ENGR337 Homework Package

Chapter 1 Circuit Basics and LTSpice

1.1 Use KCL and KVL to solve the current/voltages in the following circuit.

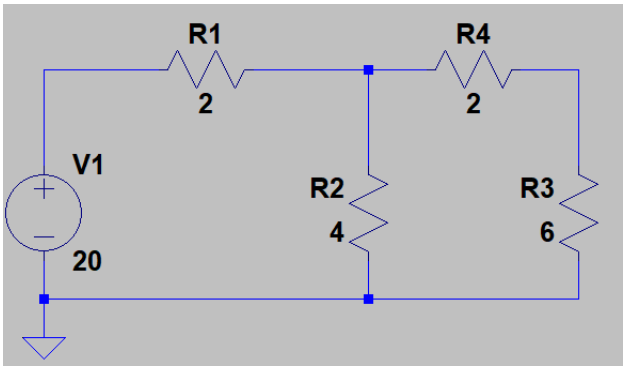
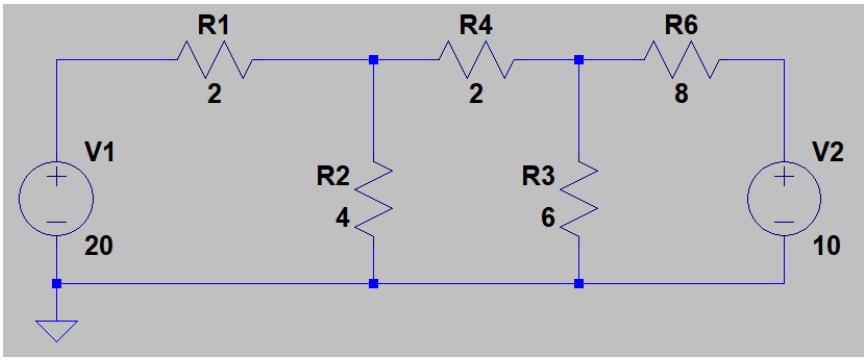


Fig. 1.1

1.2 Use the Mesh Current method to solve the current/voltages in the following circuit.

(a)



(b)

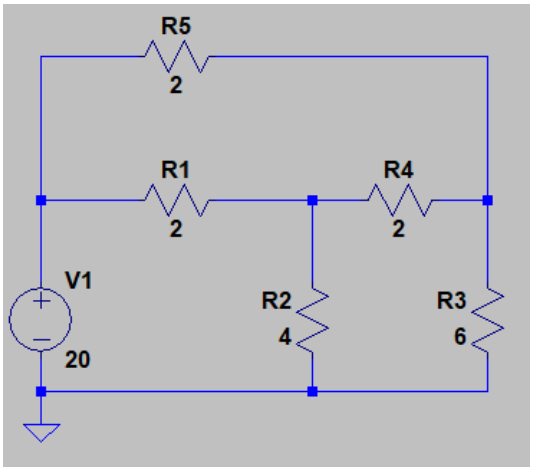


Fig. 1.2

1.3 Use the Super Mesh Current method to solve the current/voltages in the following circuit.

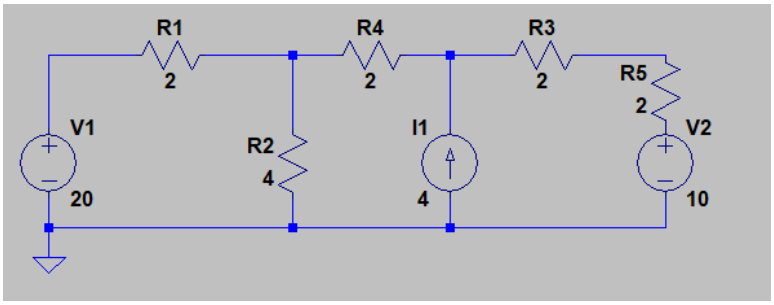


Fig. 1.3

1.4 **Convert** the following circuit into it's **Thevenin's equivalent circuit**, then **calculate** the current flows through the load resistor.

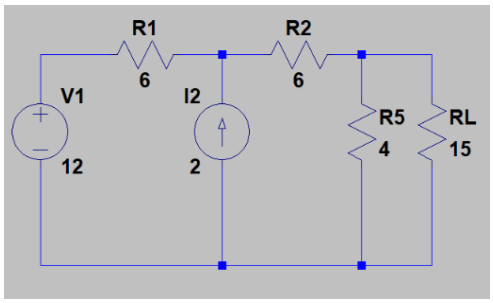


Fig. 1.4

1.5 **Convert** the following circuit into it's **Thevenin's equivalent circuit**, then **calculate** the current flows through the load resistor.

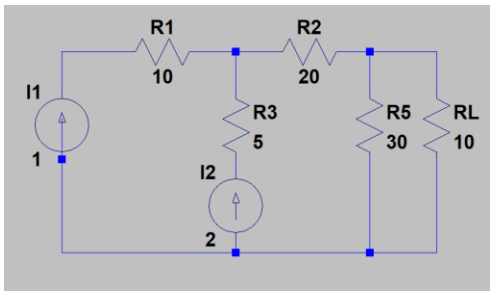


Fig. 1.5

1.6 **Convert** the following circuit into it's **Norton's equivalent circuit**, then **calculate** the current flows through the load resistor.

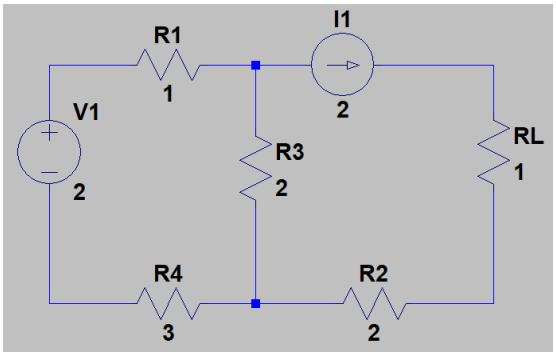


Fig. 1.6

1.7 **Convert** the following circuit into it's **Norton's equivalent circuit**, then **calculate** the current flows through the load resistor.

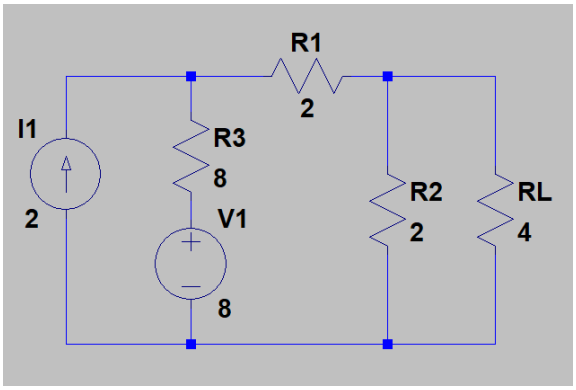


Fig. 1.7

1.8 Calculate |Vo/Vi|, and the time delay. Hand draw the input the output signal, and also show the time delay on the graph. Finally, compare the results to your simulation. V1: amplitude 1V (Vpp 2V), f=100 kHz.

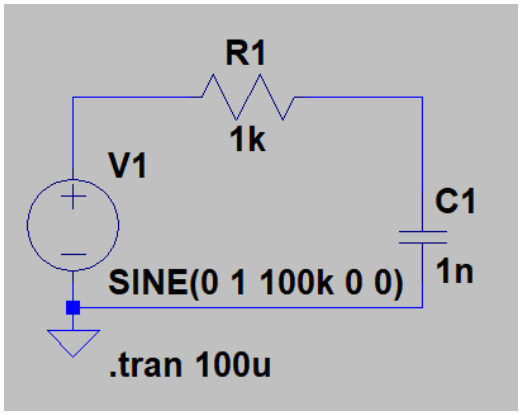
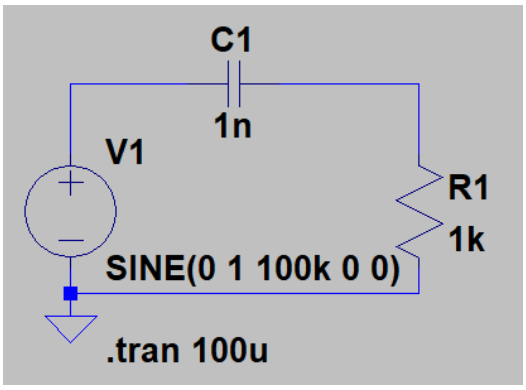


Fig. 1.8

1.9. Calculate |Vo/Vi|, and the time delay. Hand draw the input the output signal, and also show the time delay on the graph. Finally, compare the results to your simulation. V1: amplitude 1V (Vpp 2V), f=100 kHz.



1.10. Calculate |Vo/Vi|, and the time delay. Hand draw the input the output signal, and also show the time delay on the graph. Finally, compare the results to your simulation. V1: amplitude 1V, f=100 kHz.

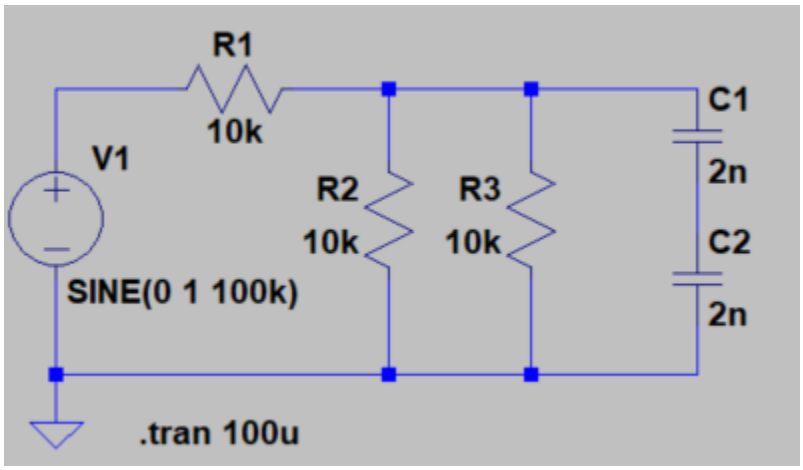


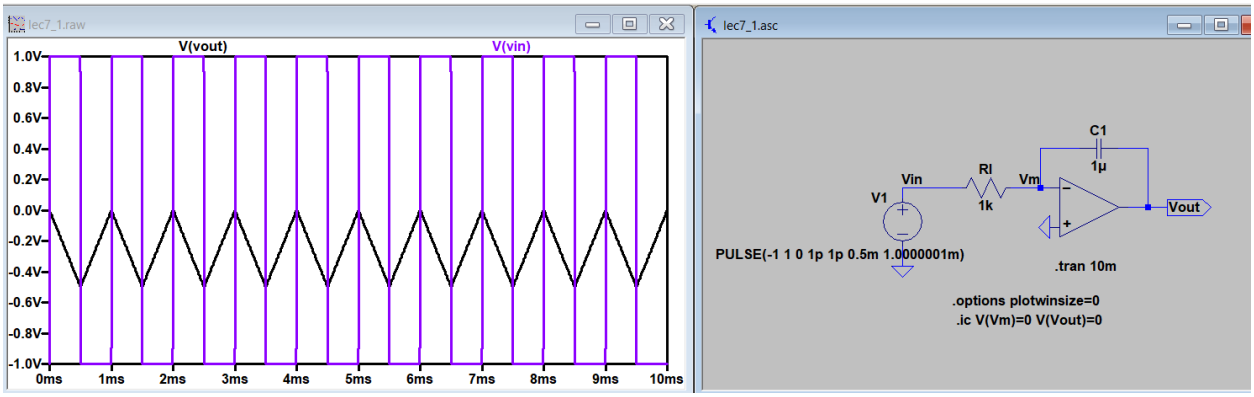
Fig. 1.10

Chapter 2 Operational Amplifiers I (Op Amp I)

2.1. 1) Use LTSpice, use the E1 voltage amplifier, create a symbol to make it in the standard OpAmp triangle shape like I showed in the lecture. 2) Use this OpAmp to build an inverting amplifier and a non-inverting amplifier. 3) Derive the voltage gain for these two amplifiers, and use LTSpice to verify the gain.

2.2. In the following integrator, Vo is charged to -0.5 V – 0 V. Modify the value of the resistor or the capacitor, to make the Vo swings between -1 V – 0V. (Do not change anything about the voltage source).

(Show the calculations and the simulations for credit).



2.3 Based on the following circuit, design the values of the capacitor and the resistor to make the cutoff frequency to be 5 kHz. Use LTSpice to verify the cutoff frequency.

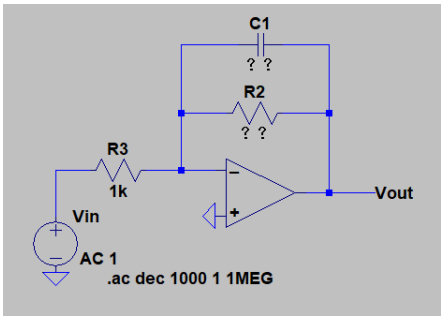


Fig. 2.3

2.4 Derive the voltage gain of both the inverting OpAmp and the non-inverting OpAmp when the open-loop gain of the OpAmp is not infinite (a finite gain of ***A***).

2.5 Derive the differential gain and the common mode gain of the following difference amplifier. Then, show the common-mode rejection ratio (CMRR). (Assume R2/R1 = R4/R3, or say, R2=R4, R1=R3).

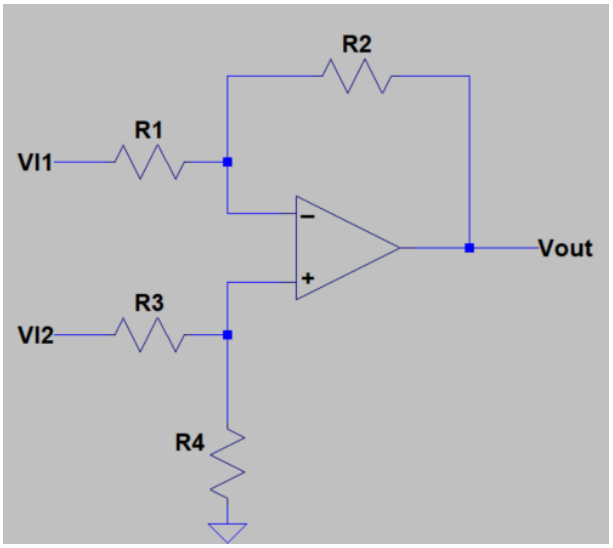


Fig. 2.5

2.6 Derive the voltage gain of the following instrumentation amplifier.

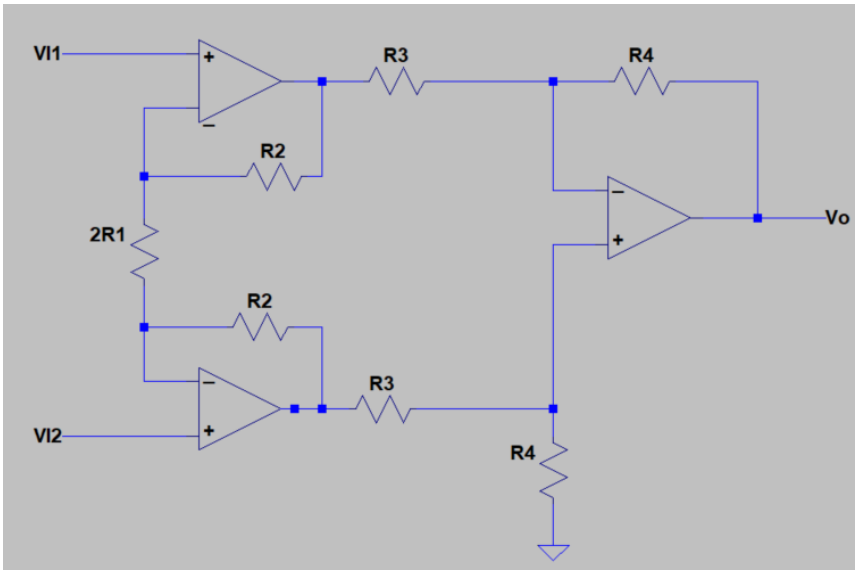


Fig. 2.6

Chapter 3 The pn Junction and the Diode

3.1 Draw the pn junction and explain:

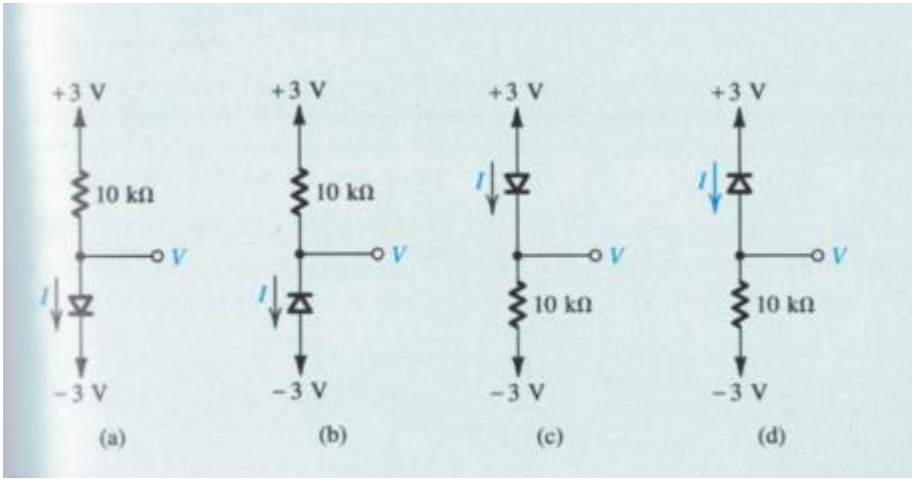
1) Without an external voltage, the formation of a depletion region, and the formation of the diffusion current and the drift current.

2) With an external voltage, explain the changes of the width of the depletion region under 'forward bias' and 'reverse bias' operations.

3) Explain how the depletion region will be changed, and how the diode capacitance will be changed with an increasing voltage under both 'forward bias' and 'reverse bias' operations?

3.2 Ideal diodes, find the values of the voltages and currents indicated in Fig. 3.2 (a-b)

(a)



(b)

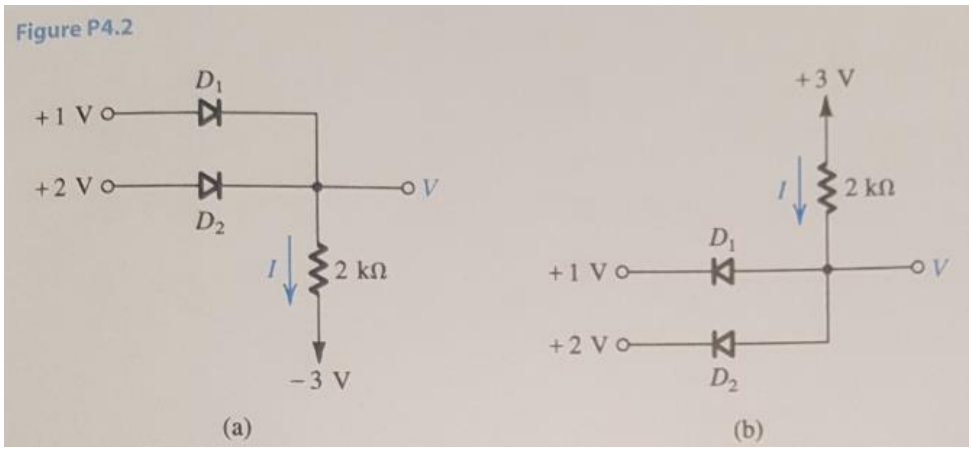


Fig. 3.2

3.3 In Fig. 3.3, Vi is a 1k Hz 5-V peak-peak sine wave (centered at 0V), sketch the waveform of Vo and label the peak values.

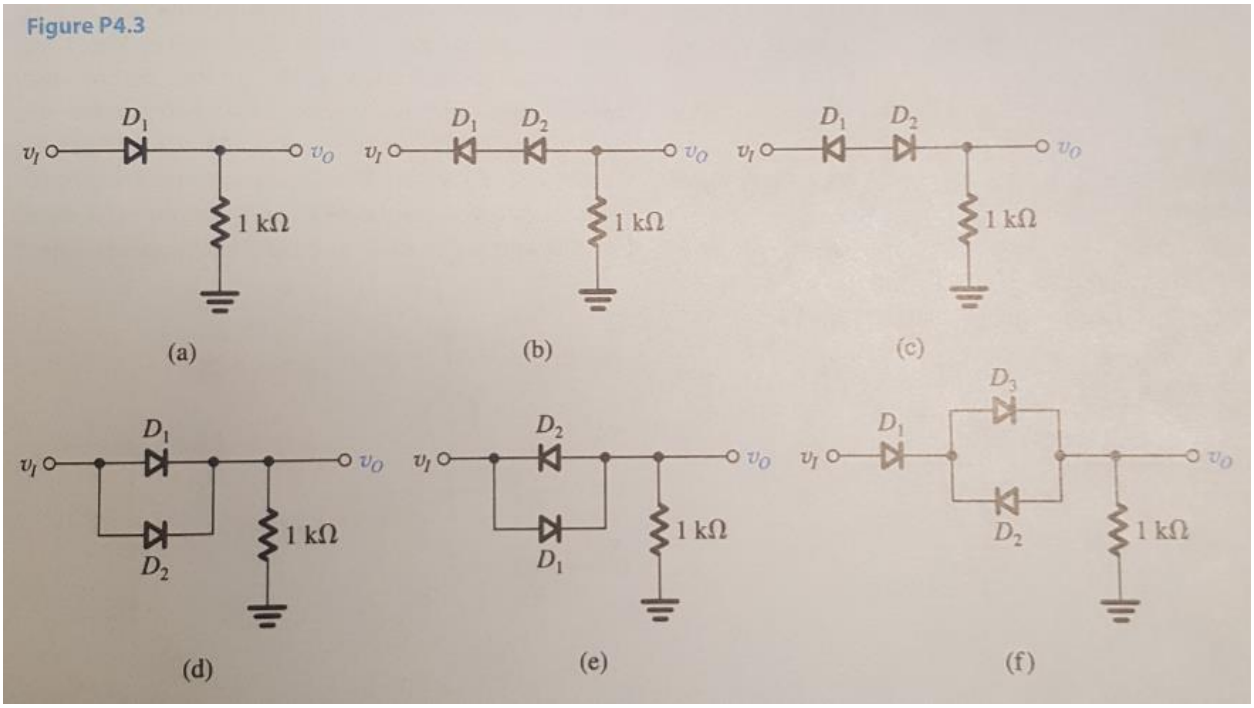


Fig. 3.3

3.4 Design a voltage multiplier (a high voltage DC power supply), the input voltage is a (-5V,5V) peak-peak square wave, the output should be at least 40 V DC. However, the working load is a 50 pF capacitor, the maximum voltage can be applied to the cap is 20 V. Use this HV power supply to drive this capacitor without burning it. (you may need a Zener diode to protect your circuit). (Use LTSpice to design the circuit and demonstrate your design).

Chapter 4 The MOSFETs

**KP and Vth can be found in model.txt

4.1. (1) Build the circuit using nmos in LTSpice, and change the VGS values to 'list 1.5 1.8 2', and simulate the 'VDS vs ID' curve. (2) Use a fixed VGS voltage and change the variable to VDS, and use '.step param VDS list 2 3 4 5', and simulate the 'VGS vs ID' curve. Show your simulation on a printed paper for credit.

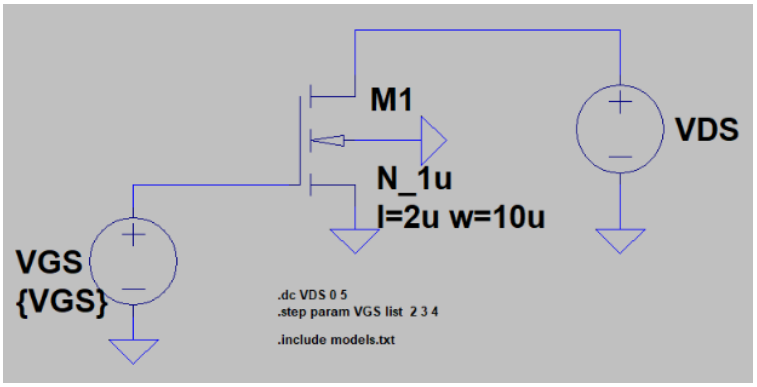
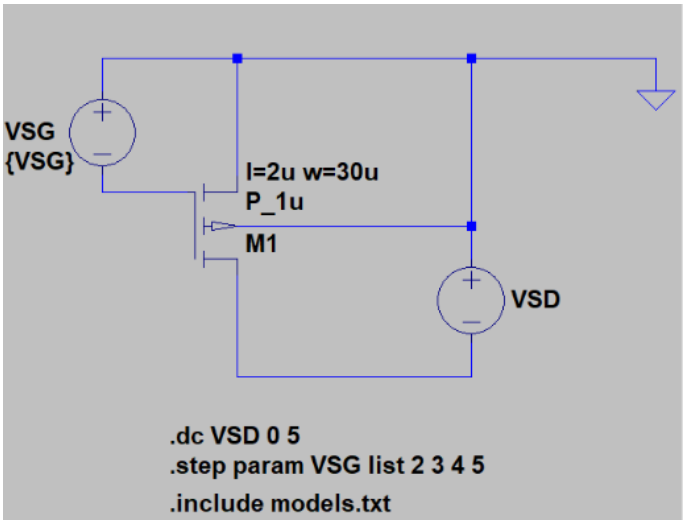


Fig. 4.1

4.2. Repeat the problem above for the PMOS circuit below: (please note that it is VSG/VSD now, and the PMOS substrate is connected to the highest potential in the circuit). Show your simulation on a printed paper for credit.



4.3. Draw a figure to explain the 'Body Effect'.

4.4. Calculate I_D of the NMOS and verify with LTSpice. Show your calculation/simulation on a printed paper for credit.

**KP and Vth can be found in model.txt. Use the N_1u model for this problem.

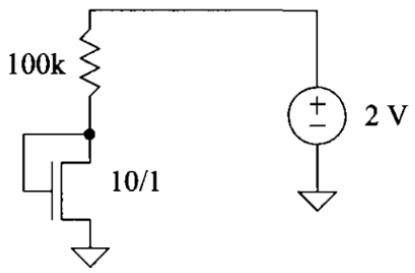


Fig. 4.4

4.5. Calculate I_D of the PMOS and verify with LTSpice. Show your calculation/simulation on a printed paper for credit.

**KP and Vth can be found in model.txt. Use the P_1u for this problem.

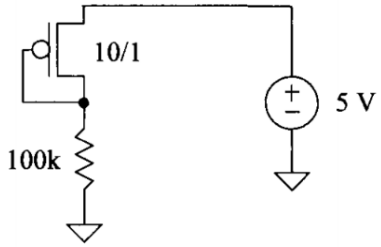


Fig. 4.5

4.6 For the circuits seen in Fig. 4.6, estimate the output DC voltage. Verify the calculation using LTSpice. (use .op)

. **KP and Vth can be found in model.txt. Use the N_1u and the P_1u for this problem.

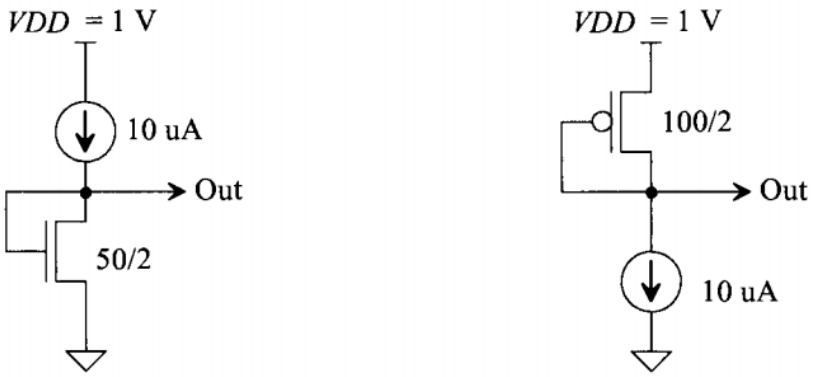
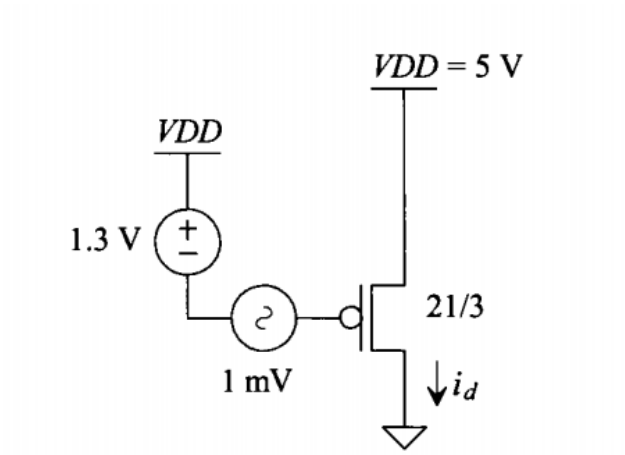


Fig. 4.6

4.7 Estimate the AC, i_d, drain current that flows in the circuit seen in Fig. 4.7. Verify your calculation with SPICE using transient analysis. **KP and Vth can be found in model.txt. Use the P_1u for this problem. The simulation schematic is shown below.



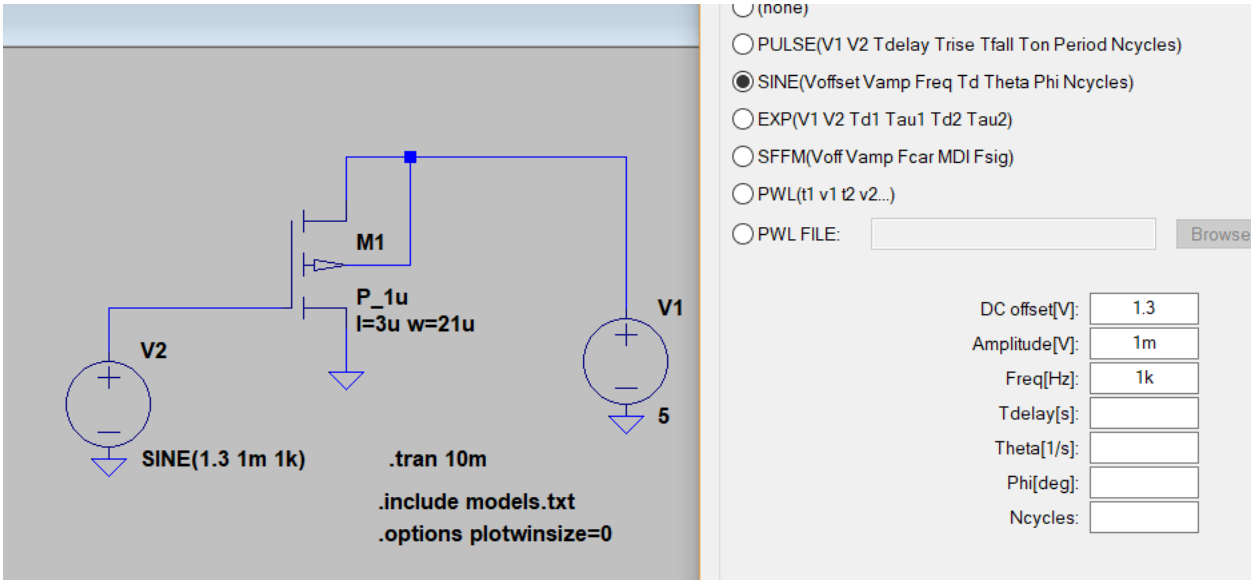


Fig. 4.7

4.8 Calculate the DC operating points (DC voltages/currents) and the AC voltages/currents in all the nodes in the following differential pair. Use LTspice to verify all of them. (Put the values in a table, two columns, one for the calculated values, one for the simulation values).

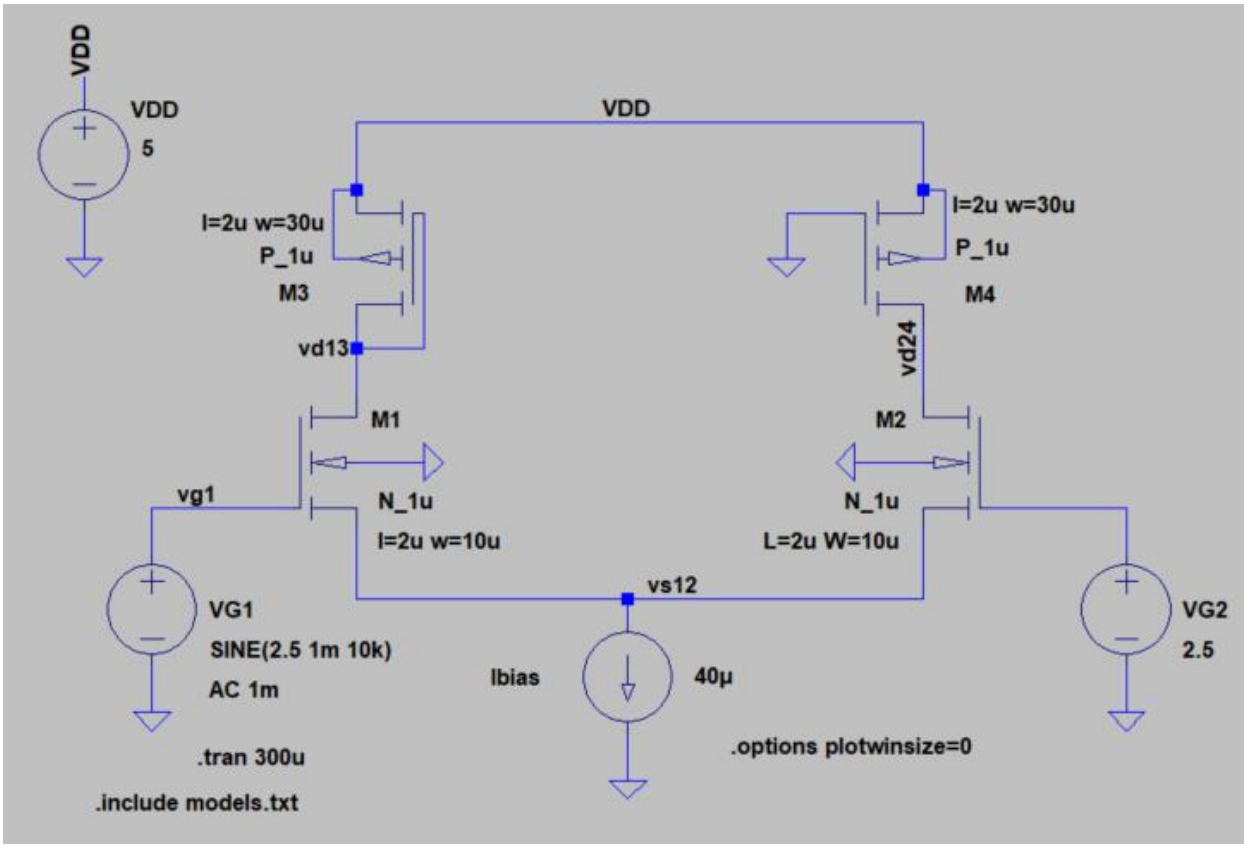


Fig. 4.8

4.9 Using the CMOS long-channel process (N_1u and P_1u), determine the current flowing in the circuit seen in Fig. 4.9. Verify your answer with SPICE. **KP and Vth can be found in model.txt.

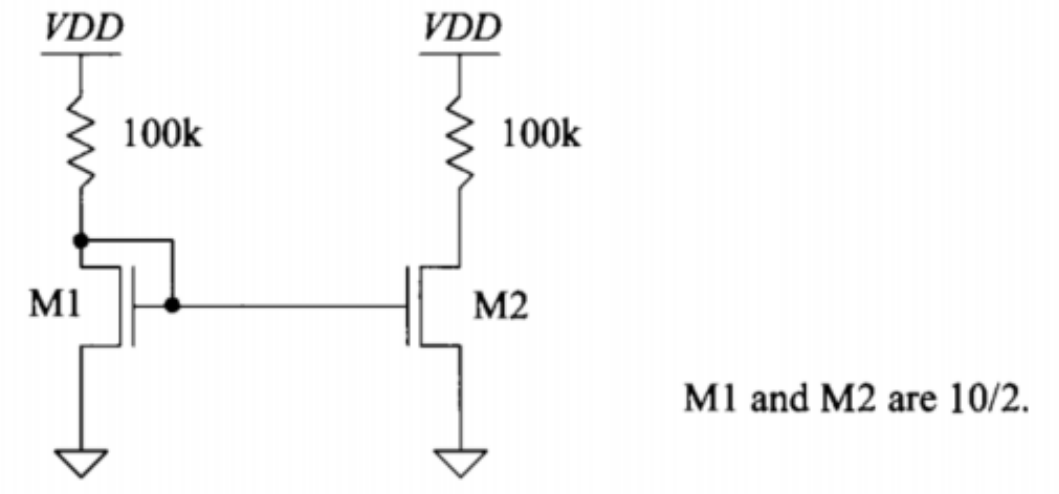
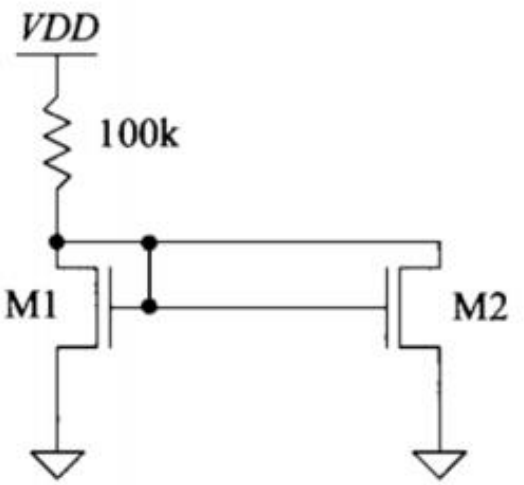


Fig. 4.9

4.10 Repeat 4.9 for the circuit in Fig. 4.10. Can M1 and M2 be replaced with a single MOSFET? If so how and what size? If not why? **KP and Vth can be found in model.txt.



M1 and M2 are 10/2.

Fig. 4.10

Chapter 5 Operational Amplifiers II (Op Amp II)

5.1. Draw the following Biasing circuit + Op Amp in LTSpice (zoom-in to see the parameters), Change the adjustable resistor R1 to 50k, 100k, 150k, 200k, and 300k, discuss the results of the DC operating point of the marked area below (V_{GS}, $V_{DS,sat}$, and I_D). What is the input swing ranges at the input of the differential pair? (Use the C5_models.txt model this time. The threshold is around 0.67V). The model file is available on the online lecture page. The differential pair is only DC biased, there is no AC input signal applied.

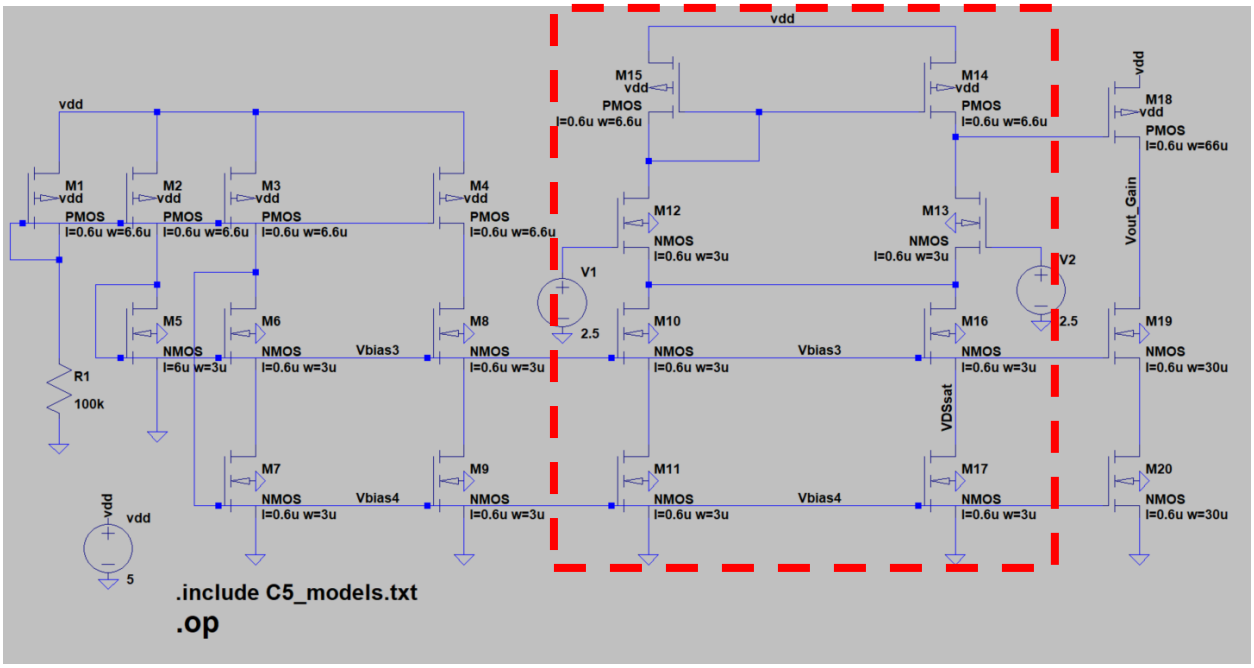


Fig. 5.1

5.2 Derive the small signal gain for the differential pair (Fig 5.2(a)) (CMOS Book P718) and the common-source amplifier (Fig 5.2(b, c)) (CMOS Book P658) in the following figure: (refer to the notes of the lecture).

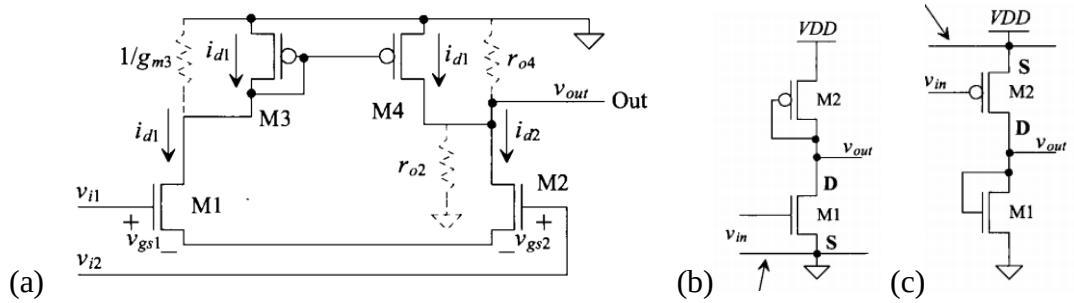


Fig. 5.2

5.3 (1) In Fig. 5.3(a), derive the gain at Vo1 and Vout, draw the gain and phase plot by hand, then verify using LTSpice.

(2) In Fig. 5.3(b), calculate the poles for the input (fin) and output (fout). Then Use LTSpice to plot the '.ac' analysis, label the poles in your figure (use your calculated values to approximate the location and then label in your figure use your pen). (Assume gm1=gm2=150 uA/V, Cgs1=23.3 fF, Cgd1=2 fF, Csg2=70 fF).

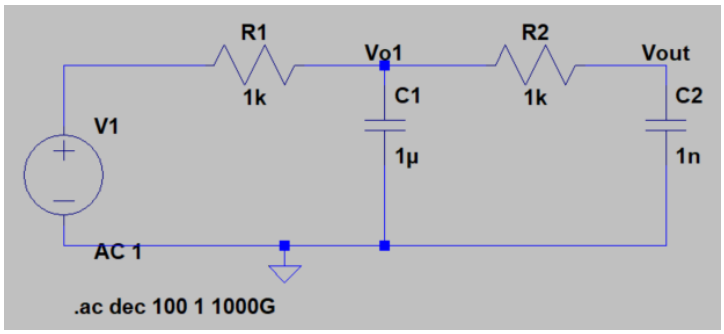


Fig. 5.3(a)

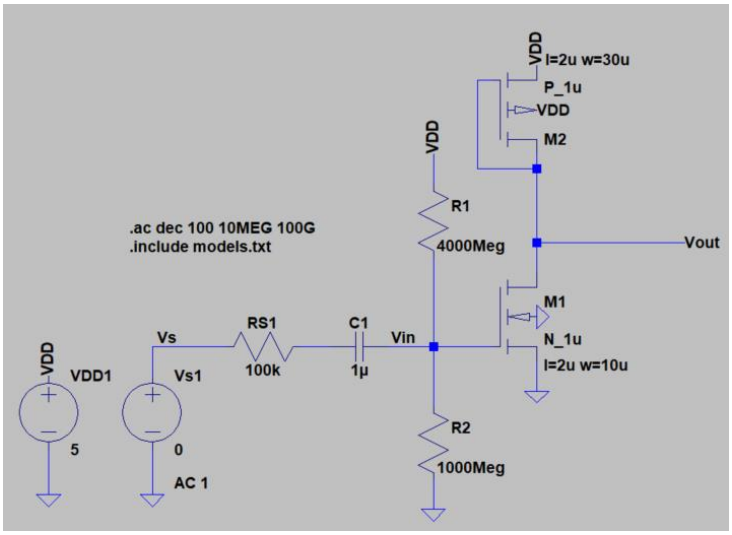


Fig. 5.3(b)

Chapter 6 BJT Transistors

6.1. Summarize the following parameters of an NPN BJT transistor.

IB =

IC =

IE =

gm=

rbe=

vbe=

6.2. Calculate the DC operating point (.op) of the following BJT amplifier, compare with your simulation. The parameters to be calculated and tested: IE, IC, IB, VB, VC, VE, gm, rbe, VCC, VEE. Beta=100. (hint: 1. DC VBE is 0.7 V as the built-in potential of a pn junction. 2. The current source 1 mA is the key to get you started).

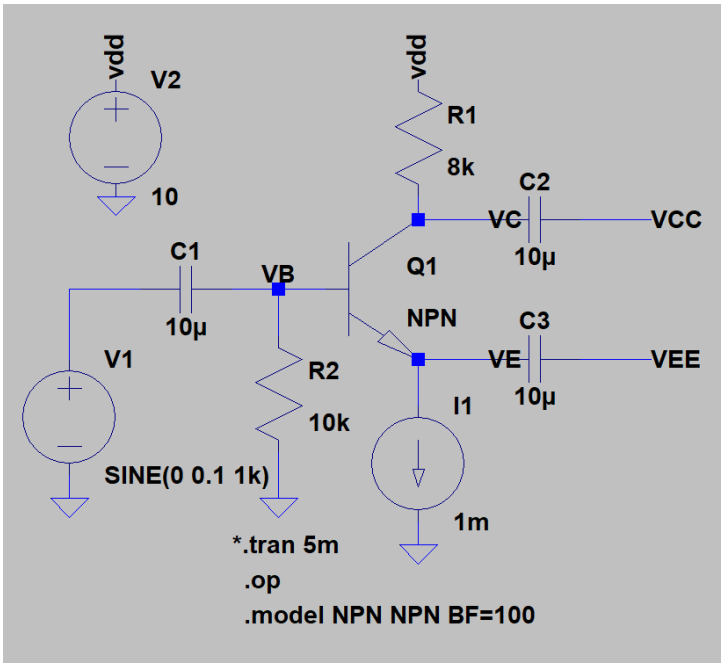


Fig. 6.1

6.3 Given that Beta=100, calculate the DC operating points and the AC gain (no simulation, just calculation). Draw the equivalent AC small signal model.

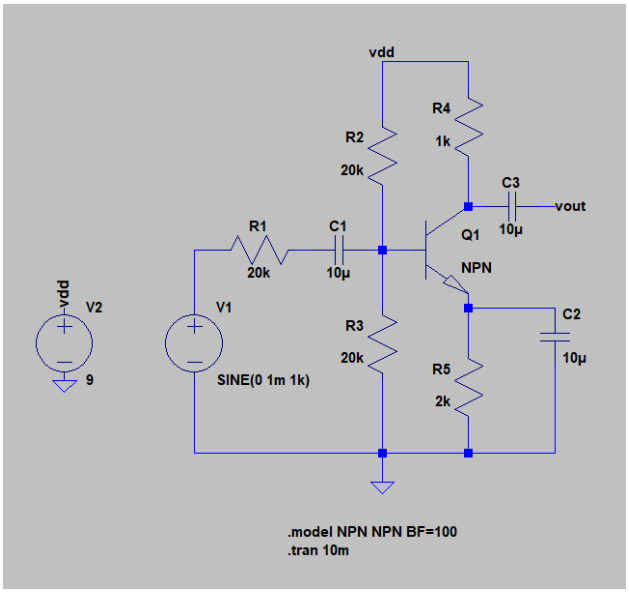


Fig. 6.3

6.4 Q1 has Beta=100, Q2 has Beta=100, VBE=0.7V. (1) Find I_{E1}, I_{E2}, V_{B1}, V_{B2}. (2) If a load resistance R_L=1k ohm is connected to the output terminal, find the voltage gain from the base to the emitter of Q2, v_o/v_{b2}, and find the input resistance R_{ib2} looking into the base of Q2. (3) Replacing Q2 with its input resistance R_{ib2}, analyze the input resistance looking into Q1's base. Calculate v_{e1}/v_{b1}. (4) Find the overall voltage gain v_o/v_{b1}. (make sure you can distinguish AC and DC notations). Vdd is 5 V, don't do simulation for this in LTSpice.

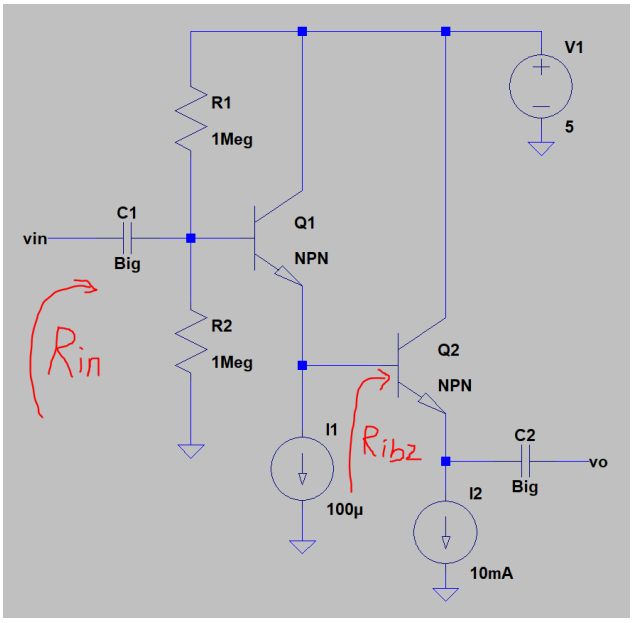


Fig. 6.4