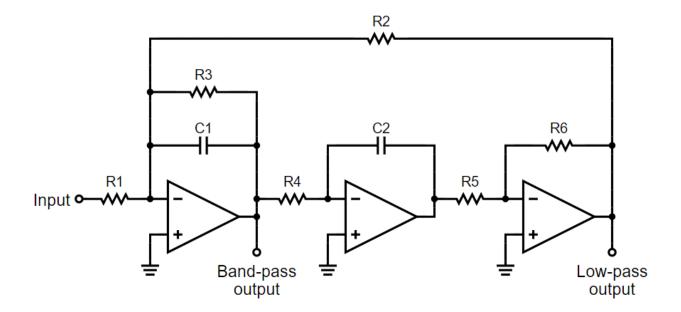# An Active Bandpass Filter

In circuit analysis classes, you can use ideal components of any value. In practice you run can run into problems because real inductors have resistance and capacitance, and real capacitors have resistance and inductance. In addition, to keep the circuit reasonably small and inexpensive, it may be necessary to use components in a narrow range of values.

If you are working at low frequencies, and low power levels, you can overcome some of these problems by using active filters. These are filters that have op-amps as an integral part of the design. For this project, you need to design a second order bandpass filter, using a particular active filter topology called the Tow-Thomas Biquad circuit. You can easily find information on this circuit by searching on-line. A schematic for the filter is shown below

The circuit can be used either as a low-pass filter or a band-pass filter (or both, if you prefer). In this project, you need to design a bandpass filter.

The center frequency of the filter is known as the natural frequency, f_0 Hz. When writing equations, it is easier to use the natural frequency in radians per second, where

$$\omega_0 = 2\pi f_0$$

The natural frequency for this particular circuit is given by the equation

$$\omega_0 = \frac{1}{\sqrt{R_2 R_4 C_1 C_2}}$$

The bandwidth of the filter, B, is usually described indirectly through something called Q, the Quality factor of the filter. B and Q are related by the equation

$$B = \frac{\omega_0}{Q}$$

You can set the quality factor of the filter by using the equation

$$Q = \sqrt{\frac{R_3^2 C_1}{R_2 R_4 C_2}}$$

The overall gain of the filter is set by the G_{bpf} parameter, which is defined as

$$G_{bpf} = \frac{-R_3}{R_1}$$

The transfer function of the filter is then given by

$$H(s) = G_{bpf} \frac{Bs}{s^2 + Bs + \omega_0^2}$$
$$R_5 = R_6$$

When designing the filter, you need to keep the following practical considerations in mind.

- Input impedance of the op-amps is about 1 MOhm, and output impedances may be a few ohms. In addtion, your wires and connections may have a few ohms of resistance also. To keep the op-amps looking close to ideal, it is important that the resistors you use are much less than the input impedance, and much greater than the output impedance or the resistance of your connections. Try to use resistors in the range of a few hundred ohms to a few tens of kiloohms if possible.
- There will be parasitic (unwanted) capacitances on your breadboard, between the conductors, and at the op-amp inputs and outputs. Try to use capacitors as large as possible, so the parasitic capacitances make an insignificant change. However for this design, it is important that you use non-polarized capacitors. That rules out the electrolytic capacitors that you normally use in the lab. Before starting your design, find the largest non-polarized capacitors available in the lab, and limit your design to those.

Procedure

You need to design a bandpass filter, determine it's theoretical performance (frequency response), built it in the lab, and test it to see how close it comes to ideal. The filter should pass frequencies from 2 kHz to 12 kHz.

You will probably want to start your design by setting the gain of your filter to 1 (or -1). That normally makes the filter easier to design than one with high gain.

Then pick the resonate frequency to near the middle of your pass band, on a log scale. In other words try to put the same number of decades between 2 kHz and f_0 as there is between f_0 and 12 kHz.

Next pick a Q value that makes the response reasonably flat (within a few dB) of 0 dB over the pass band. It would be a good idea to generate a theoretical plot of the frequency response based on H(s). You will need this when you are testing the filter.

Build the filter in the laboratory, and thoroughly test it. Test it over as wide a range of frequencies as you can easily generate in the laboratory. Also test it over a wide range of input signal levels (but not so high as to damage the op-amps).

In your tests, look at the gain, phase shift between input and output, and the harmonic content of the output. Compare all these to what the theory predicts.

Write up a high quality technical report describing your design, construction and test results for the circuit. There should be a title page, table of contents, introduction, main body, and conclusion section to the report. All figures, schematics, graphs, tables, charts, etc. should be computer generated (not hand drawn). Do not simply list the component values you picked, but describe the reason behind selecting those particular values.