

ES 330: Electronics II

Laboratory Experiment No. 1**Title:** Design of a Multistage Amplifier using BJT/MOSFET

Objective: The main objectives of this lab session will be

- 1) Learn how to bias a BJT/MOSFET for amplifier application
- 2) Design, analyze and implement a multistage amplifier using BJT/MOSFET

N. B. This lab experiment will be a part of your **project 1**. SO YOU MUST SAVE YOUR DESIGN.

Introduction:

Transistors are the core devices for developing any electronics circuits. They can work either as an amplifier or a switch depending upon the bias condition. Switch mode application of transistor is usually used for logic gate design. In analog circuit design part, transistors are mainly used for linear and non-linear circuit design such as amplifier, filter, mixer, oscillators, power amplifiers etc. There are mainly two types of transistors that we use

- 1) Bipolar Junction Transistor (BJT)
- 2) Metal Oxide Semiconductor Field Effect Transistor (MOSFET)

BJT is a current driven device whereas MOSFET is voltage driven device. So the circuit operation employing BJT or MOSFET is little bit different but their main functionality for linear or non-linear operations are pretty much same. In this session we will learn how to bias a transistor for both BJT and MOSFET. We will study different parameters like input resistance, output resistance, gain, frequency response etc of both single and multi-stage amplifier. The multi-stage amplifier will be required for your project work.

DC Biasing:

Among the various types of biasing schemes, self-bias scheme with emitter or source resistance is the most suitable one for amplifier application. Here we will design a self-bias circuit with emitter or source resistance for a BJT or MOSFET, respectively.

For BJT, follow the steps as below

1. Set the Q-point value of your amplifier. This will give the information about output voltage (V_{ce}) and output current (I_c).
2. Set the supply voltage (V_{cc}) and find out the remaining voltage ($=V_{cc} - V_{ce}$) for two output resistances (R_c and R_e).
3. Divide the remaining voltage equally between R_c (V_{RC}) and R_e (V_{RE}).
4. Now find collector terminal resistance $R_c = V_{RC}/I_c$.

5. Find emitter terminal resistance $R_e = \alpha V_{RE}/I_c$. Here $\alpha = \beta/(\beta+1)$. The gain of the BJT device is β . You can get the value of β from the data sheet.
6. Find base current $I_b = I_c/\beta$
7. Assume $I_1 = 10I_b$ to simply the design and at the same time to reduce bias current consumption.
8. Voltage at the base terminal can be found as $V_B = V_{RE} + |V_{BE}|$. The value of $|V_{BE}| = 0.7$ for the active region of operation.
9. Now find the value of $R_1 = (V_{CC} - V_B)/I_1$.
10. The value of R_2 will be $R_2 = V_B/(I_1 - I_b)$.

For MOSFET, follow the steps as follows

1. Set the Q-point value of your amplifier. This will give the information about output voltage (V_{ds}) and output current (I_d).
2. Set the supply voltage (V_{dd}) and find out the remaining voltage ($=V_{dd} - V_{ds}$) for two output resistances (R_d and R_s).
3. Divide the remaining voltage equally between R_d (V_{RD}) and R_s (V_{RS}).
4. Now find drain terminal resistance and source terminal resistance by $R_d = R_s = V_{RD}/I_d$
5. Assume $I_1 = I_d/5$ to simply the design and at the same time to reduce bias current consumption.
6. Voltage at the gate terminal can be found as $V_G = V_{RS} + V_{GS}$. Here $V_{GS} = V_T + 250\text{mV}$
7. Now find the value of $R_1 = (V_{dd} - V_G)/I_1$
8. The value of R_2 will be $R_2 = V_G/I_1$

Design the bias circuit and simulate (using PSPICE or Multisim) the DC operating voltages. Check the result with your calculation.

Single Stage Amplifier Design:

In this section we will develop a single stage amplifier. Draw the circuit amplifier circuit with coupling capacitors (coupling capacitors are needed to confine the dc bias voltage within the bias circuit). Simulate the input resistance, output resistance, gain, and frequency response of the single stage amplifier.

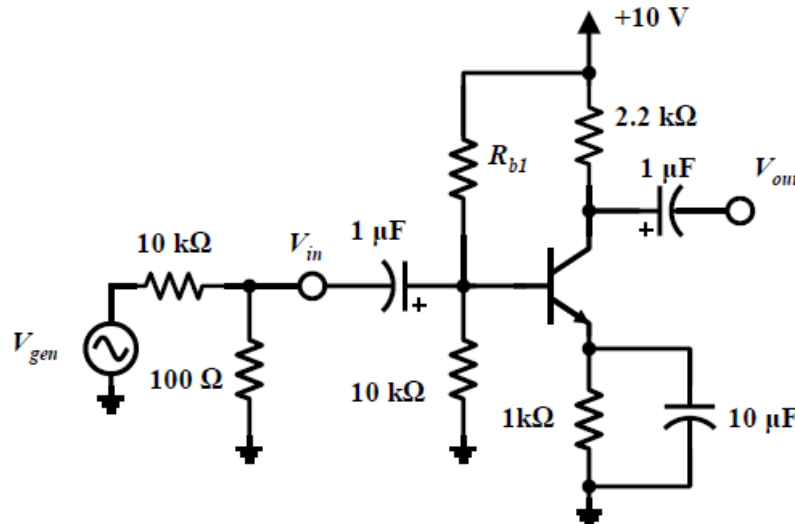


Fig 1: Circuit schematic of a CE amplifier

For the simulation, PSICE will be used to simulate the following defining parameters of our amplifier.

Simulation of Input Resistance

Draw the circuit as shown in the figure. Remove any output load resistance. Now apply a known (1V) voltage source at the input. You will simply use an AC voltage source at the input and do the AC analysis of your circuit.

Simulation of Gain and Overall Gain

With the same setup perform the AC analysis and find the ratio of output voltage and input voltage.

Simulation of Frequency Response

You can also see the frequency response of the amplifier with the same AC analysis.

Simulation of Output Resistance

Draw the circuit as shown in the figure. Deactivate the input signal source. Now apply a known (1V) voltage source at the output. You will simply use an AC voltage source at the output and do the AC analysis of your circuit.

Multistage Amplifier Design:

Draw the circuit as shown in Figure 2 and simulate input resistance, output resistance, amplifier gain, overall gain and frequency response.

Construct the circuit in circuit Figure 2. Measure and record the mid-band AC voltage gain with a generator signal of 0.1V sinusoid at 10 KHz. It may be difficult to see V_{in} on your oscilloscope because of the large step-down ratio of the voltage divider at the input (nominally a divide-by-1000). If so, increase the input signal briefly to establish the

actual voltage division ratio, then use this information to calculate the for a 0.1V generator input.

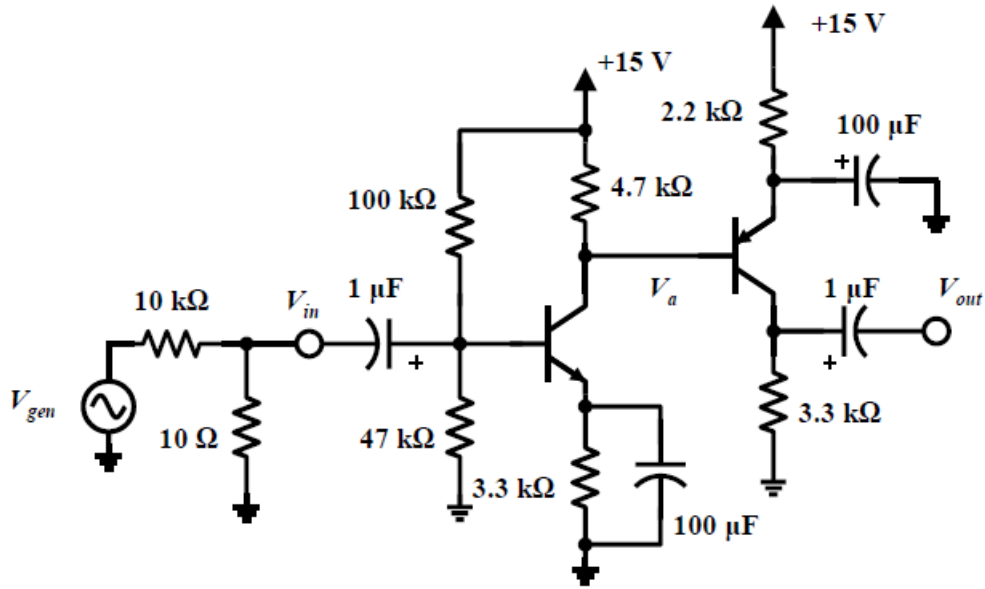


Fig 2: Multistage Amplifier

Compare the measurement result with the simulated one.