OPAMPs and Filters

General Hints: For all simulations include the schematic and the output waveform showing a selection of most relevant voltages/currents. The waveforms should look as you would expect them to look like on an oscilloscope. If appropriate, the circuit netlist or parts of it should be included in the report. All choices of component values or parameters you are making should be justified. All simulation results should be put into context with theoretical calculations followed by a brief discussion.

Problem 1: Inverting Amplifier

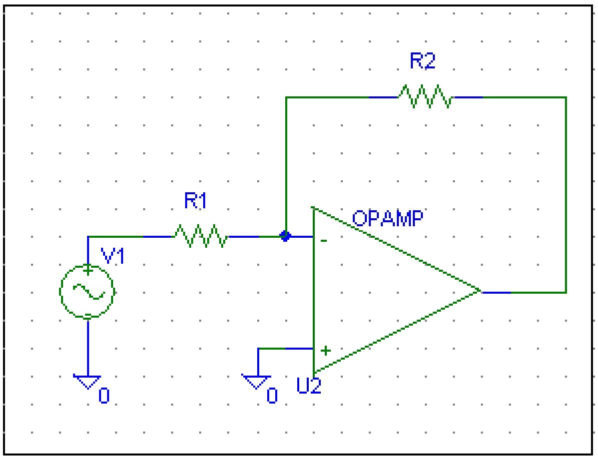

Fig.1: Inverting amplifier.

Consider the inverting amplifier in fig. 1. The exercise is aimed at investigating this amplifier by simulation.

For exercises a) to c) use an ideal opamp (Partname: *OPAMP*)

a) Choose R1 and R2 so that you get a gain of 10. Perform a transient simulation with an input sinusoidal source (f=1kHz, Amp.=1V).
b) Perform a parametric sweep of R2 and repeat the transient simulation. Explain what happens above a certain value of R2. What can be done to solve the problem?
c) For suitable values of R2/R1, what are the input currents into the amplifier? (*HINT*: Simply placing a current marker on the input terminal of ideal opamp symbol does NOT give the correct answer). What is the voltage across the negative and positive terminal of the opamp? Explain their values.

Now replace the ideal opamp with the model of a 741 opamp (Partname: uA741)

d) Repeat exercises a) – c).
e) From suitable simulations derive the DC open loop gain, offset voltage, input resistance and output resistance of the opamp. Compare your results with values from the 741 datasheet. (*HINT:* You may find it useful to connect the amplifier output to a load resistance.)
f) Investigate the frequency response of the closed loop amplifier. What are the open loop -3dB corner frequency, the unity gain frequency and the closed loop -3dB frequencies for different closed loop gain values. (*HINT:* In PSPICE, do

an AC analysis with a parametric sweep of R2 and present the output voltage as a Bode plot.) Again, compare your results with the datasheet.

Problem 2: 2nd Order Butterworth Filter with Sallen & Key Architecture

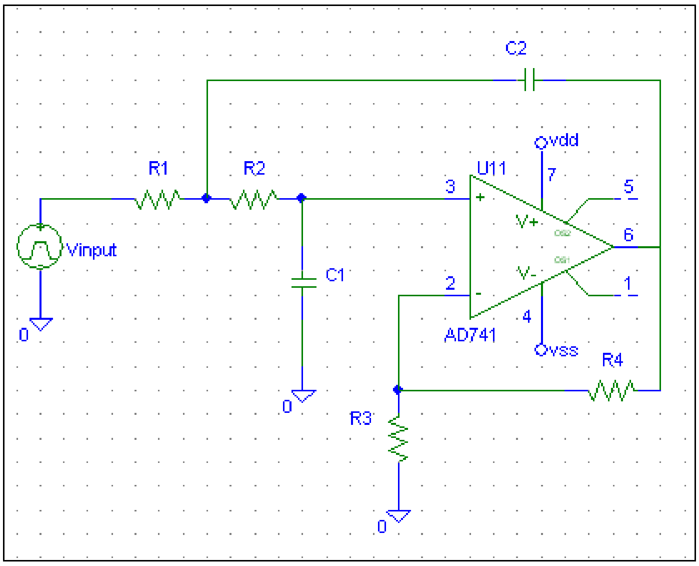

Fig. 2: Sallen and Key filter architecture.

Analyse, design and characterise a Sallen and Key (as shown in fig. 2), 2^{nd} order Butterworth filter with a cut-off frequency of fc=300Hz. (*HINT:* Compare your PSPICE simulations with ideal responses you may obtain from simulations in Matlab.)