

Experiment 4

Op-Amp Resonant Bandpass Filter

Physics 116A, D. Pellett

v. 1.2, Oct. 19, 2003

1 Introduction

In this experiment you will become familiar with a bandpass filter made with an op-amp (*active filter*). You will compare experimental results on frequency response and phase shift with analytical and SPICE analysis. An optional extra part is to measure the pulse response of the circuit as a function of time and compare with SPICE time-domain analysis.

The experimental measurements will be done in the first week. The second week's lab will provide an introduction to SPICE. The SPICE exercises can then be done in the lab or at home, since freely downloadable versions of SPICE exist for a variety of computers (see the Physics 116 web site). Note that there is an introduction to the use of SPICE and PSPICE in Ch. 16 of the text (Bobrow).

The problem solutions are to be included with your lab report.

1.1 LRC Bandpass Filter

Fig. 1 shows the frequency domain and time domain behavior of an LRC resonant bandpass filter as predicted by SPICE. Component values for this figure are $L = 47$ mH, $C = 10$ nF and $R = 200$ Ω . The figure includes the circuit diagram with node numbers and the SPICE input files used to make the plots.

AC network analysis of the circuit results in

$$\mathbf{H}(j\omega) = \frac{j\omega(\omega_R/Q)}{(j\omega)^2 + j\omega(\omega_R/Q) + \omega_R^2}, \quad (1)$$

where $\omega_R = \frac{1}{\sqrt{LC}}$ and $Q = \frac{1}{R}\sqrt{\frac{L}{C}}$. Recall that $\mathbf{H}(j\omega) \equiv \mathbf{V}_{\text{out}}/\mathbf{V}_{\text{in}}$. The response is *critically damped* when $Q = \frac{1}{2}$ and the denominator factors into $(j\omega + \omega_R)^2$.

Problem 1: Find ω_R , Q and $|\mathbf{H}(j\omega_R)|$ for those values. Compare with the Bode plot on the handout. Compare the asymptotic falloff (in dB/decade) at high and low frequencies with the expected values.

1.2 Active Bandpass Filter

Fig. 2 shows a resonant bandpass filter implemented with an op-amp and an RC feedback network instead of the LRC network.

Problem 2: Assume the op-amp is ideal and use AC network analysis to prove:

$$\mathbf{H}(j\omega) = \frac{-R_2 C_2 j\omega}{R_1 R_2 C_1 C_2 (j\omega)^2 + R_1 (C_1 + C_2) j\omega + 1}. \quad (2)$$

Eqn. 2 is of the same form as Eqn. 1 up to an overall multiplicative constant. This constant arises from the closed loop midband voltage gain of the op-amp. This becomes evident if we make the substitutions $\omega_R = 1/\sqrt{R_1 R_2 C_1 C_2}$, $1/Q = \omega_R R_1 (C_1 + C_2)$ and $K = -\omega_R^2 R_2 C_2$. Eqn. 2 is then identical to Eqn. 1 except that the factor, K , appears in the numerator in place of ω_R/Q . K is negative since the op-amp is in an inverting amplifier configuration (180° phase shift relative to the LRC circuit).

For the experiment, we will use the following component values: $R_1 = 100 \Omega$, $R_2 = 47 \text{ K}\Omega$ and $C_1 = C_2 = 0.01 \mu\text{F}$

Problem 3: Find ω_R , Q and K for the R and C values given above. Compare with the results of Prob. 1 (including midband gain, $|\mathbf{H}(j\omega_R)|$).

2 Experiment

Wire up the active bandpass filter of Fig. 2 using the component values given in the last section. Be sure to connect the $\pm 15 \text{ V}$ power connections as well. Refer to the op-amp experiment (Lab 2) for the pinout.

Measure the voltage gain, $A_V = |V_{out}/V_{in}|$, and output phase as a function of frequency as you did for the RC filter circuits in Lab 3. Then make a Bode plot, plotting voltage gain in dB ($20 \log_{10} A_V$) and phase vs. log frequency. Collect enough data at large and small frequencies so you can see the fall off on the Bode plot and compare with the expected linear asymptotes. Collect enough data near the peak, A_{Vmax} , so you can also make a linear plot of gain vs. frequency in the region of the maximum. Use this to estimate Q from the peak width, $\Delta\omega$, at the half-power point (where $A_V = 0.707 A_{Vmax}$; $Q \approx \omega_R/\Delta\omega$). Compare with your circuit analysis results in the previous section.

Observe the following precautions to get good results.

1. Be sure to use capacitors with no worse than $\pm 20\%$ tolerance (marked “.01M,” for example; the “M” indicates the tolerance in this case). Use $\pm 5\%$ capacitors if available (marked “103J,” read as 10×10^3 , with units assumed as pf, and “J” tolerance, or 5%).
2. The source impedance of the signal generator must be low ($\ll R$). Otherwise, it contributes significantly to the resistance of the series LRC circuit.
3. When you are near the resonance peak, the input signal, V_{in} , must be small ($\approx 10 \text{ mV}$ amplitude) to avoid overloading the op-amp. A larger signal is needed when you are far away from the peak.
4. The signal generator output may depend on frequency, so V_{in} needs to be measured at each frequency.

Many of the above requirements will be met if you drive the circuit with a voltage divider as shown in Fig. 3, so do that. Try $R_A = 100 \Omega$ and $R_B = 1 \Omega$ ($R_A = 1 \text{ K}\Omega$ and $R_B = 10 \Omega$ would be maximum values).

2.1 Pulse Response (Optional)

If time permits, look at the pulse response (time domain) of your circuit (use a narrow pulse with a long repetition rate). Compare with the time-domain SPICE results for the LRC circuit in Fig. 1.

3 SPICE Analysis (may be done outside of lab period)

1. Use SPICE to simulate the circuit in Fig. 2 and produce the Bode plot showing the frequency domain response. You can start from the SPICE input file for the Bode plot in Fig. 1. Note that the phase was plotted in degrees. On the department computer, the angle will be in radians. Model the op-amp as a simple voltage-controlled voltage source as follows, using the node numbers of Fig. 2:

```
E 4 0 0 3 1G
```

(see Example 16.6 on p. 1047 of the text). Compare with your experimental result and calculations.

2. Modify the SPICE analysis to use a more realistic op-amp simulation, such as the one in Example 16.37 on p. 1115 of the text or the one here (including an example of its use):

```
* subcircuit definition of a real op-amp, type 741
* 1 = non-inverting input: +
* 2 = inverting input: -
* 3 = output
* open loop gain = 100000
* ft = 1 Mhz (freq. at which open loop gain = 1)
* call: Xyyyyy n1 n2 n3 op741
.subckt op741 1 2 3
E1 4 1 1 2 1
R 4 5 1K
C 5 1 15.9U IC=0V
E2 3 1 5 1 100K
.ends

* usage:
VIN 1 0 AC 1
X 1 2 3 op741
```

The point here is that the open-loop response of the 741 op-amp is not flat. Gain falls with frequency, reaching unit gain at $f_T = 1$ MHz or so. Does this op-amp model give better agreement with your experimental results?

3. Perform the time-domain SPICE analysis for your circuit (use the time-domain analysis of Fig. 1 for an example of a pulse input and transient analysis).

Problem 4 (extra credit): Use SPICE to simulate the critical damping case for the LRC circuit ($Q = \frac{1}{2}$) and compare the time response with Fig. 3.38 in the text. Explain why the behavior of V_{out} looks like $i(t)$. Also make the Bode plot for this case.

3.1 Minimal Notes on SPICE Usage on Physics Linux Computers

You are free and in fact encouraged to try SPICE on your own computer. A certain number of computers running a Windows version of PSpice will also be available in the Physics 116 lab room.

You may also use one of the Linux computers in Room 106 Physics or lifshitz.ucdavis.edu remotely.

Login accounts are available for students in the Physics 116 class. Keys to Room 106 may be checked out from the Physics office. The default shell is `bash`, although `tcsh` is available. Various editors are available including `vi`, `emacs` and `pico`. The instructions below refer to the Linux version of SPICE.

Try entering the following SPICE commands into a file called `LRC.cir`:

```
Resonant Bandpass (LRC) filter
*****
VIN 1 0 AC 1
L 1 2 47M
C 2 3 10N
R 3 0 200
.AC DEC 500 100 99k
.control
run
plot db(V(3)/V(1)) (180*ph(V(3))/3.14) xlog ydel 20 xdel 2000 ylimit -180 180
.endcontrol
.END
```

Then you should get a Bode plot for the LRC circuit when you issue the following command from the shell:

```
spice3 LRC.cir
```

Unfortunately, the “print” command button in the plot window does not produce useful printer output. You must use the command (inside SPICE),

```
shell import LRC.ps
```

and click on the plot window title bar to produce a postscript file of the window for later printing. To get a postscript file using the `import` program, you must use the `.ps` extension. The name of the file otherwise is up to you, of course. You can print this after quitting SPICE using the Linux command,

```
lpr LRC.ps
```

and fetch the result from the printer.

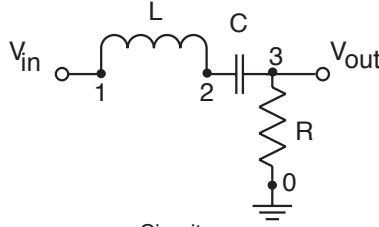
Another quirk of the system is that the editor *within SPICE* seems to require that you give the name of your file when entering edit:

```
edit LRC.cir
```

If you just issue `edit`, all appears well until you exit the editor, at which point the program crashes.

Additional hints: the first line of the SPICE program is considered to be a comment, even if it doesn't begin with "*" (it is meant to be a title of the file). Also note the use of the control statements to run the simulation and plot the results. There are differences from PSPICE. In particular, there is no .PROBE command although there are interactive analysis commands described in the interactive "help" facility.

Response of LRC Bandpass Filter in Frequency and Time Domains



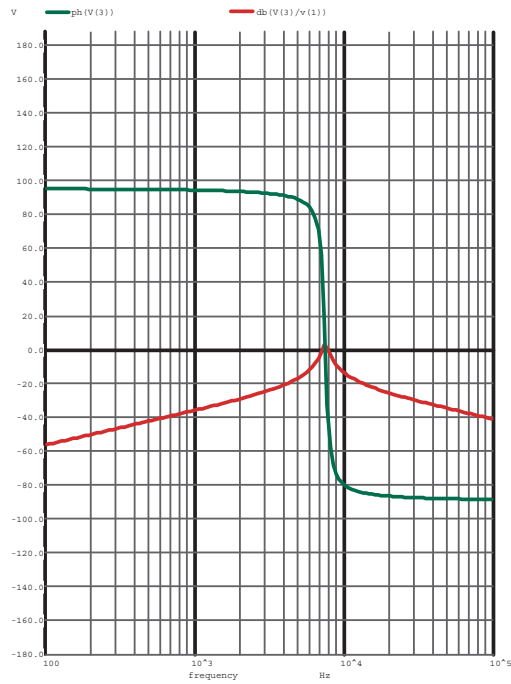
D. Pellett
Physics 116A
1/27/98

Circuit

```
Resonant Bandpass (LRC) filter
*****
VIN 1 0 AC 1
L 1 2 47M
C 2 3 10N
R 3 0 200
.AC DEC 50 100 99k
.control
run
plot db(V(3)/v(1)) ph(V(3)) xlog ydel 20 xdel 2000 ylimit -180 180
.endcontrol
.END
```

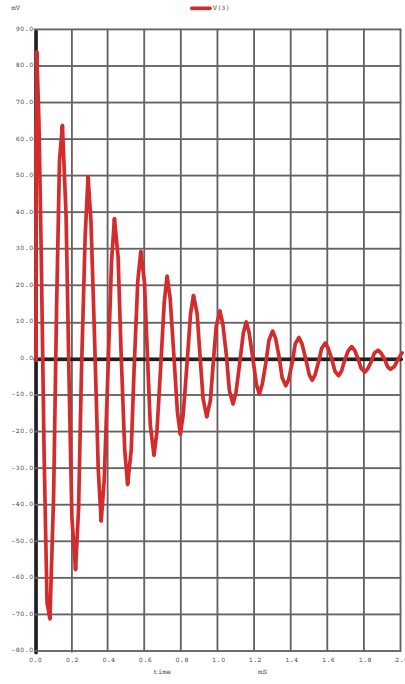
```
Pulsed Resonant Bandpass (LRC) filter
*****
VIN 1 0 PULSE ( 0 0V 1us 10V 2us 10V 3us 0V)
L 1 2 47M
C 2 3 10N
R 3 0 200
.TRAN 1us 2ms
.control
run
plot V(3)
.endcontrol
.END
```

Frequency Domain Spice Input



Frequency Domain Response
(Bode Plot)

Time Domain Spice Input



Time Domain Response
(Short Pulse Input)

Figure 1. Resonant bandpass filter frequency and time domain response. The frequency domain plot shows both the phase shift in degrees and the voltage gain in dB on the same graph.

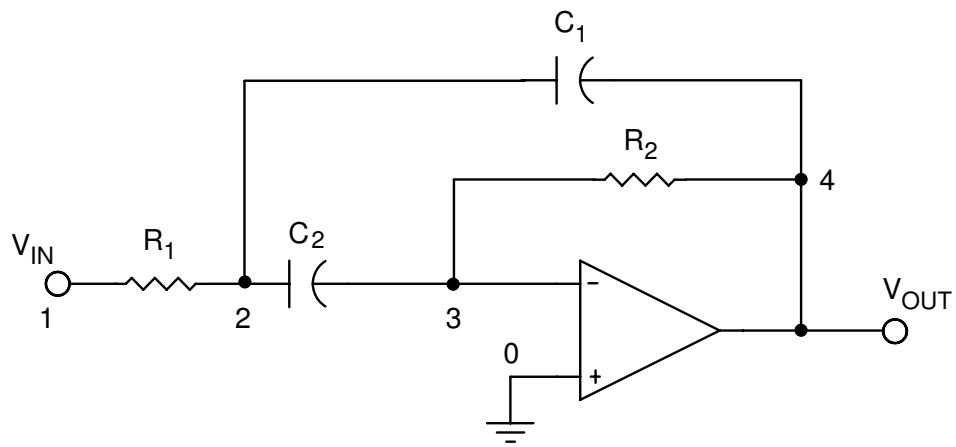


Figure 2: Bandpass filter made with an op-amp and RC networks (*active filter*).

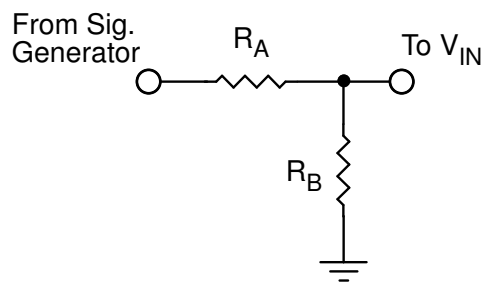


Figure 3: Voltage divider used to couple the signal generator to the circuit input.