EECS 211
CAD Assignment 1: Low Pass Filter

Due with Lab Report #2

Goal: To stimulate a band−pass filter which is similar to Experiment #2 and to determine its time and frequency domain response using Accusim. To see that the design equations for K, f_o, and Q given in the lab are approximate, and that when you take the finite gain and bandwidth of the LM741 opamp, and its finite input and output impedances, you must do some design "tweaking" as in CAD Assignment 1 to get the correct component values for the desired filter specifications (K, f_o, Q).

1. Design a band−pass filter with f_o ~ 5 KHz, ±0.2 KHz, Q ~ 15 ±2 and K ~ 50 ±5. The low frequency input impedance should be greater than 5 KΩ. In order to minimize the effect of the input capacitance at the negative input terminal of the LM 741, choose C_1=C_2 > 1 nF.

   When you enter the schematic in DA, delete the 1 μ F 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 F capacitors connected to the DC power supply inputs of the opamp. You do not need the capacitors since the sourve 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 on the same page write your calculations for the different component values.

2. Model this filter using Accusim and check that f_o, Q, K, agree fairly well with the calculations in part 1 above. 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_o, 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_o and Q.

   Given <u>one</u> plot of the transfer function (amplitude and phase) versus frequency.

   Zoom in around f_o, and use the cursors to mark the −3 dB points and determine f_0 and therefore Q.

   Give <u>one</u> plot of the zoomed area around f_o, and your cursor placements for determining Q.

3. Determine the time domain response using Accusim for a 100 Hz square wave (V_{ppk}= 1V, V_{avg} = 0V). Remember, the time step should be around one hundredth of the time period.

   Give <u>one</u> plot (t=0 to t=0.02 s) showing V_{in} and V_{out} on the same chart.

4. Zoom in on the time domain response and use the cursor feature in Accusim to determine K, Q, and f_o.

   Give <u>one</u> plot of the zoomed in time response showing the final voltage of a cycle and the first 6−8 ripple peaks of the next cycle. On the same page, label the first peak voltage, ripple frequency and the final voltage. Show your calculations of K, Q, and f_o on the same page 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 square wave with a frequency equal to <u>your</u> f_o (around 5 kHz) with V_{ppk} = 0.1V and V_{avg} = 0V.

   Give <u>one</u> plot showing V_{in} and V_{out} on the same chart. Label the peak voltage of the output and compare with your hand calculations using the Fourier Series components of V_i and the filter gain at f_o.

6. Determine the time domain response using Accusim for a triangular wave with a frequency equal to your f_o (around 5 kHz) with V_{ppk} = 0.1V and V_{avg} = 0V.

   Give <u>one</u> plot showing V_{in} and V_{out} on the same chart. Label the peak voltage of the output and compare with the hand calculations using the Fourier Series components of V_{in} and the filter gain at f_o.

You should submit 7 plots (7−8 pages total). Any CAD having more than 8 pages will be rejected.

$$x(t) = \sum_{x=1,n,odd}^{\infty} 8\frac{A}{n^2\pi^2}\sin\left(n\frac{\pi}{2}\right)\sin\left(nw_{0t}\right)$$

Setting up a triangular wave of period T in Accusim:

1. Go to TIME MODE
2. Click an ADD FORCE
3. In the dialog box that opens up, click PWL in the **Force Type** field. (PWL stands for piece wise linear)
4. Click on Time−Value Pairs
5. Enter 0 in **Time** and −0.5 in **Value**. Once you do that, another Time/Value pair box will appear below.
6. In the nest **Time /Values** boxes, enter T/2 and 0.5 for time and value respectively.
7. In the **repeating Period** box, enter T.

This sets up a triangular wave of 1 V ppk with a frequency = 1/T. So knowing the frequency you can enter the appropriate values for T/2 and T in steps above.