Purpose

The purpose of the Part I of this experiment is to consider the sinusoidal response of several first order filters. The effect of loading (applying a load resistor to the output of the circuit) on the frequency response will be investigated. Part II of the experiment will consider the use of Fourier Series in circuit analysis.

Part I Sinusoidal Steady State Frequency Response

This part of the experiment will use the circuits shown in Figure 1. The RC circuit is referred to as a passive circuit and the circuit with the op-amp is known as an active circuit.

1. Measure the low frequency (about 100Hz) gain and phase shift at the output for a sinusoidal input. This should be done on both circuits as well as for each circuit with a 1K load resistor across its terminals (output to ground). (Note that this is a total of 4 measurements.)

2. Slowly increase the frequency until the output is reduced by 3db (half-power point, .707 point, and corner frequency are other names given to this point). Measure the gain and phase at this frequency for each of the four circuits.

3. Use PSPICE to plot the frequency response (gain and phase) of each of the four circuits.

4. The theoretical (expected value), experimental and PSPICE data generated in sections 1 through 3 should be displayed in a table that clearly shows low frequency gain and phase and 3db frequency data for each of the four circuits.

Part II Square Wave Input/Fourier Series Analysis

This part of the experiment will consider the output response for a square wave input to the passive RC circuit without a load resistor. The input signal will be the square wave shown in Figure 2.
1. **Calculations**: Find the Fourier series of the input ($V_{\text{in}}$). Find the Fourier series of the output by putting the Fourier series of $V_{\text{in}}$ through the RC circuit one term at a time. Superposition may be used to find the Fourier series of ($V_{\text{out}}$).

2. **Experimental**: The function generator must be adjusted to give the desired form shown in Figure 2. The oscilloscope should be used for this adjustment. The output and input waveforms should be observed. The GPIB program, PLOT, should be used to obtain a hard copy of the waveforms. An informative representation of this waveform is obtained by showing the output waveform and the input waveform superimposed on top of each other. Be sure that both the input and the output traces have the same amount of DC offset on the oscilloscope.

3. **PSPICE**: For this PSPICE simulation the VPULSE function is used for the input. The following parameter should be used for VPULSE: delay time = 0, period = 0.2ms, rise_time = 1us, fall_time = 1us, Pulse_width = 0.1ms, pulsed voltage = 10 and initial voltage = 0. These numbers correspond to the square wave shown in Figure 2. The rise time and fall time in PSPICE cannot be zero. Therefore, a time so small (1µs) that it has no effect on the solution is chosen. A transient analysis should be used with a final time of .6ms. Use PROBE to plot the input and output waveforms.

The Fourier (FOUR) analysis is used in conjunction with the transient analysis. The Fourier series for the square wave input and the output waveform should be obtained. The Fourier analysis is enabled in the TRAN setup window. The center frequency is 5Khz (i.e. 1/T). Eight harmonics should be sufficient. (If the eighth harmonic does not appear to be small enough, more harmonics may be used.) The input and output nodes are entered in the space for output variables. The Fourier series coefficients are found in the output file generated by PSPICE. DO NOT USE ANY OTHER TERMS THAT MAY LOOK LIKE FOURIER COEFFICIENTS.

4. **Comparison**: The Fourier series coefficients that were calculated in part 1 above should be compared with the Fourier series coefficients that were found by PSPICE.

**Discussion Items**

1. Compare the various gains and phase shifts found in Part I.

2. Compare the steady state response of the active circuit to the response of the passive circuit when a load is added.

3. Compare the calculated Fourier series coefficients with the PSPICE generated coefficients.

4. Compare the experimental output to the PSPICE simulated output for the square wave input of Part II.

---

**Pre-Lab (10 points)**

**Week 1 (10 points)** Prior to the first laboratory meeting calculate the transfer functions of the four circuit used in Part I. The theoretical part of the table that is required in Part I should also be completed. In addition, the op-amp circuit should be built.

**Week 2 (10 points)** Prior to the second laboratory meeting calculate the two Fourier series required in Part II.