Birla Institute of Technology and Science Pilani, Pilani Campus (Raj.)
Department of Electrical and Electronics Engineering
EEE/INSTR F244 Microelectronic Circuits
Second Semester 2019-20

Weightage: 15 marks **SPICE Assignment** **Date: 09/03/2020**

Instructions for submission

1. Students can use LTSPICE simulator for solving the problems.
2. A final hand written report has to be submitted to respective tutorial instructors.
3. Use only A4 sheets with one-inch margin on each side. You can write on both sides of a sheet.
4. On page-1 of your report, please mention your name, ID No. and Tut Section at the top,
5. Also mention your Name and ID No. on the top right corner of each sheet.
6. Your assignments shall be collected on *03rd April 2020* as per schedule to be updated soon.
7. A penalty of FIVE marks per day shall be imposed for late submissions.
8. Neatly sketch and label all the plots. Proper units have to be mentioned for all the quantities.

Problem-1 (DC Analysis and parametric sweep): For the two port network shown in figure below, obtain the Z-parameters, ABCD (transmission) parameters and h (hybrid) parameters. Cross check your results obtained from simulation against hand calculations. What should be the value of load resistance connected at Port-2 for maximum power transfer when a voltage source of 25V is connected at Port-1.
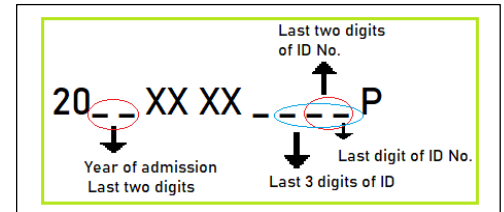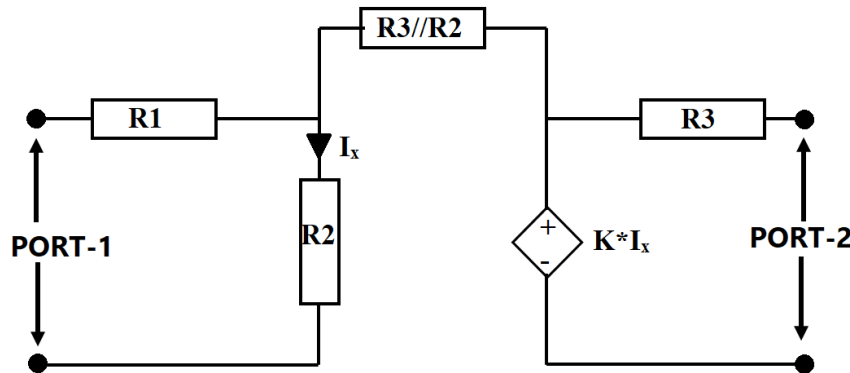

S.No.	Resistor Name	Value in kΩ
1.	R1	(Last digit of your ID + 3) multiplied by your tut section no.
2.	R2	Product of last two digits of your ID + tut section no.
3.	R3	(Sum of last three digits of your ID % 8)*Tut Section no.
4.	K	Last two digits of your year of admission – Tut section number

In your submission, redraw the circuit showing the values of each resistor and CCVS. Include the SPICE code, hand calculations. At the last present a table comparing the hand calculation Vs SPICE results.

Problem-2 (Q Point, Bode Plot, Transient Analysis): A schematic of common base amplifier (CB) is shown in the figure below. Obtain the DC Operating point, plot the Bode plot for voltage gain (A_V) and current gain (A_I). Also observe the output waveform for a 1KHz 10mV sine wave as excitation. Also calculate the input impedance (R_{in}) and output impedance (R_{out}) of the amplifier.
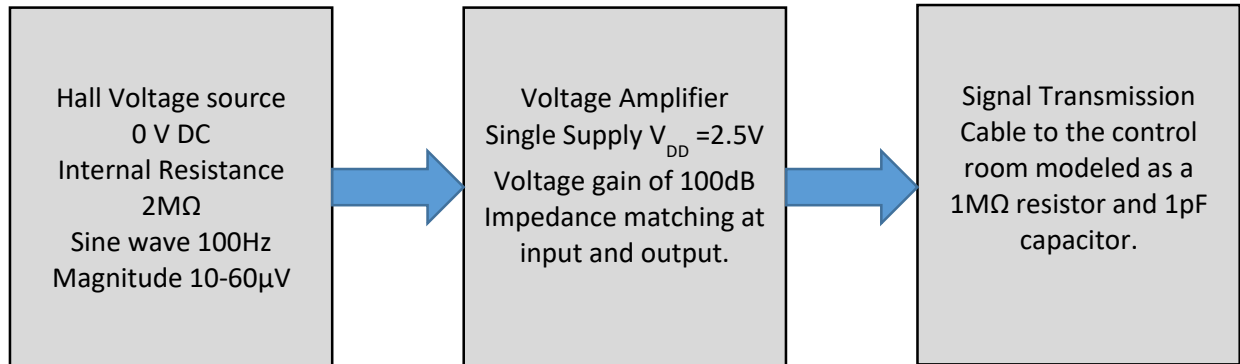
- **a.** What is the impact of R3 on A_V, A_I, R_{in} and R_{out}?
- **b.** What is the impact of RE on A_V, A_I, R_{in} and R_{out}?
- **c.** What is the impact of Bias voltage V_{EB} on A_V, A_I, R_{in} and R_{out}?
- **d.** Design the circuit to achieve the target specifications (as per the table) to be achieved by choosing proper values for Resistors and Bias Voltage.
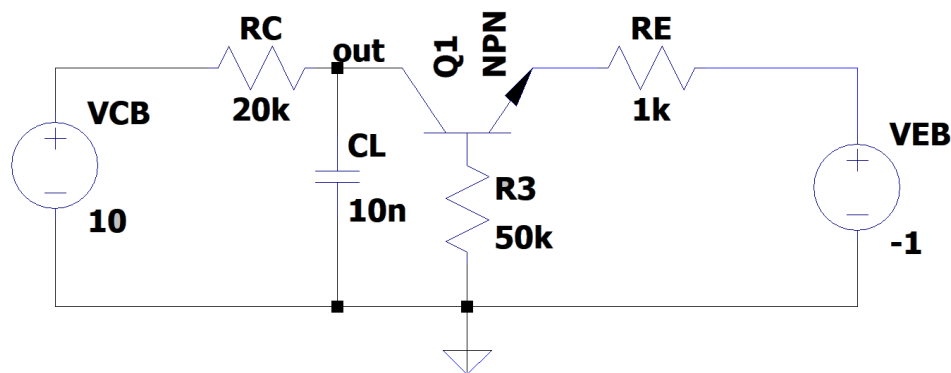
S.No.	Specification	Value in kΩ
1.	Input Resistance in kΩ	Last digit of your ID + Tut section number.
2.	Voltage gain in dB	Last two digits of your year of admission + Tut Section number
3.	3 dB Band width in kHz	(Sum of last four digits in your ID % 12) * Tut section number.
4.	BJT Model to be used	Single degree A3 : BC107A , A8 : BC108A Dual degree A3 : BC107B , A8 : BC108B

In your submission: Include the SPICE code. For part-a, b & c show the variations in form of graphs. You may use Excel spread sheets to generate these graphs. For part-d re-draw the circuit diagram which satisfies the given specifications. List the specifications and what you have achieved in the simulation in tabular form below the circuit.

Problem-3 (Design and Analysis): A MOSFET based circuit has to be designed for providing suitable signal amplification to a signal obtained from a Hall voltage sensor. The range of voltage obtained from the sensor is in the order of 10-60µV. The signal can be modeled as a low frequency (<100Hz) sinusoidal wave. The Thevinin resistance of the Hall sensor is approximately 2MΩ. The amplified signal has to be routed to a control room situated at a remote distance via a long transmission line. The transmission line can be modeled as 1MΩ resistor with a 1pF capacitance seen to ground.

Hall Voltage source 0 V DC Internal Resistance 2MΩ Sine wave 100Hz Magnitude 10-60µV	Voltage Amplifier Single Supply V_{DD} =2.5V Voltage gain of 100dB Impedance matching at input and output.	Signal Transmission Cable to the control room modeled as a 1MΩ resistor and 1pF capacitor.

The following are the restrictions on your circuit design:

1. Single rail power supply 2.5V.
2. Maximum power dissipation 1mW.
3. You can't use resistors anywhere in the circuit except for R_{source} and R_{load}.
4. One stable current source of 10µA is available.

Use the 500nm Library file provided. L_{min} is 600nm.

In your submission: Include the SPICE code. Show all the design calculations. Final circuit along with W/L values and R, C values. Report the gain (dB), 3-dB frequency, transient output, power from the SPICE simulation.

*** The End ***

Note: You need not include any screenshots or printouts for any part of this assignment report.

-Compiled by
Mr. K. Babu Ravi Teja
Assistant Professor, EEE Department
BITS Pilani, Pilani Campus
e-mail: baburaviteja.k @pilani.bits-pilani.ac.in