IJARSCT

ISSN (Online) 2581-9429

International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)

International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal

Impact Factor: 7.301

Volume 3, Issue 2, September 2023

Application of Multisim Simulation Software in Teaching of Applied Electronics

Aphasana Mulla[1] and Wrushali Deshmukh[2]

Lecturer, Department of Electronics & Telecomm[1,2]

Bharati Vidyapeeth Institute of Technology, Navi Mumbai, India

Abstract: *In this paper, the application of Multisim in electronic design is studied through examples, and the specific steps of simulation analysis are discussed. Using virtual simulation software to simulate the circuit can not only be free from the influence of the experimental site and the instrument, but also avoid the risk of component damage and personal injury, and improve safety. At the same time, through virtual simulation instruments, students can observe the signal waveforms of various positions of electronic circuits in real time,deepen their understanding of the principles,*
and thus improving the effective method of classroom teaching

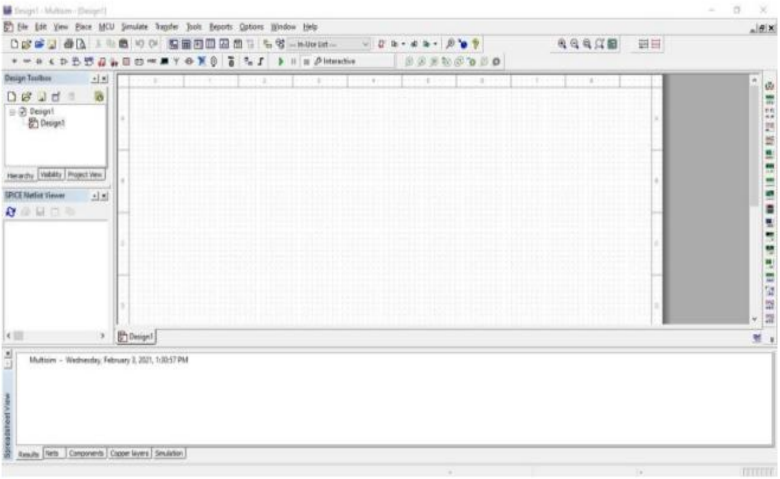
Keywords: Multisim, Applied Electronics, Application

I. INTRODUCTION

Multisim is an electronic design automation software launched by National Instruments, which is used to simulate analog and digital circuits [2-6]. Multisim software has an intuitive and easy-to-use operation interface, convenient component callout, intuitive component labelling, strong simulation reality, and high similarity to the actual experimental platform. The software has a full range of component libraries, a variety of test instruments, such as multimeters, oscilloscopes, signal generators, logic converters, logic analyzers, comprehensive simulation analysis methods, and rich simulation capabilities.[1]

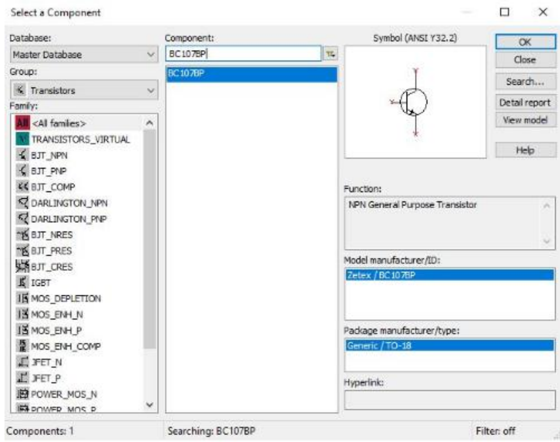
II. PROCEDURE

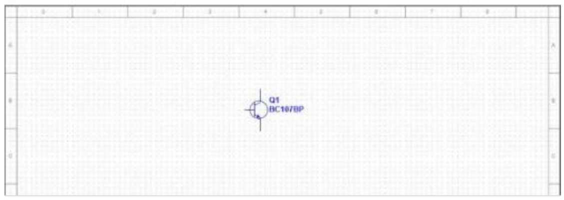
1) Start multisim tool, for the common emitter amplifier circuit we require the following components. Components include resistors, capacitors, transistors, voltage source, power source, and ground connection for designing this circuit.



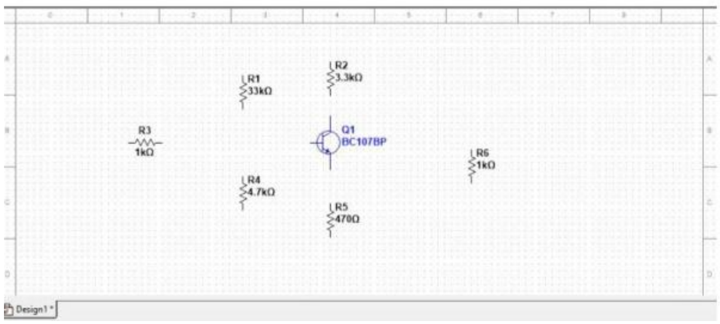
IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
Impact Factor: 7.301
Volume 3, Issue 2, September 2023

2) Click on the place or place transistor icon, then select a component pop-up that appears

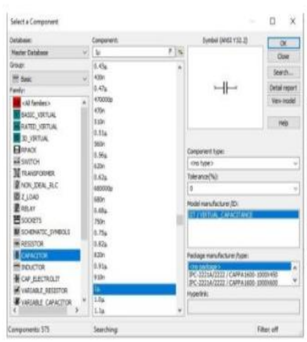
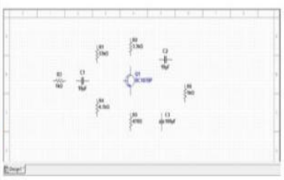




3) Now click on place, then select component, in group section select basic and then select resistor. In this multisim tutorial, we require the following resistor value 33k, 3.3k 1k, 47k,0.47k ohm for the circuit design.Place all the resistors as shown in the figure below on the multisim design window.



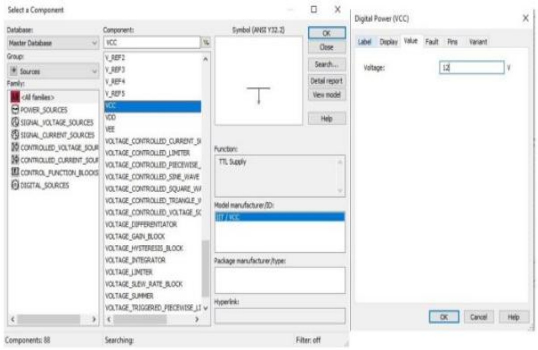
4) The next step in this multisim tutorial to place the capacitors of the following values 10uf and 100uf on the multisim design window. Click on place, select component, in group section select basic then select the capacitors.

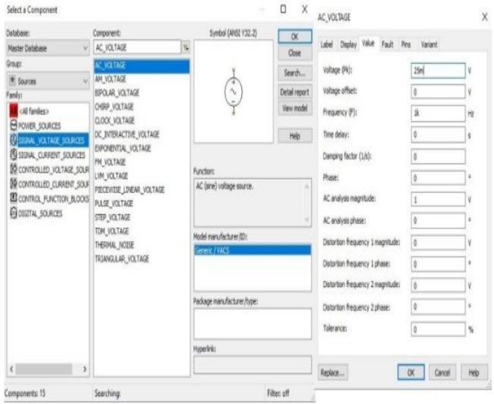
Copyright to IJARSCT
www.ijarsct.co.in
DOI: 10.48175/568
ISSN 2581-9429 IJARSCT

IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
Impact Factor: 7.301
Volume 3, Issue 2, September 2023

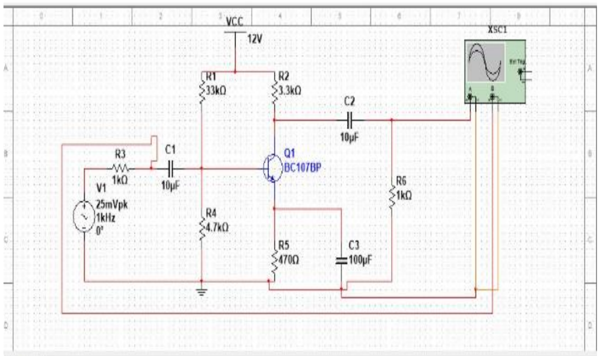
5) We require input source and TTL supply, click on place, and select component, then select the group as source and Vcc (TTL supply) click ok



6) Place the Vcc on the multisim design window and double click on Vcc to change the value to 12V. Double click on the AC power source, and change the voltage to 25mV and frequency to 1khz.

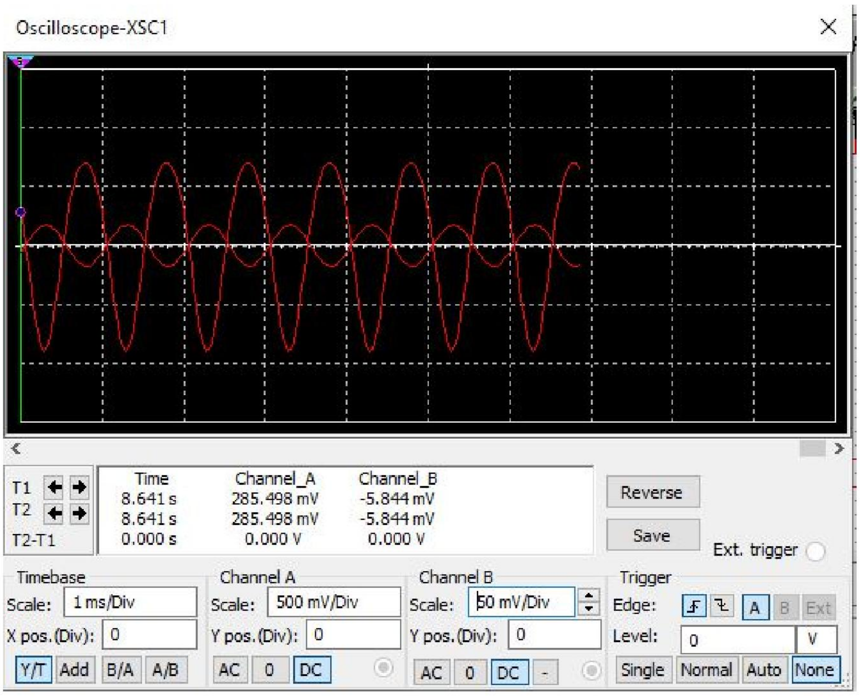


7) Connect all the components with proper wiring and also ensure that nodes are formed at the interconnection points.In this tutorial, we must place the ground and the two-channel oscilloscope to simulate the input and output of the circuit.

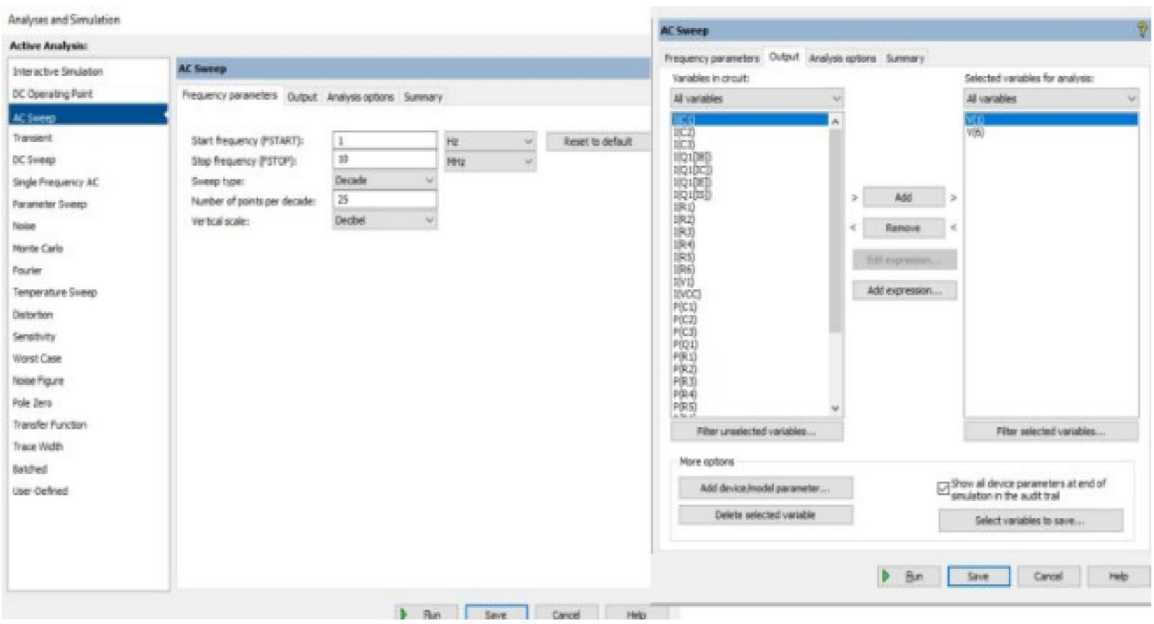


8)Now to determine the gain of the common emitter amplifier, click on the run button. Double click on oscilloscope. Common emitter amplifier multisim output simulation. Make the following changes on Time-base, channel A, and channel B

IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
Impact Factor: 7.301
Volume 3, Issue 2, September 2023

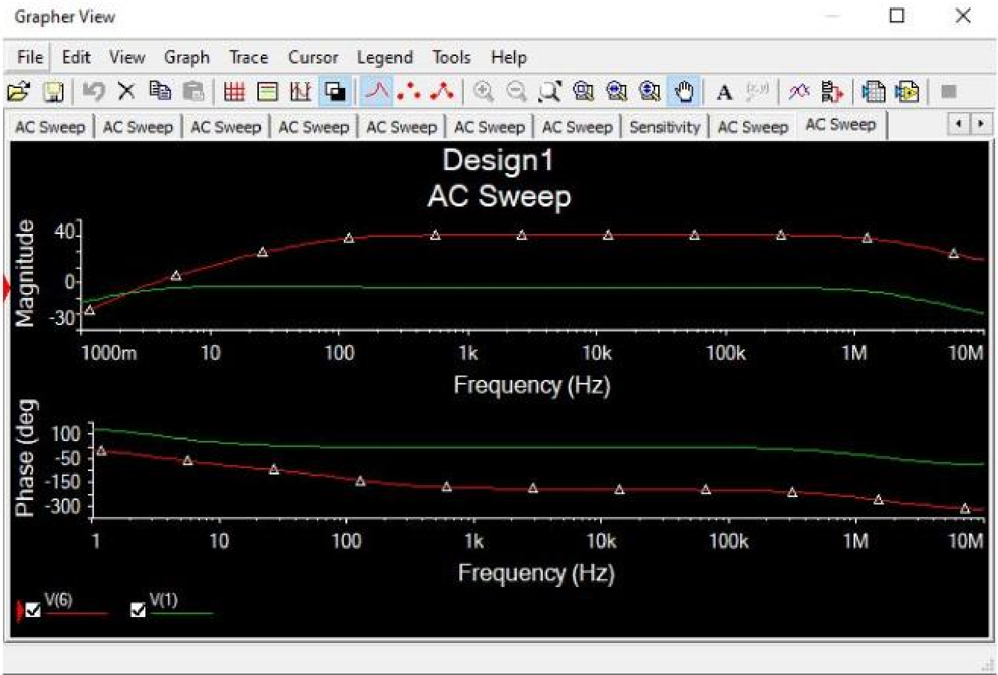


9) Select AC sweep and enter the following parameter.
Start Frequency- 1Hz
Stop Frequency- 10MHz
Sweep type: Decade
Number of points per decade: 25
Vertical sale: Decibel



10) In the output, add variable V(1) and V(6), run the simulation.
You can see the bandwidth of a common emitter amplifier based on the frequency response curve.

IJARSCT
ISSN (Online) 2581-9429
International Journal of Advanced Research in Science, Communication and Technology (IJARSCT)
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
Impact Factor: 7.301
Volume 3, Issue 2, September 2023



IV. CONCLUSION

The students of Electronics And Telecommunication Engineering nowadays have more theoretical knowledges than practical ones in the subjects of their primary interests. This is the motivation for this paper, to give the students more hands-on experience with real hardware devices and their usage through hardware and software simulations

REFERENCES

[1]Zhengdong Li 1 ,Xiuling Li2*, Decai Jiang3 , Xingzong Bao1 , Yan He1"Application of Multisim Simulation Software in Teaching of Analog Electronic Technology"Journal of Physics: Conference Series March 2020

[2] Hou Y.Y., Chen B., Li T.L.(2018) Teaching Research and Practice on "Analog Electronic Technology Basis" Based on Hybrid Teaching Mode. Education Teaching Forum, 33: 172-173.

[3] Zhang X.W., Si Y.Q.(2019) Application of Multisim9 in small bulb series-parallel circuits. Journal of Hubei Normal University(Natural Science), 39(4): 84-88

[4] Zhang J.L., Li K.R.(2019) Simulation Experiment Study on Inductance Filtering of HalfWave Rectification Based on Multisim 10. Physical Experiment of College, 32(6): 104-107

[5] Zhu J.N., Guo X.F., Lyu Y., et al(2018). Circuit Design of Flickerless Direct-current LED Lamp Based on Multisim China. Illuminating Engineering Journal, 29(5): 120-123

[6] Li Y., Li X.H., Guo W.L.(2019) Design and implementation of low-frequency virtual laboratory based on LabVIEW-Multisim. Modern Electronics Technique , 42(6): 72-75