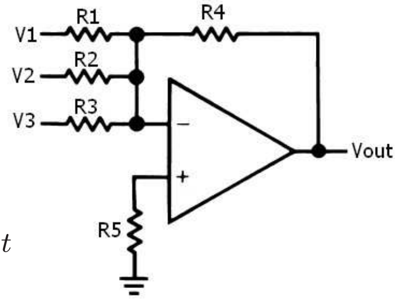
The choice $R_1 = R_2 = R_3$ reduces

$$V_{out} = -R_4 \left(\frac{V_1}{R_1} + \frac{V_2}{R_2} + \frac{V_3}{R_3} \right)$$

to

$$V_{out} = -\frac{R_4}{R_1} (V_1 + V_2 + V_3)$$

simply amplifying the sum of the signals.

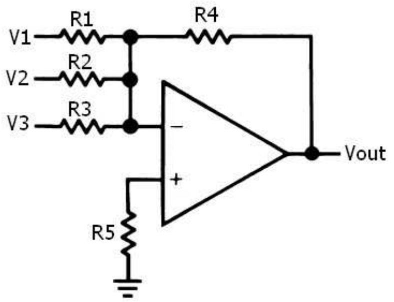


Oftentimes extra resistor R_5 is added:

$$\frac{1}{R_5} = \frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} + \frac{1}{R_4}$$

to account for possible small (bias) input currents $i_- \neq 0$, $i_+ \neq 0$. This ensures V_{out} remains sum, without bias/offset.

Background Theory - inverting summing amplifier



Inverting summing amplifier:

$$V_{out} = -R_4\left(\frac{V_1}{R_1}+\frac{V_2}{R_2}+\frac{V_3}{R_3}\right)$$

and for $R_1 = R_2 = R_3$:

$$V_{out} = -\frac{R_4}{R_1}(V_1+V_2+V_3)$$

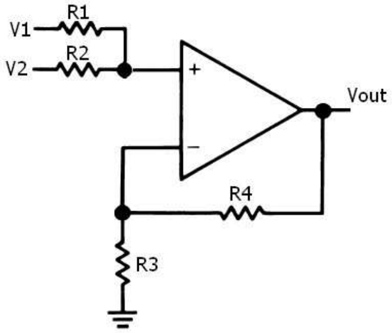
has a limitation for single source voltage supplies:

Single voltage power supply with $V_{S+} = V_{cc}$ and $V_{S-} = 0$:

- Limits use of inverting summer ($V_{out} < 0$ not possible)
- Limits use of large gain R_4/R_1 ($V_{out} > V_{cc}$ not possible)

'clipping' of V_{out} will occur if sum of input signals is positive.

Background Theory - non-inverting summing amplifier



Based on a non-inverting amplifier signals can also be summed:

$$V_{out} = \left(1+\frac{R_4}{R_3}\right)\frac{R_1R_2}{R_1+R_2}\left(\frac{V_1}{R_1}+\frac{V_2}{R_2}\right)$$

Analysis:

- due to $i_- = 0$ we have $V_- = \frac{R_3}{R_3+R_4}V_{out}$
- Due to $V_+ - V_-$ and $i_+ = 0$ with Kirchhoff's Current Law:

Hence

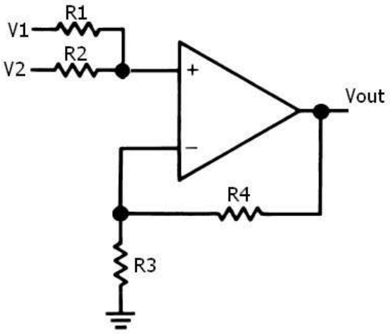
$$\frac{V_1 - \frac{R_3}{R_3+R_4}V_{out}}{R_1} + \frac{V_2 - \frac{R_3}{R_3+R_4}V_{out}}{R_2} = 0$$

$$\frac{V_1}{R_1}+\frac{V_2}{R_2} = \left(\frac{1}{R_1}+\frac{1}{R_2}\right)\frac{R_3}{R_3+R_4}V_{out}$$

and

$$V_{out} = \left(1+\frac{R_4}{R_3}\right)\frac{R_1R_2}{R_1+R_2}\left(\frac{V_1}{R_1}+\frac{V_2}{R_2}\right)$$

Background Theory - non-inverting summing amplifier



The choice $R_1 = R_2$ reduces

$$V_{out} = \left(1 + \frac{R_4}{R_3}\right) \frac{R_1 R_2}{R_1 + R_2} \left(\frac{V_1}{R_1} + \frac{V_2}{R_2}\right)$$

to

$$V_{out} = \left(1 + \frac{R_4}{R_3}\right) \frac{V_1 + V_2}{2}$$

indicating amplifciation of the sum V_1 and V_2 if $R_4 \geq R_3$.

Further choice of also $R_1 = R_2 = R_3 = R_4$ leads to

$$V_{out} = V_1 + V_2$$

indicating a simple summation of V_1 and V_2.

Unlike inverting summing amplifier, no extra resistor can be added to compensate for bias input current.
Not desirable: source impedance part of gain calculation. . .

Background Theory - filtering

So far, all circuits were build using op-amps and resistors.

When building filters, mostly capacitors are used as negative, positive or grounding elements.

Interesting phenomena: resistor value of capacitor depends on frequency of signal.

Analysis for capacitor: capacitance C is ratio between charge Q and applied voltage V:

$$C = \frac{Q}{V}$$

Since charge $Q(t)$ at any time is found by flow of electrons:

$$Q(t) = \int_{\tau=0}^{t} i(\tau) d\tau$$

we have

$$V(t) = \frac{Q(t)}{C} = \frac{1}{C} \int_{\tau=0}^{t} i(\tau) d\tau$$

Background Theory - filtering

Application of **Laplace transform** to

$$V(t) = \frac{Q(t)}{C} = \frac{1}{C}\int_{\tau=0}^{t} i(\tau)d\tau$$

yields

$$V(s) = \frac{1}{Cs} i(s)$$

Hence we can define the impedance/resistence of a capacitor as

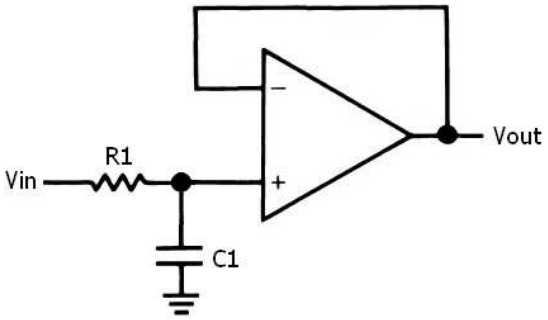
$$R(s) = \frac{V(s)}{i(s)} = \frac{1}{Cs}$$

With Fourier analysis we use $s = j\omega$ and we find the frequency dependent 'equivalent resistor value of a capacitor':

$$R(j\omega) = \frac{1}{jC\omega}$$

This value will allow analysis of op-amp circuits based on resistors (as we have done so far)

Background Theory - filter (1st order RC)



Consider simple 1st order RC-filter with voltage follower op-amp.

Due to $i_+ = 0$ we have

$$V_+(j\omega) = \frac{\frac{1}{C_1 j\omega}}{R_1 + \frac{1}{C_1 j\omega}} V_{in}(j\omega)$$

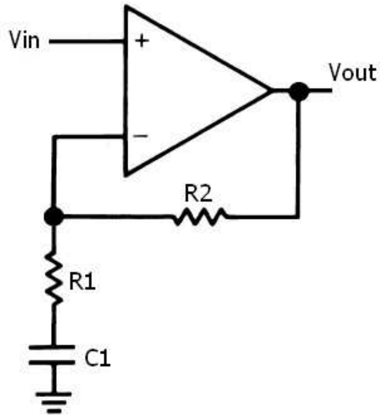
With $V_{out} = V_- = V_+$ we have

$$V_{out}(j\omega) = \frac{1}{R_1 C_1 j\omega + 1} V_{in}(j\omega)$$

This is a 1st order low-pass filter with a cut-off frequency

$$\omega_c = \frac{1}{R_1 C_1} \text{ rad/s or } f_c = \frac{1}{2\pi R_1 C_1} \text{ Hz}$$

Background Theory - filter (audio amplifier)



Consider circuit of non-inverting amplifier where R_1 is now series of R_1 and C_1. Equivalent series resistance is given by

$$R_1 + \frac{1}{jC_1\omega}$$

Application of gain formula for non-inverting amplifier yields:

$$V_{out}(j\omega) = \left(1 + \frac{R_2}{R_1 + \frac{1}{jC_1\omega}}\right) V_{in}(j\omega)$$

We can directly see:

- For low frequencies $\omega \to 0$ we obtain a Voltage follower with $V_{out} = V_{in}$
- For high frequencies $\omega \to \infty$ we obtain our usual non-inverting amplifier $V_{out}(j\omega) = \left(1 + \frac{R_2}{R_1}\right) V_{in}(j\omega)$

Background Theory - filter (audio amplifier)

This transition between Voltage follower to amplifier as a function of the frequency is also called *audio amplifier*.

The transition between low and high frequency can be studies better by writing $V_{out}(s) = G(s)V_{in}(s)$ where $G(s)$ is a transfer function.

This allows us to write

$$V_{out}(s) = \left(1 + \frac{R_2}{R_1 + \frac{1}{C_1 s}}\right) V_{in}(s)$$

as

$$V_{out}(s) = \left(1 + \frac{R_2 C_1 s}{R_1 C_1 s + 1}\right) V_{in}(s) = \frac{(R_1 + R_2)C_1 s + 1}{R_1 C_1 s + 1} V_{in}(s)$$

making

$$G(s) = \frac{(R_1 + R_2)C_1 s + 1}{R_1 C_1 s + 1}$$

Background Theory - filter (audio amplifier)

The transfer function

$$G(s) = \frac{(R_1 + R_2)C_1 s + 1}{R_1 C_1 s + 1}$$

has the following properties:

- single pole at $p_1 = -\frac{1}{R_1 C_1}$ and found by solving $R_1 C_1 s + 1 = 0$.
- single zero at $z_1 = -\frac{1}{(R_1+R_2)C_1}$ and found by solving $(R_1 + R_2)C_1 s + 1 = 0$.
- DC-gain of 1 and found by substitution $s = 0$ in $G(s)$. Related to the final value theorem for a step input signal $v_{in}(t)$:

  $$\lim_{t\to\infty} V_{out}(t) = \lim_{s\to 0} s \cdot V_{out}(s) = \lim_{s\to 0} s \cdot G(s) \cdot \frac{1}{s} = \lim_{s\to 0} G(s)$$

  where $\frac{1}{s}$ is the Laplace transform of the step input $v_{in}(t)$.
- High frequency gain of $\frac{R_1+R_2}{R_1} = 1 + \frac{R_2}{R_1}$ and found by computing $s \to \infty$.

Background Theory - filter (audio amplifier)

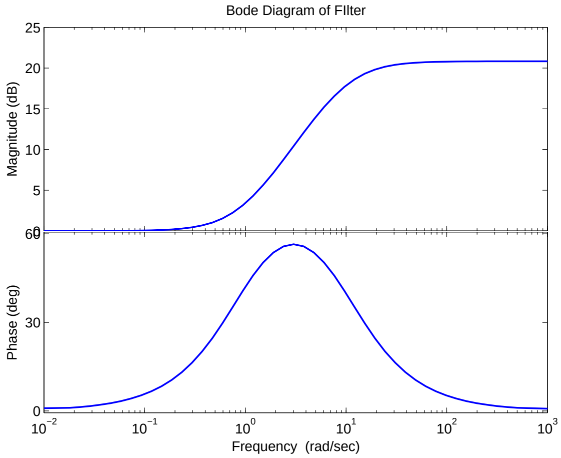
The transfer function

$$G(s) = \frac{(R_1 + R_2)C_1 s + 1}{R_1 C_1 s + 1}$$

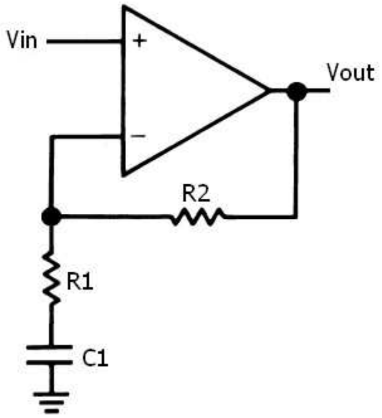
is a first order system where zero $z_1 = -\frac{1}{(R_1+R_2)C_1} < p_1 = -\frac{1}{R_1 C_1}$. This indicates $G(s)$ is a lead filter.

Easy to study in Matlab:

```
>> R2=100e3;R1=10e3;C1=10e-6;
>> G=tf([(R1+R2)*C1 1],[R1*C1 1]);
>> bode(G)
```



Background Theory - filter (audio amplifier)



Amplification lead filter circuit with

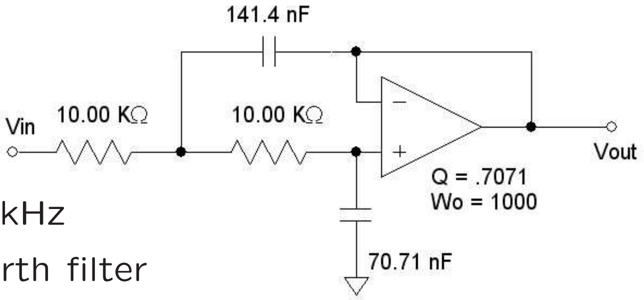
$$V_{out}(j\omega) = \left(1 + \frac{R_2}{R_1 + \frac{1}{jC_1\omega}}\right) V_{in}(j\omega)$$

will be used to strongly amplify a small high frequent signal but maintain (follow) the DC-offset.

From the previous analysis we see:

- Gain at DC ($\omega = 0$) is simply 1.
- Gain at higher frequencies approaches $1 + \frac{R_2}{R_1}$

Background Theory - filter (2nd order Butterworth)



Another fine filter:

- 2nd order low pass Butterworth filter
- Pass-band frequency of 1kHz
- 2nd order 1kHz Butterworth filter is a standard 2nd order system

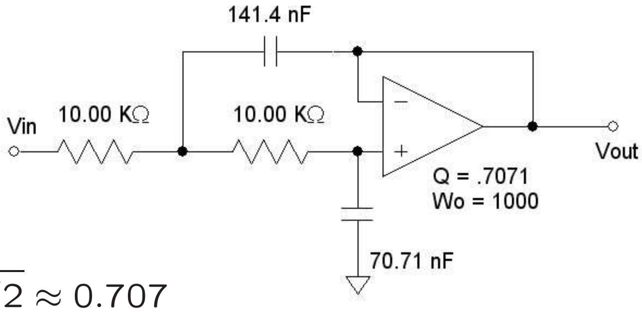
$$V_{out}(s) = G(s)V_{in}(s)$$

where

$$G(s) = \frac{\omega_n^2}{s^2 + 2\beta\omega_n s + \omega_n^2}$$

with $\omega_n = 2\pi \cdot 1000$, $\beta = \sqrt{1/2} \approx 0.707$.

Background Theory - filter (2nd order Butterworth)



$$V_{out}(s) = G(s)V_{in}(s)$$

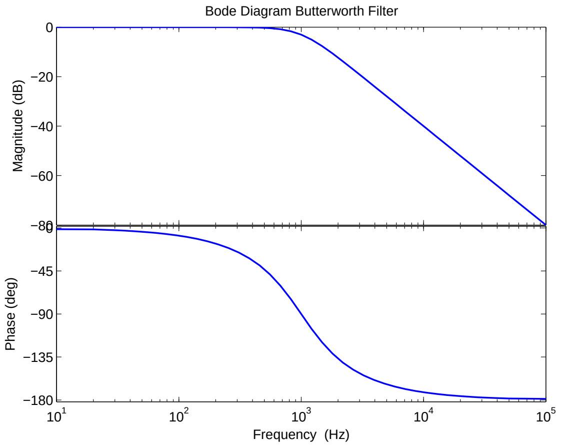
where

$$G(s) = \frac{\omega_n^2}{s^2 + 2\beta\omega_n s + \omega_n^2}$$

with $\omega_n = 2\pi \cdot 1000$, $\beta = \sqrt{1/2} \approx 0.707$ means:

- well damped filter
- -40dB/dec above 1kHz

```
>> [num,den]=butter(2,2*pi*1000,'s');
>> G=tf(num,den);
>> bode(G)
```



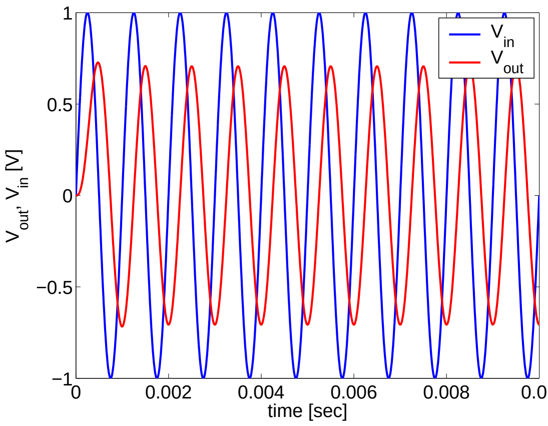
Background Theory - simulation

For some of the circuits we can write down transfer functions (see for example Slide 32 and 35)

As a result you can also do a *linear* time domain simulation of your circuit via Matlab:

Example Matlab code for our Butterworth filter in Task 3-1:

```
>> [num,den]=butter(2,2*pi*1000,'s');
>> G=tf(num,den)
>> t=linspace(0,0.01,1000);
>> Vin=sin(2*pi*1000*t);
>> Vout=lsim(G,Vin,t);
>> plot(t,Vin,t,Vout,'r')
```



Drawbacks:

- only linear simulation
- you would have to derive the 'transfer function' for each signal in your circuit

Background Theory - simulation

- SPICE (Simulation Program with Integrated Circuit Emphasis) is an open source electronic circuit simulator.
- Originated from Electronics Research Laboratory of the University of California, Berkeley by Laurence Nagel

Key to circuit simulation is a good GUI that allows you:

- Draw, edit and modify circuits
- Uses (non-linear) models of components
- Easily load new models of components
- Has good interface with SPICE to perform DC-, AC- and transient-analysis of your circuit

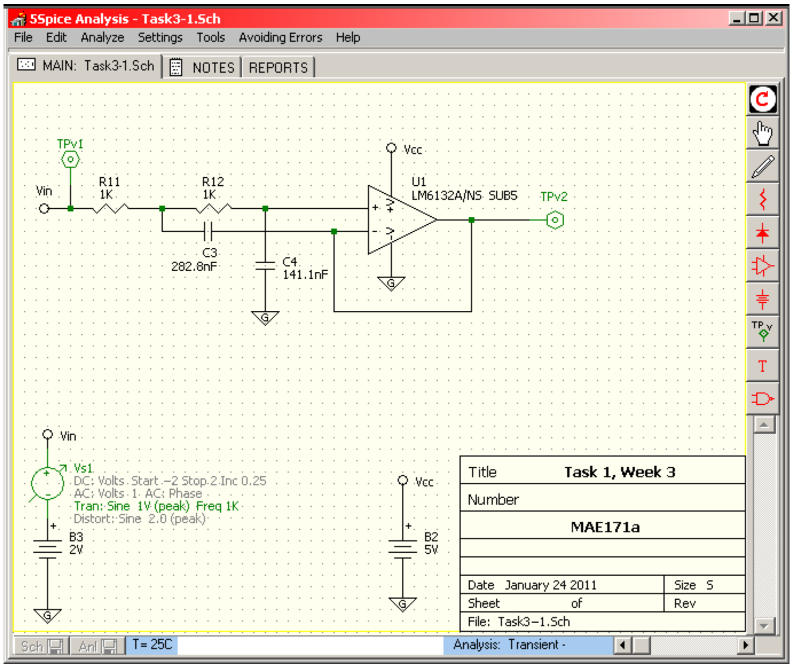
Good (limited version) for the PC (Windows) is 5Spice.

Optional, but try to use it for your circuit simulations!

We provide links to circuits already drawn for 5Spice.

Background Theory - simulation

Our Butterworth filter demo in 5Spice:



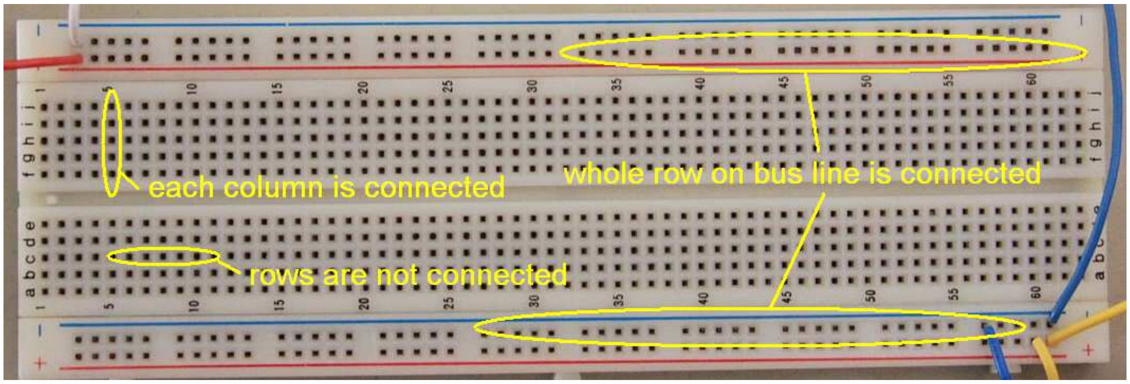
DC: Volts Start -2 Stop 2 Inc 0.25
AC: Volts 1 AC: Phase
Tran: Sine 1V (peak) Freq 1K
Distort: Sine 2.0 (peak)

Title Task 1, Week 3
Number
MAE171a
Date January 24 2011 Size S
Sheet of Rev
File: Task3−1.Sch

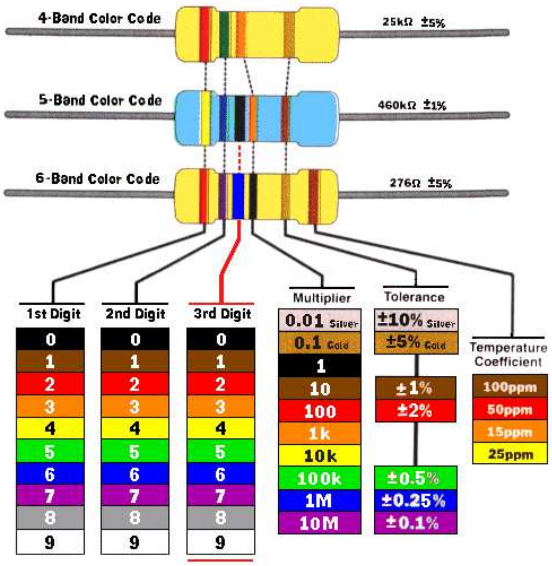
Laboratory Work - breadboard

- Build/debug/test circuits on a *breadboard* or *protoboard*: a construction base for (prototype) electronic circuit.
- Allows you to connect components via *tie points* and avoid soldering (*solderless breadboard* or *plugboard*)
- Although useful for prototyping, mechanical connection in tie points sometimes fails. . . Hard to debug when it fails.

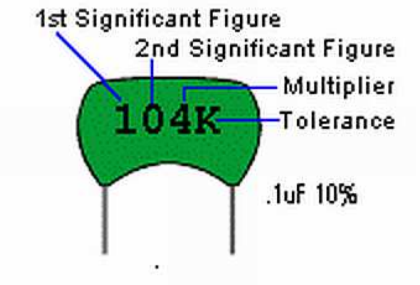
Important for our experiment: important to know how tie points are connected on breadboard:



Laboratory Work - Resistor and Capacitor coding



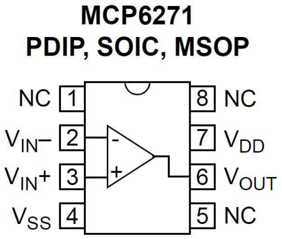
1st Digit	2nd Digit	3rd Digit	Multiplier	Tolerance	Temperature Coefficient
			0.01 Silver	±10% Silver	
			0.1 Gold	±5% Gold	
0	0	0	1		
1	1	1	10	±1%	100ppm
2	2	2	100	±2%	50ppm
3	3	3	1k		15ppm
4	4	4	10k		25ppm
5	5	5	100k	±0.5%	
6	6	6	1M	±0.25%	
7	7	7	10M	±0.1%	
8	8	8			
9	9	9			

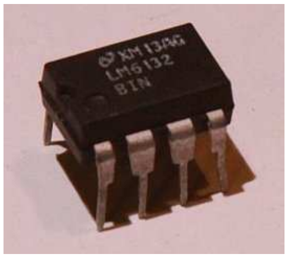


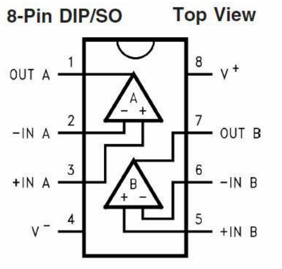
Laboratory Work - rail-to-rail OpAmp

Rail-to-rail OpAmp used during laboratory work can be either:

- MCP6271 (single OpAmp in 8 pin package)
- LM6132 (dual OpAmp in 8 pin package)







Laboratory Work - week 1

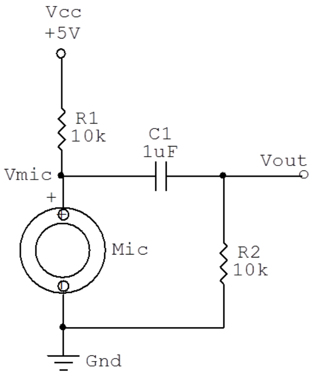
Two independent tasks that are combined in a final task 1-c:

1-a: create/test signal conditioning for electrec microphone

1-b: create/test non-inverting amplifier

Task 1-a:

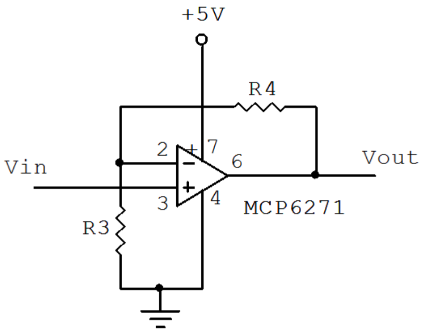
- Audio application: generate input signal via MIKE-74 electret microphone
- build a DC-bias circuit for the microphone to measure (sound) pressure variations
- Measure the DC-bias (offset) voltages
- Display and analyse the time plots generated by the microphone
- Compare experiments and task 1-a simulation with 5SPICE.



Laboratory Work - week 1

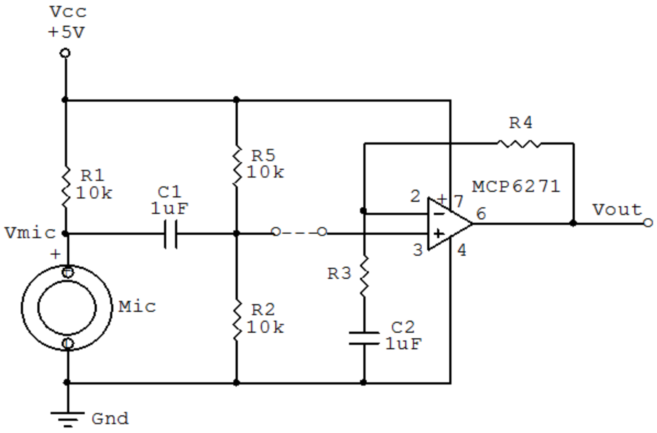
Task 1-b (independent of task 1-a):

- Build non-inverting amplifier using op-amp
- Measure bias voltages
- Experiments:
  determine gain for different resistor values and avoid clipping on output signal V_{out}.
- Measure the frequency response of your amplifier.
- Compare experiments and task 1-b simulation with 5SPICE.



Laboratory Work - week 1

Task 1-c: connect non-inverting amplifier to microphone signal conditioning circuit

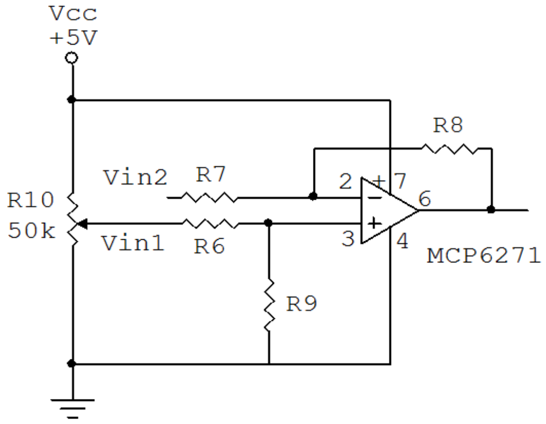


- Additional resistor R5 to off-set of microphone signal.
- Additional capacitor C2 in series with R3 to create desired **audio amplifier** (see earlier slides in these lecture notes).
- Change R3 to 1K and R4 to 100K. Result: DC-gain of 1 and (high frequency) gain of 101.

Laboratory Work - week 2

Designing a mixer (differential amplifier) and combining it with the Week 1 circuit (microphone + audio amplifier).

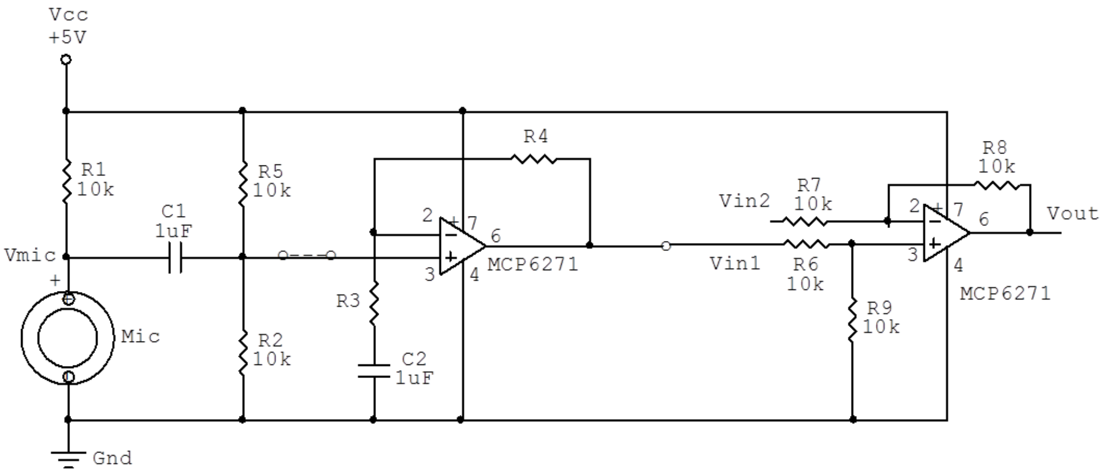
Task 2-a: difference (differential) amplifier:



- Build/test the mixer (difference amplifier)
- Verification of operation (difference of signals)
- Bias voltage adjustment via pot meter `R10`.
- Gain adjustments and experimental verification of gain and linearity of circuit
- Compare experiments and task 2-a simulation with 5SPICE.

Laboratory Work - week 2

Task 2-b: combine microphone + audio amplifier from Week 1 with difference amplifier to allow mixing of signals.



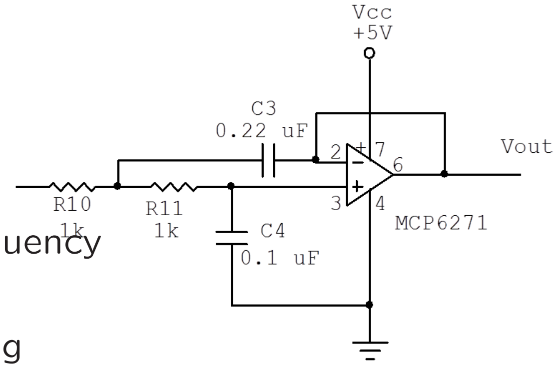
- Connect `Vin1` to `Vout` of Week 1 circuit - elimination of pot meter `R10`. Why?
- Verify operation of circuit - do sinals of microphone nicely mix with signals applied to `Vin2` without ‘clipping’.

Laboratory Work - week 3

3-a: Create 2nd order Butterworth filter

3-b: Combine/test all parts of your circuitry

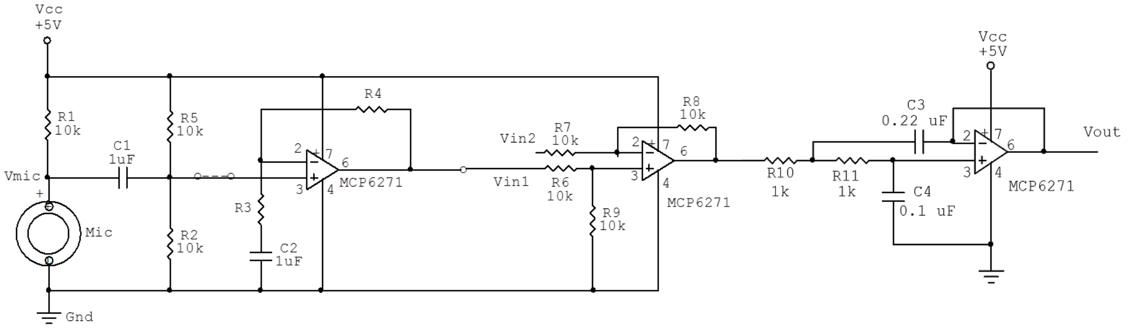
Task 3-a:



- Create active second order low pass filter with cut-off frequency around 1kHz (for speech).
- Demonstrate filter by measuring amplitude of output signal for sine input with different frequencies
- Compare frequency measurements with Bode plot in Matlab (see Slide 35 of this lecture)
- Compare experiments and task 3-a simulation with 5SPICE.

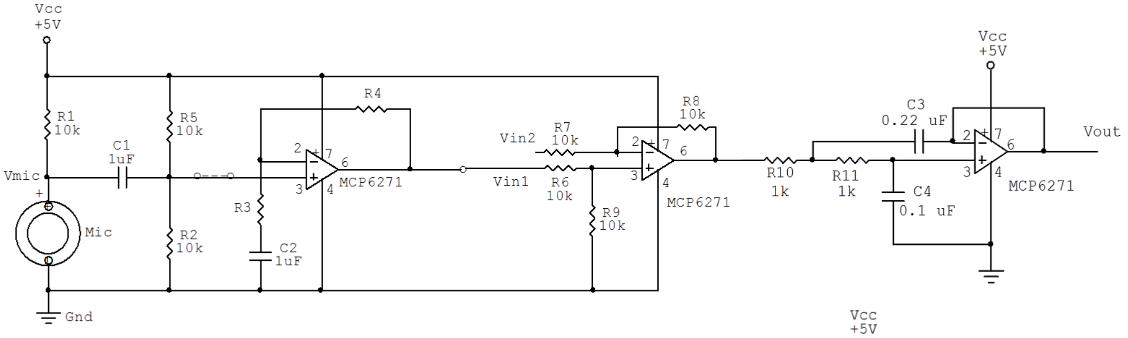
Laboratory Work - week 3

Task 3-b: Combine/test all parts of your circuitry



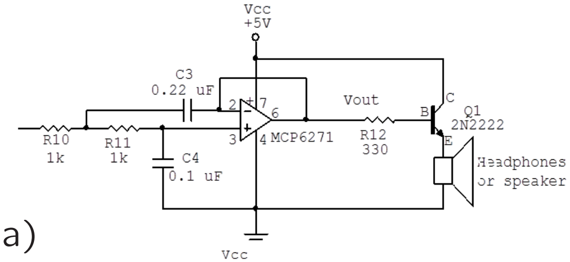
- All circuits combined.
- Should be able to filter and amplify microphone signal.
- Should be able to mix in and filter an addition signal at `Vin2`.
- Should be able to hear mixing (microphone and signal at `Vin2`) via `Vout` via set of additional capacitor and headphone.
- Measure signal at different locations in your circuit for your report to show ‘clipping’.

Laboratory Work - week 3

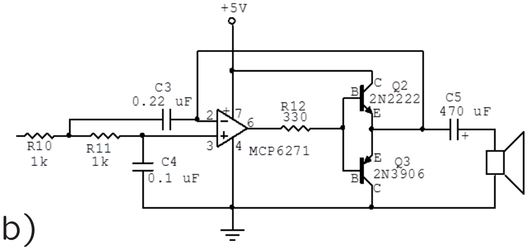


Optional:

- Add a power boost to your circuit
- Power boost can be
  (a) single NPN transistor
  (b) double ‘push-pull’ or NPN/PNP transistor pair
- Attach speaker to hear result



(a)



(b)

Summary

- (relatively simple) signal conditioning algorithms: *amplification*, *adding/difference* and basic (most 2nd order) *filtering*
- audio application on a single voltage power supply for amplification and filtering while maintaining DC (off-set)
- challenge: single source power supply of 5 Volt. Avoid clipping/distortion of amplified, mixed and filtered signal.
- insight in op-amp based linear circuits by build/debug
- compare theory (ideal op-amp) with practice (build and test)
- experimentally verify signals, gain and filtering of circuitry
- optional: compare experiments with simulation generated by 5SPICE simulation software
- for error/statistical analysis: measure signals multiple times and estimate circuit parameters (gain, cut-off frequency) from experiments

What should be in your report (1-2)

- Abstract
  Standalone - make sure it contains clear statements w.r.t motivation, purpose of experiment, main findings (numerical) and conclusions.
- Introduction
  - Motivation (why are you doing this experiment)
  - Short description of the main engineering discipline (circuit design, amplification, filtering)
  - Answer the question: what is the aim of this experiment/report?
- Theory
  - Summary of relevant Op-Amp circuits to analyze your data
  - Summary of filtering, frequency response analysis
  - Summary on how you did simulation of your circuitry

What should be in your report (2-2)

- Experimental Procedure
  - Short description of experiment
  - How are experiments done (detailed enough so someone else could repeat them)
- Results
  - Measured input/output signals to demonstrate microphone/amplification/filtering)
  - Frequency Response estimation
  - Effect of rail-to-rail conditions (saturation)
  - Parameter estimation (gain, cut-off frequency, etc.)
- Discussion
  - Why are simulation results different from experiments?
  - Do the Op-Amps behave as expected?
- Conclusions
- Error Analysis
  - Mean, standard deviation and 99% confidence intervals of estimated parameters from data
  - How do errors propagate?

GOOD LUCK