2. BAND-PASS NOISE MEASUREMENTS

2.1 Object

The objectives of this experiment are to use the Dynamic Signal Analyzer or DSA to measure the spectral density of a noise signal, to design a second-order band-pass filter, to use the swept sine wave feature to measure its frequency response, and to measure its noise bandwidth with both the rms voltmeter and the DSA.

2.2 Laboratory Procedure

2.2.1 Filter Design

Figure 2-1 shows the circuit diagram of a second-order infinite-gain multi-feedback band-pass filter preceded by a unity-gain buffer having an attenuator and low-pass filter at its input. The transfer function for the input buffer circuit is given by

$$\frac{V_i'}{V_i} = \frac{R_B}{R_A + R_B} \frac{1}{1 + (R_A \| R_B) C_A s} \tag{2.1}$$

This is a first-order low-pass filter function with a gain constant less than unity. The purpose of the voltage divider is to obtain a gain of unity from the overall circuit at the resonance frequency of the band-pass filter. The purpose of C_A is to suppress possible rf interference at the input.

The transfer function of the band-pass filter is given by

$$\frac{V_o}{V_i'} = -K \frac{(1/Q)(s/\omega_0)}{(s/\omega_0)^2 + (1/Q)(s/\omega_0) + 1} \tag{2.2}$$

where ω_0 is the resonance frequency, K is the gain at resonance, and Q is the quality factor. These are given by

$$K = \frac{R_3 C_1}{R_1 (C_1 + C_2)} \tag{2.3}$$

$$\omega_0 = \frac{1}{\sqrt{(R_1 \| R_2) R_3 C_1 C_2}} \tag{2.4}$$

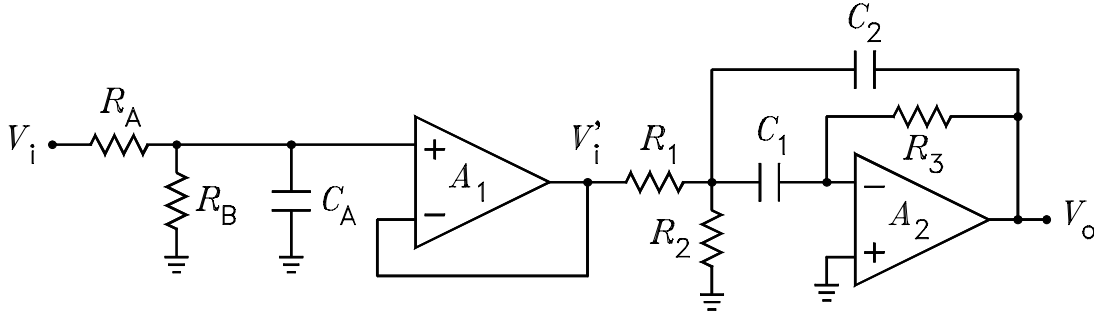$$Q = \frac{\sqrt{R_3 C_1 C_2 / (R_1 \| R_2)}}{C_1 + C_2} \tag{2.5}$$

Figure 2-1   Band-pass filter with input buffer.

If we let $C_1 = C_2 = C$, the values of R_1, R_2, and R_3 for a specified K, ω_0, and Q are given by

$$R_1 = \frac{Q}{K\omega_0 C} \qquad R_2 = \frac{R_1}{(2Q^2/K) - 1} \qquad R_3 = 2KR_1 \qquad (2.6)$$

The -3 dB bandwidth $\Delta\omega_3$ of the band-pass filter is related to the resonance frequency ω_0 and the quality factor Q by

$$\Delta\omega_3 = \frac{\omega_0}{Q} \qquad (2.7)$$

Let the lower and upper -3 dB cutoff frequencies, respectively, be denoted by ω_a and ω_b. These frequencies are related to the bandwidth and the resonance frequency by

$$\Delta\omega_3 = \omega_b - \omega_a \qquad \omega_0 = \sqrt{\omega_a \omega_b} \qquad (2.8)$$

$$\omega_a = \frac{\omega_0}{2}\left[-\frac{1}{Q} + \sqrt{\frac{1}{Q^2} + 4} \right] \qquad \omega_b = \frac{\omega_0}{2}\left[\frac{1}{Q} + \sqrt{\frac{1}{Q^2} + 4} \right] \qquad (2.9)$$

Specifications

The band-pass filter is to be designed for the following specifications:

$$Q = 10 \qquad f_0 = \frac{\omega_0}{2\pi} = 3 \text{ kHz} \qquad 1 \le K \le 10 \qquad (2.10)$$

Note that f_0 is the geometric mean of 100 Hz and 1 kHz, so that it lies half way between these frequencies on a log scale. As a possible design procedure

- Assume a value for C and K. A recommended range for C is 100 pF to 0.1 μF.

- Calculate R_1, R_2, and R_3. To minimize any problems with the filter realization, it is recommended that these lie in the range from 1 kΩ to 100 kΩ. However, values in the range from 100 Ω to 1 MΩ should give acceptable results.

By judicious choice of C and K, it should be possible to obtain acceptable values for the resistors. The filter elements should be calculated so that the required numerical values can be obtained within an error of $\pm 5\%$. Because precise capacitor values are not available in the laboratory stock, use the RCL meter to measure the value of all capacitors used in the filter. Note that two reasonably matched capacitors are required. After the gain constant K has been chosen, choose R_A and R_B such that the gain of the input voltage divider is $1/K$. This gives an overall gain of unity at the resonance frequency of 316 Hz. An error of $\pm 5\%$ is acceptable. Choose C_A so that the -3 dB cutoff frequency of the rf filter is in the range from 1 MHz to 2 MHz.

2.2.2 Filter Assembly

Assemble the filter circuit on the solderless breadboard. Use a RC decoupling network on each power supply rail consisting of a 100 Ω resistor and a 100 μF capacitor.

2.2.3 Frequency Response

Oscilloscope

With the oscilloscope connected to the circuit output, drive its input from the function generator and determine if it exhibits the correct frequency response. Determine the frequency f_0 at which the gain is a maximum, the gain at this frequency, and the two frequencies f_a and f_b at which the gain is down by a factor of $1/\sqrt{2}$. Calculate the quality factor from

$$Q = \frac{f_0}{f_b - f_a} \tag{2.11}$$

Use the oscilloscope to observe the sine wave and square wave responses at several frequencies, e.g. at f_0, at f_0/k, and at kf_0, where k is suitably chosen to obtain the most interesting waveforms for the square wave input case. Note any ringing on the waveform.

DSA Swept Sine Wave Measurement

Measure the dc voltage at the input and output nodes of the circuit with the DMM. It should be in the millivolts range. If it isn't something is seriously wrong with the circuit. Correct the problem and proceed.

- Set the inputs for Channels 1 & 2 on the **Agilent 3570A** DSA for ac coupling. To do this, press "Input," select the menus for Channels 1 & 2 front end setup, and select "ac coupling."

- Connect Channel 1 to the circuit input (V_i) and Channel 2 to the output (V_o).

- Connect the "Source" out to the input node (V_i).

- Press the "Source" button and set the level to 0.1 V.

- Press "Inst Mode" and select swept sine wave.

- Press "Freq" and set the start frequency to 100 Hz, the stop frequency to 30 kHz, and the "Steps" to "log."

- Press "Trace Coord" and switch the frequency axis to "log."

- Press "Disp Format" and select "Bode." Set the marker to 3 kHz (or the measured value at which the gain is a maximum). Print the display.

- Press "Disp Format" and select "Single." Print the display.

Use the frequency response data that you obtain to determine f_0 and Q. Compare the results with those obtained using the function generator and the oscilloscope.

Disconnect the circuit from the DSA.

2.2.4   Noise Analysis

Noise Source Setup

The filter circuit is not connected to the DSA for this part.

- Press "Inst Mode" on the DSA and select FFT.

- Set the DSA for One Channel measurements.

- Set the "Source" output to produce a random noise voltage having a rms value of approximately 0.1 V. Turn the source on.

- Measure the source output voltage with the **HP 34401A** DMM, and the DSA programmed to measure the voltage over the extended audio range ($0 - 100$ kHz). For this step, connect the "Source" output to the Channel 1 input of the DSA.

Noise Spectral Density Measurement

The filter circuit is not connected to the DSA for this part.

- Press "Avg" on the DSA, set the number of averages to 50, and turn "averaging" on.

- Set the marker or cursor on the DSA to 3 kHz (or whatever the actual center frequency of the filter is) and record the voltage level and the band over which it is measured (known as the measurement bandwidth). To determine it press "Disp Format" and select "Measurement State."

- Obtain a plot of the noise spectrum.

- Use the data to calculate the spot noise voltage at 3 kHz in $V/\sqrt{Hz}$ and the spectral density in V^2/Hz.

Spectra of Filtered Noise

The filter circuit is connected to the DSA for this part.

- Apply the random noise signal at the "Source" output of the DSA to the input of the filter.

- Connect the filter output to the Channel 1 input of the DSA.

- Program the signal analyzer to measure the spectrum of the filter output signal. Because the spectral density of the input noise signal is flat (white) over the filter bandwidth, the DSA display should resemble the shape of the frequency response of the filter.

- Program the analyzer to measure the total rms noise voltage over the frequency interval from 0 to 100 kHz. Obtain a plot of the spectrum. Also measure the noise output with the the **HP 34401A** DMM and compare the measurements with the signal analyzer value.

- Use the results of the measurements to calculate the noise bandwidth B_n of the bandpass filter. Compare this with the theoretical value given by $B_n = (\pi/2) B_3$, where B_3 is the -3 dB bandwidth.

2.3   Laboratory Report

1. Turn in all plots properly labeled, i.e. title, axes, scales, and data identified.

2. Perform all calculations asked for in the procedure section.

3. Derive the transfer function of the active filter used in the experiment. Derive the equations for its two -3 dB frequencies and use these equations to verify that $f_0 = \sqrt{f_a f_b}$ and $Q = f_0/(f_b - f_a)$. From the transfer function, derive the noise equivalent bandwidth of the filter.

4. Compared the measured and/or calculated values obtained with the different procedures.

5. Perform SPICE simulations to obtain the transfer function (magnitude and phase versus frequency) of the filter. First, assume that the op-amp can be modeled by a voltage-controlled voltage source having a gain of 2×10^5. Second, perform the simulation using an appropriate ac macromodel for the particular op-amps used. Compare these to each other and to the experimental Bode plot.

6. Discuss anything interesting, unusual, or new that occurred in the experiment.

SPICE Op-Amp Macromodel Circuits

Quasi-Ideal Macromodel

Macromodel circuits are used to represent the op-amp in SPICE simulations. The simplest macromodel is a voltage-controlled voltage source having a voltage gain equal to the dc gain of the op-amp. Such a circuit is illustrated in Fig. **??**(b). In SPICE, a voltage-controlled voltage source is modeled by an "E amplifier." The SPICE subcircuit code for such a macromodel with a gain $A = 2 \times 10^5$ is given below. This is referred to as the *quasi-ideal macromodel subcircuit* in the following. The non-inverting input is node 1, the inverting input is node 2, and the output is node 3. This subcircuit model is useful in SPICE simulations of most op-amp circuits where the op-amp is considered to be ideal in the design of the circuit.

```
*QUASI-IDEAL OP-AMP SUBCIRCUIT
.SUBCKT OPAMP 1 2 3
EOUT 3 0 1 2 2E5
.ENDS OPAMP
```

The node numbers in the op-amp subcircuit code are "dummy" node numbers. The actual node numbers in a circuit may are used in the subcircuit call. For example, the code line

```
X1 0 4 9 OPAMP
```

refers to op-amp X1 with non-inverting input node 0, inverting input node 4, and output node 9.

Small-Signal AC Macromodel

A more elaborate macromodel is required if the ac frequency response behavior of the op-amp is to be modeled. Such a model is given in Fig. 2-2. This circuit models the small-signal input resistance, output impedance, and voltage-gain transfer function of the op-amp. Only a single pole is modeled in the voltage-gain transfer function. The SPICE subcircuit codes for the 741 and the LF351 op-amps are given below. These are referred to as the *small-signal AC macromodel subcircuits*. The non-inverting input is node 1, the inverting input is node 2, and the output is node 3. The ac macromodel is useful in SPICE simulations of most op-amp circuits where the small-signal frequency response of the op-amp is to be modeled.

```
*741 OP-AMP SUBCIRCUIT   *LF351 OP-AMP SUBCIRCUIT
.SUBCKT OA741 1 2 3      .SUBCKT OA351 1 2 3
RIN 1 2 2E6              RIN 1 2 2E12
GM1 4 0 1 2 1.38E-4      GM1 4 0 1 2 2.83E-4
R1 4 0 1E5              R1 4 0 1E5
CC 4 5 20E-12           CC 4 5 15E-12
GM2 5 0 4 0 106         GM2 5 0 4 0 283
RO1 3 5 150             RO1 3 5 50
RO2 5 0 150            RO2 5 0 25
.ENDS OA741             .ENDS OA351
```
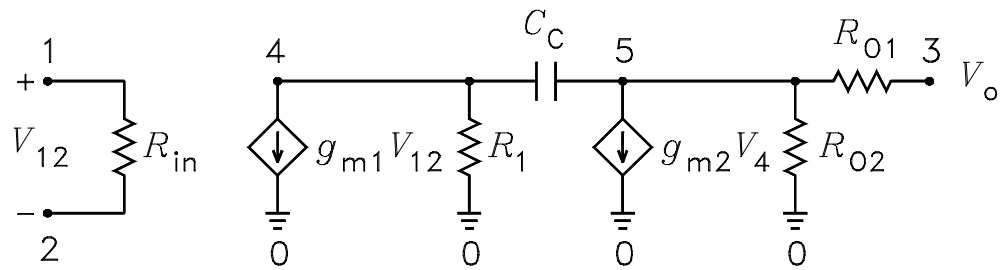
Figure 2-2   Op amp ac macromodel.

The resistor R_O shown connected in series with the output of the op-amps in this experiment is included for practical reasons, viz. to reduce the possibility that the circuit will oscillate when a capacitive load such as an oscilloscope probe is connected to it. It should not be included in the theoretical derivations, calculations, or simulations. For these analyses the output of the circuit should be taken directly at the output node of the of the appropriate op-amp.