Department of Design Engineering and Mathematics

Middlesex University

Transistor Amplifiers

PDE2431 Analogue and Digital Electronics

BEng Electronics Engineering

November 2020

Introduction

A Transistor is a three terminal semiconductor device that can amplify the signal. Transistor is formed by connecting the two diodes back-to-back. It consists of N-type and P-type doped semi-conducting material. The common name of the terminals is base (B), collector (C) and emitter (E).

Transistors can be used in diverse ways depending on the region of operation it has three regions: -

- Cut-off region – in cut off region it works as switch 'off' by insufficient of base current and stops flow of collector current.
- Saturation region-in this region the transistor works as switch 'no' as sufficient base current allow the flow of collector current.
- Active region-in this region the transistor act as amplifier by rising the strength of the collector current.

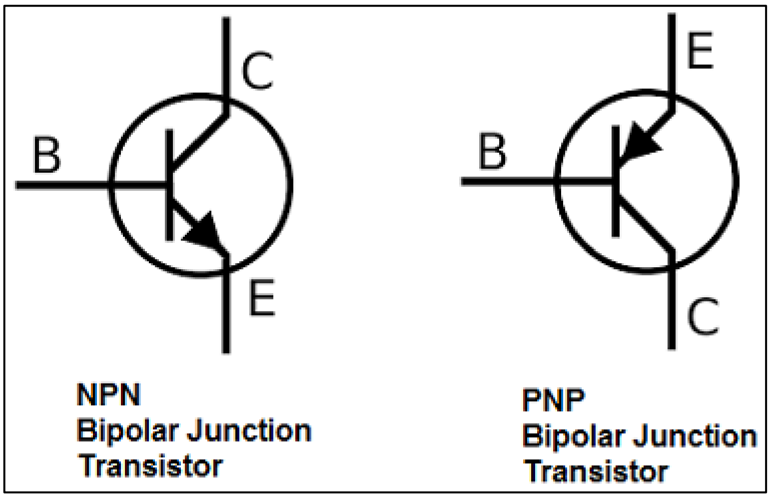

Figure 1:- Type of Bipolar Junction Transistor

Transistors are used in three basic configurations to amplify the signals.
- Common collector amplifier(emitter follower)
  - Used for voltage buffer or current gain
  - Base is input, emitter is output and collector is common
- Common base amplifier
  - Used as current buffer or voltage gain
  - Emitter is input, collector is output and base is common
- Common emitter amplifier
  - Used as power gain
  - Base is input, collector is output and emitter is common

From this different configuration we can use the transistors in varies applications as voltage regulator, microphone preamp and voltage amplifier.

The Aim of the experiment is to calculate and measure the voltage and current gain of various transistor amplifier circuit. We will also look at other factors effecting the gains (like emitter resistor in emitter follower circuit).

Procedure

Section 1

- The link contains Multisim Live common-emitter transistor amplifier circuit https://www.multisim.com/content/LpZjfREW4Tir7MEbPeTY2Z/interactive-trannsistor-amplifiers-circuit/open
- Run the simulation in interactive and get a graph.
- From the graph calculate the voltage gain of the circuit using
  - Voltage gain = AV = V output/V source(input).
- Now again run the circuit in ac sweep simulation having 10Hz as starting frequency and 5k Hz as ending frequency.
- From the graph of ac sweep measure the gain at 100Hz.
- Now with the help of Tinkarcad built the circuit as in figure 2 by placing the f components given in table 1: -

Components	**properties**
Breadboard	Small
Function generator	Square wave
Oscilloscope ×2	Read current
Resistor(Ω) ×4	8.2k, 18k, 1k, 560
Capacitor (nano faraday) ×3	1, 22, 100
BJT transistor	NPN

Table 1:- List of components for tinkarcad circuit

- Wire the components as shown in image below
  - Red for positive
  - Black for negative
  - Green for data
  - Blue for Oscilloscope data

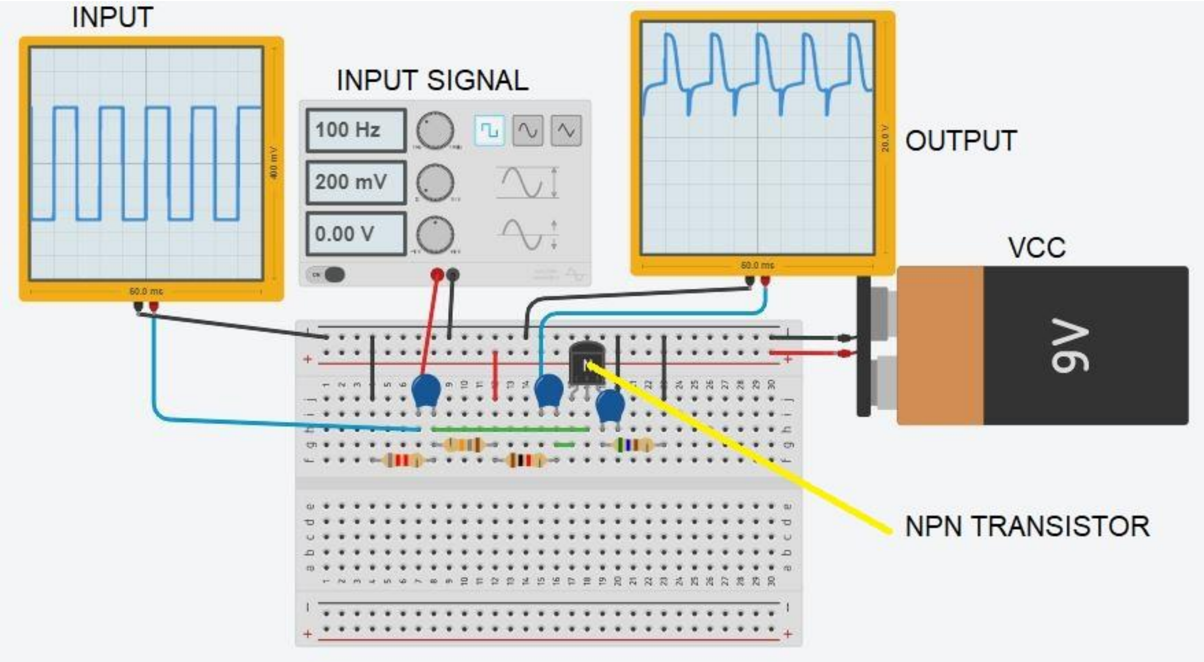

Figure 2:- Tinkarcad circuit

- Now set the function generator (INPUT SIGNAL) to the following value:
  - Output signal waveform at sine wave
  - Frequency of 100Hz
  - Amplitude at 0.2V peak to peak
  - DC offset of 0V
- Now step both the oscilloscope's time per division to 5 microseconds
- Set the same circuit in the NI ELVIS breadboard as shown in figure3 and then connect the board to the computer

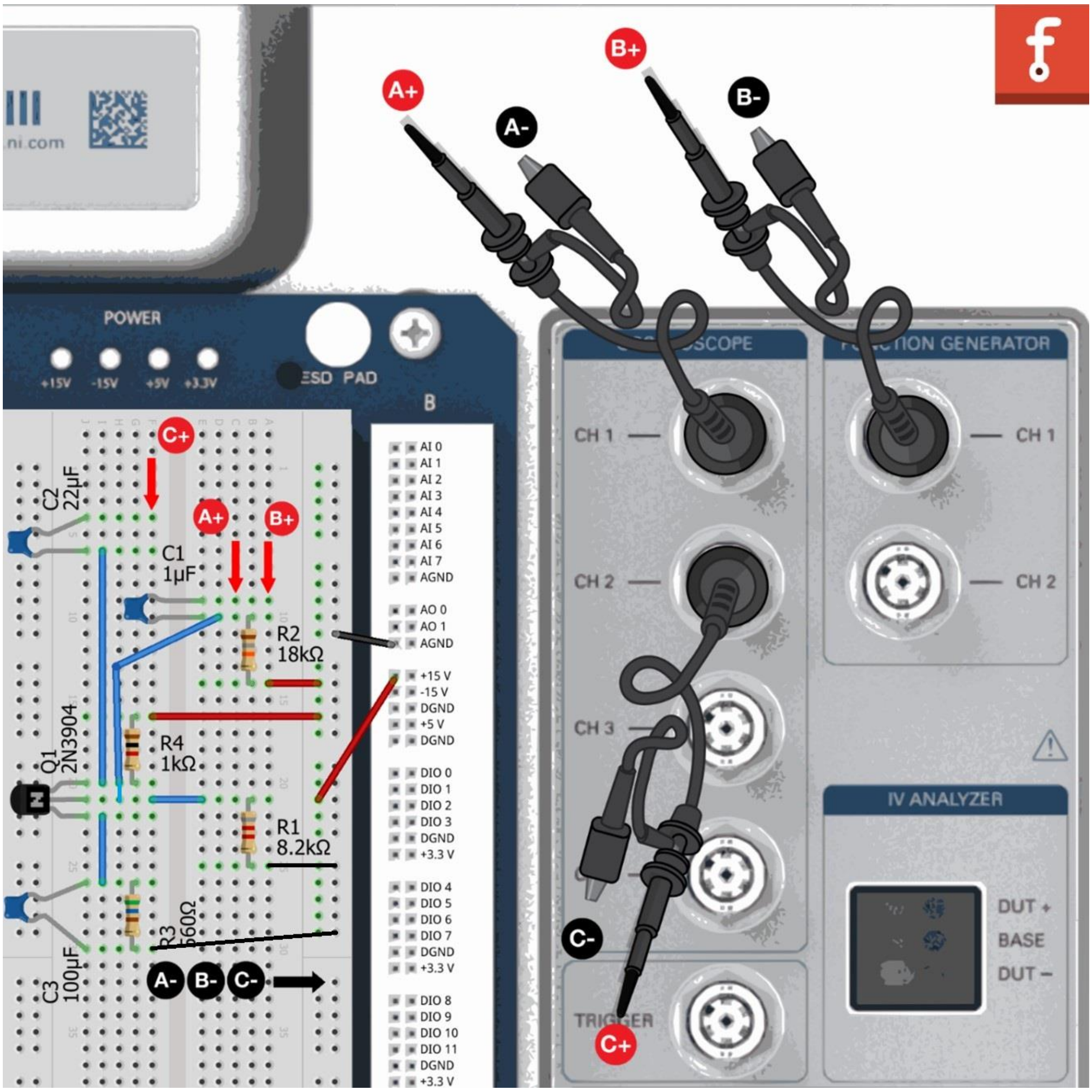

Figure 3 circuit on Elvis board

- ➢ Add a function generator from instrument lab of Measurements Live and set frequency at 100Hz and amplitude of 0.2Vpp
- ➢ Calculate base current and emitter current for β is 100 base voltage 0.7 for the circuit.

$V_{TH} = V_{cc} \times R_2 / R_1 + R_2$

$V_{TH} = 9 \times 8.2/8.2 + 18$

$V_{TH} = 2.816\ V$

$R_{TH} = R_1 \times R_2 / (R_1 + R_2)$

$R_{TH} = 18k\Omega \times 8.2k\Omega / (18+8.2)\ k\Omega$

$R_{TH} = 5633\Omega$

$I_E = V_{TH} - V_{BE} \ / R_E + (R_{TH} / \beta)$

$I_E = 2.816 - 0.7 / 560 + (5633/100)$

$I_E = 3.433$ mA

$I_B = V_{TH} - V_{BE} \ / (\beta \times R_E) + R_{TH}$

$I_B = 2.816 - 0.7 / 100 \times 560 + 5633$

$I_B = 34.3\mu A$

- Measure the current by modifying the circuit as below

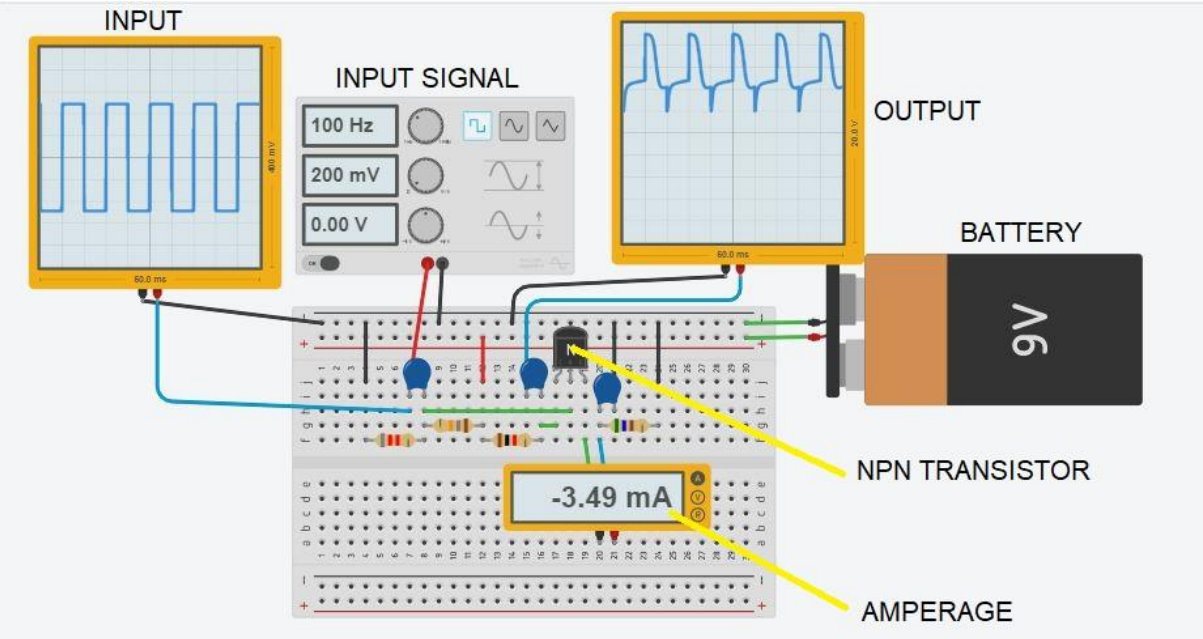

Figure 4 Tinkarcad circuit to measure emiter current

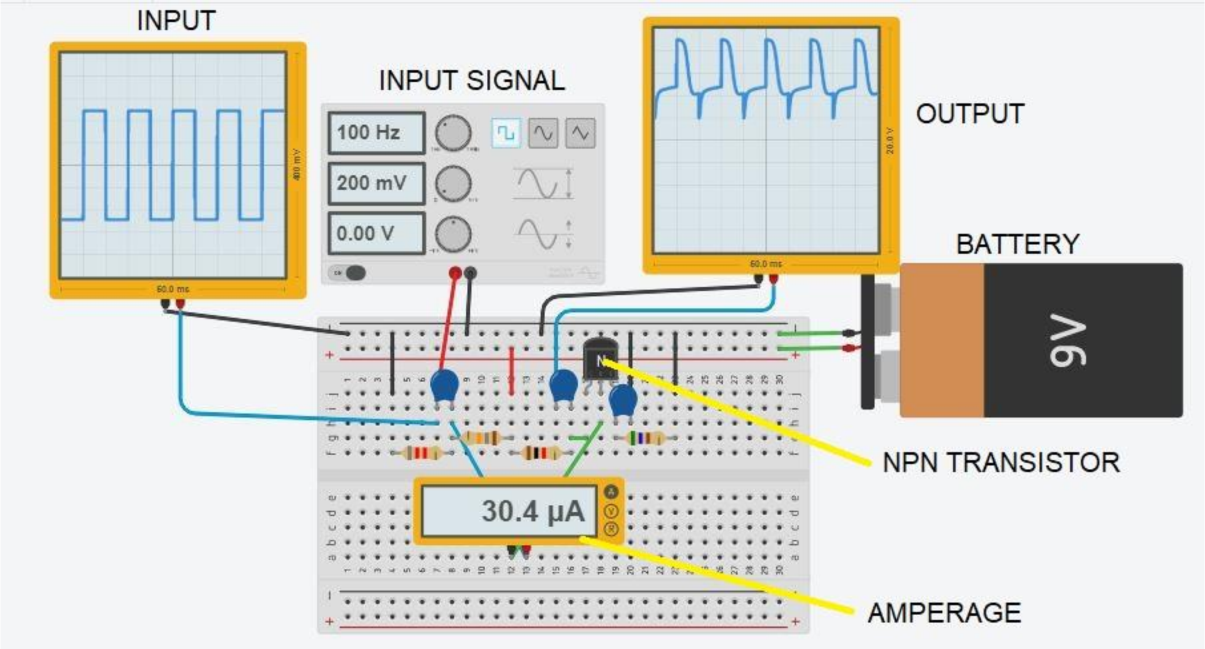

Figure 5 Tinkarcad circuit to measure the base current

- Add oscilloscope from the instrument lab of measurements Live. And configure Time/div to 2ms in Horizontal and Acquisition
- Run the function generator and oscilloscope and with the help of the graph calculate the linear gain in decibel.
- Open the bode analyzer from measurement Live and configure as the frequency range between 10 to 5 kHz
- From the measurement graph and Use the cursors to find the gain at 100Hz.

Section 2

- The link contains the multisim live simulation for the emitter follower Amplifier circuit for voltage response. https://www.multisim.com/content/zGWUhynstWEfCHexu6GREc/emitter-follower-v/open
- Run the stimulation and observe the graph of a voltage follower circuit.
- The link contains the multisim live simulation for the emitter follower Amplifier circuit for current response. https://www.multisim.com/content/B2DuX5YtreVZMtNVj6r2gW/emitter-follower-a/open
- Run the circuit and observe the current response in graph.
- With the help of graph measure the peak to peak values and calculate the gain

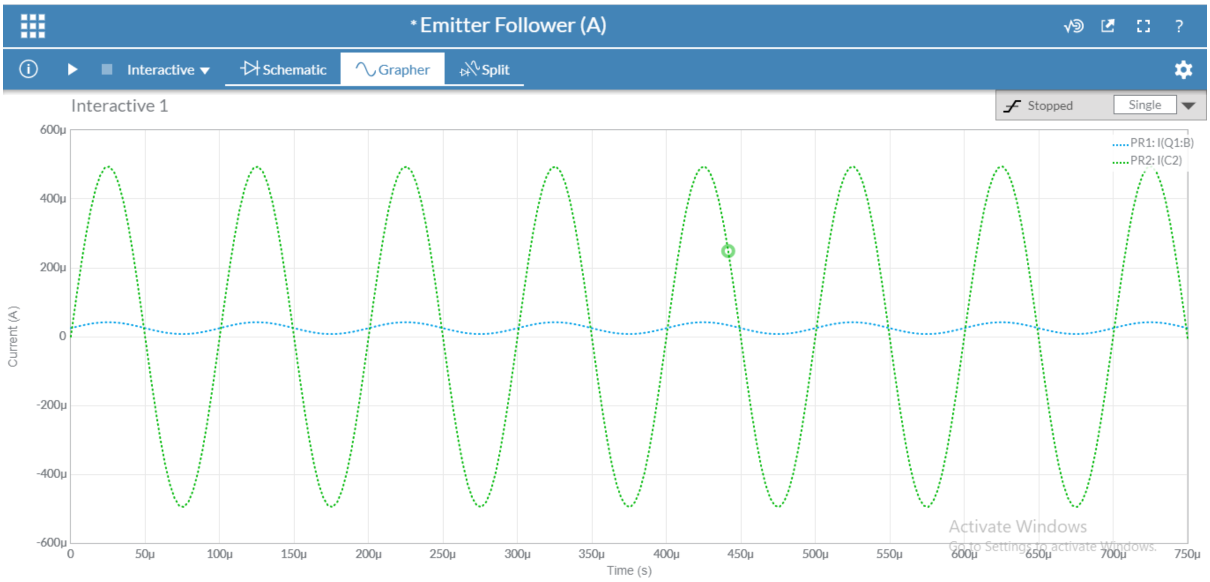

Figure 6 Current response of Emitter following circuit

From the graph the peak to peak value are
Input current is 41.7and 7.5µA giving the input current of 34.2µA
Output current is 500and -500µA giving the output current of 1mA
Using $A_{(gain)} = I_{out} / I_{in}$
$A_{(gain)} = 1000/34.2$
$A_{(gain)} = 29$

- Now try changing the value of emitter resistor at 1k,3k,5k and record the peak currents of the following.

Resistance of RE	**Probe 1 – Input Current (µA)**	**Probe 2 – Output Current (µA)**	**Gain**
1k	76-17.5=58.5	1000	17
2k	34.2	1000	29
3k	29.5-3.7=25.8	1000	38.7
5k	20-0.5=19.5	1000	51.2

Table 2 Change in current with respect to emitter resistor

Results

The graph (figure7&8)shows the difference in voltage in input and the output of a stimulation transistor amplifier circuit in multisim live. From the graph the input voltage(Vpk) is 0.1V and the output voltage is 3V. Using the formula, the gain of the circuit is of 30 and this was also verified by the ac sweep graph.

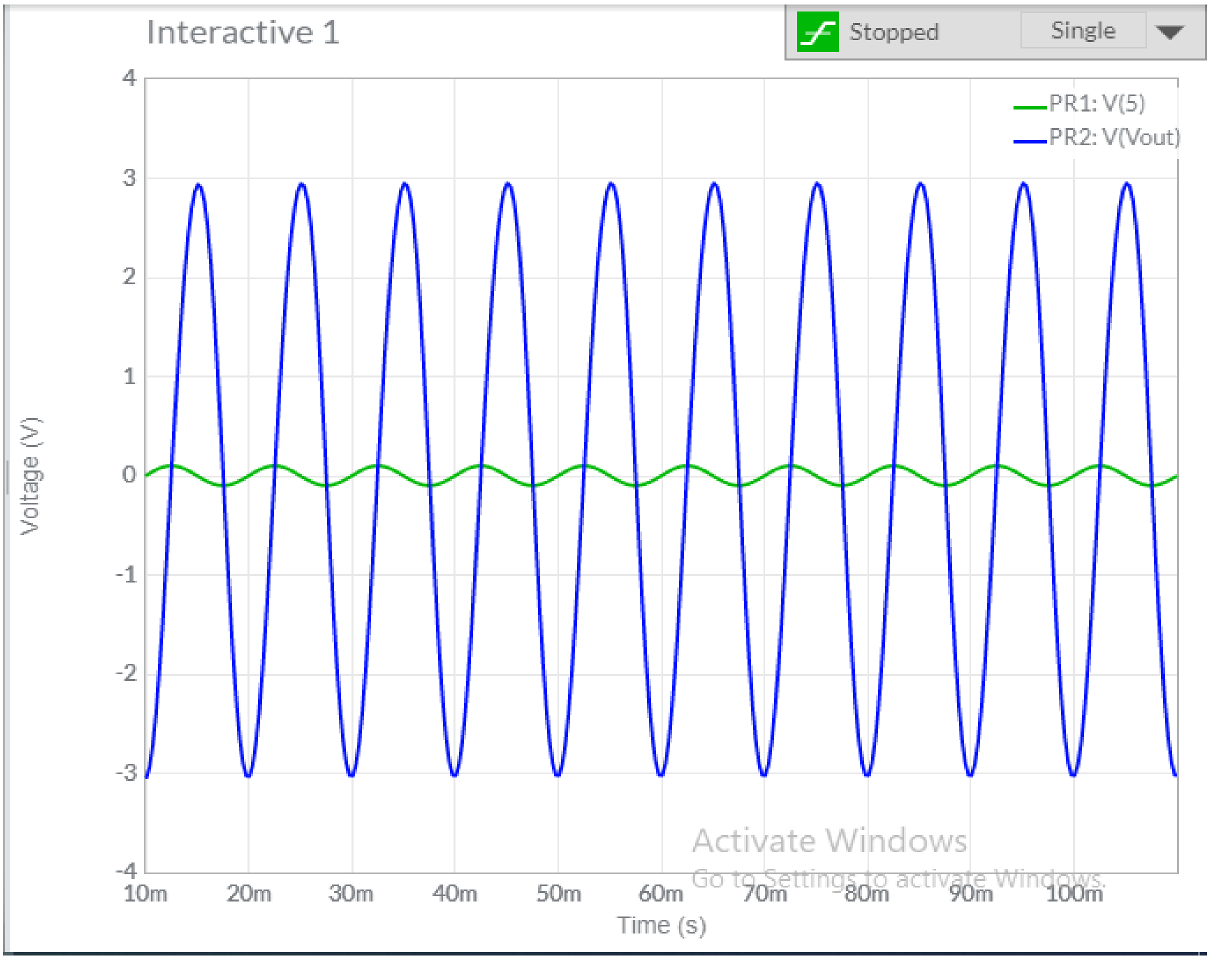

Figure 7:-simulation in interactive

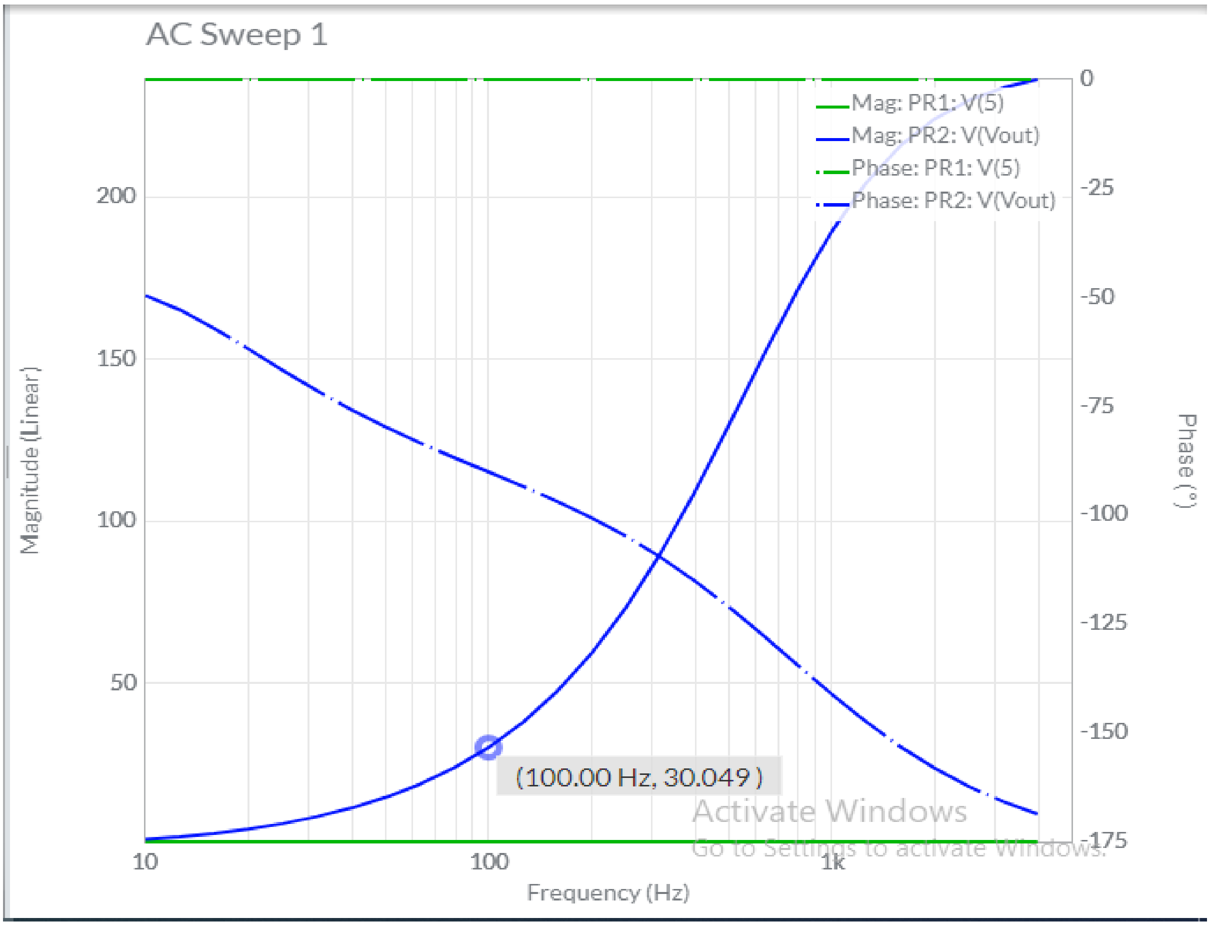

Figure 8 :- AC sweep graph of simulation

The graph(figure 9) from measurement live oscilloscope is show quite like the multisim live interactive graph. And that was also confirmed gain of 30 in bode analyser at frequency of 100Hz.

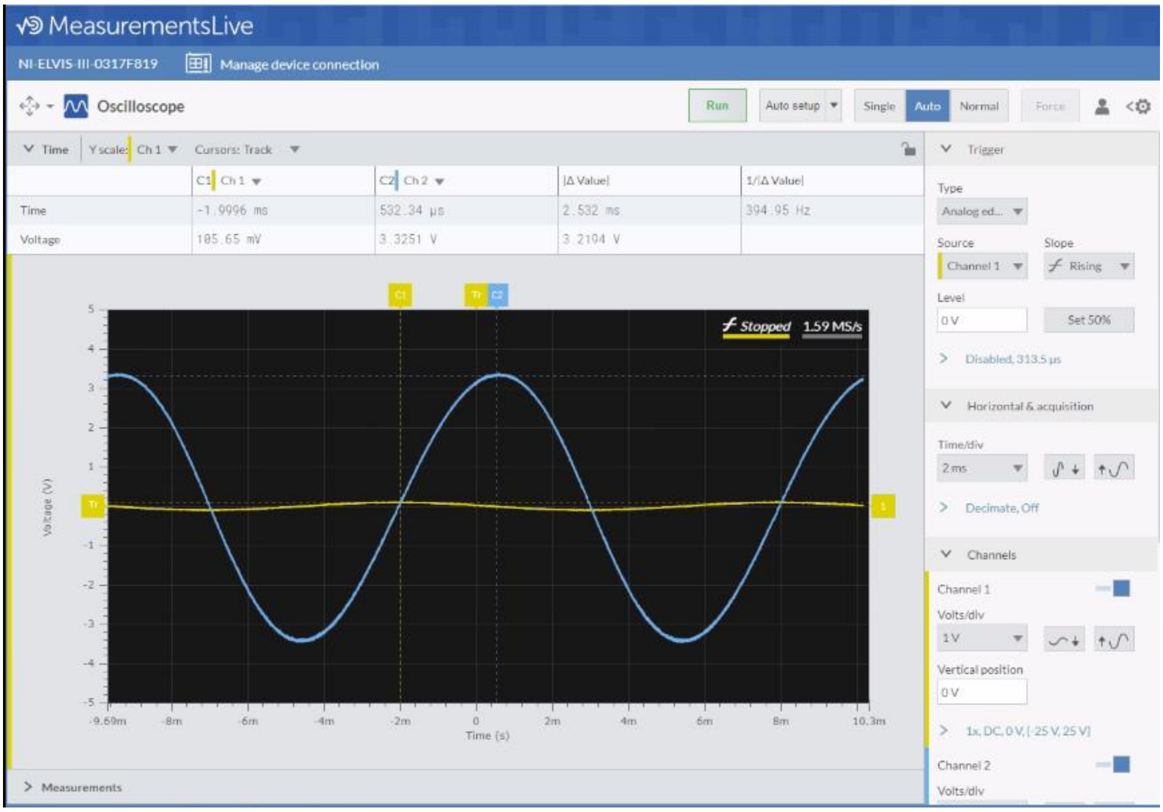

Figure 9:- Measurement live oscilloscope

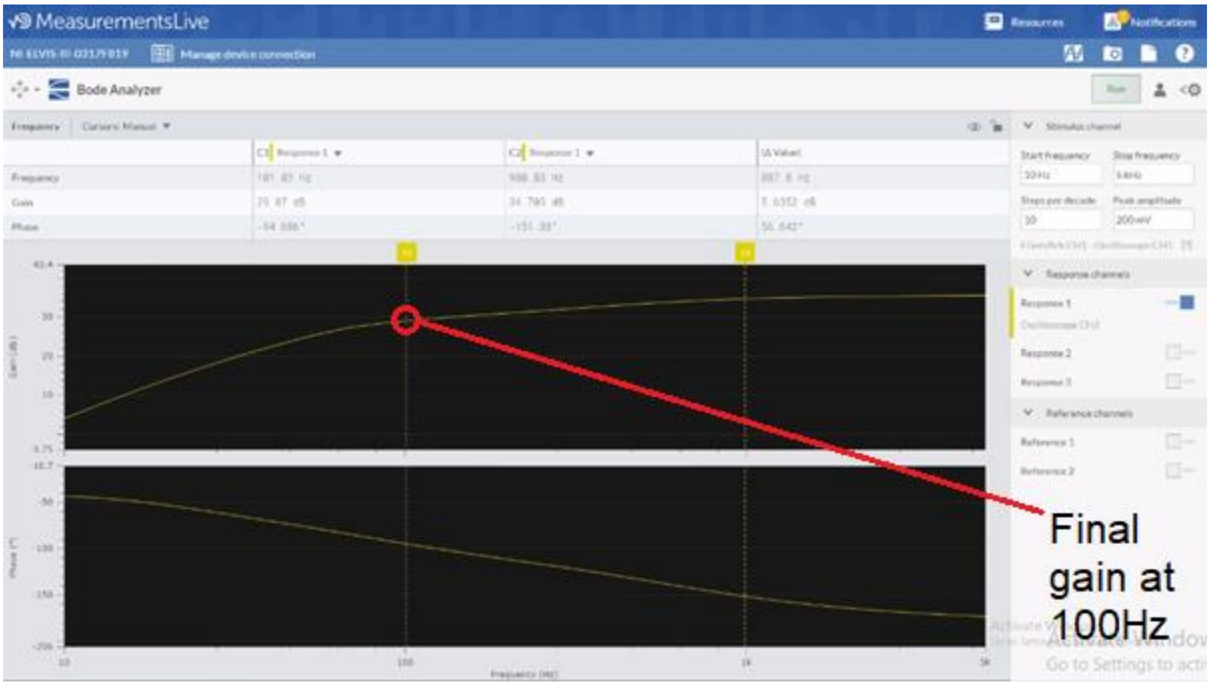

Figure 10:- Bode analyzer of Measurement live

Section 2

The voltage response of the emitter follower circuit the output signal is overlapping the input signal at any given point as the output voltage is following input voltage.

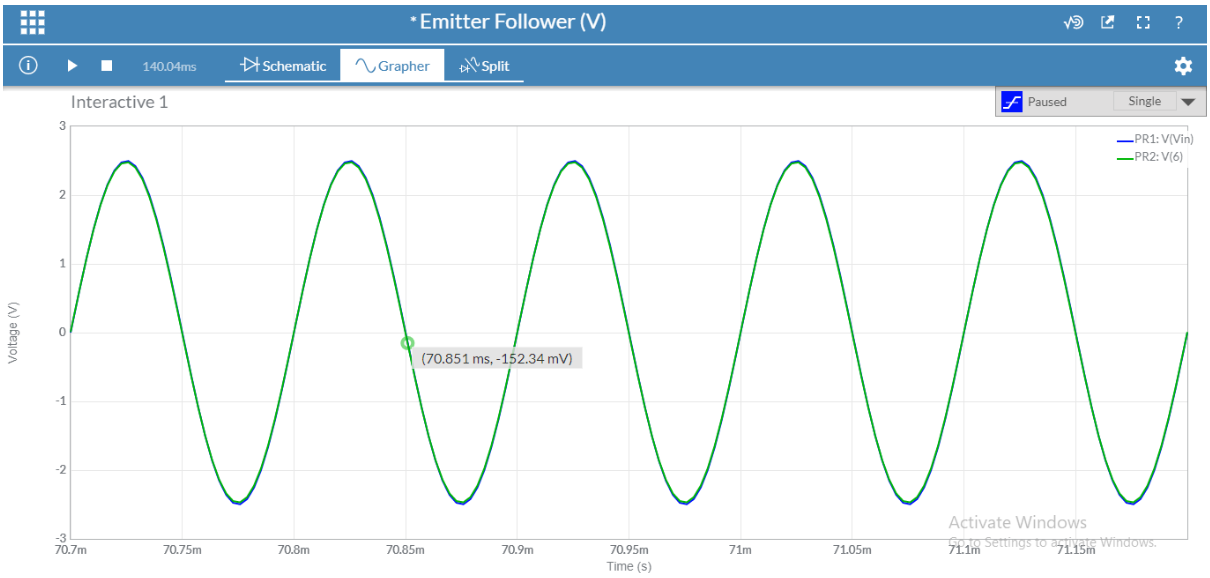

Figure 11:- Voltage response of emitter follower

The current response of the following graph was different we can see the Output current more compared to input current as shown in the graph (figure 6).We also see that the change in emitter resistor will only change the base current to lower value. But no change in output current.

Conclusion

We understand how to use transistor as amplifier in 3 separate ways depending on what is required to amplify. We also got to know on what factors do the amplification depends such as resistance, V_{CC}, input current and voltage. In this lab we got the calculated gain of 30 and it was confirmed in the practical with oscilloscope and in stimulation showing the similar graph.I learnt how to increase current without changing voltage of phase. There can be an error of can be accurate power supply (Vcc value), human error and use of equipment. For this reason, we can use transistor amplifier in voltage amplifier, current amplifier, or microphone preamp.