Jacobs University Bremen

Natural Science Laboratory
Electronics Lab

Spring Semester 2019

Lab Experiment 3 – Bipolar Junction Transistor

Hayoung Kim
Mailbox No: 578

Experiment conducted by	: Hayoung Kim, Mohammed Mahfoud
Place of execution	: Research 1 EE Lab 54
Date of execution	: April. 01, 2019

1 Introduction - Prelab

1.1 Problem 1 : Biasing of Bipolar Junction Transistors

(1) (a) Calculate V_B, V_E, V_{CE} and V_C. (b) Calculate I_B, I_E and I_C.

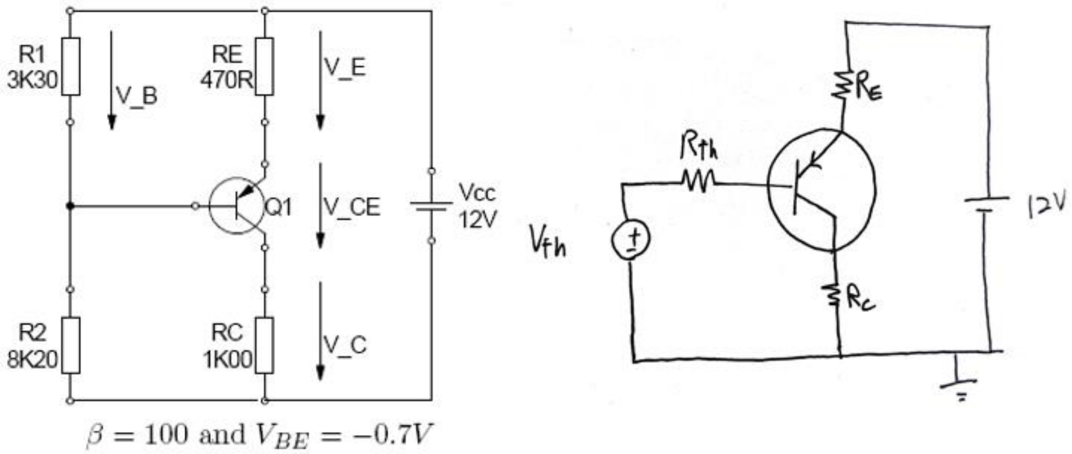


Figure 1 (left) BJT circuit (right) Equivalent circuit with Thevenin voltage

$$I_C = \beta I_B \quad ; \qquad I_E = I_C + I_B$$
$$V_{th} = V_{cc}\frac{R_{B2}}{R_{B1}+R_{B2}} = 12\cdot\frac{8200}{3300+8200} = 8.556\,V$$
$$R_{th} = \frac{R_{B1}R_{B2}}{R_{B1}+R_{B2}} = \frac{3300\cdot 8200}{3300+8200} = 2353\Omega$$

$$V_{th} + I_B R_{th} + V_{EB} + I_E R_E - V_{CC} = 0$$
$$V_{CC} - V_{th} + V_{BE} = I_B(R_{th} + (\beta+1)R_E)$$
$$\boldsymbol{I_B} = \frac{V_{CC} - V_{th} + V_{BE}}{R_{th} + (\beta+1)R_E} = \frac{12 - 8.556 - 0.7}{2353 + 101\cdot 470} = 0.00005507A = \ 55.075\mu A$$
$$\boldsymbol{I_C} = \beta I_B = 100\cdot 0.000055075 = 0.0055075A = 5.5075mA$$
$$\boldsymbol{I_E} = I_B + I_C = 0.000055075 + 0.0055075 = 0.005562575 = 5.5626mA$$

$$\boldsymbol{V_C} = R_C\cdot I_C = 1000\cdot 0.0055075 = 5.507V$$
$$\boldsymbol{V_E} = V_{CC} - R_E\cdot I_E = 470\cdot 0.005562575 = 2.614V$$
$$\boldsymbol{V_{CE}} = V_{CC} - V_E - V_C = 12 - 2.614 - 5.507 = 3.879V$$
$$\boldsymbol{V_B} = V_E - V_{BE} = 2.614 + 0.7 = 3.314V$$

(2) Use the same circuit. Calculate the necessary biasing resistors (R_1, R_2, R_C, and R_E) under the assumption that $V_{CEQ} = 8V$ and $I_{CQ} = 8mA$. The transistor operates in the active mode.
Other parameters: $V_{CC} = 20V, \beta = 150, V_E = 4V, R_{th} = 0.1\beta R_E$.

$$\boldsymbol{R_C} = \frac{V_C}{I_C} = \frac{V_{CC} - V_E - V_{CEQ}}{I_{CQ}} = \frac{20 - 4 - 8}{0.008} = 1000\Omega$$
$$I_B = \frac{I_C}{\beta} = \frac{0.008}{150}0.000053333A$$
$$\boldsymbol{R_E} = \frac{V_E}{I_C + I_B} = \frac{4}{0.008 + \frac{0.008}{150}} = 496.689\Omega$$
$$R_{th} = 0.1\cdot\beta\cdot R_E = 0.1\cdot 150\cdot 496.689 = 7450.335\Omega$$

$$V_{th} = V_{CC} - V_E + V_{BE} - I_B R_{th} = 20 - 4 - 0.7 - 0.000053333 \cdot 7450.335$$
$$V_{th} = 14.9026V$$

$$V_{th} = V_{cc}\frac{R_{B2}}{R_{B1}+R_{B2}} \qquad R_{th} = \frac{R_{B1}R_{B2}}{R_{B1}+R_{B2}}$$

Using equation from above

$$\boldsymbol{R_{B_1}} = R_{th} \cdot \frac{V_{cc}}{V_{th}} = 7450.335 \cdot \frac{20}{14.9026} = 9998.7049\Omega = 9.9987\text{k}\Omega$$
$$\boldsymbol{R_{B_2}} = \frac{R_{th}V_{CC}}{V_{CC} - V_{th}} = \frac{7450.335 \cdot 20}{20 - 14.9026} = 29231.90\Omega = 29.232\text{k}\Omega$$

1.2 Problem 2 : Constant Current Source

(1) Given is following constant current circuit:

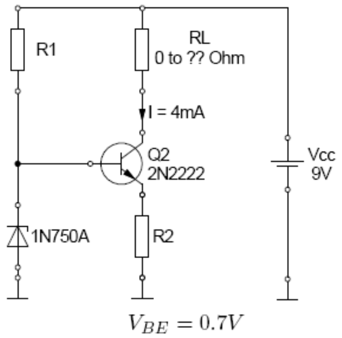


Figure 2 Constant current circuit

(2) Find the values for R1 and R2 to get a constant current of $I_C \approx 4mA$?
Hint: The 1N750A diode is a Zener diode! See data sheet for electrical Properties

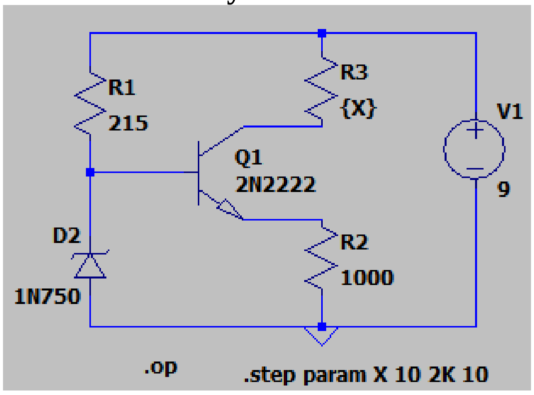
1N750A diode : 4.7V , Assume $I_1 = 20mA$

$$R_1 = \frac{V_{CC} - V_Z}{I_1} = \frac{9 - 4.7}{0.020} = 215\Omega$$
$$R_2 = \frac{V_E}{I_E} = \frac{V_Z - V_{BE}}{I_C} = \frac{4.7 - 0.7}{0.004} = 1000\Omega$$

(3) What is the maximum value for R_L to still get $I_C \approx 4mA$??

$$R_L = \frac{V_C}{I_C} = \frac{V_{CC} - V_{CE} - V_E}{I_C} = \frac{9 - 0.7 - 4.0}{0.004} = 1075\Omega$$

(4) Implement the circuit in LTSpice and verify your calculations! Use the .step command to vary RL.



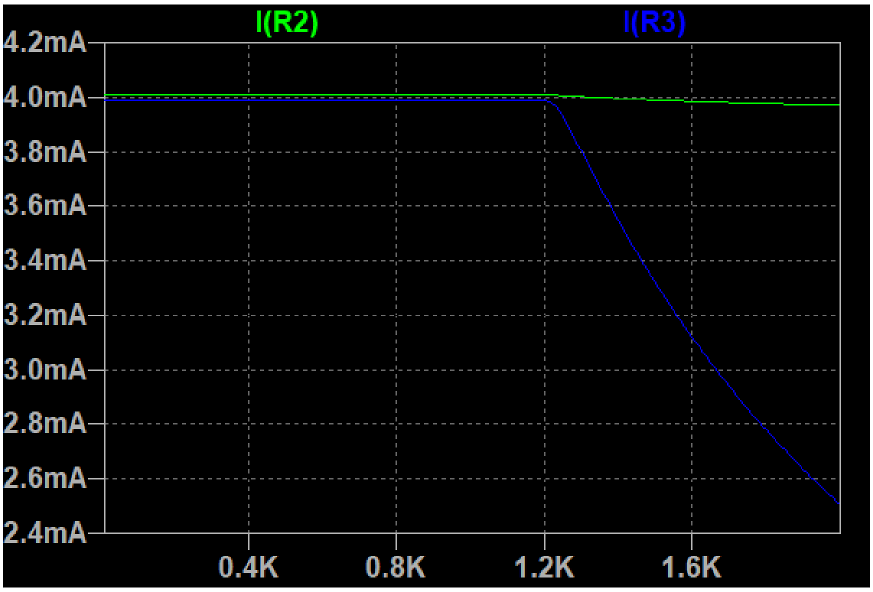


Figure 3 Current I(R3) depending on the load resistor

The current remains at 4.0mA till R_L reaches 1.2kΩ

(5) Explain the function principle of the circuit!

The circuit keeps the current I_C constant, therefore can be used as a constant current source. This feature is obtained because of the Zener diode which keeps the voltage of the emitter constant.

1.3 Problem 3 : Amplifier circuit

(1) Use 'LTSpice IV' to implement the following circuit:

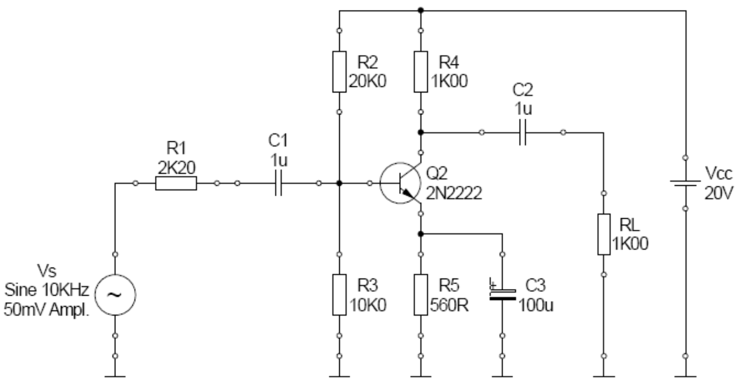


Figure 4 Amplifier circuit

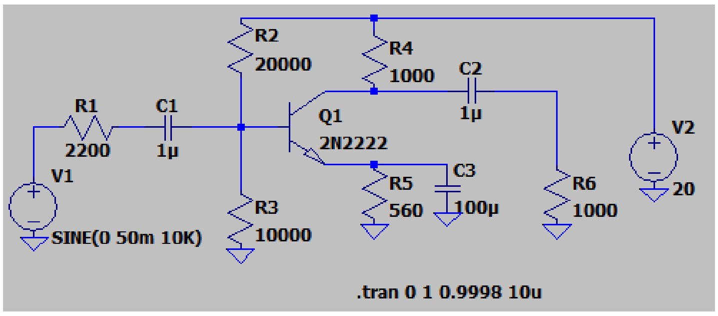


Figure 5 LT spice circuit diagram

(2) Determine the DC operation point values for $V_B, V_C, V_{CE}, V_E, I_C$ *and* I_B.

$$\begin{aligned} V_B &= 6.3354\ V \\ V_C &= 10.0169\ V \\ V_{CE} &= 4.39852\ V \\ V_E &= 5.61838\ V \end{aligned} \qquad \begin{aligned} I_C &= 0.00998314\ A \\ I_B &= 4.96897 \cdot 10^{-5} \end{aligned}$$

(3) Perform a transient analysis for about 2 cycles of a sinusoidal input signal. The input signal V_s *exhibits a frequency of 10KHz and an amplitude 50mV peak. Display the voltage at the base and across the load resistance* R_L*. Determine the voltage gain* V_C/V_B *and* V_C/V_S.

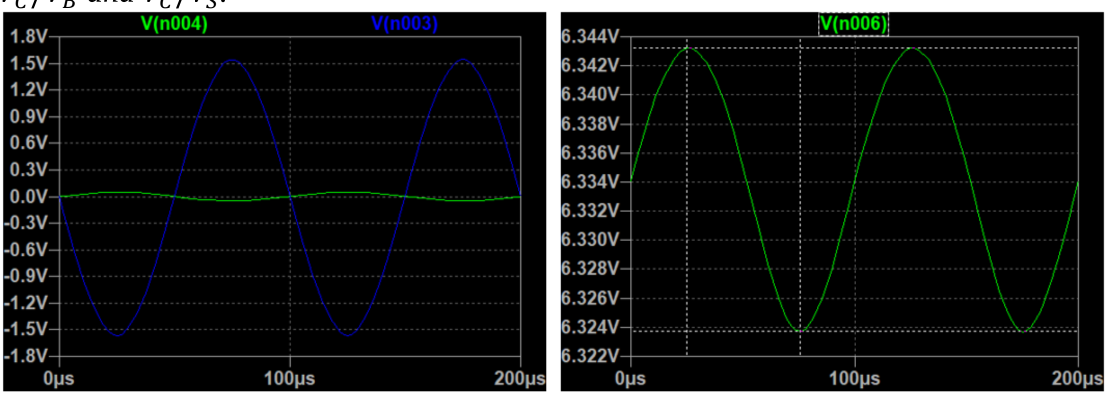


Figure 6 (left) Green: V_S, Blue: V_L (Right) V_B

V_L : Voltage across the load resistor

V_S: Voltage across the source

$$V_L = 1.573V$$
$$V_B = \frac{19.48}{2}\text{mV} = 9.77\text{mV} = 0.00977\text{V}$$
$$V_S = 50mV = 0.050V$$

$$\frac{V_L}{V_B} = \frac{1.573}{0.00977} = 161.00$$
$$\frac{V_L}{V_S} = \frac{1.573}{0.050} = 31.46$$

(4) Perform an AC analysis. Keep the amplitude of the input signal constant at 50mV .Vary the frequency from 100Hz to 1MHz with 10 points per decade and display the voltage across the load resistance RL.

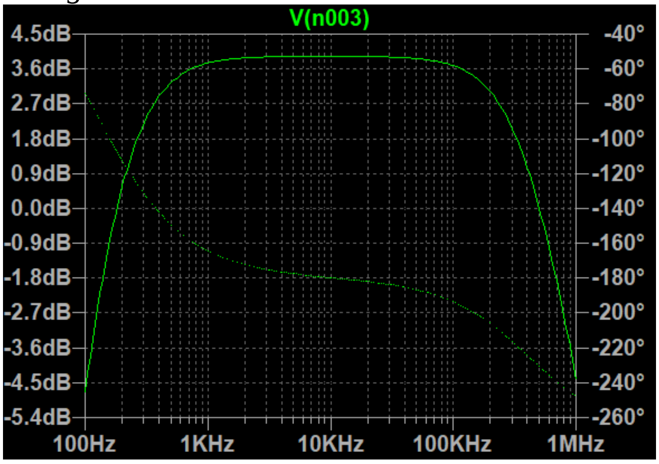


Figure 7 AC analysis

(5) Use the LTSpice '.MEASURE' command (see help file and example 'MeasureBW.asc') to determine the lower and upper -3dB frequencies and the bandwidth.

Added
.MEAS AC tmp max mag(V(n003));
.MEAS AC BW trig mag(V(n003))=tmp/sqrt(2) rise=1 targ mag(V(n003))=tmp/sqrt(2) fall=last

Result:
```
tmp: MAX(mag(v(n003)))=(3.9266dB,0°) FROM 100 TO 1e+006
bw=414205 FROM 215.786 TO 414421
```

Bandwidth: 414205 Hz
Lower cut-off frequency: 215.786Hz
Upper cut-off frequency: 414421Hz

2 Execution, Evaluation

2.1 Experiment Setup

Used tools and instruments:

- Breadboard, Tools box from workbench
- Oscilloscope Tektronix, Function Generator

2.1.1 Experiment Part 1 : Determine Type and Pin Assignment of BJT – Setup

- Objective

Understand Bipolar Junction Transistor (BJT) and their terminals

2.1.2 Experiment Part 1 – Execution and Results

- Description of the measurement procedure.

Find the base, emitter, and collector terminal by diode checking across the pins of the transistor

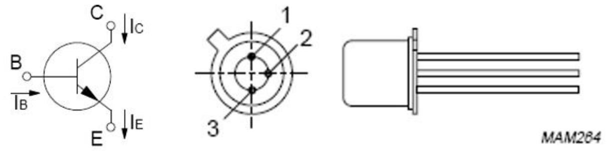


Figure 8 Terminals of transistor

Results:

(1) Finding the base terminal

Table 1 Diode check of BJT

Multimeter Leads connected to BJT		Diode Check value
+ Terminal	GND Terminal	[L]
1	2	0.0
2	1	0.716
1	3	0.0
3	1	0.0
2	3	0.709
3	2	0.0

When 1, 3 is connected, diode check shows 0 V regardless of the direction. This shows that the base terminal is number 2.

(2) Determine the type of the transistor

The diode check value has a value when the base is connected to the + terminal and the other terminals connected to the ground terminal. Which implies that the transistor has an NPN characteristic.

(3) Determine the emitter and collector terminal

When terminal 1 is connected to the ground terminal, it shows a higher value than when terminal 3 is connected. Therefore terminal 1 is the emitter junction. Which gives terminal 3 as the collector terminal.

Table 2 Terminal type of BJT

Transistor Type	NPN
Base Terminal	2
Emitter Terminal	1
Collector Terminal	3

In the lab report:

1. In Problem (1), explain why the remaining terminal is the base when the other two terminals give overload :0L with both polarities of the multimeter applied?

If the terminal C and E is connected together the diodes of the transistor is facing towards each other, blocking the current flow. Therefore, the two terminals that gives current flow of 0 is the Collector and Emitter terminal. And the other terminal is the base terminal.

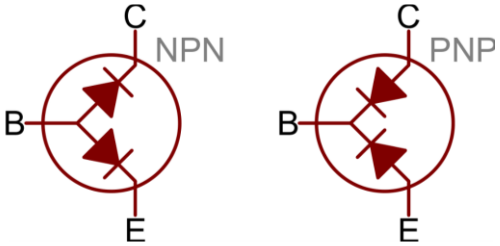


Figure 9 Transistors as two diodes

2. Explain why Problem (2) can be used to determine whether the transistor is 'NPN' or 'PNP'.

Refer to figure 9. If the voltage value shows in the direction of B->C and B->E (if the diode test result is not 0) , it is NPN transistor as in the experiment. If the voltage value shows in the direction of C ->B and E->B, than it is PNP transistor.

3. Explain why Problem (3) can be used to determine the collector and the emitter terminals.

The PN junction between Emitter and Base has higher voltage drop than the Collector and Base PN junction. This is because the emitter semiconductor layer has a heavier doping producing a higher forward voltage drop.

2.1.3 Experiment Part 2 – Setup

- Objective

Understand AM modulated signal

- Test Circuit:

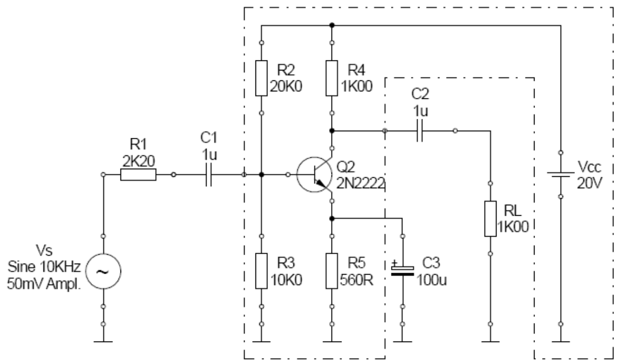


Figure 10 Circuit of part 2

2.1.4 Experiment Part 2 – Execution and Results

- Description of the measurement procedure.

Understand how to check the bias of the BJT

Results:

(1) Recorded voltages

V_{cc} = 19.94 V
V_B = 6.38 V
V_E = 5.73 V
V_C = 9.73 V
V_{CE} = 4.003 V
V_{BE} = 0.661 V

In the lab report:

1. Compare the measured values with the theoretical ones. Discuss the differences.

Table 3 Measured and Theoretical values

	Measured [V]	Theoretical [V]
V_{cc}	19.94	20
V_B	6.38	6.3354
V_E	5.73	5.61838
V_C	9.73	10.0169
V_{CE}	4.003	4.39852
V_{BE}	0.661	0.71702

The difference between the measured and theoretical values comes from the accuracy of the instruments and circuit elements used for the measurement. Also, the LT spice transistor characteristics might be different from the transistor used in the experiment.

2. Calculate the common emitter current gain β . Use only measured values!

$$\beta = \frac{I_C}{I_B} = \frac{(V_{cc} - V_C)/1000}{\frac{V_{CC} - V_B}{20000} - \frac{V_B}{10000}} = \frac{\frac{19.94 - 9.73}{1000}}{\frac{19.94 - 6.38}{20000} - \frac{6.38}{10000}} = 255.25$$

3. Determine the error sources with approximate values and -CALCULATE- the relative error of the calculated β. Check the plausibility of your previous calculated value by comparing it to the simulation. If the error is too high, what is the reason and is there a way to avoid it?

Simulation $\beta = \frac{I_C}{I_B} = \frac{0.00998314}{4.96897 \cdot 10^{-5}} = 200.91$

Relative error $\frac{\Delta\beta}{\beta} = \frac{|200.91 - 255.25|}{200.91} \cdot 100 = 27.04\%$

The error sources can be the accuracy of the instrument and the accuracy of the circuit elements used. The resistors used have error of 1%. The error of the voltage source can also influence the error. In this case, the source input was $\frac{20-19.94}{20} \cdot 100 = 0.3\%$. The error of 27.04% is quite high. This is also because of the current range used for calculation. The current range used is very small, decreasing precision of the calculation.

2.1.5 Experiment Part 3: Common emitter circuit – Setup

- Objective

Demonstrate correctly biased BJT amplification of small signals.

- Test Circuit:

Same as part 2

2.1.6 Experiment Part 3 – Execution and Results

- Description of the measurement procedure.

Set up the circuit. Use the frequency f = 10KHz. Increase V_s starting from 50mV. Reduce V_s to find undistorted output signal.

Results:

(1)Common emitter circuit – BJT amplification of small signals

Channel 1: Over the base

Channel 2: Over R_L

Active state

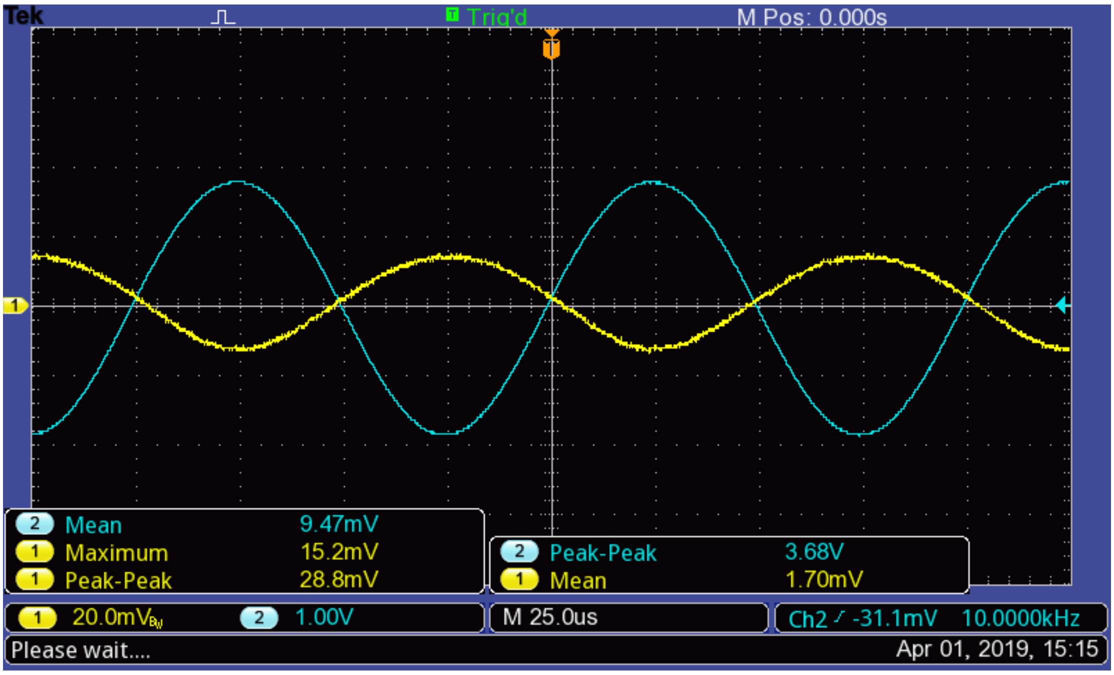


Figure 11 Common emitter circuit active mode

Distorted state

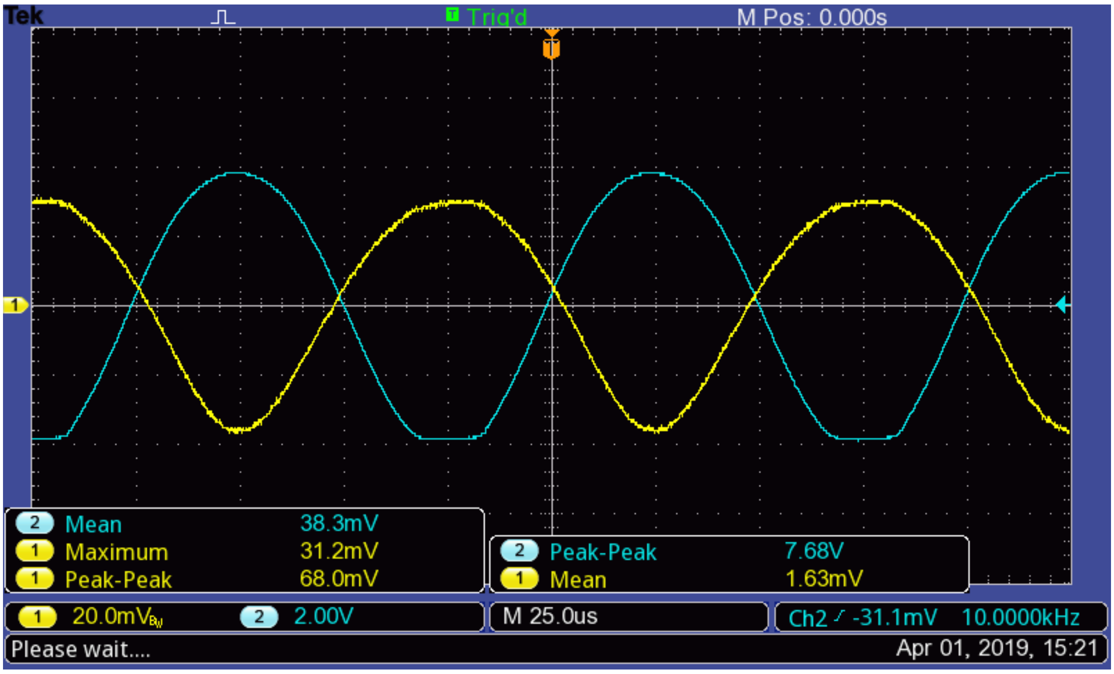


Figure 12 Distorted state

The distorted sine wave is visible in the figure above.
Peak to peak over base: 68.0mV
Peak to peak over R_L: 7.68V

(2) Maximum undistorted output signal

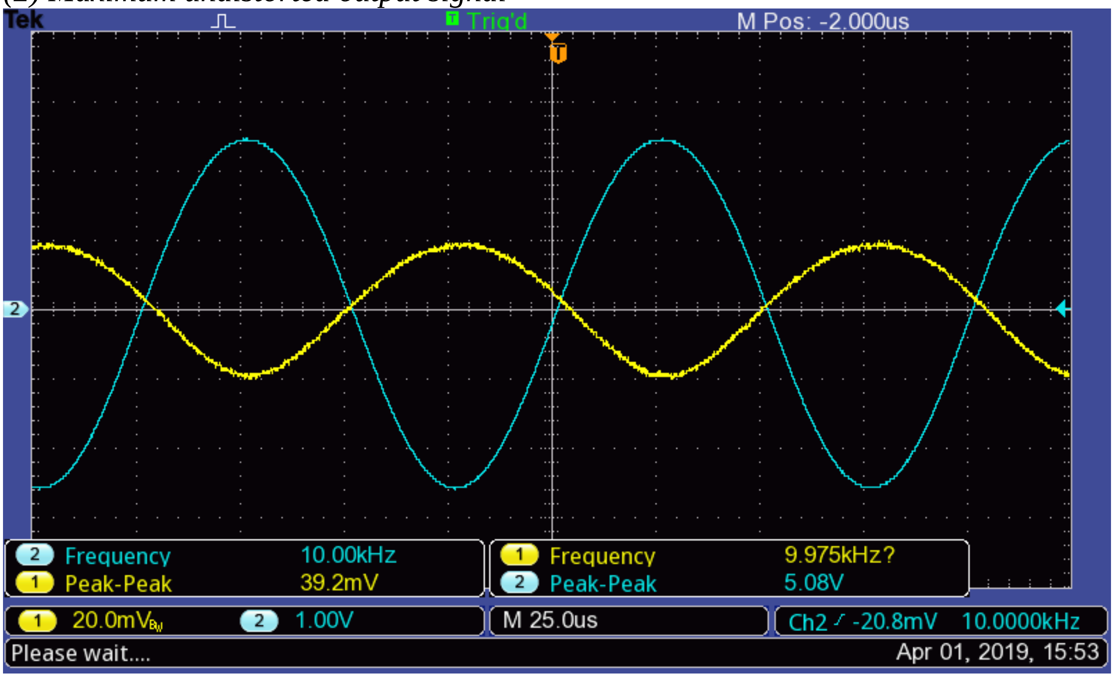
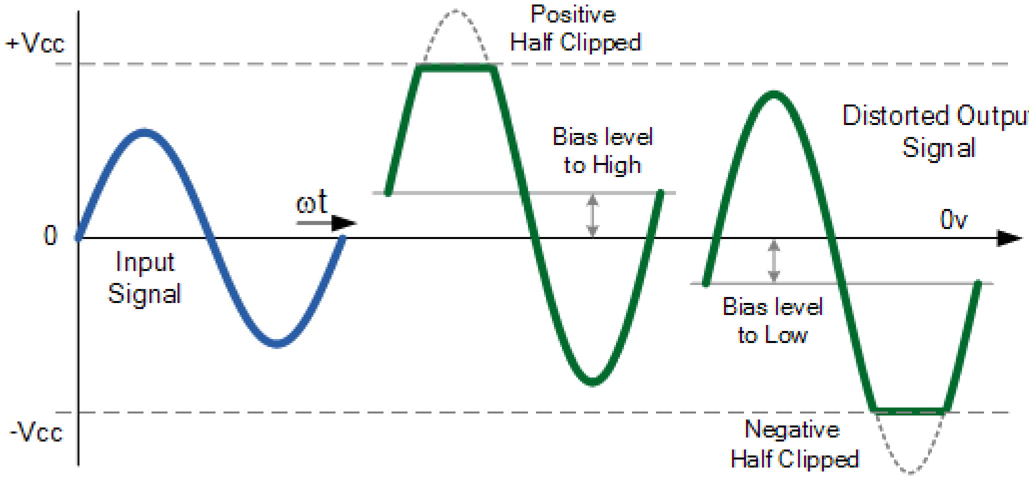


Figure 13 Maximum undistorted state

Peak to peak over base: 39.2mV
Peak to peak over R_L: 5.08V

In the lab report:
1. In what region of the output characteristic is the circuit for a distorted positive or negative amplitude? Explain! What is the condition in your case?



If the bias voltage is small and the Q voltage (DC component) is in the lower half of the load signal, the output would have a negative clipped waveform. Which, the transistor is in saturation point. If the bias is big and the Q point is in the upper half of the load line, the output would have positive clipped waveform. This shows that the transistor is in the cutoff point.

2. Using the measurements taken in the lab, calculate the voltage gain A_V *of the Amplifier.*
Peak to peak over base: 39.2mV
Peak to peak over R_L: 5.08V

$$A = \frac{V_L}{V_B} = \frac{5.08}{0.0392} = 129.6$$

3. Determine from the hard copies, what is the phase relationship between the input and the output signals?
There is a phase difference of 180º

4. Explain the reason for such a relation.

The phase difference of 180º shows that the output and input has a different sign. (Positive and negative). When the voltage of the input increases, the current flowing through the base increases resulting in more current going through the collector. This leads the output current goes small showing inversion.

2.1.7 Experiment Part 4 : Bandwidth of amplifier circuit – Setup

- Objective

Understand how to determine the bandwidth of BJT amplifier

2.1.8 Experiment Part 4 – Execution and Results

- Description of the measurement procedure.

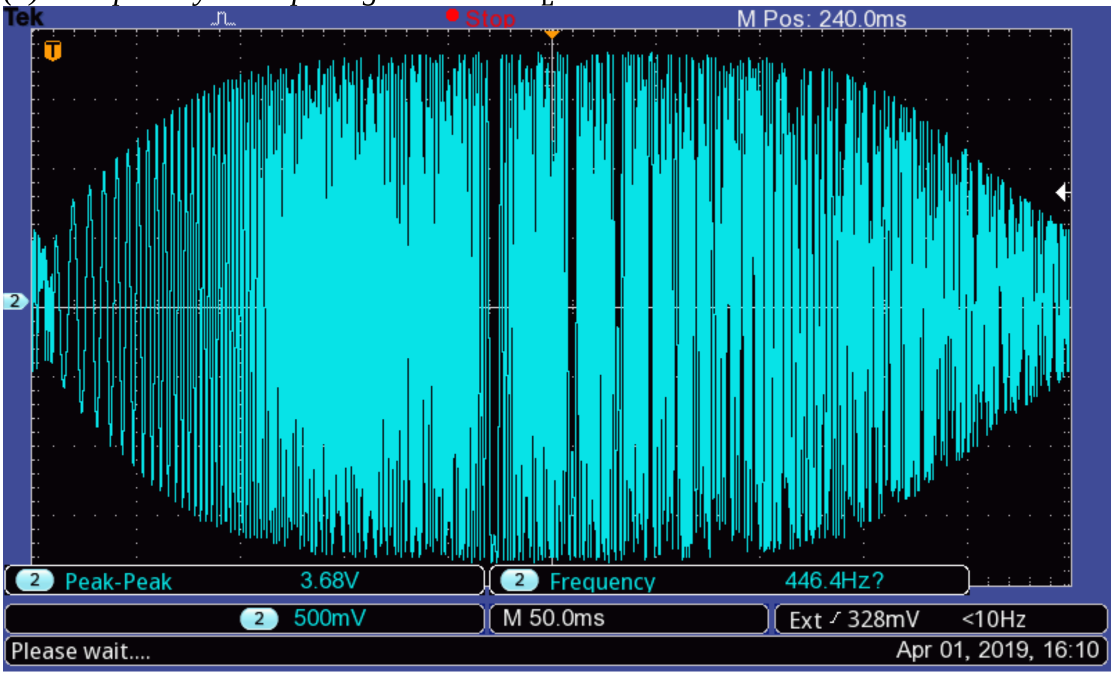
Set up the circuit shown and set the function generator:

1: START F : $100Hz$
2: STOP F : $1MHz$
3: SWP TIME : $500ms$
4: SWP MODE : logarithmic

Record the signal values and change the mode of the generator and get the cut-off frequency.

Results:

(1) Sweep analysis output signal across R_L



(2) -3dB cut-off frequency

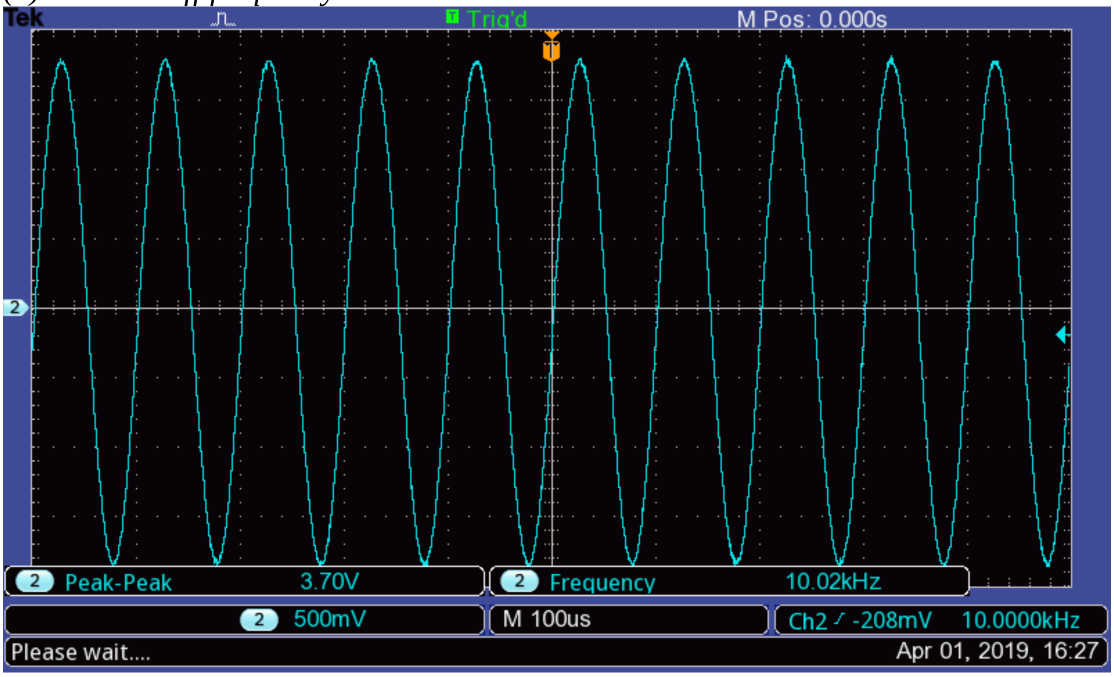


Figure 14 Signal at center freuqency

Center frequency amplitude: V_{center} = 3.70V

$$V_{cutoff} = \frac{V_{center}}{\sqrt{2}} = \frac{3.70}{\sqrt{2}} = 2.62$$

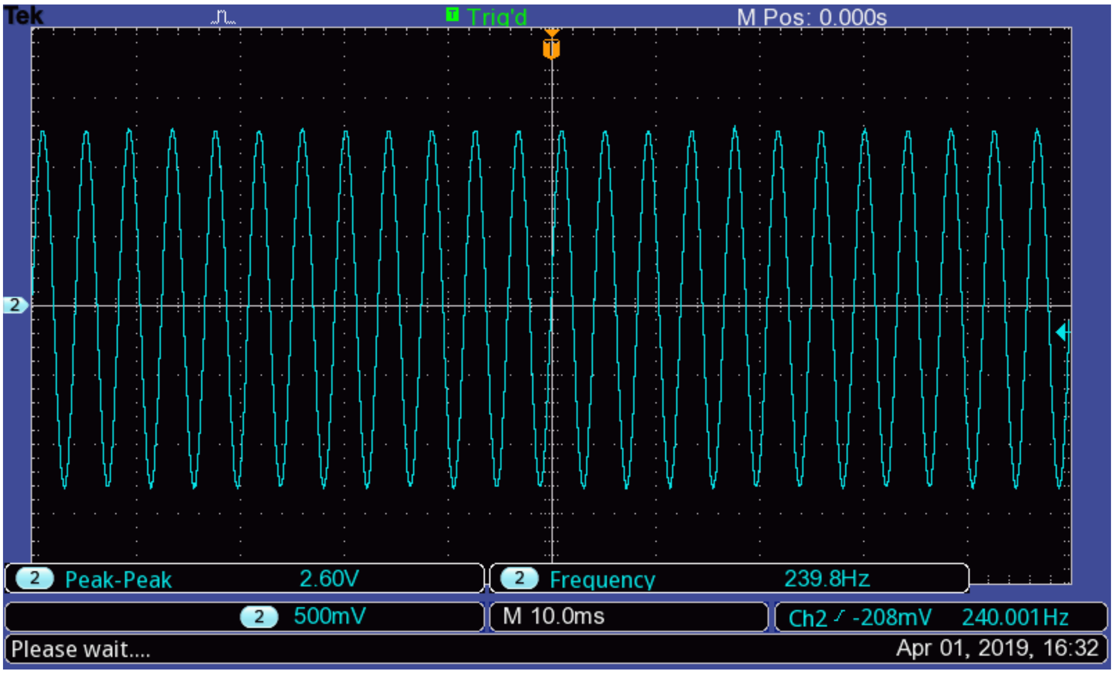


Figure 15 Lower cut-off

Lower cut-off frequency $f_{cutoff-low} = 239.8Hz$

(3) upper -3dB cut-off frequency

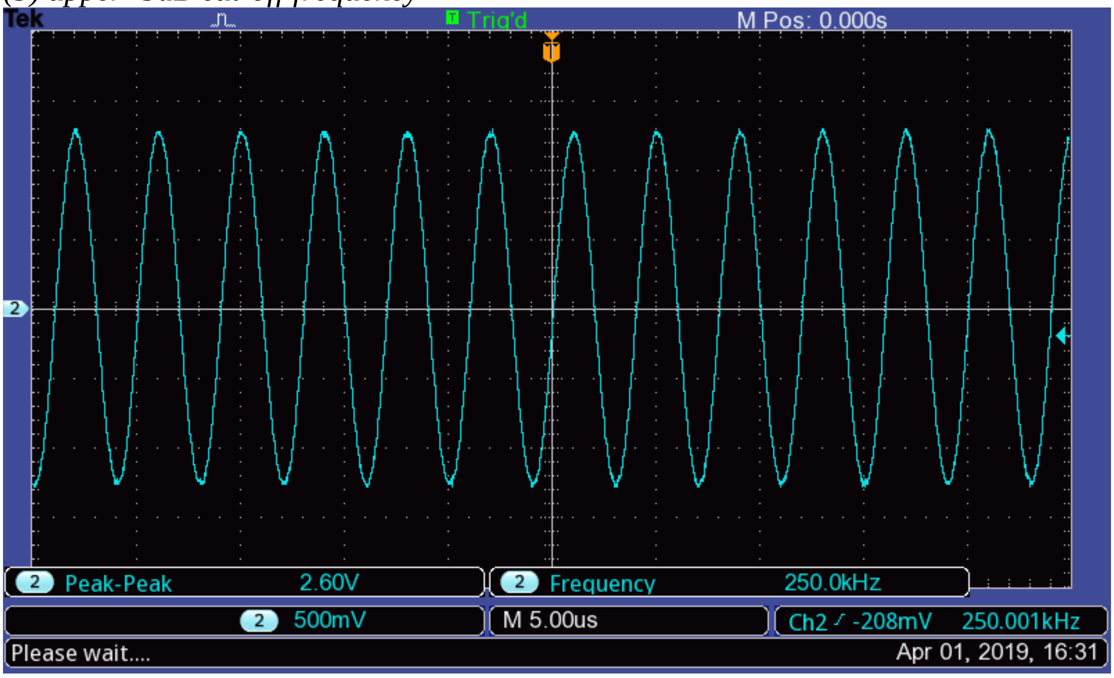


Figure 16 Upper cut-off

Upper cut-off frequency $\mathrm{f_{cutoff-up}} = 250.0\mathrm{kHz}$

In the lab report:

1. Explain your observation.

It can be seen that the circuit works as a band pass filter from 100Hz to 1MHz. In low frequency range, the output capacitor filters the low frequency components and in high frequency, the transistor cannot fallow the change in the base current.

2. Using the measurements taken in the lab, calculate the amplier bandwidth.

$$BW = f_{cutoff-up} - f_{cutoff-low} = 250000 - 239.8 = 249760.2\ Hz$$

3. Compare to the simulation!

The simulation bandwidth is 414205Hz and the measured bandwidth was 249760Hz. Which gives an error of $\frac{414205-249760}{414205} \cdot 100 = 39.7\%$. The difference might have come from the errors of the transistor and the instrument.

3 Conclusion

In the experiment, bipolar junction transistor characteristics were observed. The terminal type was decided in part one by performing the diode check. The biased of the BJT were examined in part 2 and the current gain and error were calculated. The error between the measured and calculated resulted in 27% which was relatively high and was caused by the accuracy of the instrument, resistors and calculations. Amplification of small signals were studied using common emitter circuit are studied in part 3, calculating the voltage gain and the distortion conditions. Also, the input and output signal had a phase shift of 180º due to the current flow amount difference. Lastly in part 4 the bandwidth of the BJT amplifier were determined. The measured bandwidth had an error of 39.7%. The reason for this error came from the errors of the circuit elements and the instrument. Especially on the accuracy of the transistor had caused the main errors.

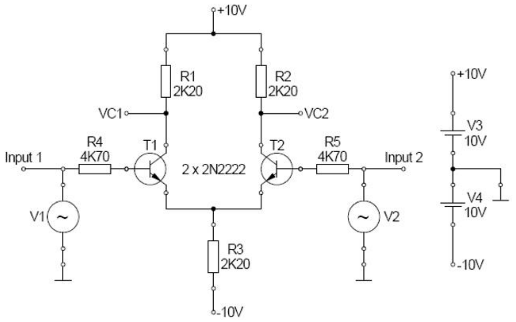
5 References

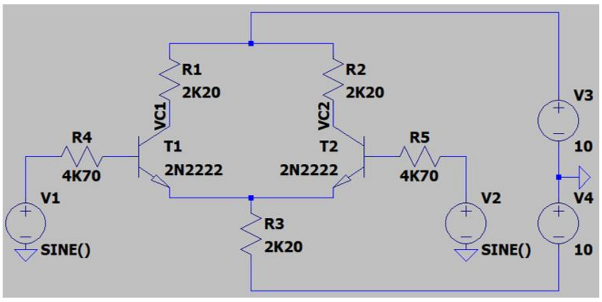
[1] Inst. Uwe Pagel: ‘Electronics Lab’ (2019), ‘Bipolar Junction Transistor’, 37-52
[2] Transistors, learn.sparkfun.com/tutorials/transistors/all.
[3] “LATEST.” All About Circuits, www.allaboutcircuits.com/textbook/semiconductors/chpt-4/meter-check-transistor-bjt/.
[4] “Amplifier Distortion in Transistor Amplifiers.” Basic Electronics Tutorials, 13 June 2018, www.electronics-tutorials.ws/amplifier/amp_4.html.

6 Appendix - Data and Prelab of Experiment 4

6.1 Prelab Operational Amplifier

6.1.1 Problem 1 : Simulate a Differential Amplifiier

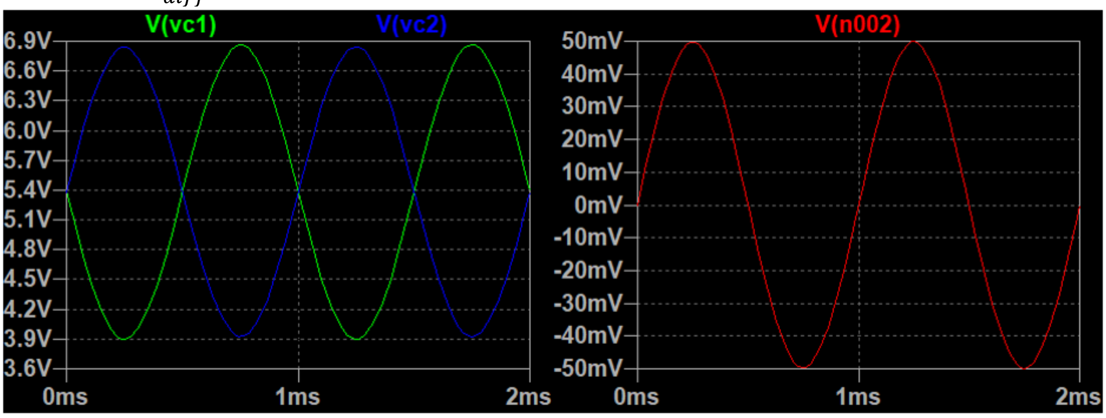




1. Perform a DC operation point analysis for the above circuit. Determine the values for $V_{BE}(T1,T2)$, $V_C(T1,T2)$, $I_C(T1,T2)$, $I_E(T1,T2)$ and I_{RE}. What would happen with the values in the two branches if the transistors are not absolute identical.
Hint: To get an idea, replace one 2N2222 transistor -temporarily- with a 2N3904 from the library.

$$V_{BE}(T1) = V_{B1} - B_{E1} = (-0.047094859) - (-0.72071642) = 0.673621561\,V$$
$$V_{BE}(T2) = V_{B2} - B_{E2} = (-0.047094859) - (-0.72071642) = 0.673621561\,V$$
$$V_C(T1) = 5.3824\,V$$
$$V_C(T2) = 5.3824\,V$$
$$I_C(T1) = 0.00209891\,A$$
$$I_C(T2) = 0.00209891\,A$$
$$I_E(T1) = 0.00210893\,A$$
$$I_E(T2) = 0.00210893\,A$$
$$I_{RE} = I_{R3} = 0.00421786\,A$$

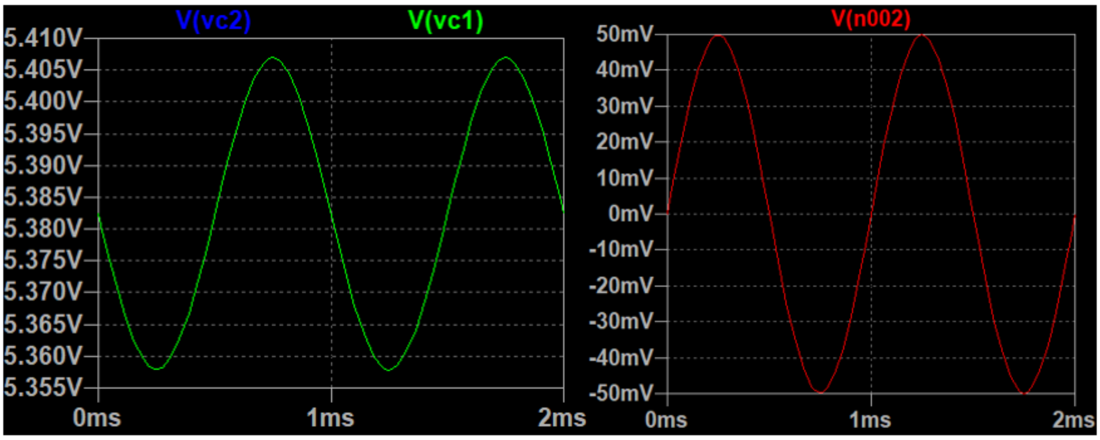
2. Perform a transient analysis. Use single ended input mode. Set V1 to Sine, f = 1KHz, $\hat{u}$ = 50mV, and V2 to Gnd. Display and measure the two collector voltages! Calculate $A_{V_{diff}}$in dB.



(left) Voltage over the two collectors (right) Input voltage

$$A_{V_{diff}} = 20\log\left(\frac{V_C}{V_1}\right) = 20\log\left(\frac{2.9609886}{0.100}\right) = 29.43dB$$

3. Perform a simulation with common mode input. Set V1 and V2 to Sine, f = 1KHz, $\hat{u}$ = 50mV. Display and measure the two collector voltages! Calculate $A_{V_{cm}}$ in dB.

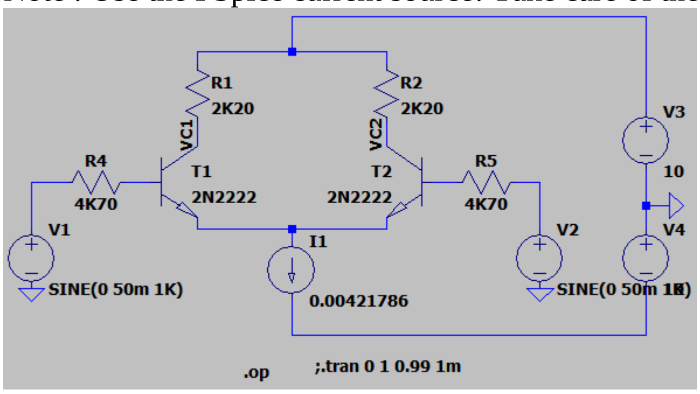


$$A_{V_{cm}} = 20\log\left(\frac{V_C}{V_1}\right) = 20\log\left(\frac{0.049058764}{0.100}\right) = -6.1856679545dB$$

4. Calculate the common-mode rejection ratio in dB!

$$A_{V_{diff}} - A_{V_{cm}} = 29.43dB - (-6.1856679545dB) = 35.616\ dB$$

5. Replace the resistor R3 by a current source. Use the same current you got from the .op analysis in step 1.
Note : Use the PSpice current source! Take care of the flow direction of the current.



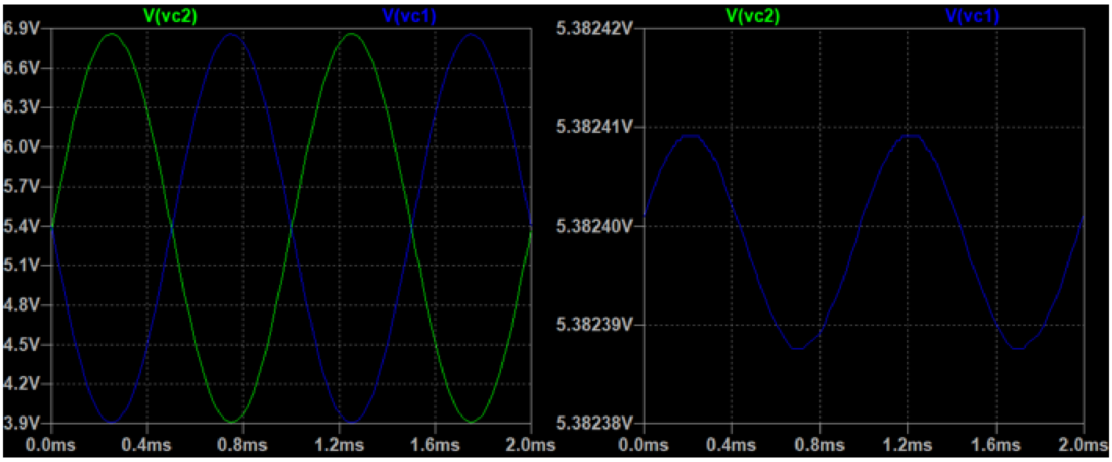
6. Repeat step 1 - 4 for the new circuit.

Step 1:

$$V_{BE}(T1) = V_{B1} - B_{E1} = (-0.0470949) - (-0.720717) = 0.6736221\ V$$
$$V_{BE}(T2) = V_{B2} - B_{E2} = (-0.0470949) - (-0.720717) = 0.6736221\ V$$
$$V_C(T1) = 5.3824V$$
$$V_C(T2) = 5.3824\ V$$
$$I_C(T1) = 0.00209891A$$
$$I_C(T2) = 0.00209891\ A$$
$$I_E(T1) = 0.00210893A$$
$$I_E(T2) = 0.00210893\ A$$
$$I_{RE} = 0.00421786\ A$$

Step 2:

$$A_{V_{diff}} = 20\log\left(\frac{V_C}{V_1}\right) = 20\log\left(\frac{2.9449929}{0.100}\right) = 29.381dB$$



(left) Step2, single ended input mode (right) Step 3, common mode input

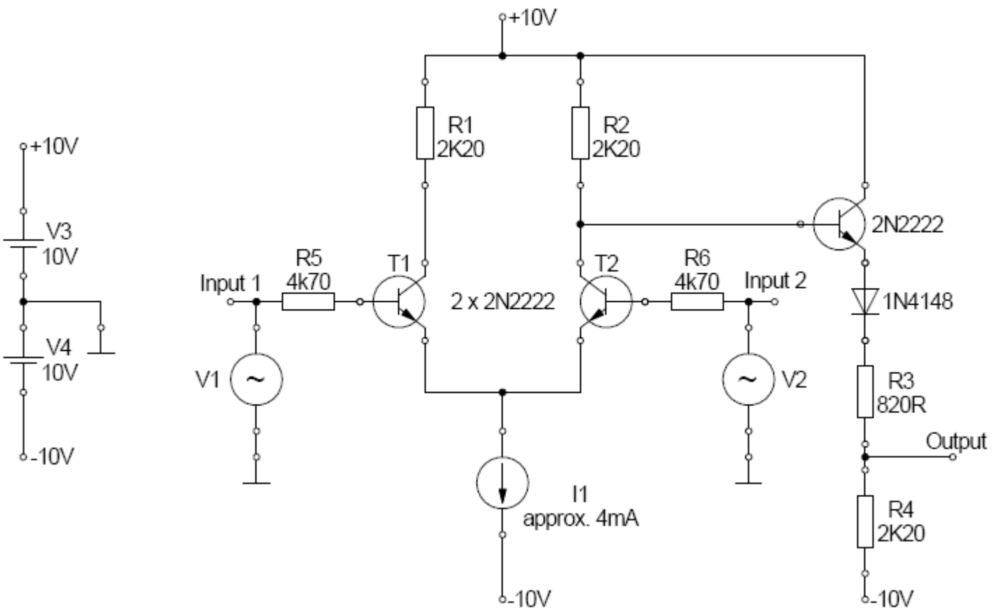
<u>Step 3:</u>

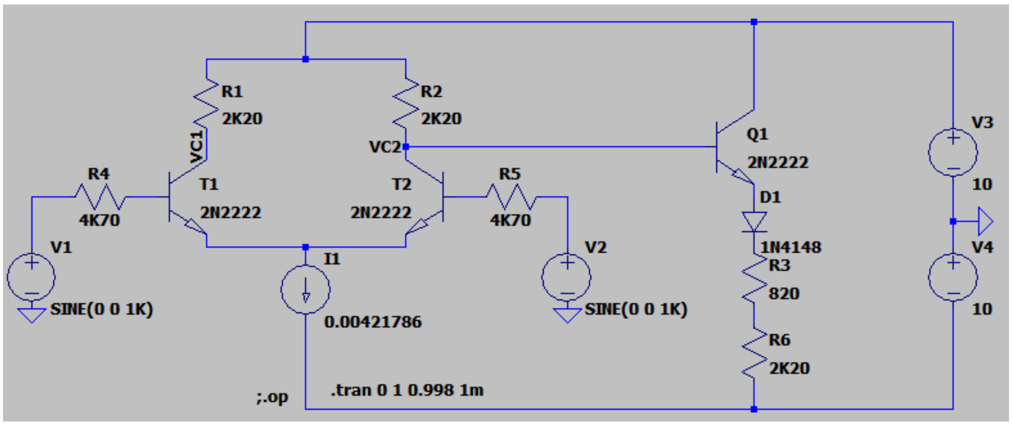
$$A_{V_{cm}} = 20\log\left(\frac{V_C}{V_1}\right) = 20\log\left(\frac{21.457672\mu}{0.100}\right) = -73.368dB$$

<u>Step 4:</u>

$$A_{V_{diff}} - A_{V_{cm}} = 29.381dB - (-73.368dB) = 102.749\ dB$$

6.1.2 Problem 2





1. Calculate the voltage at the output of the emitter follower when both inputs are connected to ground. Assume $\beta = 200$ *and* $U_{BE} = 0.7\ V$. *The forward voltage drop of the diode is 0.7V.*

$$V_{R_6} + V_{R_3} + 0.7 + 0.7 + 2200 \cdot \frac{I_C}{2} = 20$$

$$V_{R_6} + V_{R_3} = 13.9603V$$

$$\mathrm{V_{R_6}} = 13.9603 \cdot \frac{2200}{2200 + 820} = 10.1697V$$

$$V_{out} = -10 + V_{R_6} = -10 + 10.1697 = 0.1697\ V$$

2. Perform a dc operation point analysis. Determine the output voltage and compare to your calculation.

$\mathrm{V_{out}} = 0.186334$ V

The calculated voltage was 0.1697 showing not much difference in the operation and calculation.

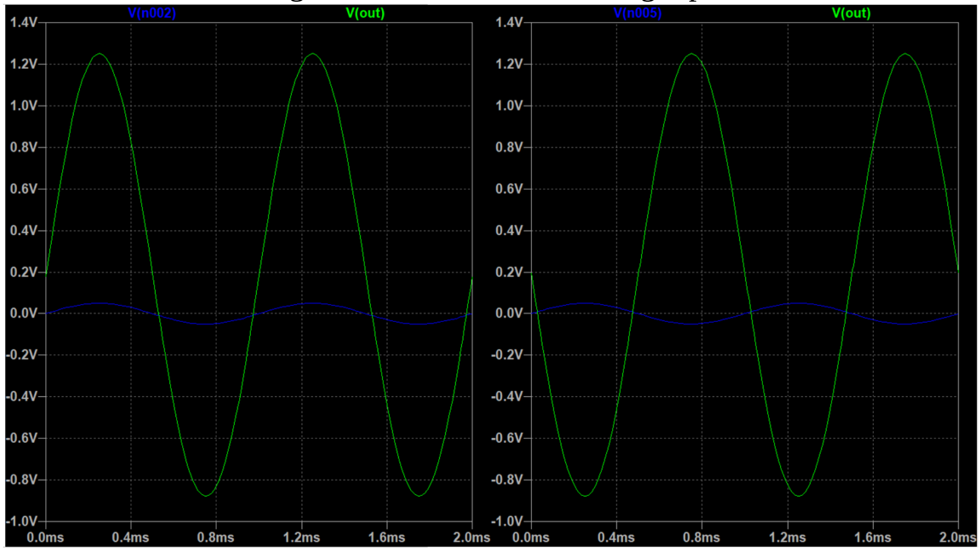
3. Determine $A_{V_{diff}}, A_{V\,cm}$ and the common- mode rejection ration in a similar way like problem 1.

$$A_{V_{diff}} = 20\log\left(\frac{V_C}{V_1}\right) = 20\log\left(\frac{2.9487902}{0.100}\right) = 29.392dB$$

$$A_{V_{cm}} = 20\log\left(\frac{V_C}{V_1}\right) = 20\log\left(\frac{22.888184\mu}{0.100}\right) = -75.27dB$$

$$A_{V_{diff}} - A_{V_{cm}} = 29.392dB - (-75.27dB) = 104.662\ dB$$

4. What is the inverting and what the non inverting input?

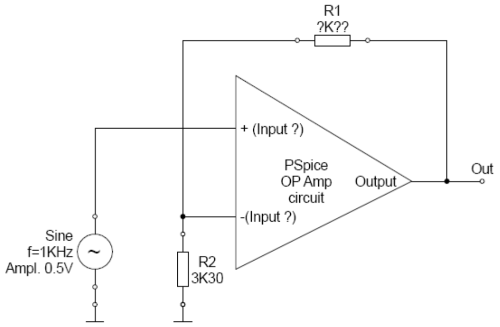


(left) input 1, (right) input 2

By the figure above, it can be seen that input 2 is the inverting input and input 1 is the non- inverting input.

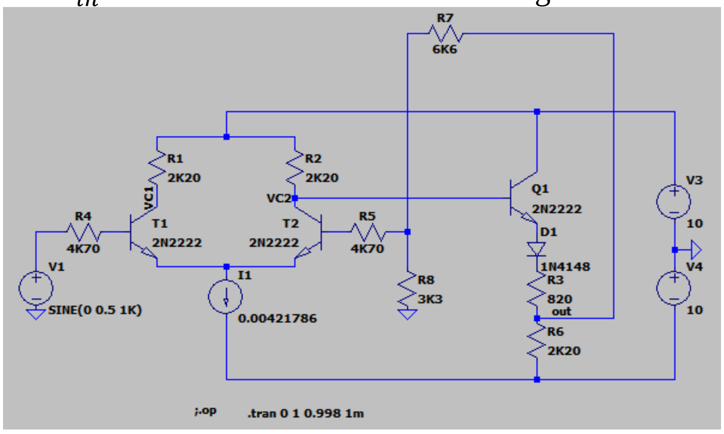
6.1.3 Problem 3

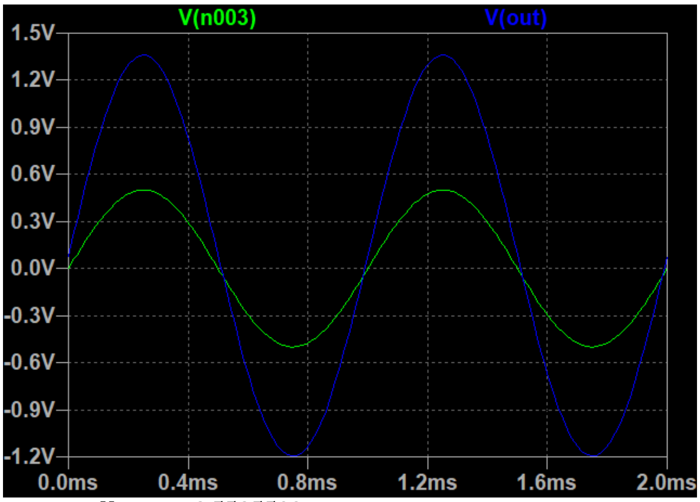
1. Design a non-inverting amplifier. Calculate R_1 to make the gain 3.



$$V_{out} = 3V_{in}$$
$$\frac{V_{out} - V_{in}}{R_1} = \frac{V_{in}}{R_2} = \frac{2V_{in}}{R_1}$$
$$R_1 = 2R_2 = 6600\Omega$$

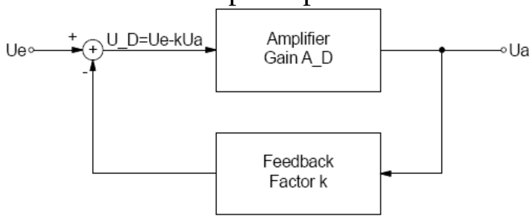
2. Use LTspice to simulate your design. Display and measure input and output signal. For U_{in} use $\hat{u} = 500mV$. Determine the gain.





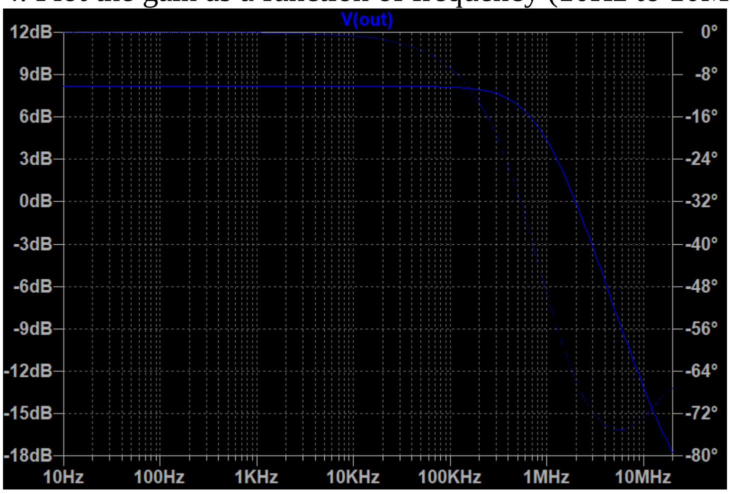
Gain = $\frac{V_{out}}{V_{in}} = \frac{2.5525593}{0.99774727} = 2.5583225099$

3. What is the reason why the measured gain is smaller than the theoretical gain.
Hint: Think of the principles of feedback!



The theoretical gain is 3 while the measured gain was 2.56. This error is due to the feedback factor k. Which gives gain as: $\frac{U_a}{U_e} = \frac{A_D}{1+kA_D}$

4. Plot the gain as a function of frequency (10Hz to 10MHz).



6.1.4 Problem 4: Properties from Data Sheet

	Data Sheet LM741C Op-Amp at 25°	Simulation
Input Offset Current	30nA	
Slew Rate	0.5 V/µs	
Input Bias Current	100nA	22.1312 µA
Input Offset Voltage	5mV	81.7381 mV
Voltage Gain	200V/mV	
CMRR	90dB	