ELECENG 2EI4: Electronic Devices and Circuits 1

Design Project 3

Gurleen Kaur Rahi

MAC ID: rahig

400377038

Table of Contents

Q1. Design Schematic

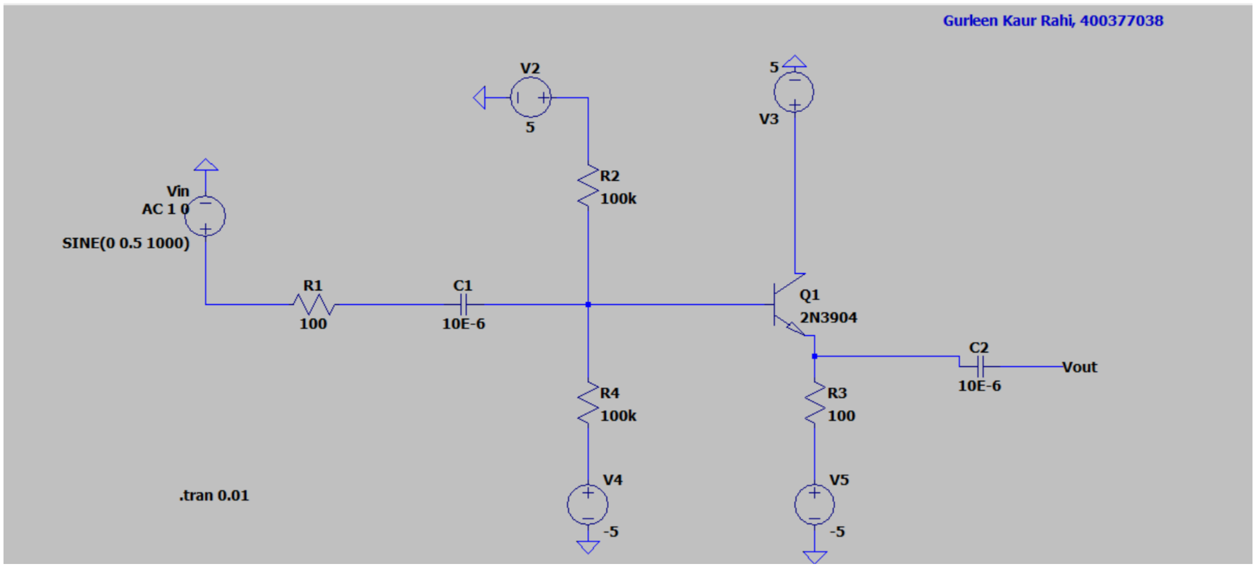

Figure 1: Complete Schematic of the design

a. What type of transistor did you choose (MOSFET/BJT)? Why?

I selected a BJT for this design. This is so that BJTs can be superior for applications requiring less current due to their generally better linearity. BJTs can provide the greater transconductance value that is required for this design. Using BJTs will also make circuit analysis simpler and compared to mosfets, BJTs are relatively faster as there is low capacitance on the control pin.

b. What amplifier topology (CE/CS/CD/etc.) did you choose? Why?

The common collector amplifier was selected as the circuit topology. The common collector topology was chosen because, due to the fact that there is almost no voltage gain, it would be simpler to observe linearity between the input and the output and because the problem's specifications generally do not favor one amplifier circuit topology over another.

c. What calculations did you use to determine the required component values?

$$V_{in} = 0.2V_T(1+g_m R_L) \geqslant 0.5$$

$V_T = 0.025$

$R_L = 100\,\Omega$

$$V_{in} = 0.2(0.025)(1+100g_m) \geqslant 0.5$$

$$0.005(1+100g_m) \geqslant 0.5$$

$$1+100g_m \geqslant 100$$

$$100g_m \geqslant 99$$

$$g_m \geqslant 0.99\,S$$

$$\Rightarrow \boxed{g_m \geqslant 990\,mS}$$

$$\frac{g_m R_L}{g_m R_L + 1} \cdot \frac{R_{in}}{R_S + R_{in}} \geq 0.9$$

$$R_L = 100\,\Omega \quad g_m = 0.99$$

$$\frac{99}{1+99} \cdot \frac{R_{in}}{R_{in} + 0.1} \geq 0.9$$

$$0.99 \cdot \frac{R_{in}}{0.1 + R_{in}} \geq 0.9$$

$$\frac{R_{in}}{R_{in} + 0.1} \geq 0.91$$

$$R_{in} \geq 0.9 R_{in} + 0.09$$

$$0.091 R_{in} \geq 0.091$$

$$R_{in} \geq 1000\,\Omega$$

$$R_L = 100\,\Omega = 0.1\,k\Omega$$

$$r_\pi = \frac{\beta}{g_m} = \frac{100}{990} = 0.10\,k\Omega$$

$$R_{in}' = r_\pi\left(1 + g_m R_L\right)$$

$$= 0.101\,(1 + 990)$$

$$= 10.1\,k\Omega$$

$$R_{in} = R_1 \parallel R_2 \parallel R_{in}'$$

$$1k = \frac{1}{1/R_1 + 1/R_2 + 1/10.1}$$

$$1k = 1/R_1 + 1/R_2 + 1/10.1$$

$$1/R_1 + 1/R_2 = 0.90099$$

$$\frac{R_1 + R_2}{R_1 R_2} = 0.90099$$

$$\boxed{R_1 \parallel R_2 \geq 1.11\,k\Omega}$$

Figure 2: Calculations required to determine the required component values

Q2. Simulations

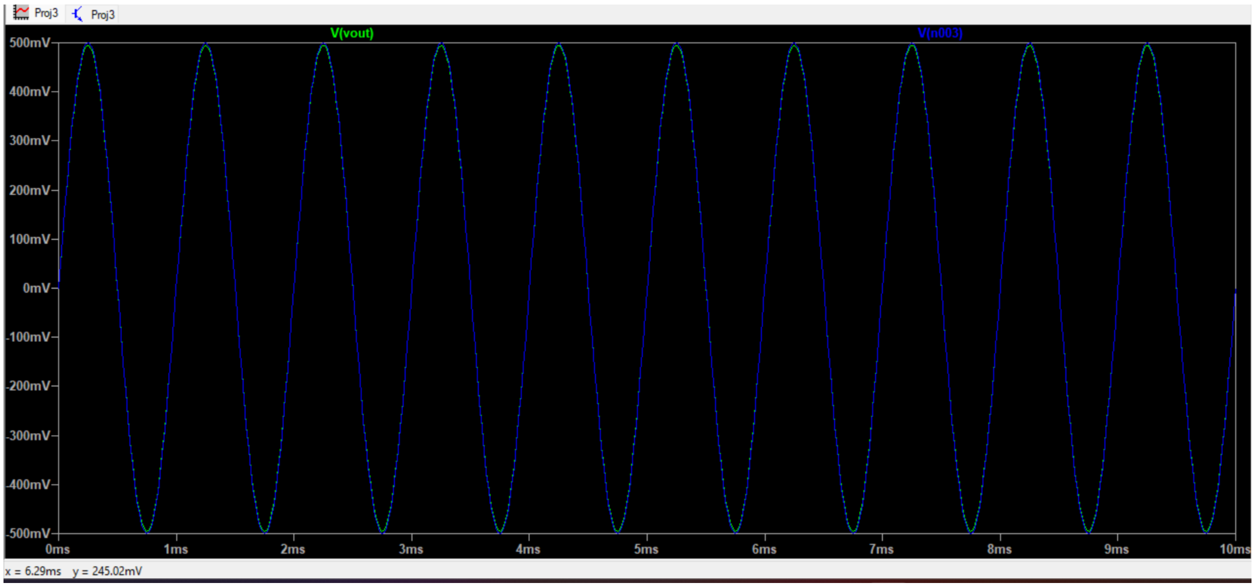

Figure 3: LTSpice simulation for input and output voltage

a. How did you model the transistor in the simulator you used?

To model the NPN BJT 2N3904, I used LTSpice software. I made the schematic diagram using that BJT. This BJT can be found in our kits so I used this transistor to model the simulation.

b. What settings did you use for each simulation (transient/dc sweep/ frequency sweep/etc.)?

For each simulation, I used a transient analysis with a stop time of 0.01 ms. For the input voltage, I have used a sinusoidal function with a frequency of 1KHz followed by a small signal analysis of AC amplitude of 1V and phase of 0V. Overall, I have used the transient analysis for both voltages.

c. Explain the overall gain determined from the simulations.

$$V_{in} = 500mV$$

$$V_{out} = 496\,mV$$

$$A_v = \frac{V_{out}}{V_{in}} = \frac{496\,mV}{500mV} = 0.992$$

$$\therefore \boxed{A_v = 0.992}$$

Figure 4: Calculation of overall gain from the simulations

Therefore, the figure above shows that the gain from the simulation is equal to 0.992 which is very close to the expected value of the gain which is 1.

d. Explain the other performance parameters determined from the simulations.

$$R_{in} = \frac{V_{in}}{I_{in}} = \frac{0.5V}{26\mu A} = \frac{0.5V}{0.026} = \boxed{19.23k\Omega}$$

Measured from the Simulation

Figure 5: Calculation of input resistance

Q3. Physical Circuit

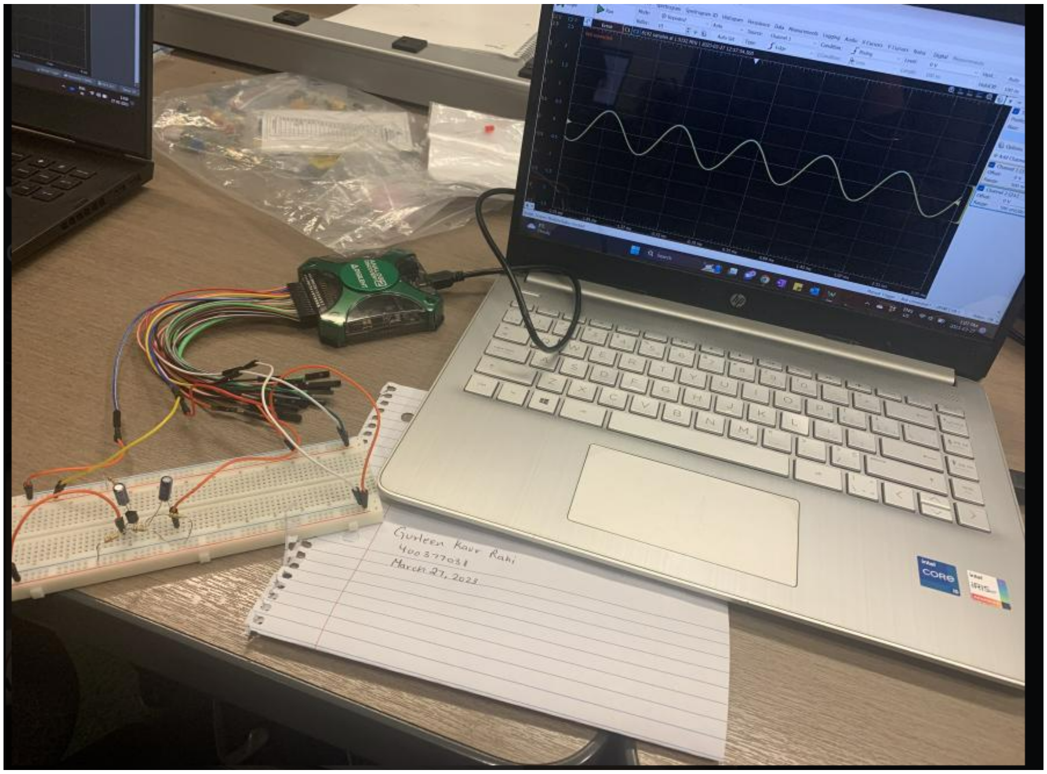

Figure 6: Physical circuit and waveform simulation

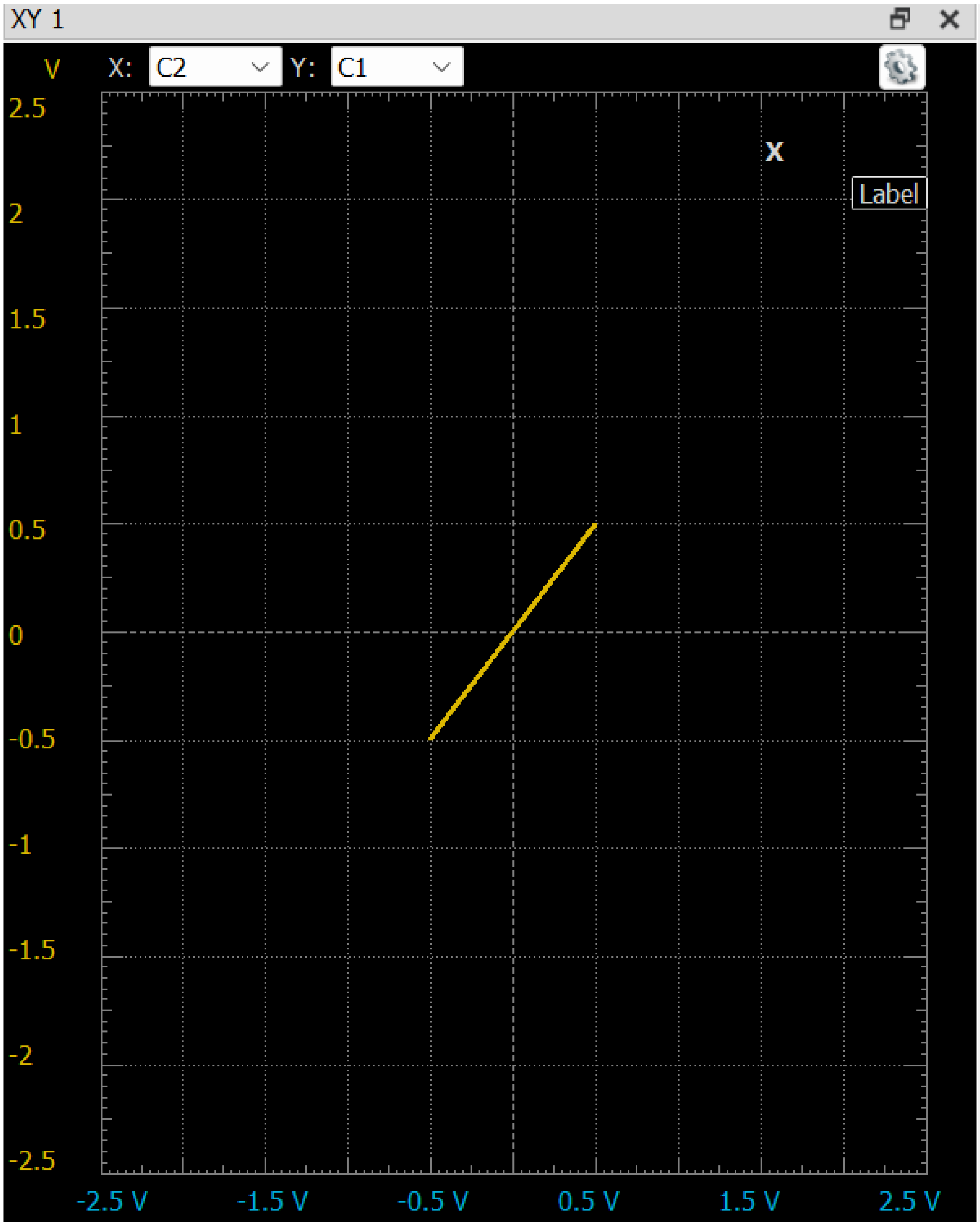

Figure 7: Linearity of the physical circuit

Q4. Waveform Measurements

a. What measurements did you do? (E.g. waveforms as functions of time, XY plot, network analyzer, etc. You decide which measurements are beneficial and explain why you used them.)

For this circuit, I employed time-varying waveforms. I examined the peak-to-peak voltages to see if the circuit met the criteria of less than 10% attenuation. Because the source voltage was 0.5V and the output voltage was also 0.5V and this criterion was met.

b. How did you determine the midband gain for your amplifier? What value did you obtain? Compare your value with what was expected from calculations and simulations.

I calculated the midband gain for this amplifier by dividing the output voltage amplitude by the input voltage amplitude. This was accomplished with the oscilloscope and a custom math function. The function, as seen in Figure 7 after calculating the slope, maintains close to a value of 1V, which is extremely close to the simulated value of 1V. Both of them are close to the required value of 1.

c. Demonstrate linearity at an input amplitude of 0.5V.

To show the linear nature of the waves, I used a spectrum analyzer. This also showed the similarity of both waves.

Figure 8: Spectrum analyzer to show the linearity of the circuit