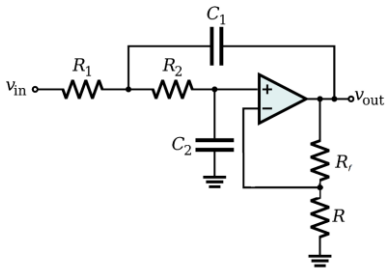
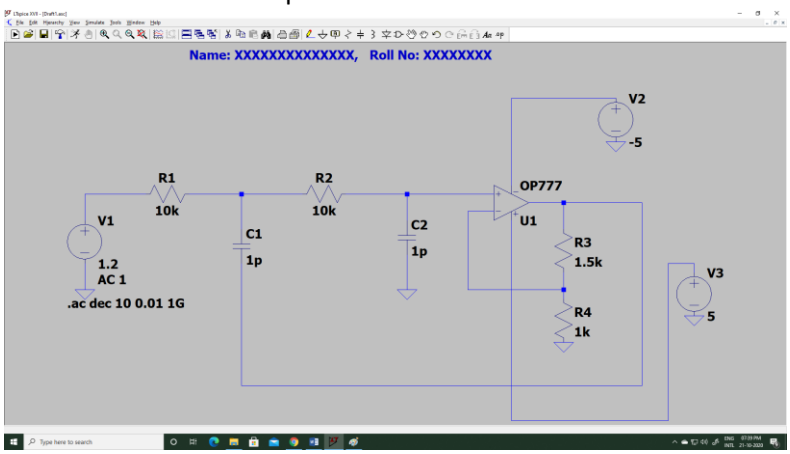
Laboratory Assignments on Experiment 4: Active low pass filter

- General Instructions:
  1. Download and install LTSpice from the following Link: https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html
  2. Experiment how to use this tool to develop a second order passive RC filter. Then try to get frequency or AC response of the circuit.
  3. Once you are comfortable, proceed for doing the following experiment.
  4. Please submit soft copy of the reports in Moodle by November 9th, 2020.

- Simulation Assignment:



1. Draw neatly the above circuit in LTSpice. All resistors and capacitors should be used as ideal components. However, OP777 op-amp should be used from the component library. Attach the screen-shot of the schematic after entering your Name and Roll No as a text on it. Please see the reference example below. (10)



2. Use R_1=R_2=10 KΩ and C_1=C_2=1 nF. For setting a Q factor of 1, calculate the value of R_f and R. Also, calculate the cut-off frequency theoretically and show the frequency response (gain and phase plots). Attach the screen shot of the plots. (10)
3. Now, change the Q value to 1.5 and 2.5 and show the frequency response (gain and phase plots). Attach the screen shot of the plots. (10)
4. Next choose R_1=R_2=10 KΩ and C_1=C_2=1 pF and Q=2.5 and calculate the value of R_f and R. Also, calculate the cut-off frequency theoretically and show the frequency response (gain and phase plots). Attach the screen shot of the plots. Please comment on this results. (10)