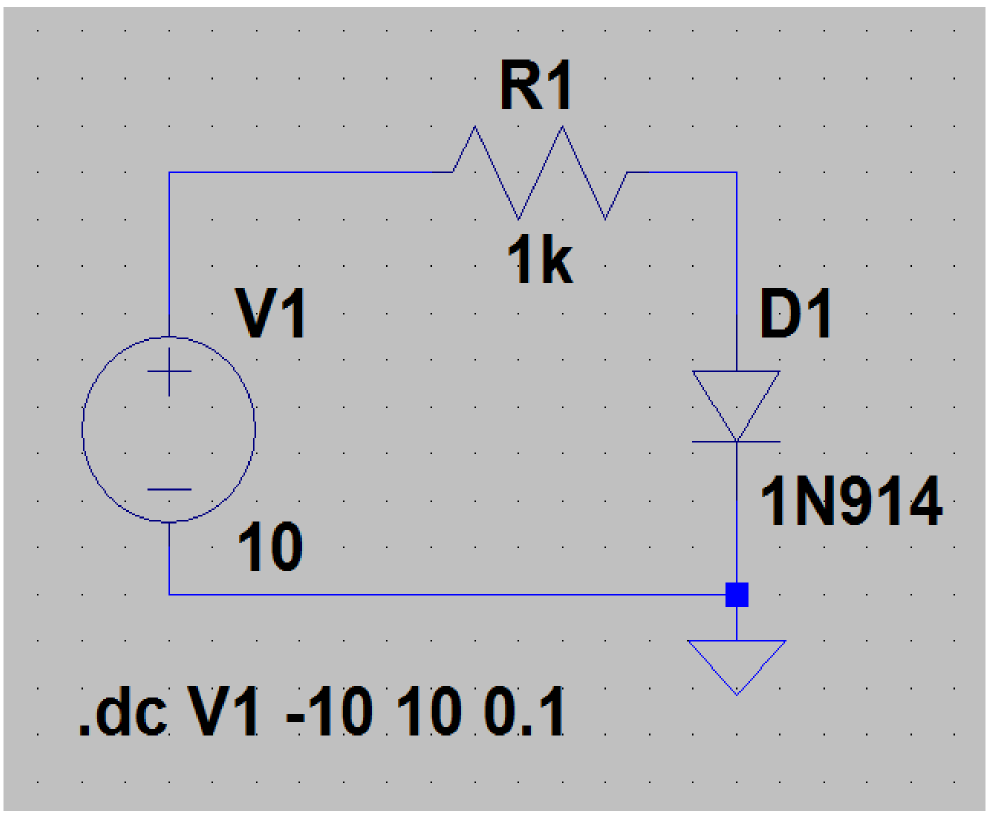
REPORT FOR ASSIGNMENT 1

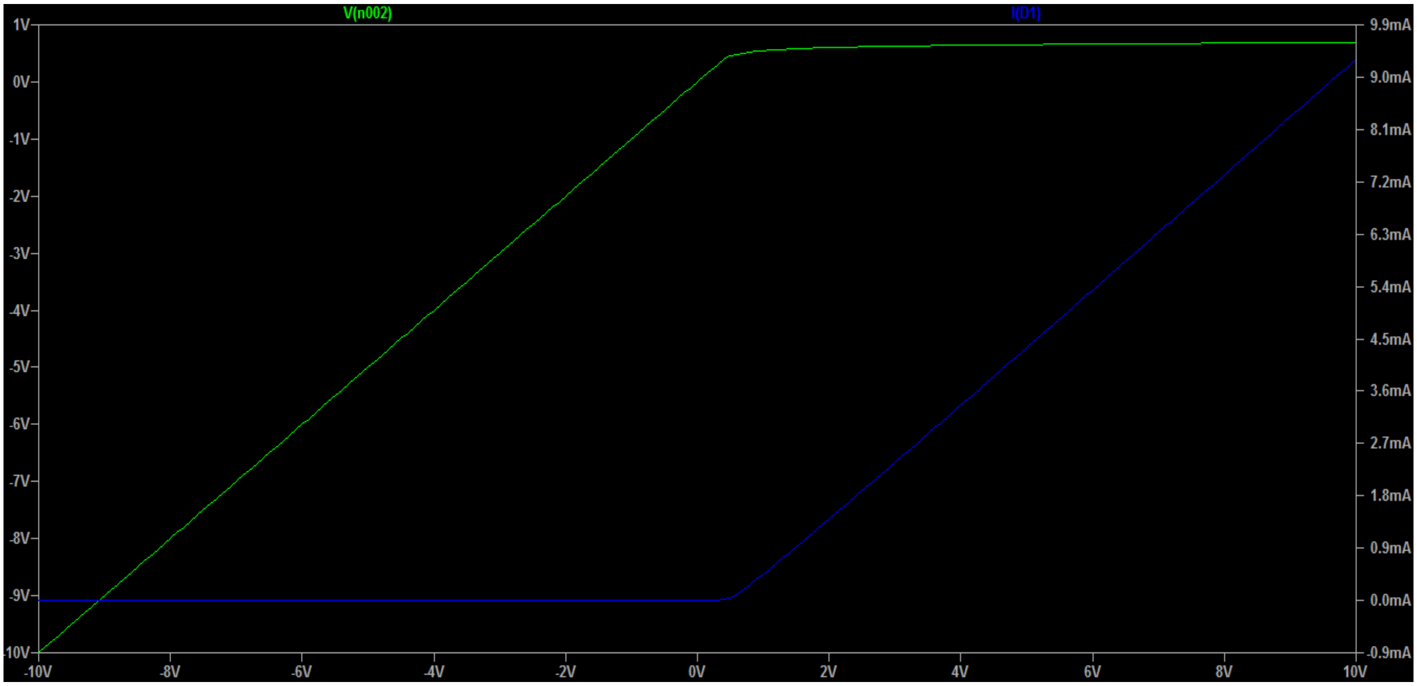
EEE2045F
Analog Electronics
RONAK MEHTA
MHTRON001

April 23, 2018

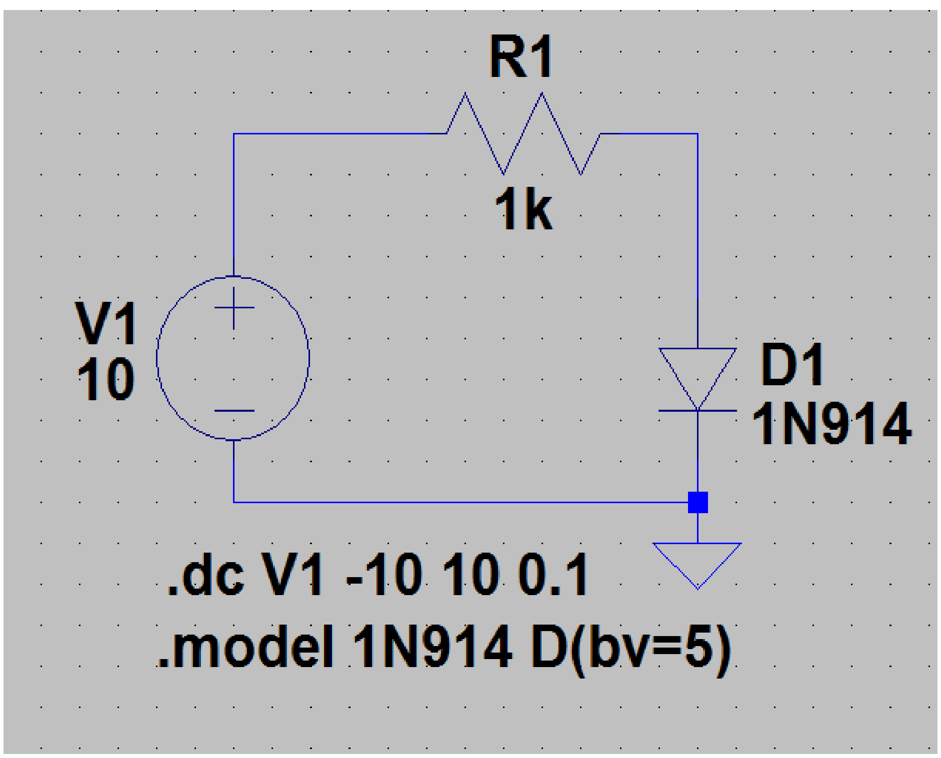
1. VI Characteristics

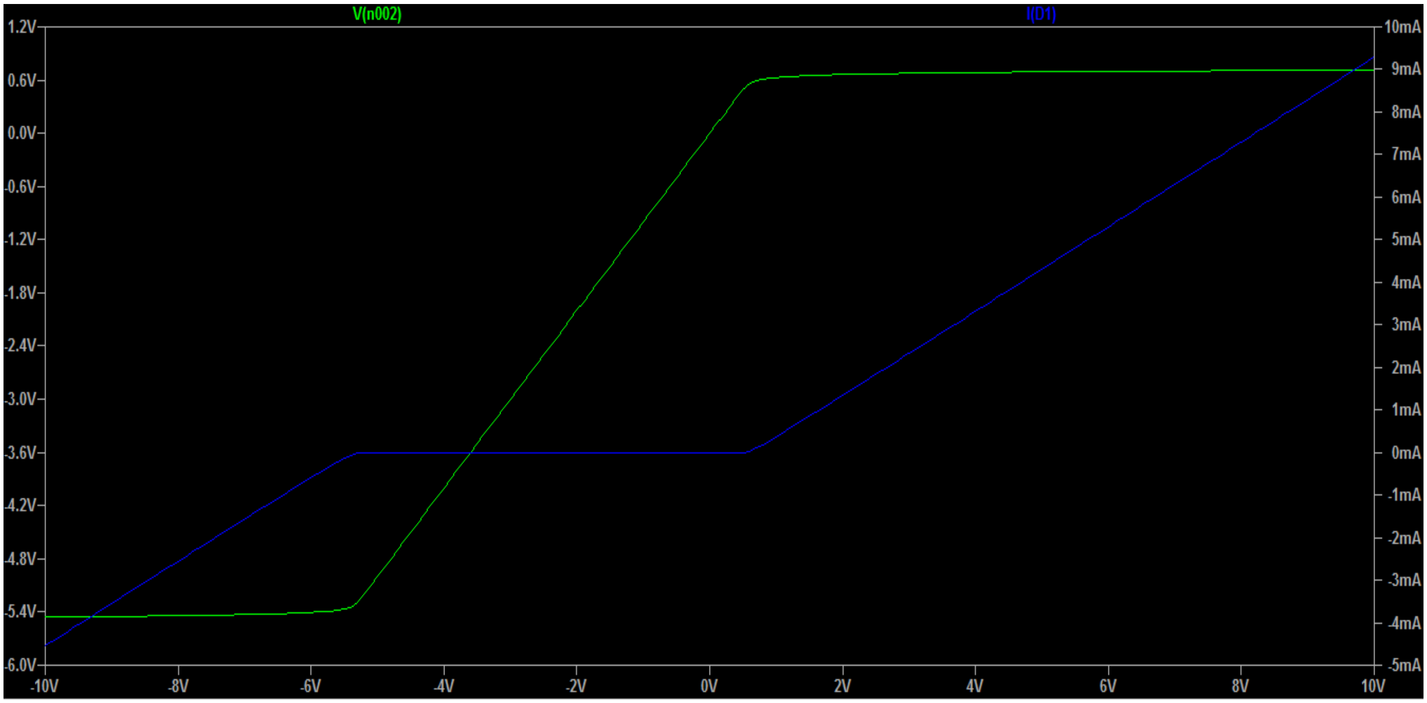
(a) 1N914 diode VI Characteristics





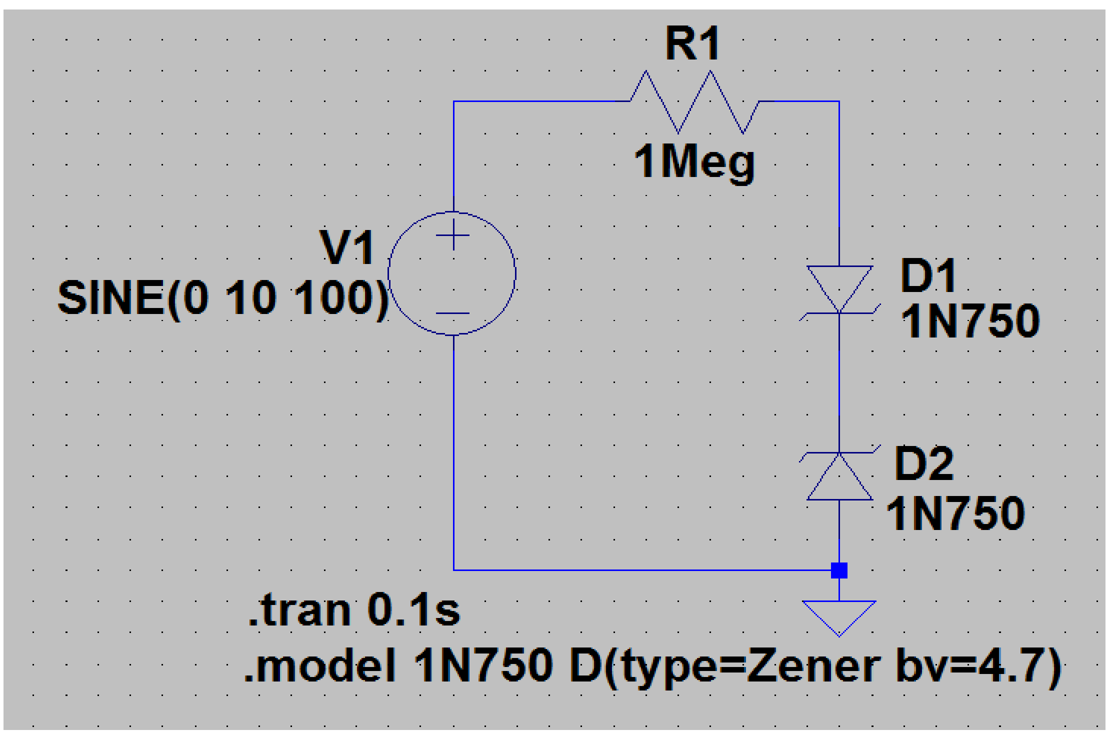
(b) 1N914 diode with reverse breakdown voltage at 5V V-I characteristics.

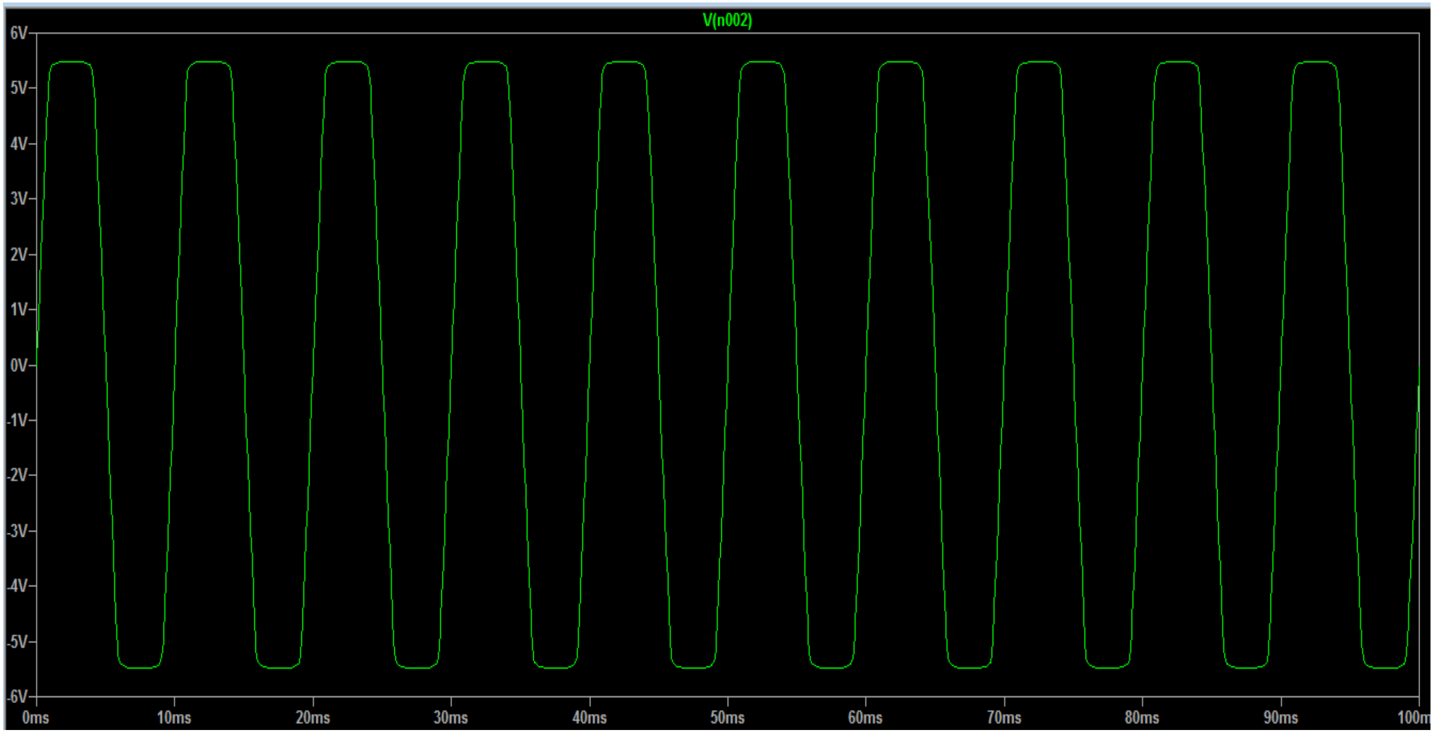




2. Clamping Circuits

(a) Results obtained

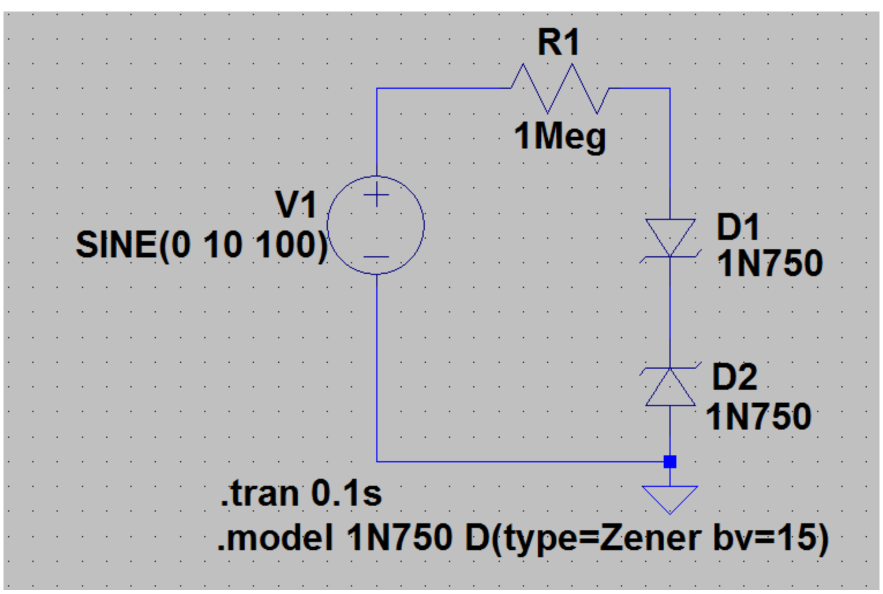




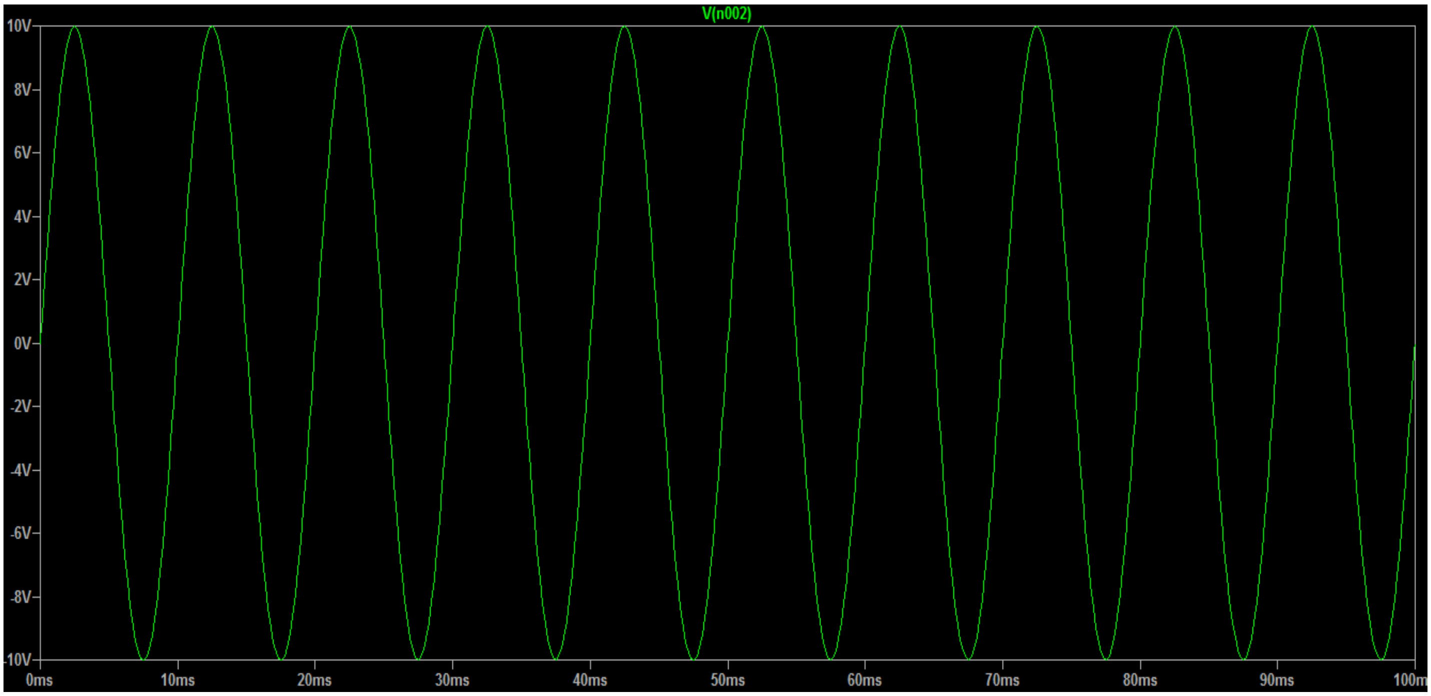
(b) Explain why the results shown in (a) were obtained.

These results were obtained as the output voltage is clipped due to the Zener diodes. For positive output voltage, V^+ = (0.7V from D1) + (4.7V from breakdown voltage of D2) = 5.4V and for V^- = -0.7V – 4.7V = -5.4V

(c) Result after varying the breakdown voltage to 15V



If the Reverse breakdown voltage decreases from 4.7V, the clipping of the output voltage increases but if the breakdown voltage increases from 4.7V, then the output voltage clipping decreases and we get a more sine wave-like structure. Once a specific breakdown voltage is obtained (around 9.3V), the output voltage will no longer be clipped be will look as below:

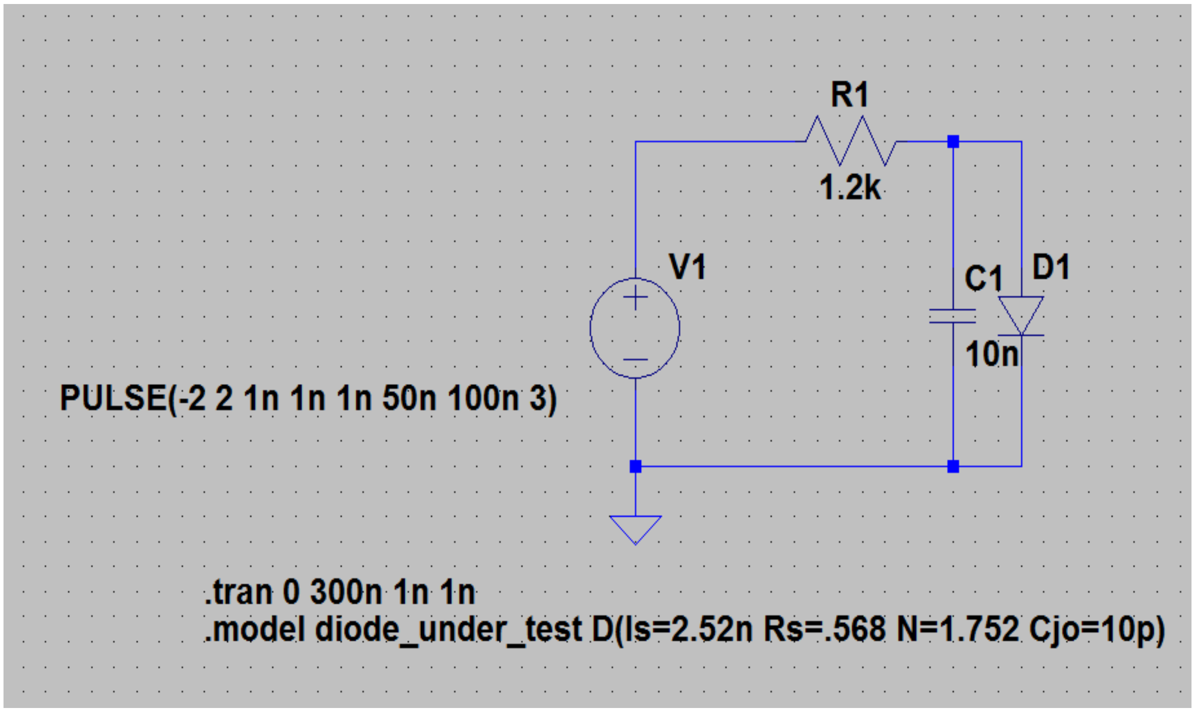


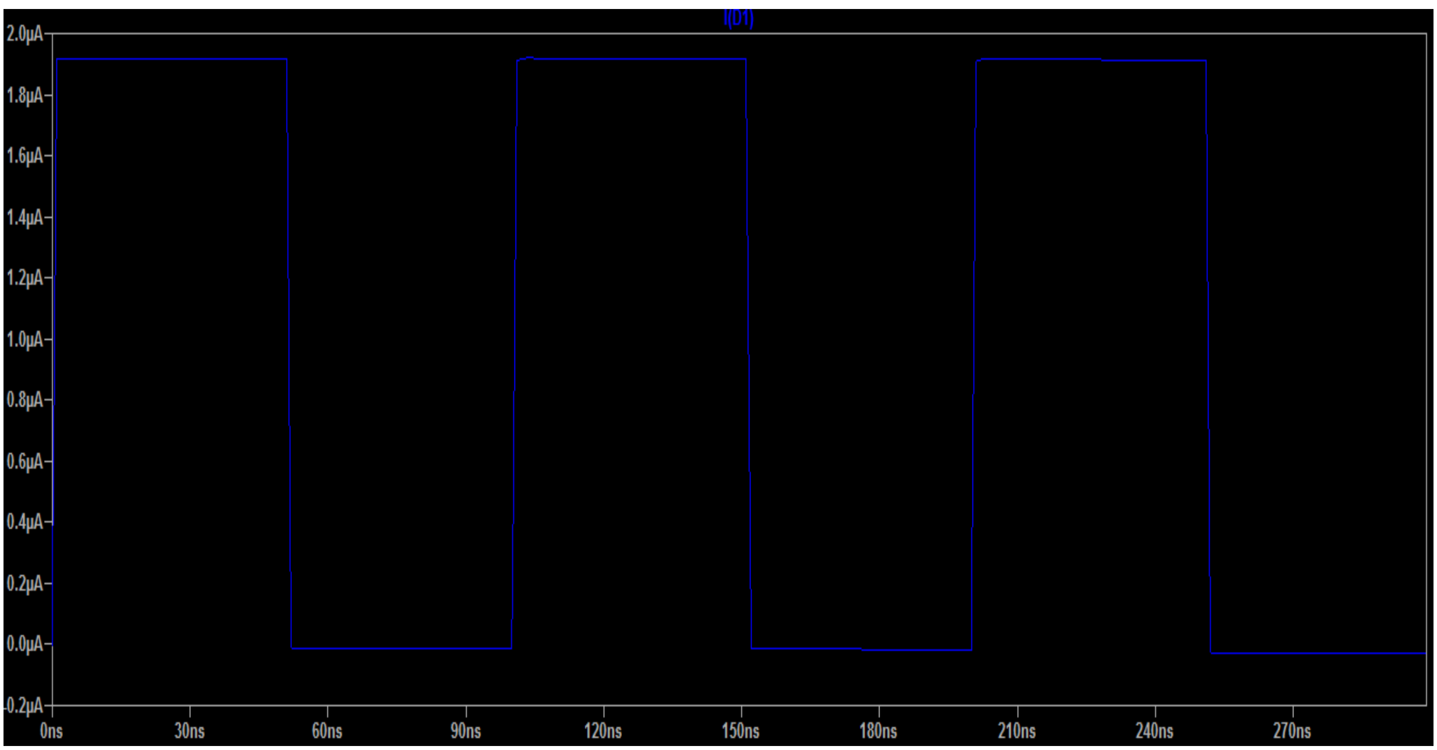
3. Diode SPICE Parameters

(a) Which model parameter of the diode is causing this uncharacteristic waveform?
=> The **Junction Capacitance** is causing this uncharacteristic waveform.

(b) Explain how to rectify this problem and prove this with the 'fixed' output waveform.
=> We can rectify this problem by connecting a Capacitor in parallel to the diode.

Proof:


R1
1.2k
V1
C1
10n
D1
PULSE(-2 2 1n 1n 1n 50n 100n 3)
.tran 0 300n 1n 1n
.model diode_under_test D(Is=2.52n Rs=.568 N=1.752 Cjo=10p)


I(D1)
2.0µA
1.8µA
1.6µA
1.4µA
1.2µA
1.0µA
0.8µA
0.6µA
0.4µA
0.2µA
0.0µA
-0.2µA
0ns
30ns
60ns
90ns
120ns
150ns
180ns
210ns
240ns
270ns

4. (a) From your understanding, when would you use each of the following simulation commands in LTspice?

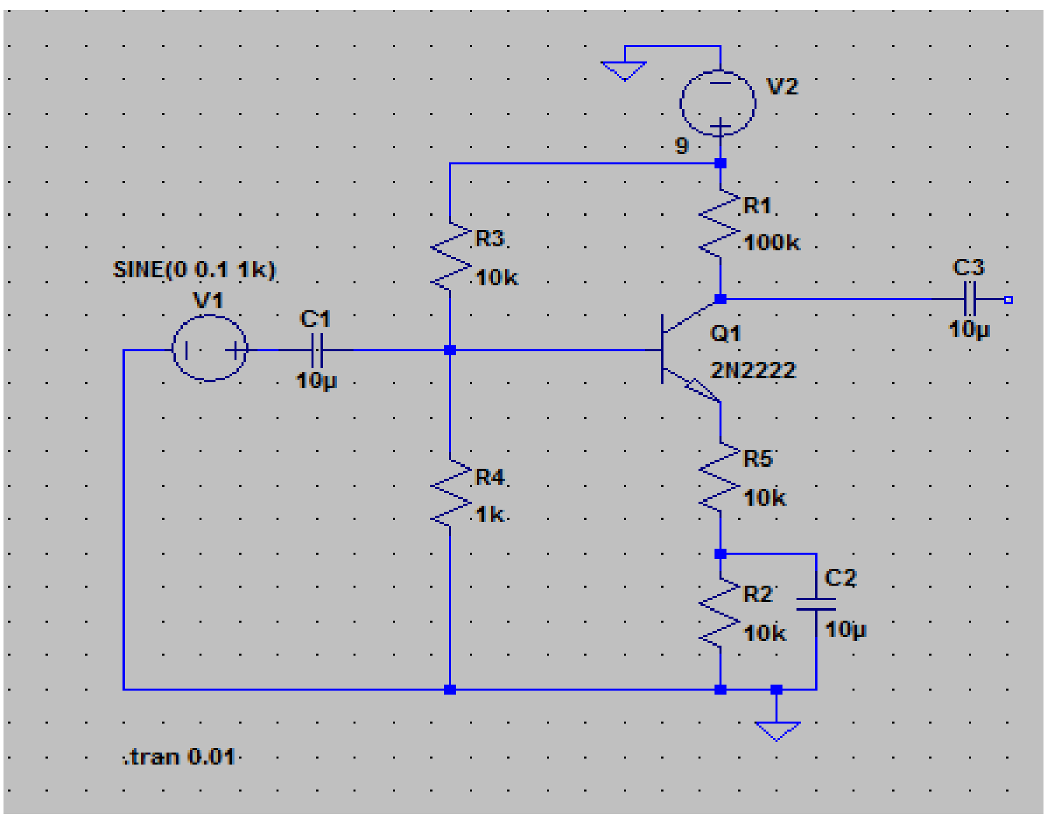
i. DC Operating Point – Can use this command when I want to show/find out the DC currents and DC Voltages in all nodes through every device in my circuit. While calculating this value, it interprets all Capacitors open circuits and all Inductors as short circuits.

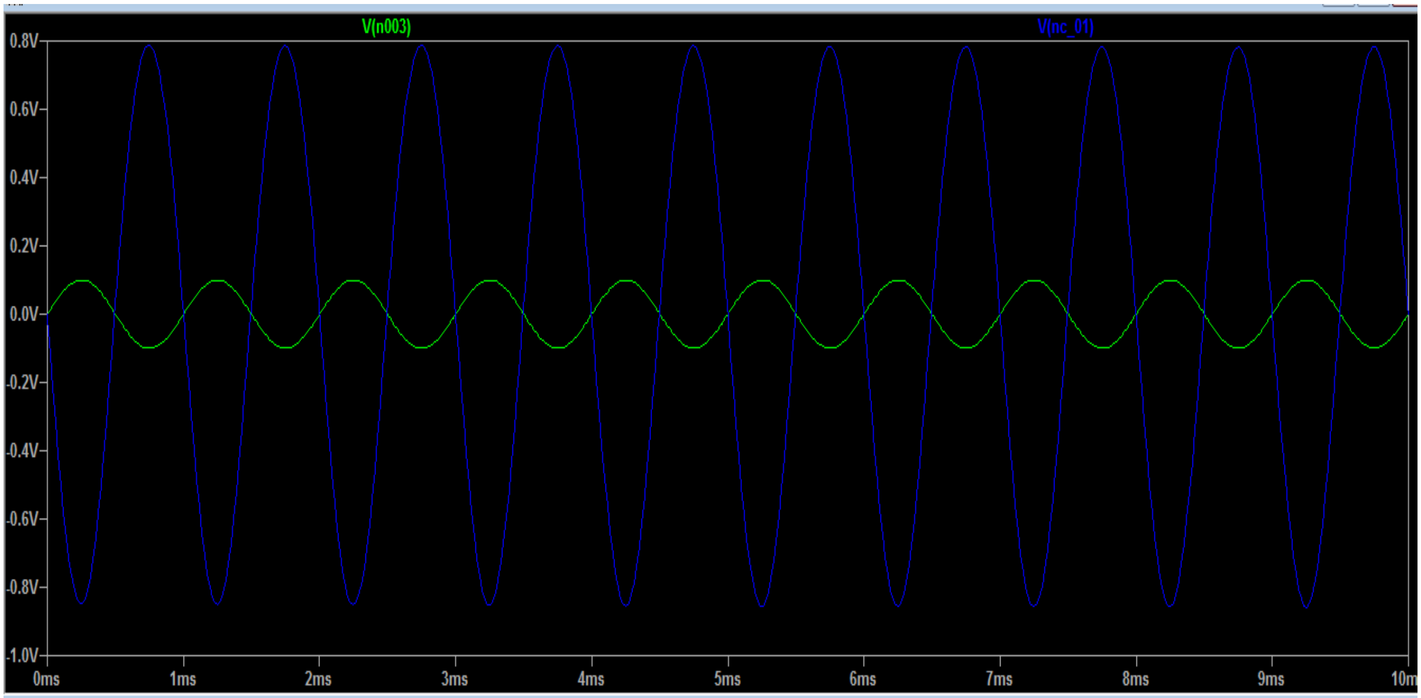
ii. **Transient** – This helps to calculate different values in your circuit over a range of time. We can look at any signal at any node at any specified time.

iii. AC Analysis – We can use this command when the supply voltage is an AC source and it is used to calculate AC behavior of signals which is linearized about its DC operating point. We can use this function to establish a relationship between an amplitude or phase of a sine wave with its frequency. In an AC analysis, all AC sources are thought of as sine waves.

iv. DC Sweep – We can use this command when we want to change DC voltages from one value to another to see how it reacts and responds to various conditions. We have to allocate a start value, an end value and a value for increment.

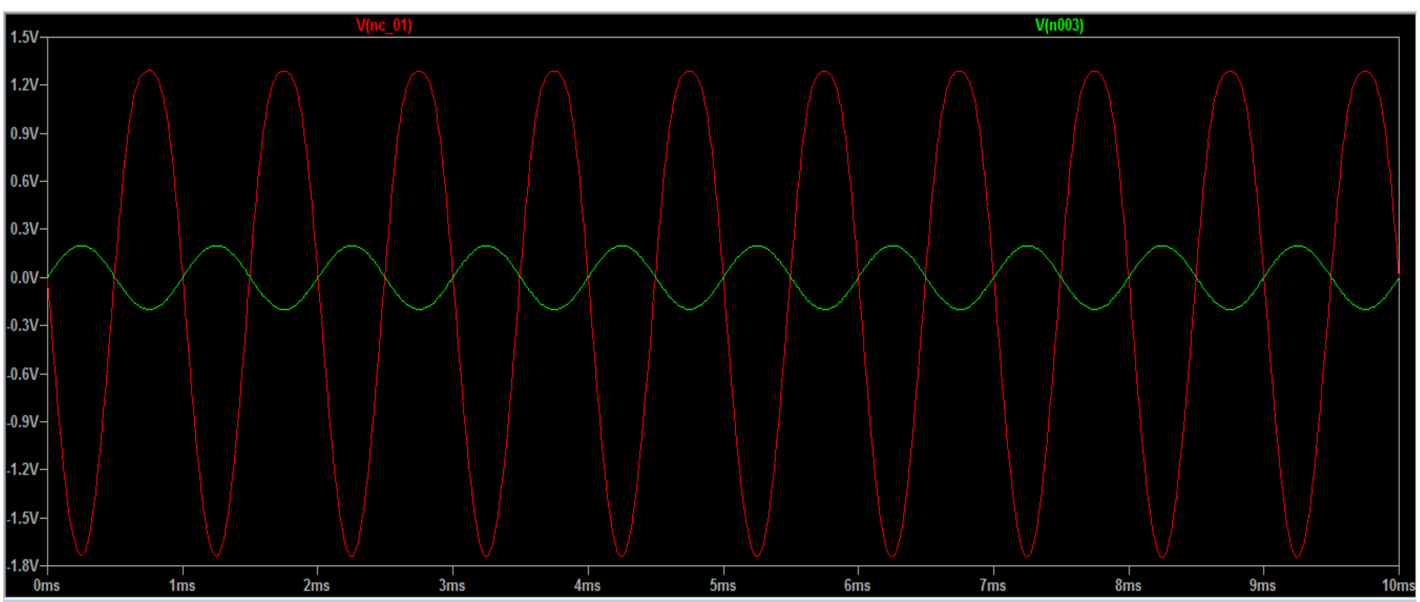
5.





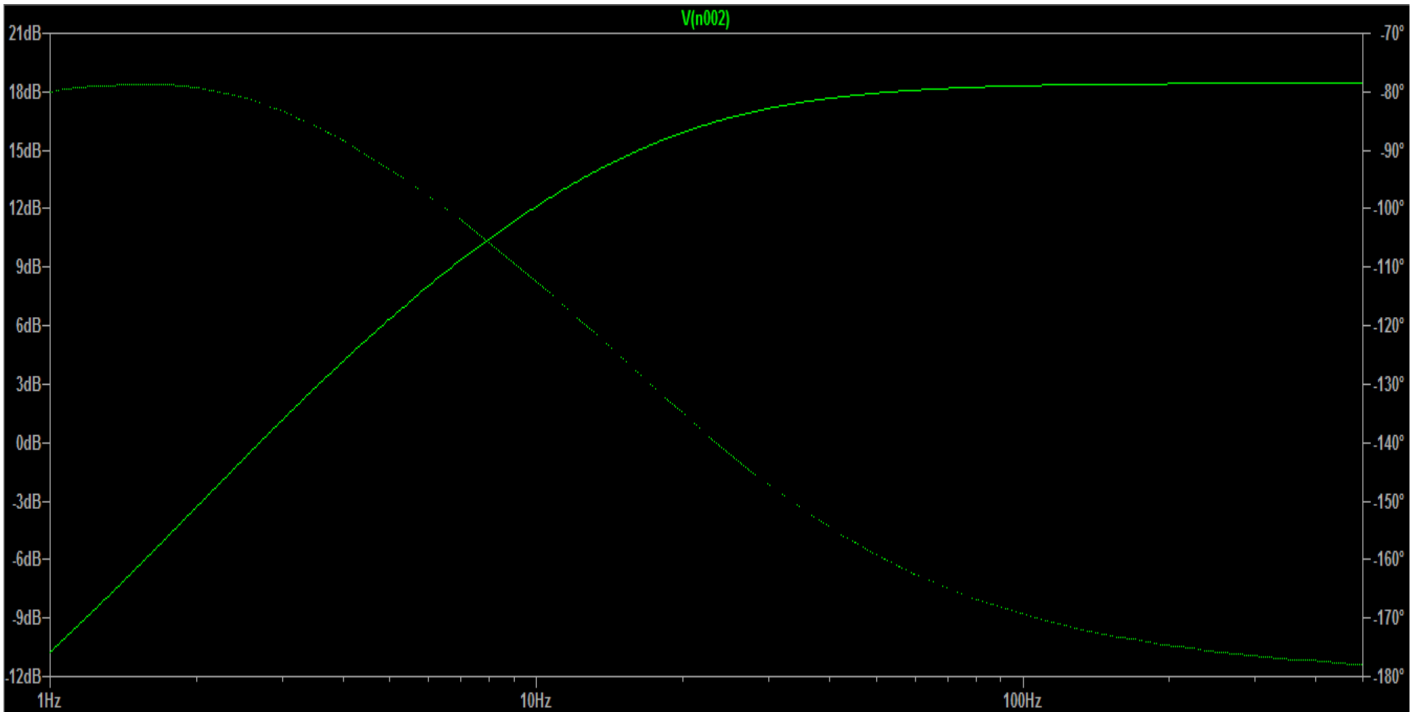
i. How can you increase the gain without affecting the DC bias?
=> We can do this by removing the bypass capacitor in the emitter region.

ii. How far can you drive the output before the voltage starts to distort? HINT: Vary the input sine wave to a higher value to observe output variations. Its normally very small in mV.
=> According to my resistor values, the highest voltage I can drive the output before distorting the signal is 0.2V



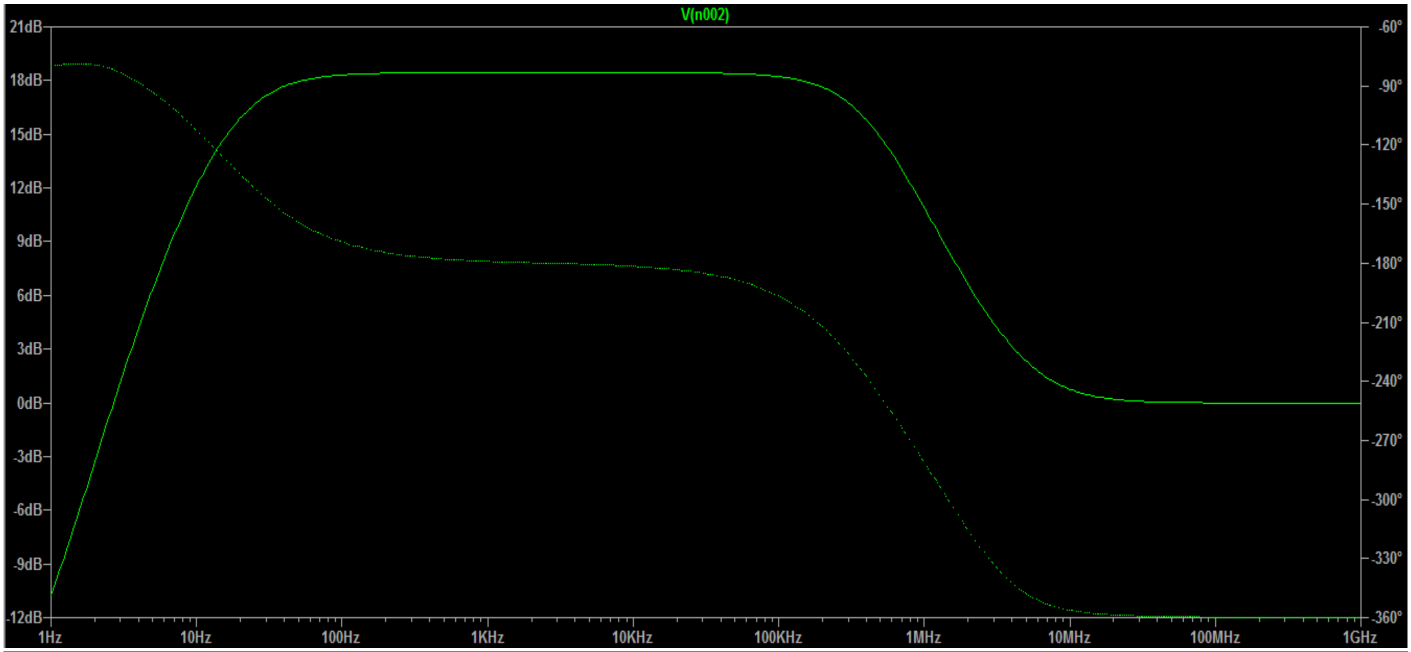
iii. Run AC simulations to observe the frequency response What is the frequency range of this amplifier you designed? What are the upper and lower cutoff frequencies? How can you change the lower cutoff frequency of your amplifier?

LOWER CUT OFF FREQUENCY:



The Lower cut off frequency would be:
18db – 3db = 15db;
then the frequency at 15db is **20Hz**
Thus, the Lower cut off frequency is 20Hz

UPPER CUT OFF FREQUENCY:



The Upper cut off frequency is **300KHz**

Thus, the frequency range of my amplifier is **20Hz <= f <= 300KHz**