EE230 Filter Laboratory

"Build Your Own Bose WaveRadio® Bass Preamp"
Active Filter Design

Objectives
1) Design an active filter on paper to meet a particular specification
2) Verify your design using Spice and Matlab simulations
3) Build the filter in hardware and test on real-world signals

---

Situation:

You are a junior engineer working in Sony's Sound Division. You are working for a team responsible for designing a small portable boom-box capable of playing mp3's recorded on the Sony Memory Stick flash EEPROM. At the request of the marketing department, the senior project manager has tasked you to design the stereo preamplifier section to power the bass speakers in such a way that mimics the response of the famous Bose WaveRadio. She requests that you implement the circuit using LMC660 opamps powered from ±5V power supplies (this voltage is available because the power amplifier team requires this much voltage be able force a 1Ω speaker to carry 100W of power, $V^2/R=P$, and is supplied from 4 standard 1.5V cells by using widely-available solid-state DC-DC converters). She also requests you implement the circuit using a Butterworth filter for sound quality using the Sallen-Key circuit configuration to save the expense of the extra capacitor needed in the multiple-feedback configuration.

You began your work by connecting a frequency generator to the line-in jack of a WaveRadio and discovered that the bass preamplifier has a uniform 0dB gain at all low frequencies and drops to a -3dB gain at 200Hz. The filter is so good that by 2kHz it attenuates the signal by –79dB (almost dividing the original signal by 10,000!).

Part I: Laplace Analysis
Report and checkoff are due by the START of Lecture 15 (i.e it is a 2 lecture assignment). You may only work within your lab group; HR received on any problem from other groups begins at half credit.

For your lab writeup you only need to answer the following questions 1 - 9

1. What filter order is required? Note that a filter's order is the same as the number of poles it has, and is a guide to how complex the circuit will be.

2. From the given specifications, what is the expected gain (in dB) of the Bose base preamplifier at 20kHz?

3. One simplistic way to design the filter is to connect multiple first-order circuits in series. This places however many poles are required by the filter order (each filter order = one pole = 20dB/decade drop) at the cutoff frequency. Using this filter design method,

   a) determine the transfer function H(s) of a 4^{th} order filter with a 200Hz cutoff frequency (make sure it has unity gain in the passband) and

   b) evaluate its magnitude response in dB at 2Hz, 20Hz, 200Hz, and 2kHz.

You will find good agreement with the desired filter response for frequencies that are well above and below the cutoff frequency, but a poor agreement near the cutoff (i.e. the gain at the cutoff is not near 3dB, but instead 3dB x the filter order). Never build a filter like this! It is discussed in this problem only to show why we go to the pain of using a standard filter design (Bessel, Butterworth, Chebyshev, or elliptic) which all have exactly a 3dB drop at the critical frequency (e.g. a much sharper cutoff than, for instance, a fourth-order real repeated pole which would start attenuating much earlier, have a 12dB drop at the critical frequency, and have a much more rounded-looking frequency response graph)

4. Use the AFD software to design a Butterworth filter to meet the given specifications. Determine the filter's response in the frequency domain and print out the Bode plot of the filter. Label the cutoff frequency using a vertical line on the Bode plot.

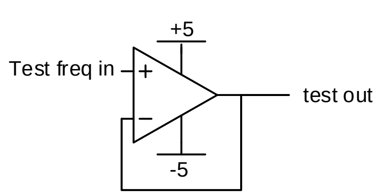
5. Determine the filter's response in the time domain to $1V_{pp}$ sinusoidal waves of 100Hz and 1kHz and print out a plot of the response vs. time (ie: simulate an oscilloscope's output to a $1V_{pp}$ input of 100Hz and 1kHz). You can do this either using AFD (read the manual or experiment with different menu options) or use Spice (Voltage source, Advanced, Sine). Whatever you use, make sure they are labeled. If you use Spice, the circuit's initial response (especially evident on the 1kHz case) will look odd because of startup transients; see if you can avoid plotting the output until the startup transients are settled. You'll know if you succeeded because a 1kHz input must yield a 1kHz output (with a much lower gain), but startup transients will look like an exponential decay.

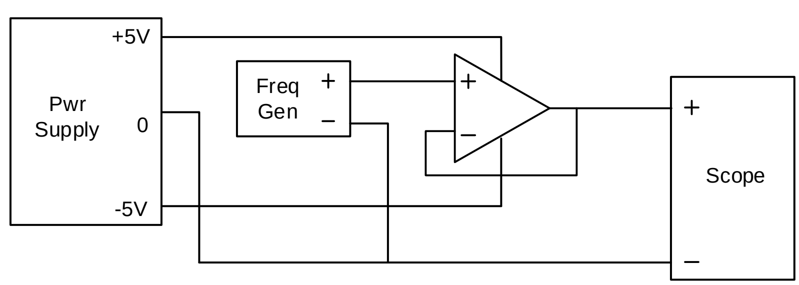
6. Design the hardware implementation of the filter. Keep capacitor values in the 100pF to 0.1uF range and resistor values in the 1kΩ to 250kΩ range.

7. Simulate the filter using Spice. Use an AC voltage source (Voltage → Advanced →AC) as the input (the maximum voltage present in a "line-in" preamp jack), and setup the analysis to include an AC sweep using a decade (i.e. logarithmic) scale from 10Hz-10kHz. Since this circuit has many nodes, you may find it helpful to label your output node (double-click the wire to do this). Attach a printout of the AC sweep. See the Spice chapter from EE120 for help using opamps and the AC Analysis option.

8. Construct the filter. Experimentally determine the Bode plot(magnitude and phase) using your frequency generator as an input. Set the frequency generator to provide a 1Vpp signal and record your circuit's output at 10Hz, 30Hz, 100Hz, 300Hz, 1kHz, 3kHz, and 10kHz. Plot the response (in dB gain on a log frequency scale) on graph paper, using the approximate rule that on a log scale, 3 falls almost exactly between 1 and 10. How does this compare with your expected gain at each frequency? Show in table form your measured, expected, and % difference. Note that random noise may interfere with small-signal readings; turn on the scope's 20MHz filter (called BW) to reduce this.

Hardware troubleshooting tip: If you have trouble getting the hardware circuit to work, try first building a simple buffer as shown below on the left (which is shorthand for the circuit on the right) and making sure a 1kHz input gives a 1kHz output. Then just build the first stage of your filter and make sure it passes100Hz but attenuates 1kHz. Then build the final stage.





9. Get your circuit checked off by the instructor – hook up the scope's trace 1 to the frequency generator and trace 2 to your filter's output. Want to see it in action while waiting? I'll have a sound source line out and boom box line in set up in the lab.

10. Write up your findings in an abbreviated lab format. You only need to include the answers to the above 9 questions. If you put in papers as appendices, label each page (e.g. "Problem 5") and put them in order. You may work with one lab partner, and only need to turn in one lab report for both partners. This lab counts for two homework grades.

How to use the Agilent Function Generator

1) Turn the Agilent function generator on. It takes a minute to boot up. Press "Channel" button over the output jack, "output load", and "set to high z". You must do this every time you start the generator, or all voltages will be off by a multiplicative factor of 2.

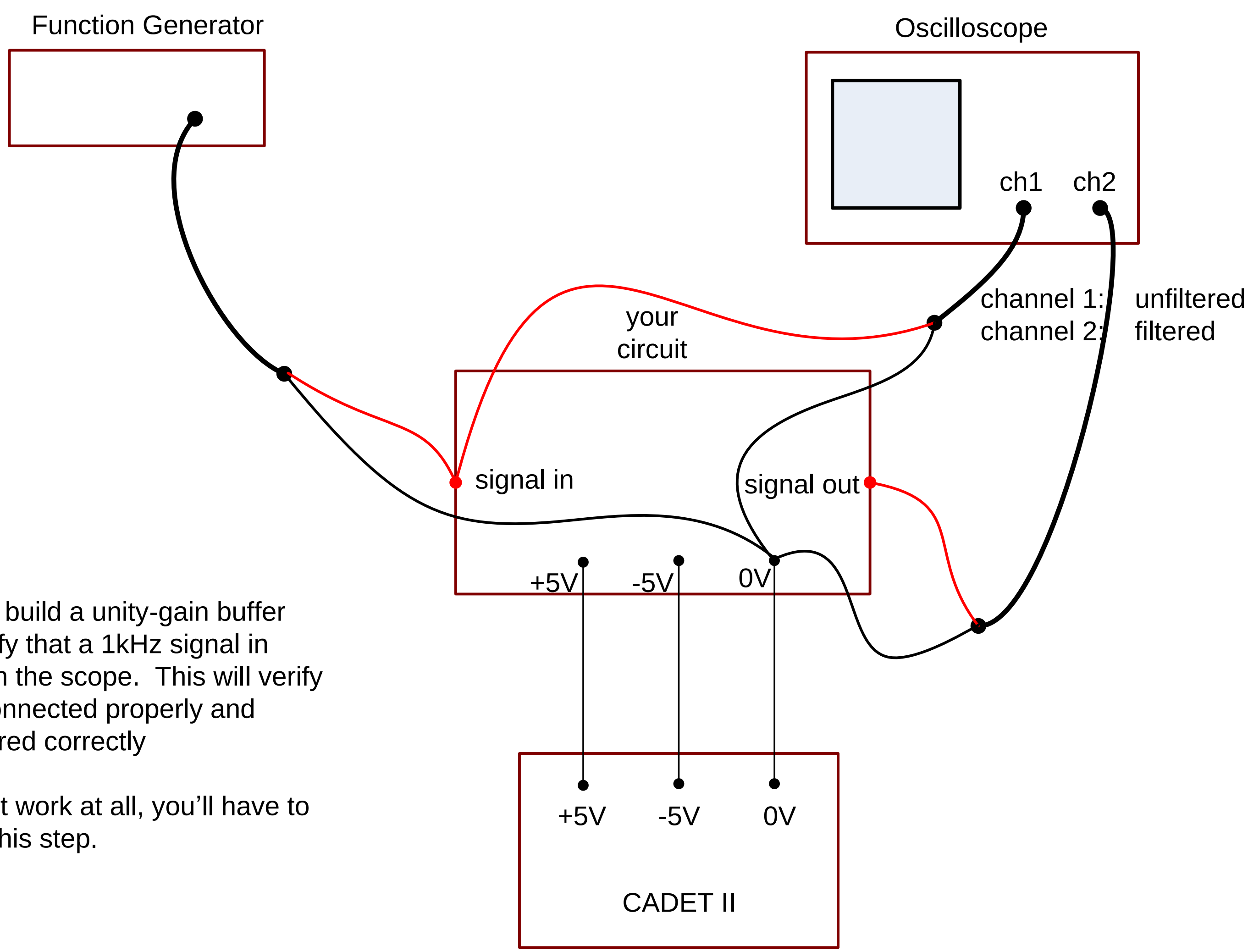
2) Press "output" to turn the output on (otherwise they are dead).

3) To choose a waveform and its parameters, click the dedicated "Waveforms" button. Press the "Sine" waveform, then press "frequency" and enter the frequency on the numeric pad.

4) Press the "Amplitude" button at the bottom, then press "3" and "Vpp" (for a ±1.5V signal).

EE230 Filter Design Lab
Signal Hookup

(This is the mandatory part – build this for me to checkoff your circuit's performance)

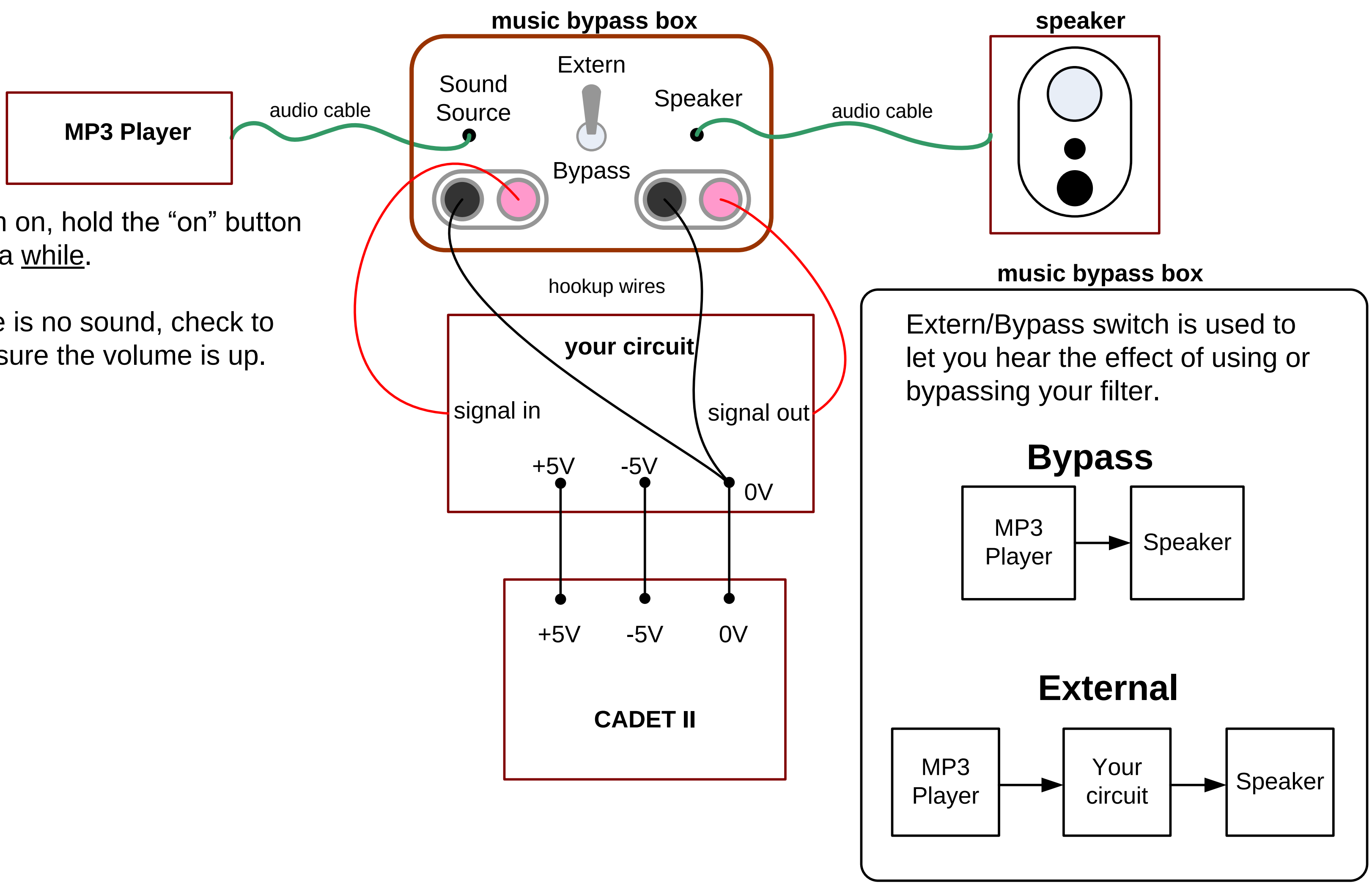


Strong **suggestion:**
Before building the circuit, build a unity-gain buffer using one opamp and verify that a 1kHz signal in yields a 1kHz signal out on the scope. This will verify
1) your signal wires are connected properly and
2) that the opamp is powered correctly

If your final circuit does not work at all, you'll have to rip it up and start back at this step.

EE230 Filter Design Lab
Optional part – hear the filter’s effects on music

(This is the optional part – build this if you want to hear what a lowpass filter sounds like. I’ll have it and the music bypass box set up on the demo bench in NEB428))



To turn on, hold the “on” button on for a <u>while</u>.

If there is no sound, check to make sure the volume is up.

August 2000

National Semiconductor

LMC660
CMOS Quad Operational Amplifier

General Description

The LMC660 CMOS Quad operational amplifier is ideal for operation from a single supply. It operates from +5V to +15V and features rail-to-rail output swing in addition to an input common-mode range that includes ground. Performance limitations that have plagued CMOS amplifiers in the past are not a problem with this design. Input V_{OS}, drift, and broadband noise as well as voltage gain into realistic loads (2 kΩ and 600Ω) are all equal to or better than widely accepted bipolar equivalents.

This chip is built with National's advanced Double-Poly Silicon-Gate CMOS process.

See the LMC662 datasheet for a dual CMOS operational amplifier with these same features.

Features

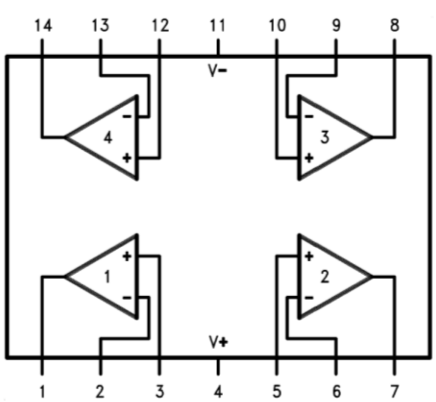
- Rail-to-rail output swing
- Specified for 2 kΩ and 600Ω loads
- High voltage gain: 126 dB
- Low input offset voltage: 3 mV
- Low offset voltage drift: 1.3 μV/°C
- Ultra low input bias current: 2 fA
- Input common-mode range includes V^-
- Operating range from +5V to +15V supply
- I_{SS} = 375 μA/amplifier; independent of V^+
- Low distortion: 0.01% at 10 kHz
- Slew rate: 1.1 V/μs
- Available in extended temperature range (−40°C to +125°C); ideal for automotive applications
- Available to Standard Military Drawing specification

Applications

- High-impedance buffer or preamplifier
- Precision current-to-voltage converter
- Long-term integrator
- Sample-and-Hold circuit
- Peak detector
- Medical instrumentation
- Industrial controls
- Automotive sensors

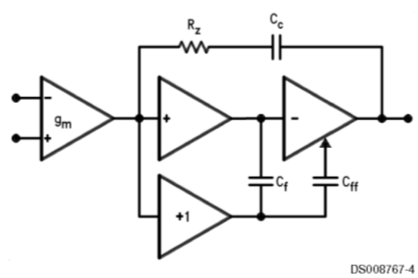
Connection Diagram

14-Pin DIP/SO



DS008767-1

LMC660 Circuit Topology (Each Amplifier)



DS008767-4

