LT-SPICE PROBLEM BANK

1.) Using SPICE, analyze the following Full Wave (Bridge) Rectifier Circuit by plotting the input waveform across AC source V1 and output waveform across resistor R2:
(a) Without Smoothing Capacitor C1
(b) With Smoothing Capacitor C1
Using these two simulations, explain the role of the Smoothing Capacitor. Can you
also identify the path that the current traces during the positive wavecycle and the
negative wavecycle?
Now, plot the output waveform by changing the capacitance of C1 to:
(a) 20 nF
(b) 20 uF
Can you now understand the relation between the capacitance value and the AC source frequency? Explore further by changing the source frequency.
NOTE : Use transient analysis in order to plot all waveforms.

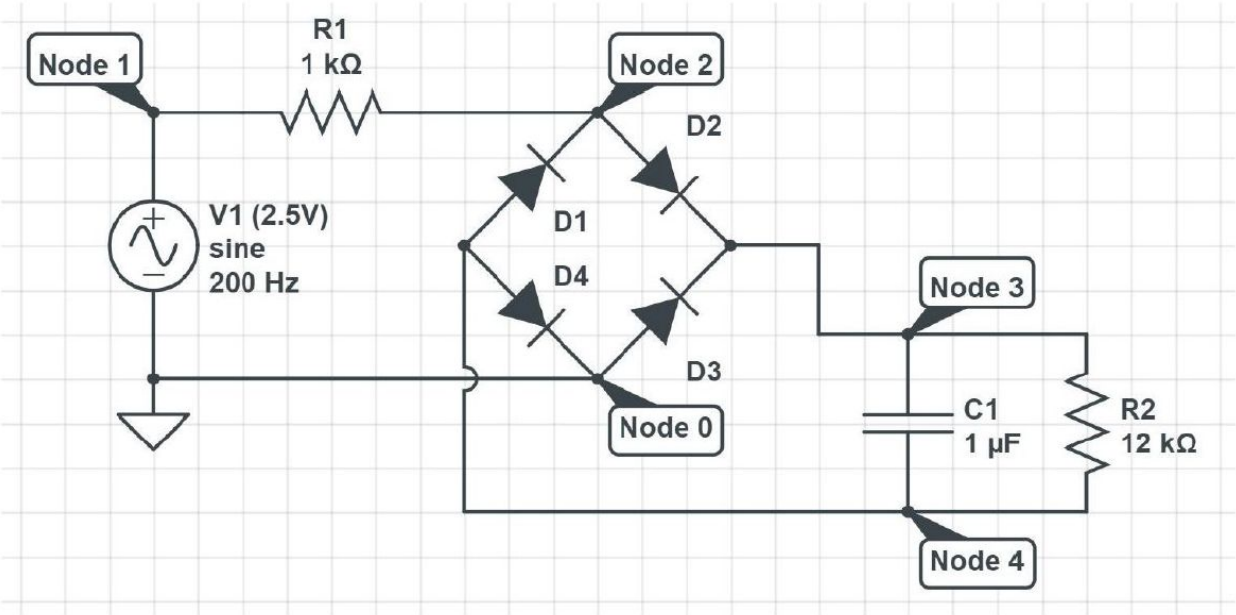


Fig Q1

2.) (i) Identify the type of filter and determine the -3dB frequency(s).
C1=1nF, C2=0.08nF, R1=R2=10kΩ.
(ii) (a) Now remove R2 and C2 and check the output across resistor R1. Find out the
type of filter if output is taken across R1.
(b) From the original circuit, remove C1 and R1, connect R2 to V1 and then find out the the type of filter by taking output across C2.
Use ac analysis.

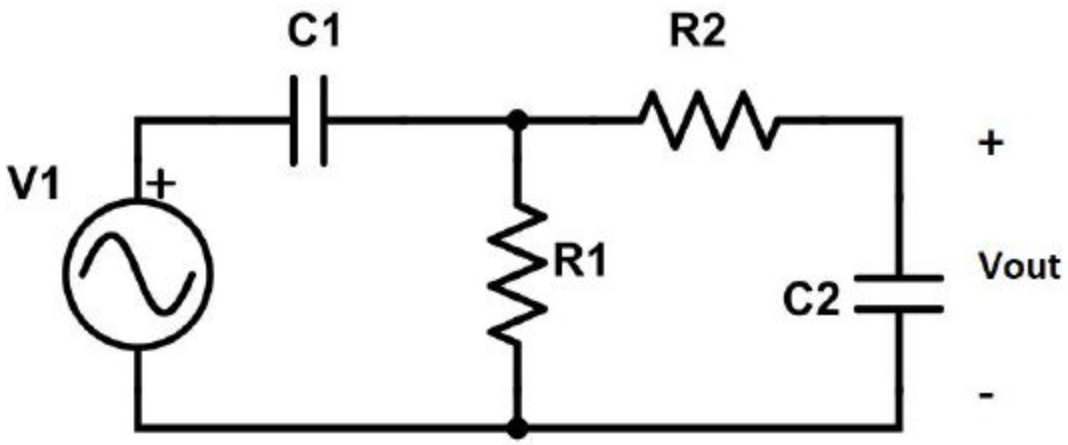


Fig Q2

3.) a) Assuming input voltage in the given figure to be Step input, perform the transient analysis. Hint: Use pulse waveform with pulse width greater than period.
S1=5*u(t) V, R1= 5KΩ, C1=15uF.
Also vary capacitance from 1uF to 50 uF in steps of 5 uF and check effect on output plot.
b) Assuming input voltage to be a square wave with 50% duty cycle and frequency of 25Hz between voltage levels 0 and 5V, plot transient analysis for the following:
i) R1= 1KΩ, C1=1uF, till t=100ms.
ii) R1= 10KΩ, C1=2uF, till t=300ms.
iii) R1= 100KΩ, C1=20uF, till t=1s.

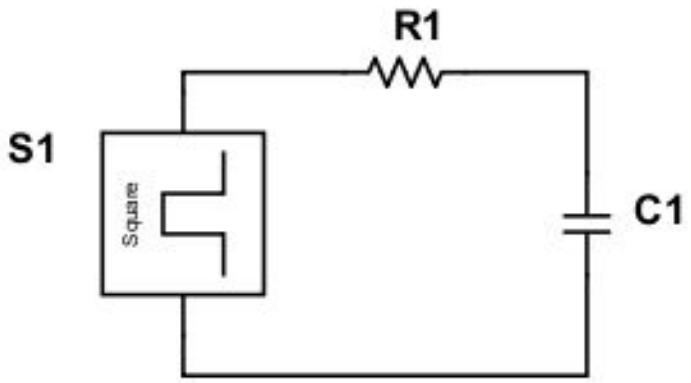


Fig Q3

c) Assuming input to be ACsine of 10mV amplitude, 1KHz frequency, riding on 2V dc level. Perform DC operating point, transient and AC analysis. Find -3dB frequency and bandwidth. Identify the type of filter. Find dc transfer function.

Take R1=5KΩ and C1=1uF.

Now observe changes in time response and Bode plot on sweeping capacitance from 1uF to 50 uF in steps of 5 uF.

4.) With SPICE build a current mirror circuit which has a reference current of 10 μA and and output current of 100 μA. The load resistance can be any value upto 500 kΩ. Sweep the resistance value and plot the output current and show that it is constant.

Leave the output branch hanging and see the current/voltages in that branch. Can you explain the results with the charging/discharging phenomena?

5.) For the given common-emitter amplifier configuration, Vcc=10v,R1=3.5kΩ,R4=1kΩ,C3=100μF,C2=100μF,C1=10μF,R1=100kΩ Determine

a) Base current, I_B

b) Find the voltage gain using SPICE simulation.(Draw ac and transient analysis)

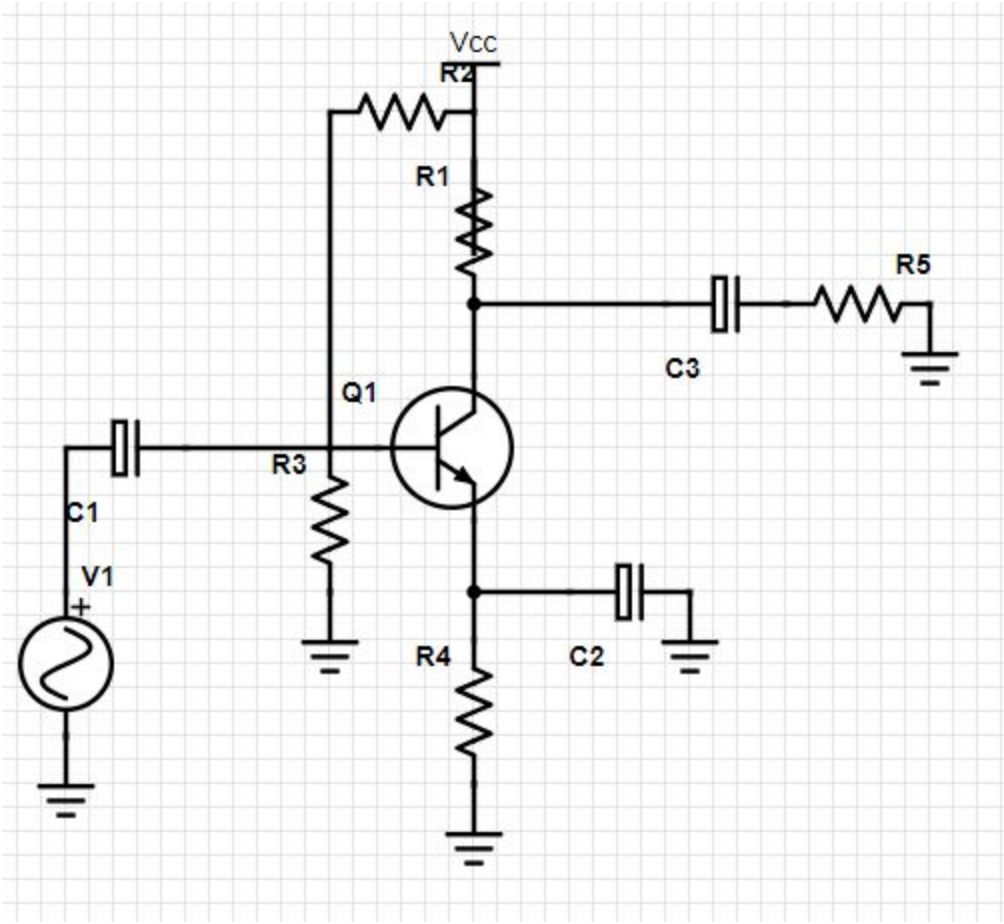


Fig Q5

6.) Design a common source amplifier which fulfills the following requirements,

a) Load should be PMOS.
b) Gain should be of 30 dB.
c) Unity gain bandwidth should be of 100 MHz.

7.) Use Spice to analyze the given circuit by plotting the output voltage. Using the similar concept of clipper circuit, get waveform A.
NOTE: Use transient analysis in order to plot all waveforms.

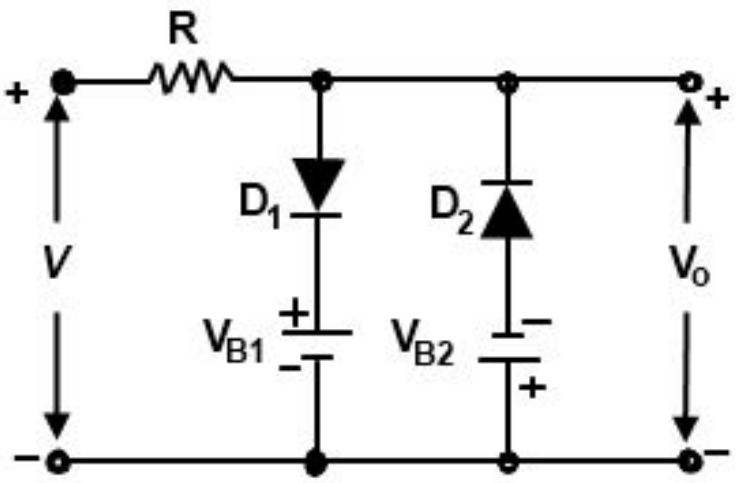
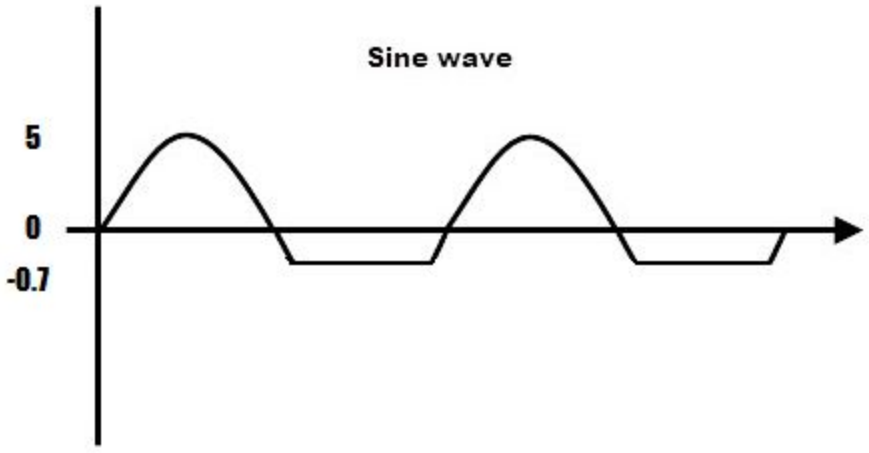


Fig Q7

Take
V= 5v (peak),1KHz
R= 1kΩ
Vb1=2v
Vb2=3v

Waveform A:



8.) (i). Plot I d vs V ds Characteristic curve for different gate voltages for NMOS in LTSpice.
(ii). Calculate the threshold voltage of the MOSFET through LTSpice simulation.
(iii). Find out the active resistance of NMOS for gate voltage of 1.1V.

Hint. 1) Connect the gate and drain of NMOS with separate DC voltages. Perform DC analysis for drain voltages and sweep the gate voltage between appropriate interval and get the I d vs V ds Characteristic curve.

2) Take the ratio of currents at different gate voltages and same drain voltage and calculate V T from equation given below

$$I_1 / I_2 = (V_{gs1} - V_T)^2 / (V_{gs2} - V_T)^2$$

3) Calculate the slope of I d vs V ds curve for gate voltage of 1.1V in saturation region. Inverse of the slope gives resistance.

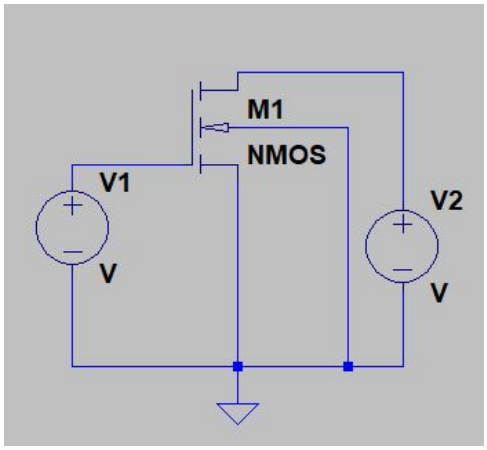


Fig Q8

9.) Welcome to the world of filters! Most filters can be realized using simple R-C circuits. This example would prove to be an effective introduction to most common filters and utilizing the common commands in LTSpice.

Low Pass Filter

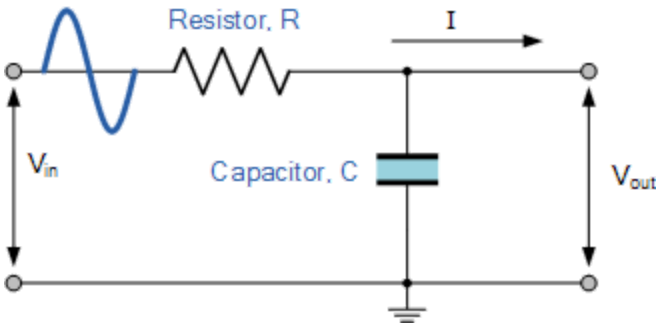
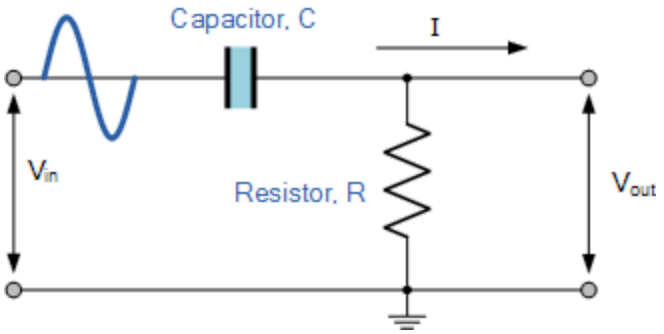


Figure SEQ Figure * ARABIC 1a

$$R = 10k\Omega,\ C = 1nF$$

a) Simulate the circuit (1a) and plot the frequency response (Vo vs frequency)
b) Given that Vin = sin(2 ρ π ft where f = (i) 2kHz (ii) 25kHz. Draw the transient response of Vo.
c) Plot the Bode plot of the circuit (Vo/Vin vs frequency) and determine the cutoff frequencies from the graph.
d) Given than Vin is a pulse of f=5kHz. Draw the transient response of Vo

High Pass Filter



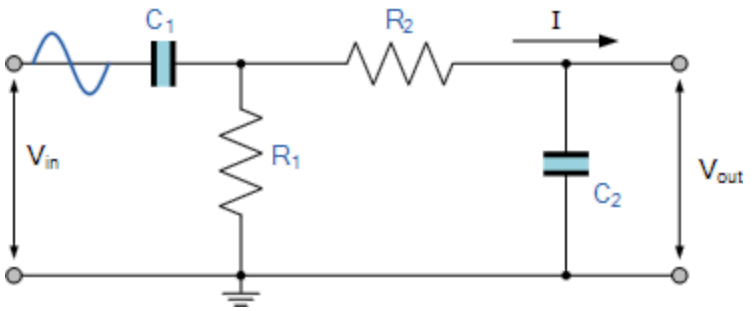
$$R = 10k\Omega,\ C = 1nF$$

a) Simulate the circuit (1b) and plot the frequency response (Vo vs frequency)
b) Given that Vin = sin(2 ρ π ft), where f = (i) 2kHz (ii) 25kHz. Draw the transient response of Vo.
c) Plot the Bode plot of the circuit (Vo/Vin vs frequency) and determine the

Figure 1b

cutoff frequencies from the graph.
d) Given than Vin is a pulse of f=5kHz. Draw the transient response of Vo

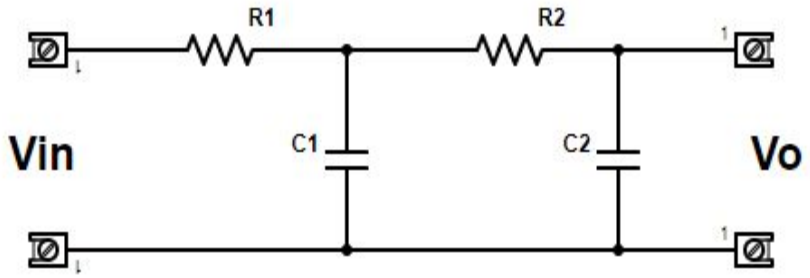
Bandpass Filter



$$R_1 = R_2 = 10k\Omega,\ C_1 = 15nF,\ C_2 = 560pF$$

a) Simulate the circuit (1c) and plot the frequency response (Vo vs frequency)
b) Given that Vin = sin(2 ρ π ft), where f = (i) 500Hz (ii) 2kHz, (iii) 35kHz. Draw the transient response of Vo.
c) Plot the Bode plot of the circuit (Vo/Vin vs frequency) and determine the cutoff frequencies from the graph.
d) Given than Vin is a pulse of f=5kHz. Draw the transient response of Vo

Bandstop Filter

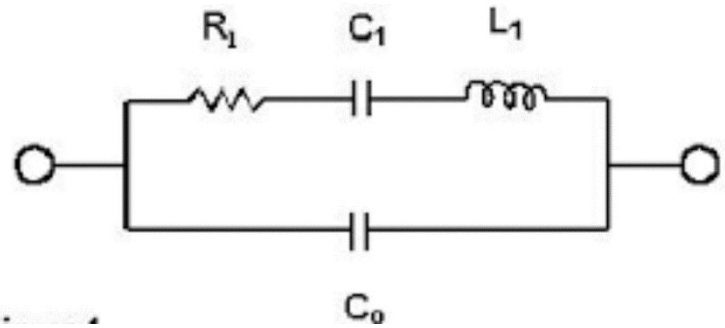


$$R_1 = R_2 = 10k\Omega,\ C_1 = 15nF,\ C_2 = 560pF$$

a) Simulate the circuit (a) and plot the frequency response (Vo vs frequency)
b) Given that Vin = sin(2 ρ π ft), where f = Hz. Draw the transient response of Vo.
c) Plot the Bode plot of the circuit (Vo/Vin vs frequency) and determine the cutoff frequencies from the graph.
d) Given than Vin is a pulse of f=5kHz. Draw the transient response of Vo

10.) Question:- Do the AC analysis of the following circuit with C = 1600pF, C1 = 80pF, L1= 312.5mH and R= 1250 ohm. Fundamental frequency for FFT is same as source frequency.

1) Find the resonant frequencies.
2) Interpret the argument plot in terms of the behaviour of circuit(region for inductive and capacitive behaviour)
3) Find the overall impedance of the circuit using ltspice.
4) Find out the components which affects resonating frequencies.
5) Find the harmonic components and total harmonic distortion.
6) Find the power factor of the circuit.



11.) For the following amplifiers,

1. Do AC,DC and transient analysis
2. Find the transfer function
3. Find the 3dB frequency
4. Perform Fourier Analysis and find the Total Harmonic Distortion (THD).

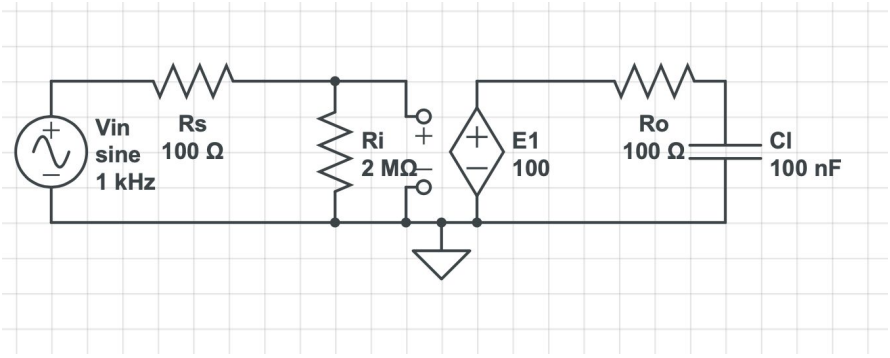


Fig 1

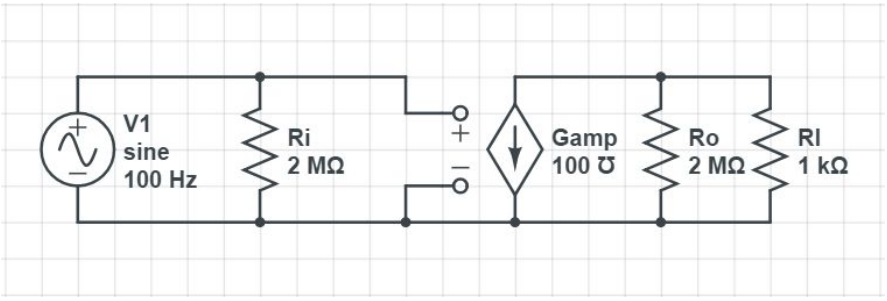


Fig 2

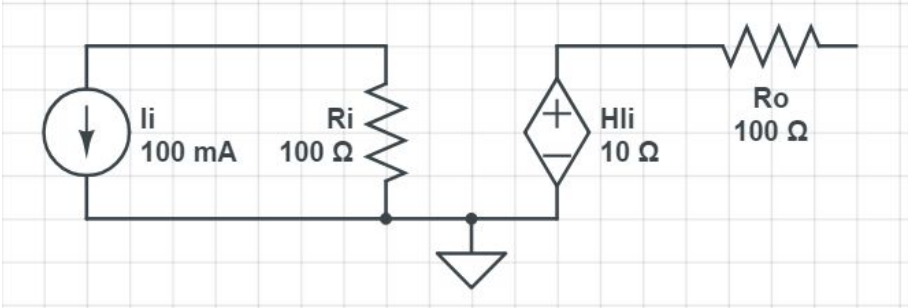


Fig 3

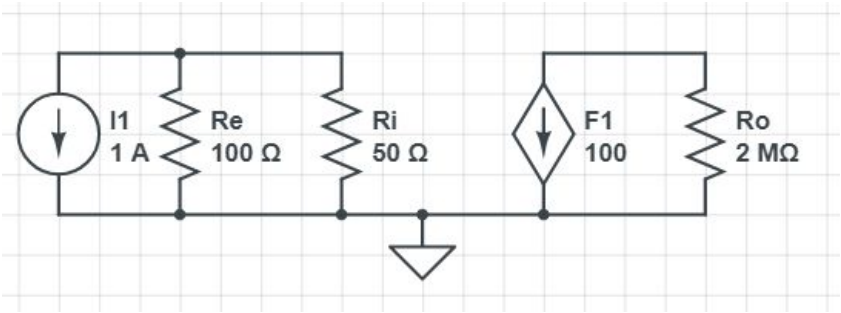


Fig 4

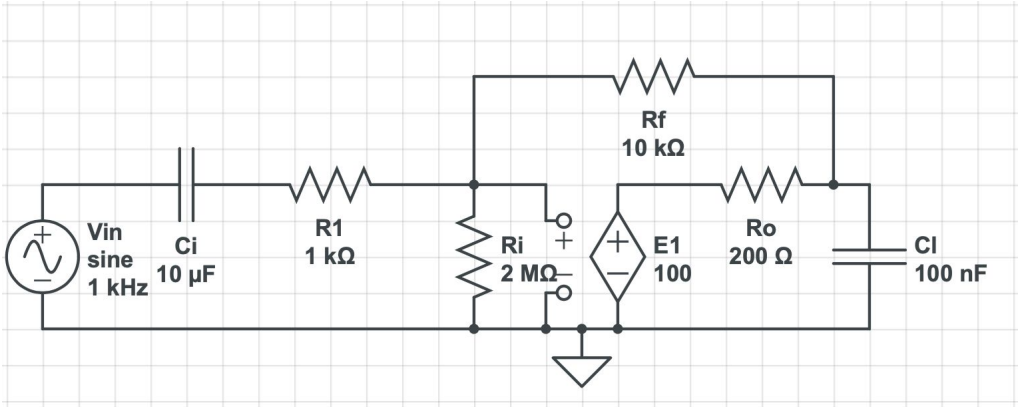


Fig 5

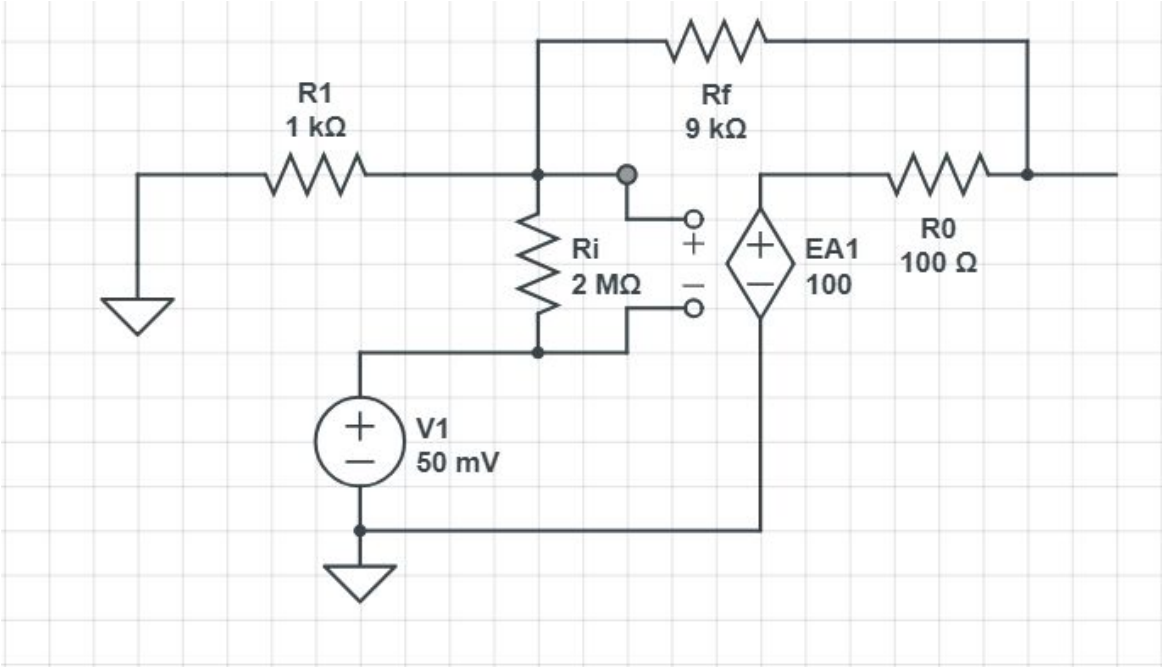


Fig 6