EECS 211
CAD Assignment 1: Low Pass Filter

Goal: To simulate a low−pass filter which is similar to Experiment #1 and to determine its time and frequency response using Accusim. To see that the design equations for K, Q and F0 given in the lab are approximate, and that when you take the finite gain and bandwidth of the LM741 opamp, and it's finite input and output impedance, then you must do some design "tweaking" to get the correct component values for the desired filter specifications (K,Q,and f_0).

Do this assignment using the techniques learnt in the section Introduction to Mentor Graphics Design Architect and Accusim at the start of this manual.

1. Design a low−pass Sallen−Key filter with f_0=20 KHz +/− 0.5 KHz, Q ~ 5 +/− 1. The low frequency input impedance should be greater then 2 KΩ. The low frequency gain should be greater than or equal to 1. In order to minimize the effect of the input capacitance of the LM 741 (at the positive input terminal), use C1 = C2 > 1.5 nF.

   When you enter the schematic in DA, delete the 1 uF DC block capacitor from the input and use an input port similar to the schematic in the introduction for the AC voltage source. Also delete the 220 uF capacitors connected to the DC power supply inputs of the opamp. You do not need the capacitors since the source you specify will be a pure AC source and you do not need a DC block capacitor. Also, make sure that you are connecting the LM 741 with the correct polarity when you hook up the various components in DA. Check it well!

   Give <u>one</u> schematic (circuit diagram entered in DA) of your design and <u>on the same page</u> write your calculations for the different component values.

2. Model this filter using Accusim. Check that K, f_0, and Q agree well with your calculations in part 1. If it does not meet the specs given above, then you will need to tweak the component values. Start with a 100 points per decade frequency mode analysis, check if f_0 and Q agree with the design specs approximately and then simulate with a 1000 points per decade to get a more accurate value for f_0 and Q. Refer to the note at the end for a tip on the design process.

   Give <u>one</u> plot of the transfer function (amplitude and phase) versus frequency (500 Hz − 500 KHz).

3. Determine the time domain response using Accusim for a 500 Hz square wave (V_{ppk} = 1V, V_{avg} = 0V). Remember, the time step should be around one hundredth of the time period (T = 1/500 = 2 ms).

   Give <u>one</u> plot (t=0 to t=0.004 s) showing V_i and V_o on the same chart. Label the final values of V_o.

4. Zoom in on the time domain response and <u>use the cursor feature</u> in Accusim to

determine K, Q, and f_0.

Give <u>one</u> plot of the zoomed in time response showing one set of ripples. On the same page, label the overshoot voltage, ripple frequency and the final voltage. Show your calculations of K, Q, and f_0 <u>on the same page</u> along with a table comparing these values with the ones from the frequency domain plot in part 2 above.

5. Determine the time domain response using Accusim for a 20 Khz (or whatever your f_0 is) square wave with V_{ppk} = 1V and V_{avg} = 0V.

   Give <u>one</u> plot showing V_i and V_o on the same chart. Label the peak voltage of the output with a cursor and compare with your hand calculations using the Fourier Series components of V_i and the filter gain at f_0.

You should submit 5 plots (5−6 pages total). Any CAD having more then 6 pages will be rejected.

Note: The Fourier Series of a square wave with amplitude A (V_{ppk} = 2A), zero dc level and frequency $f_0 (w_0 = 2\pi f_0)$ is:

$$x(t) = \sum_{x=1, n, odd}^{\infty} 4\frac{A}{n\pi}\sin(nw_0 t)$$

A note on the Design Process:
You need to keep in mind that the equations given in the lab book for Q and especially f_0 assume that the opamp is ideal. However, this is not the case, and the opamp does have an input capacitance, a limited gain at high frequencies, etc. Therefore, you will not get the Q and the f_0 that you designed for with the component values got from the ideal formulae.

After you calculate the component values to obtain f_0 and Q and simulate the circuit, you may find that f_0 shifted a bit to a lower frequency, then DECREASE the capacitance a bit (without changing the R's) and re−simulate. Remember that $w_0 = \frac{1}{C\sqrt{R1\,R2}}$, and therefore a decrease in C will increase f_0 (and vice versa).

Since $Q = \sqrt{\frac{R1}{R2}}$, the change in C will not (theoretically) affect Q. However, you will see that it will still have a small effect on Q. The reason is that this equation is valid for C1 = C2 and with the input capacitance to the opamp at the (+) and (−) terminals, and the feedback arrangements, the effective C2 is not equal to C1. Therefore, re−tune the circuit a bit to get the required f_0 and Q.

Good Luck on the design! It will get better on CAD #2.