INDIAN INSTITUTE OF TECHNOLOGY DHARWAD
Dharwad, 580 011

EE 202: Introduction to Analog Circuits
Assignment 1
Deadline: 12th Nov. 2021

Simulate all questions using ngspice. The ua741.txt model file can be downloaded from Moodle.

The assignment should be done independently.

1. Construct a model of an opamp using the equivalent circuit shown in Fig. 1. Use g_m of 10 m℧ and C value of 10 nF. Choose values of G and R for some suitable gain.

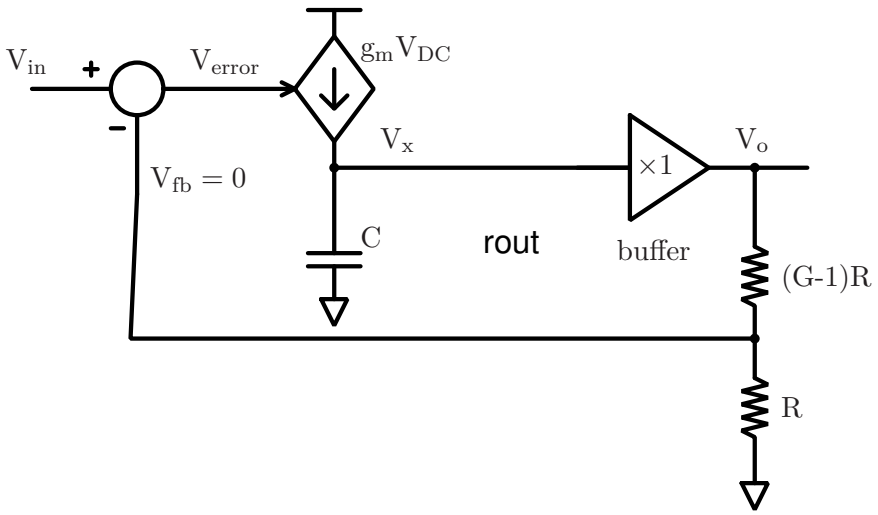

Figure 1: Model of an opamp.

   (a) Simulate the transient response of opamp model to confirm its working as an amplifier for a suitable step input. You can give a step input from 0 to 1 V.

   (b) Open the feedback loop and simulate the opamp (transient simulation) for a DC input of 1V to demonstrate the behaviour of the opamp as an integrator.

   (c) In the open loop condition, perform an AC simulation and plot the transfer function of the opamp model (V_o/V_{in}) in dB on a semilogX plot.

   (d) Next add and R_{out} of 10 MΩ in shunt with C to represent the finite output impedance of the voltage controlled current source (VCCS). Repeat above 3 sub questions (a) through (c). Compare your results obtained in part (c) with the datasheet of μA741 and comment on your observations.

2. In the opamp model of Question 1, insert an offset error at the input of the VCCS and simulate the circuit for a step input. Comment on your observations.

   (a) Next, move the offset error to the feedback path and simulate the circuit again. What do you observe?

3. Design an amplifier using an opamp with a gain of 20 and write a spice netlist for the same using the UA741.txt spice model for μA741 opamp. Calculate its bandwidth.

Simulate the circuit (transient and AC simulation). Compare the gain and bandwidth values obtained with your calculated values.

(a) Simulate the circuit for a step input of 0.1 V. Plot the voltage on the inverting terminal (the feedback node) along with the input and output and comment on your observations about working of the negative feedback loop.

For next 3 questions use UA741.txt spice model for μA741 opamp.

4. Design an AC amplifier with a gain of 12 and with a lower curtoff frequency of 10 Hz, driving a load (which is also AC coupled) of 1 kΩ. Simulate the amplifier in NGSpice, plot the frequency response and compare the results obtained with your design.

5. Design a bandstop filter that blocks all signals between 300 Hz and 3 KHz. Simulate the amplifier in NGSpice, plot the frequency response and compare the results obtained with your design.

6. Design a voltage to current converter for currents ranging from 100 nA to 10 μA, driving a grounded load. Simulate your design for loads from 100 Ω to 10 kΩ and comment on the load regulation.