بسم الله الرحمن الرحيم

Course name:

Electronics (Parctical)

Course No:

ELEE2110

Experiment 6:

Common Emitter Amplifier

Student's name:

Mohamed Hassan , Almo'taz Billah Alsousi , Amr AboSwareh

University No. :

120170672 , 120212396 , 120211836

Instructor's name:

Eng.Omar Hammad

❖ Objectives:

- To construct a common emitter amplifier circuit and make DC and AC analyze of the system.
- To find all parameters of equivalent model.
- To be familiar how can be simulate common emitter amplifier circuit using LTspice simulation.

❖ Theory:

An amplifier is an electronic circuit that can increase the strength of a weak input signal without distorting its shape. The common emitter configuration is widely used as a basic amplifier as it has both voltage and current amplification with 180 phase shift.

The factor by which the input signal gets multiplied after passing through the amplifier circuit is called the gain of the amplifier. It is given by the ratio of the output and input signals.

$$Gain = \frac{Output\ Signal}{Input\ Signal}$$

The aim of any small signal amplifier is to amplify all of the input signal with the minimum amount of distortion possible to the output signal, in other words, the output signal must be an exact reproduction of the input signal but only bigger (amplified).

To obtain low distortion when used as an amplifier the operating quiescent point needs to be correctly selected. This is in fact the DC operating point of the amplifier and its position may be established at any point along the load line by a suitable biasing arrangement.

The best possible position for this **Q-point** is as close to the center position of the load line as reasonably possible, thereby producing a Class A type amplifier operation, i.e. $V_{ce} = \frac{1}{2}V_{cc}$. Consider the Common Emitter Amplifier circuit shown below

▪ The Common Emitter Amplifier Circuit:

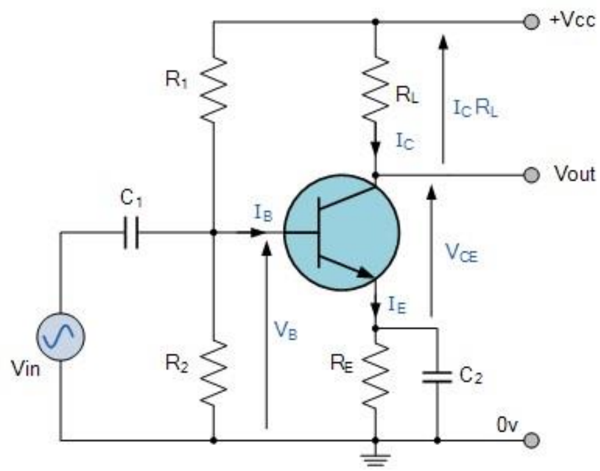


Figure 1 Common Emitter Amplifier Circuit

The single stage common emitter amplifier circuit shown above uses what is commonly called **"Voltage Divider Biasing"**. This type of biasing arrangement uses two resistors as a potential divider network across the supply with their center point supplying the required Base bias voltage to the transistor. Voltage divider biasing is commonly used in the design of bipolar transistor amplifier circuits.

▪ Amplifier Coupling Capacitors:

In Common Emitter Amplifier circuits, capacitors **C_1** and **C_2** are used as Coupling Capacitors to separate the AC signals from the DC biasing voltage. This ensures that the bias condition set up for the circuit to operate correctly is not affected by any additional amplifier stages, as the capacitors will only pass AC signals and block any DC component. The output AC signal is then superimposed on the biasing of the following stages. Also a bypass capacitor, CE is included in the Emitter leg circuit.

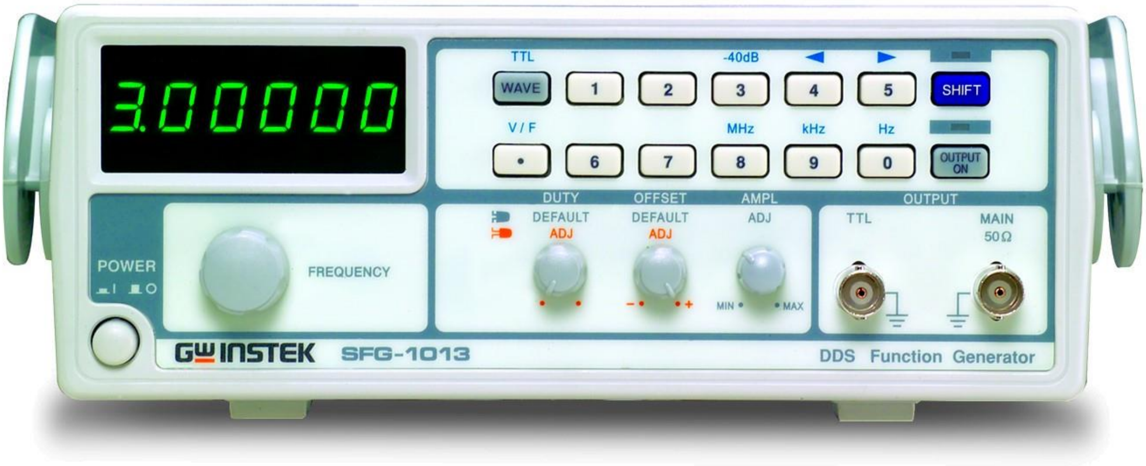
This capacitor is effectively an open circuit component for DC biasing conditions, which means that the biasing currents and voltages are not affected by the addition of the capacitor maintaining a good **Q-point** stability.

However, this parallel connected bypass capacitor effectively becomes a short circuit to the Emitter resistor at high frequency signals due to its reactance. Thus, only $\mathbf{R_L}$ plus a very small internal resistance acts as the transistors load increasing voltage gain to its maximum.

❖ Practical Work:

Tools we need for this experiment:

1. Function generator:



2. Oscilloscope:

3. Multimeter:

4. DC Power Supply:

5. Resistors:

6. Connecting Cables:

7. Transistors:

8. Capacitors:

❖ The Experiments:

Part 1: DC Analysis:

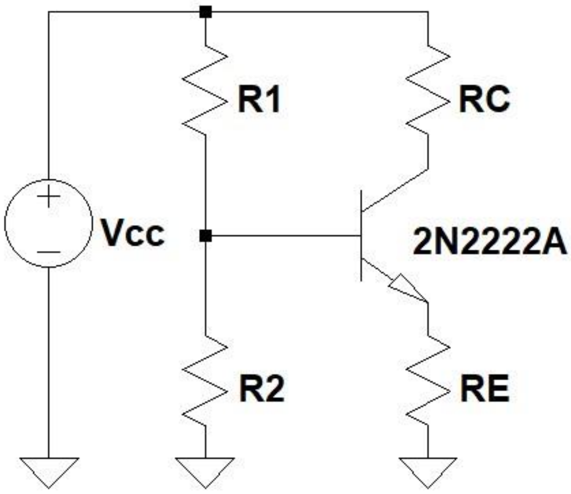


Figure 2 DC circuit for common emitter amplifier

1. Connect the circuit as shown in Figure 2.
2. Set **V_{cc} = 12 v , R_1 = 10k Ω , R_2 = 2.2k Ω , R_c = 1k Ω ,** and **R_E = 220 Ω**.
3. Measure the voltages and currents in the circuits in Table 1.
4. Simulate the circuit using LTspice program.
5. Repeat the previous step when

   i. **R_C = 4.7K Ω**

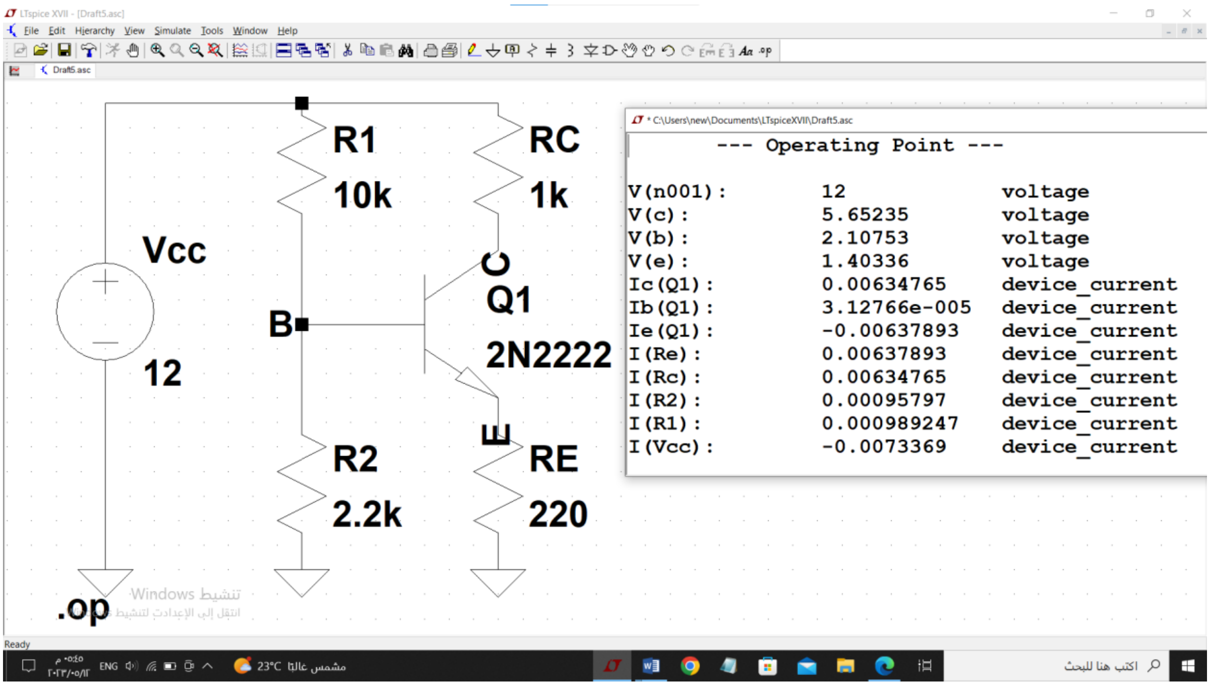
   ii. **R_C = 100k Ω**

Table 1 DC Operating Values

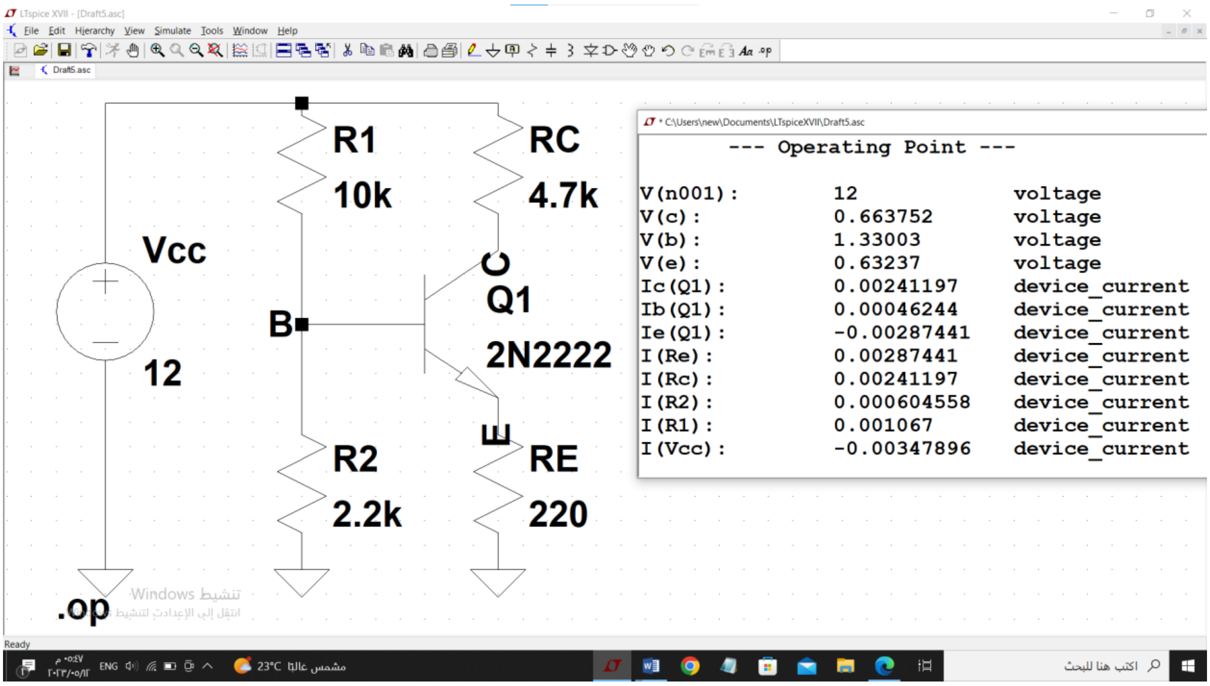
R (kΩ)	V_B (v)	V_C (v)	V_E (v)	V_{CE} (v)	I_B (μA)	I_C (mA)	I_E (mA)
1	2.06	6.4	1.24	5.15	5.59	0.005	-0.005
4.7	1.42	0.67	0.62	54.94m	0.0004	0.002	-0.002
100	0.95	0.19	0.17	20.15m	0.0006	0.0001	-0.0007

➤Simulation:

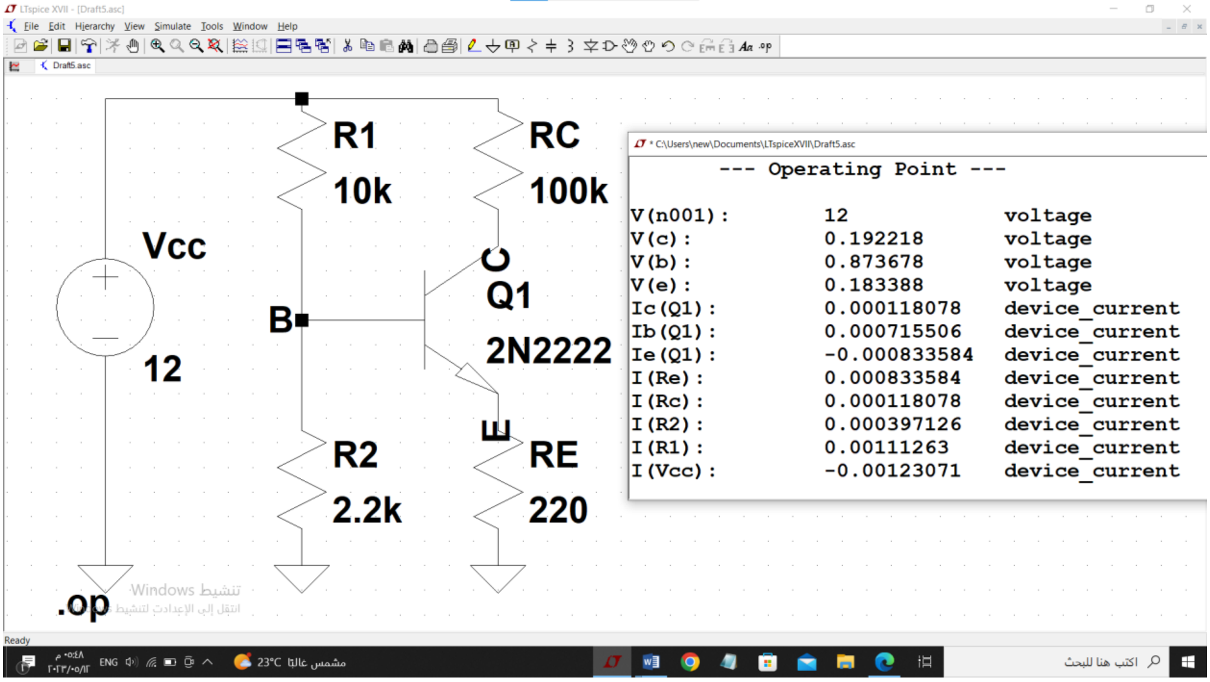
R_C = 1k Ω



R_C = 4.7k Ω



R_C = 100k Ω


LTspice XVII - [Draft5.asc]
File Edit Hierarchy View Simulate Tools Window Help
Draft5.asc
R1
10k
RC
100k
Vcc
12
B
C
Q1
2N2222
E
R2
2.2k
RE
220
.op
* C:\Users\new\Documents\LTspiceXVII\Draft5.asc
--- Operating Point ---
V(n001): 12 voltage
V(c): 0.192218 voltage
V(b): 0.873678 voltage
V(e): 0.183388 voltage
Ic(Q1): 0.000118078 device_current
Ib(Q1): 0.000715506 device_current
Ie(Q1): -0.000833584 device_current
I(Re): 0.000833584 device_current
I(Rc): 0.000118078 device_current
I(R2): 0.000397126 device_current
I(R1): 0.00111263 device_current
I(Vcc): -0.00123071 device_current
Ready
ENG
23°C

Part 2: AC Analysis:

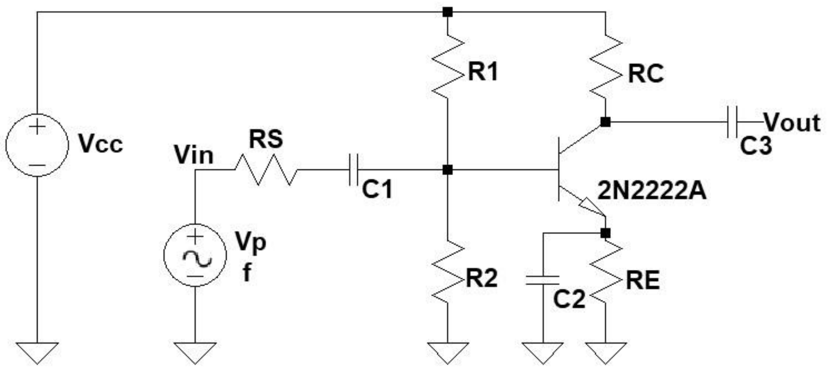


Figure 3 AC circuit for common emitter amplifier

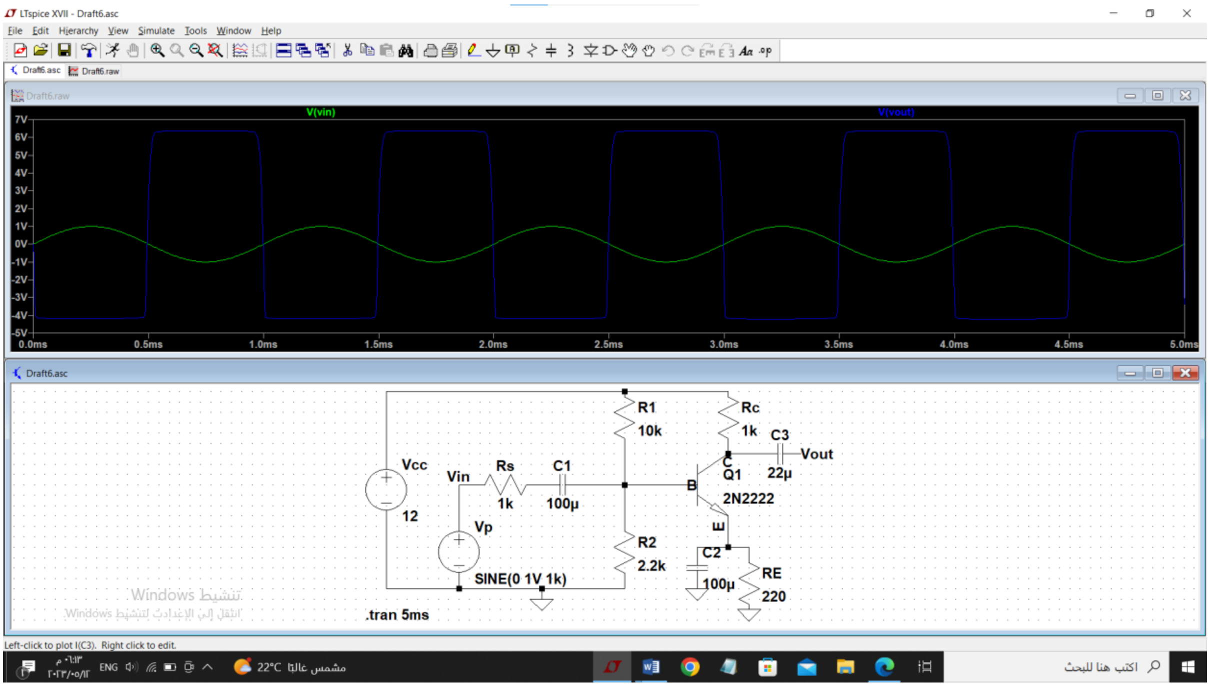
1. Connect the circuit as shown in Figure 3.
2. Set function generator to a sine wave with 500mVp and 1 kHz.
3. Set **V_{cc} = 12 v , R_1 = 10k Ω , R_2 = 2.2k Ω , R_c = 1k Ω ,** and **R_E = 220 Ω** , **C_1** = **100µ F** , and **C_3 = 22µ F**.
4. Measure the input and output voltage using oscilloscope.
5. Simulate the circuit using LTspice program.
6. Repeat the previous step when

    i. **R_C = 4.7K Ω**

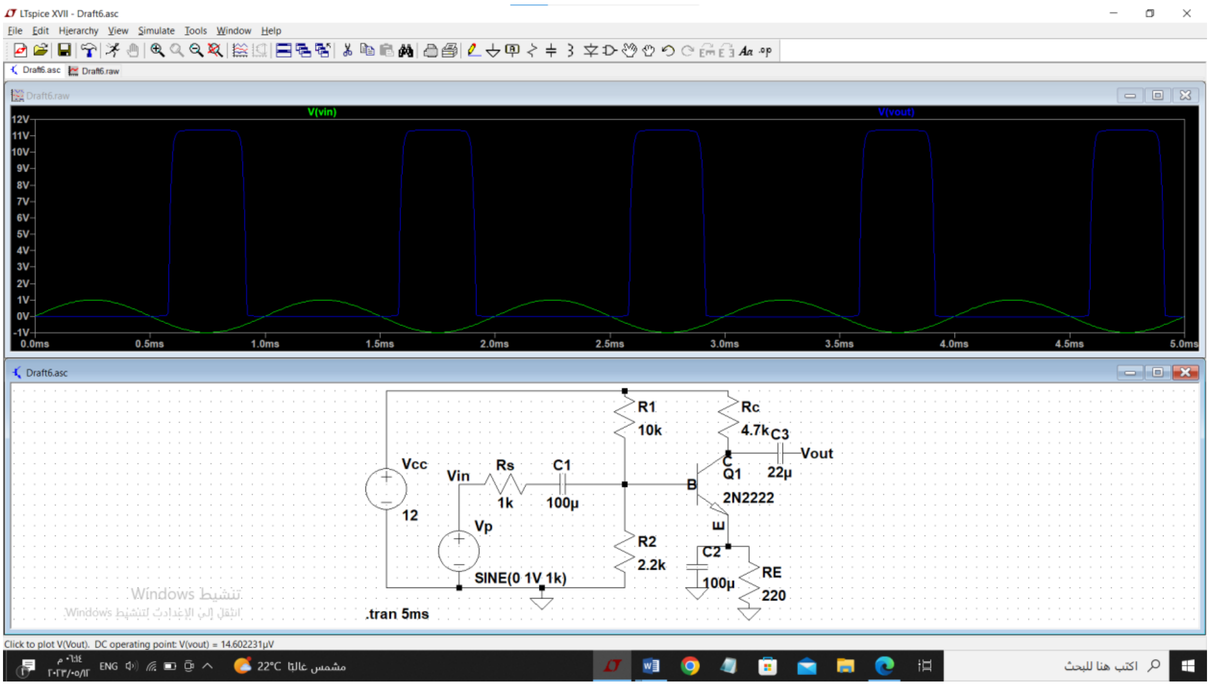
    ii. **R_C = 100k Ω**

➢Simulation:

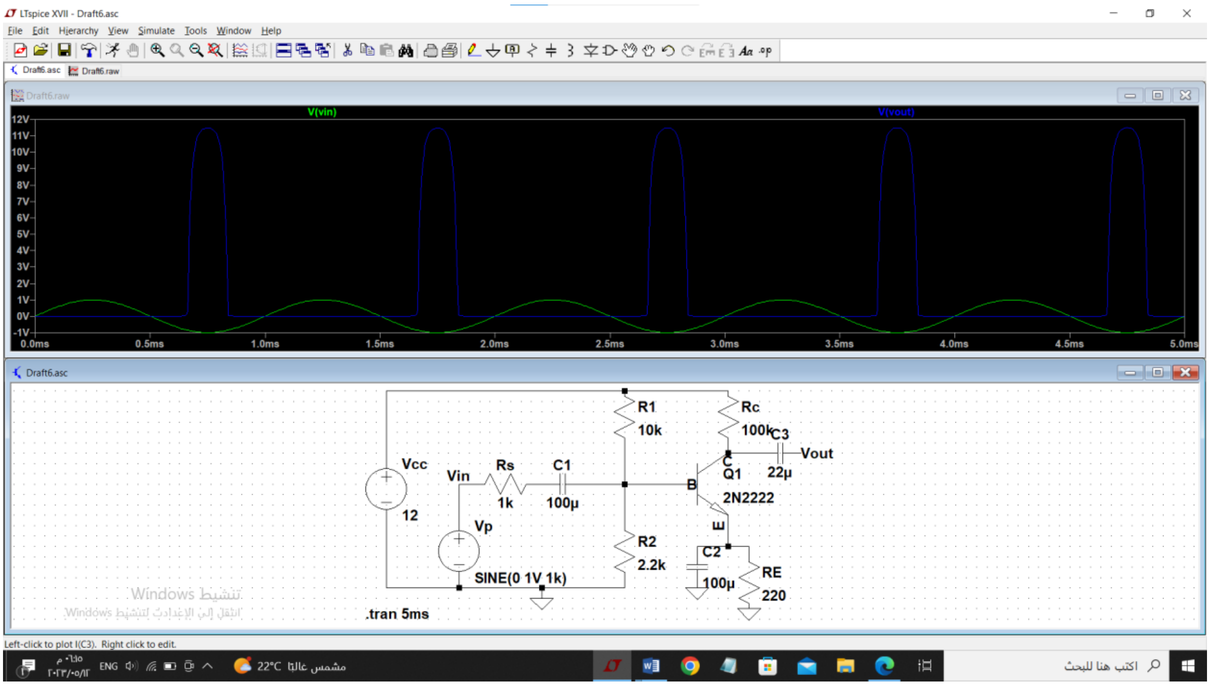
R_C = 1k Ω



R_C = 4.7k Ω



R_C = 100k Ω



LTspice XVII - Draft6.asc
V(vin)
V(vout)
R1
10k
Rc
100k
C3
22µ
Vout
Vcc
12
Vin
Rs
1k
C1
100µ
Q1
2N2222
Vp
SINE(0 1V 1k)
R2
2.2k
C2
100µ
RE
220
.tran 5ms

❖ Conclusion:

In this lab, we learned how to connect the transistor in a Circuit and we learned about of Common Emitter Amplifier and learned how to connect it in AC and DC.