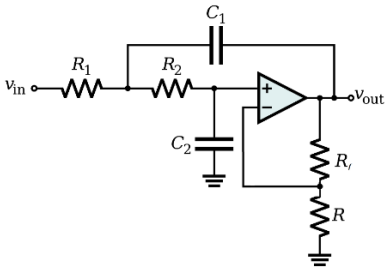
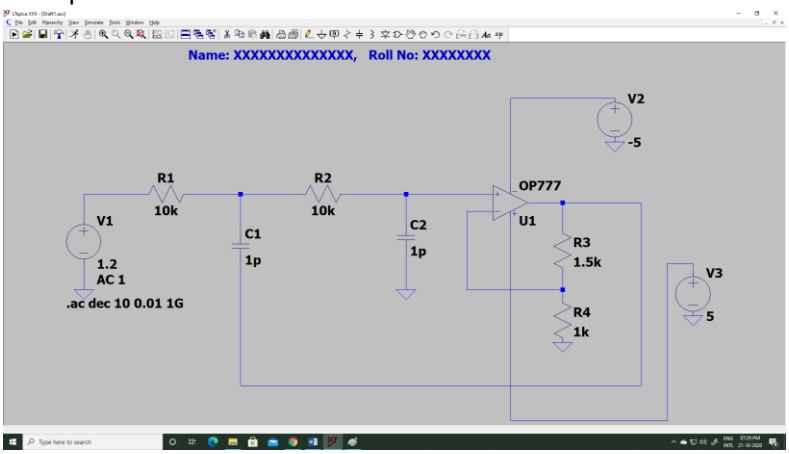
Laboratory Assignments on Experiment 4: Active low pass filter

- 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 un-damped natural frequency theoretically and show the frequency response (gain and phase plots). Attach the screen shot of the plots. Please comment on this results. (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. Please comment on this results (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 un-damped natural frequency theoretically and show the frequency response (gain and phase plots). Attach the screen shot of the plots. Please comment on this results. (10)