



AKSHAYA INSTITUTE OF TECHNOLOGY

Approved by AICTE, New Delhi, Affiliated to VTU, Belgaum, Recognized by Govt. of Karnataka,
Obalapura Post, Lingapura, Koratagere Road, Tumkur - 572 106, Karnataka



Workshop Report

“3 days hands-on Workshop on MultiSim”



Organized by

DEPARTMENT OF ELECTRONICS & COMMUNICATION ENGG

*In Association with Karunada Technologies Private Limited,
Bengaluru*

Category of Event	: Workshop
Title	: Multisim
Schedule	: 26/02/2024 to 28/02/2024
	Time: 10.00 AM to 12.00 PM and 2.00PM to 4.00PM
Venue	: DSP Lab
Duration	: 3days
Organizing Department	: Department Of Electronics And Communication Engineering
Faculty Coordinators	: Mrs. Arpitha H B
Total Number of Participants	: 61
Guest Profile	: Prof. Lokesha A M

Objective of Workshop

The main objective of this workshop is that:

- In this multi-sim workshop, students will be able to learn and implement the practical circuits and can observe and analyze the output.
- In this Workshop, Analog as well as Digital circuit implementation will be taught.
- Here, Students will be able to gain knowledge on Analog and Digital circuit theoretically and practically.
- In the final stage of this workshop, student will be able to design & execute a mini project and major project

Overview of the Workshop: -

The Multisim software workshop held by Akshaya institute of Technology, Department of Electronics and Communication Engineering on February 26, 2024, was a significant event for the 2nd-year students of the of Electronics and Communication Engineering branch. Scheduled from 10:00 AM to 4:00 PM, the workshop took place in the DSP Lab.

This workshop provided students with valuable hands-on experience in circuit design and simulation using National Instruments' Multisim software. The primary goal was to familiarize participants with the software's key features and demonstrate how to effectively simulate electronic circuits for both educational and practical applications.

Introduction to Multisim Software: -

Overview of Multisim: Students were introduced to Multisim, a powerful simulation software by National Instruments used widely in academia and industry for electronic circuit design and testing. It allows users to create and simulate circuits without needing physical components, making it an essential tool for engineers. □Importance of Simulation: The session highlighted the role of simulation in circuit design, enabling engineers to test and optimize designs virtually before physical implementation, saving time and resources.

Interface and Features: Students were walked through the software's interface, showcasing key features like schematic capture, analysis tools, and real-time simulation capabilities.

Basics of Circuit Design: -

Understanding Components

This section focused on familiarizing students with electronic components such as resistors, capacitors, inductors, diodes, transistors, etc., and their corresponding symbols used in circuit diagrams.

Creating Basic Circuits

Students were taught how to select components from the Multisim library and place them on a schematic to form simple circuits. This step was crucial in building their understanding of circuit connections.

Simulating Electronic Circuits: -

Setting Up Simulations

Students were shown how to configure simulations for their circuits. This included selecting simulation types (e.g., time-domain or frequency-domain simulations) and setting up power sources, signal inputs, and initial conditions.

Running Simulations

Practical demonstrations were provided on how to run simulations, with real-time observations of circuit behavior, including how voltage and current change in response to different inputs.

Analyzing Results

The session covered interpreting simulation outputs, helping students understand how to verify the accuracy of their circuits based on simulated data.

Circuit Analysis Tools: -

Voltage, Current, and Frequency Analysis

Students learned how to use Multisim's built-in analysis tools to measure voltages, currents, and frequencies at different points in their circuits, essential for understanding circuit performance.

AC and DC Analysis

Both AC (alternating current) and DC (direct current) circuit simulations were explored, giving students insight into how circuits behave under different types of electrical inputs and conditions.

Frequency Response and Transient Analysis

Special tools for observing the frequency response of circuits and transient behaviour (response to sudden changes in input) were demonstrated, important for designing stable and reliable circuits

Advanced Circuit Design: -

Complex Circuits

Moving beyond basic designs, students were guided through the construction of more advanced circuits that involve multiple components and subsystems, reflecting real-world applications.

Working with Integrated Circuits

This section introduced students to working with ICs (Integrated Circuits) like operational amplifiers (op-amps), timers (e.g., 555timer), and logic gates, which are commonly used in automation and robotics

Practical Examples

The advanced designs included practical circuits, such as sensor interfacing circuits or control systems used in robotics, helping students connect theory to practical applications

Troubleshooting and Optimization: -

Identifying Errors

Students were taught how to identify and resolve common design errors, such as incorrect component connections or parameter settings, which can lead to simulation failures or inaccurate results.

Debugging Techniques

Step-by-step debugging techniques were introduced to help students isolate problems in their designs by systematically testing individual parts of the circuit.

Optimizing Circuit Performance

The session also covered optimization techniques, where students learned how to tweak component values and configuration settings to improve the performance, efficiency, or stability of their circuits.

Practical Applications: -

Real-world Circuit Simulations

Students were given examples of real-world applications where Multisim is used, such as in designing power supplies, control systems, and sensor circuits in robotics, automation, and consumer electronics.

Automation and Robotics

Special emphasis was placed on how circuit simulation disintegrated into the automation and robotics fields, such as simulating motor control circuits, sensor signal conditioning, and communication interfaces.

Prototyping with Simulation

: The role of simulation in reducing prototyping costs was discussed. By simulating circuits before physically building them, engineers can identify issues early, leading to fewer iterations during the prototyping phase.

Q&A and Hands-on Exercises: -

Clarifying Doubts

An open Q&A session allowed students to ask questions related to any of the workshop topics, including circuit design challenges or specific Multisim features they found confusing.

Practical Exercises

Students engaged in hands-on exercises where they built and simulated their own circuits using Multisim. These tasks helped solidify their understanding by applying the concepts learned in the workshop.

Group and Individual Work

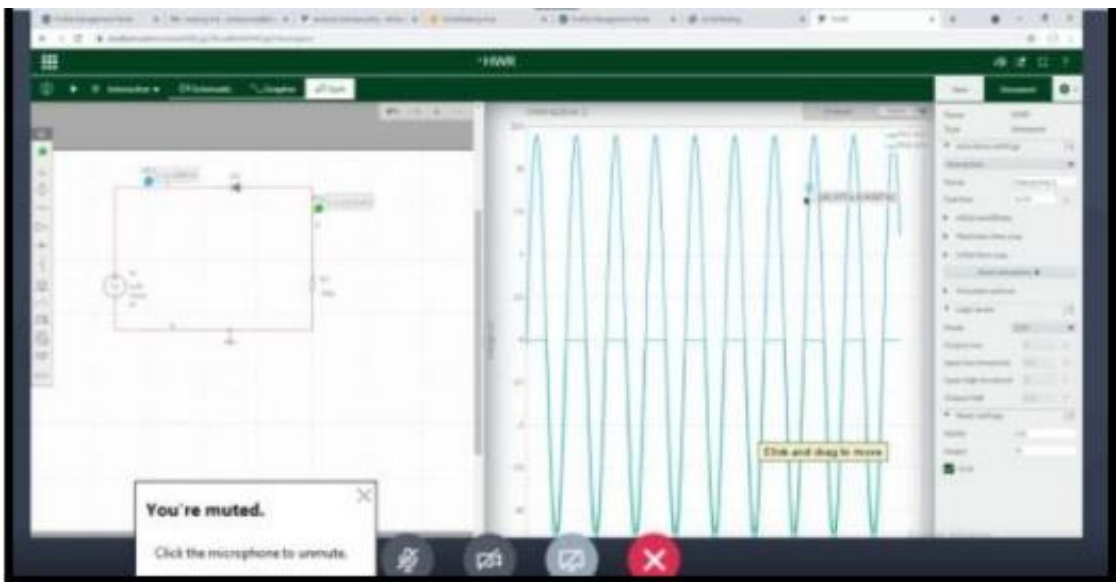
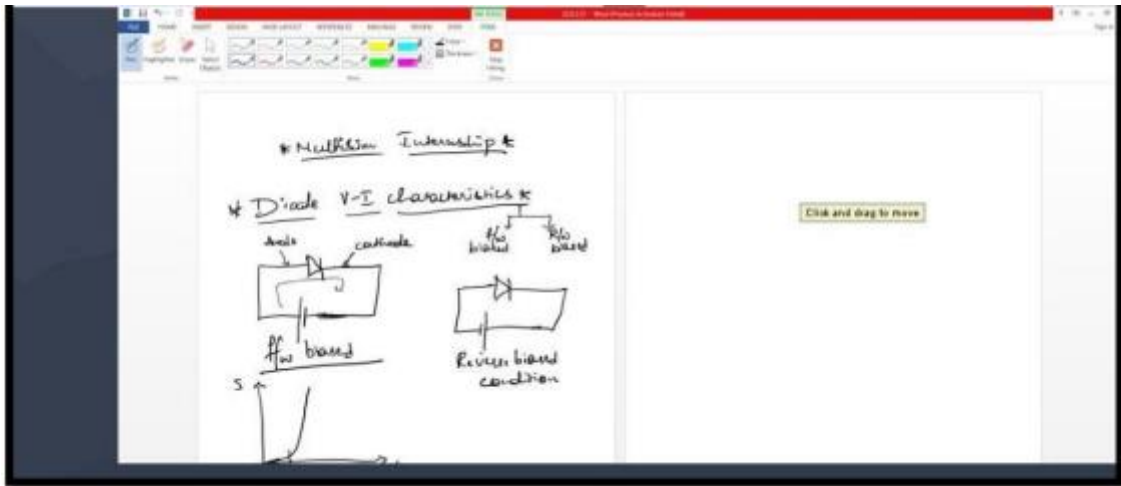
Students were encouraged to work both individually and in groups to foster collaboration and improve problem-solving skills, simulating real-world engineering teamwork.

Schedule for two Days Workshop on multi-sim

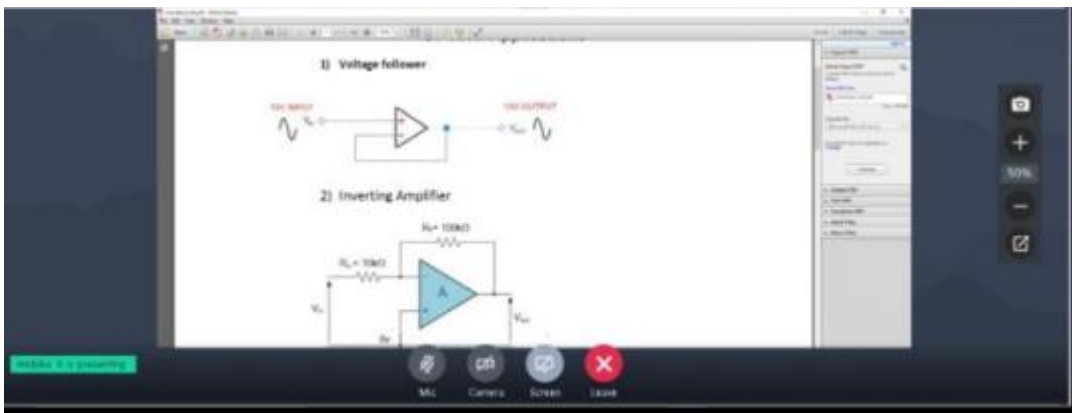
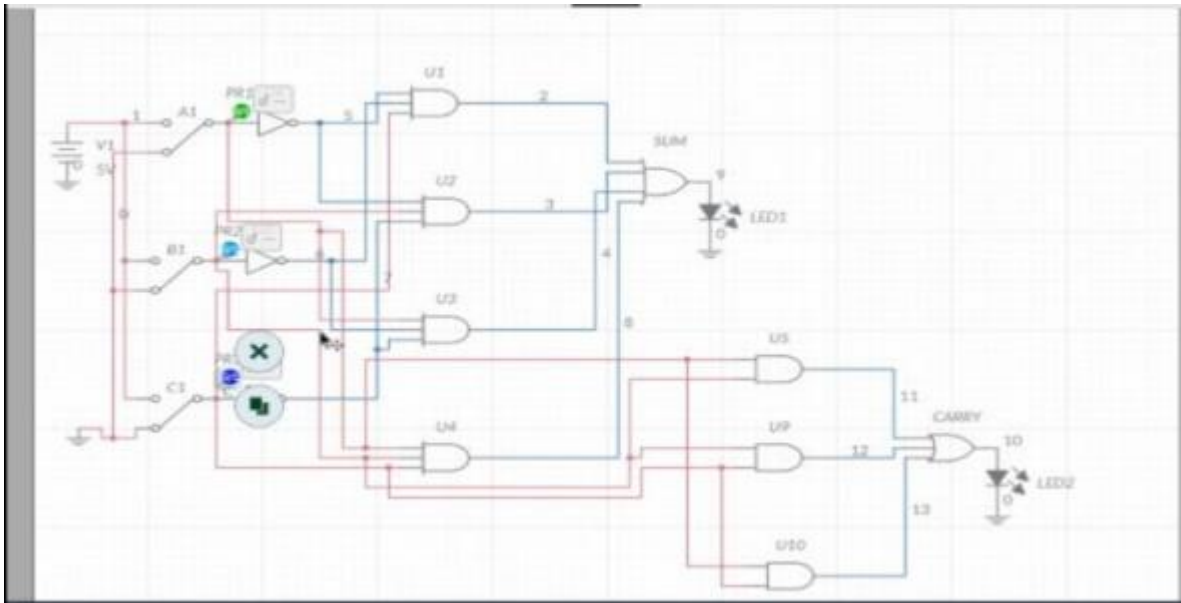
Date: 26/02/2024 – 28/02/2024

Sl.No	Date		Resource Person	Concept
1	Day1 26/02/2024	Morning Session	Prof Lokesha A M	1.Introduction to multi-sim live tool 2.Diode Characteristics 3.Half wave rectifier (With & Without filter)
		Afternoon Session		4.Bridge rectifier (With & Without filter) 5.CE configuration – I/P & O/P characteristics 6. Basic gates – Truth table verification 7.Half adder – Implementation using basic gates
2	Day2 27/02/2024	Morning Session	Prof Lokesha A M	8. Full adder using basic gates 9.Op-amp as voltage follower 10. First order low pass & High pass filter
		Afternoon Session		11. Wein bridge oscillator 12. Transistor as a Switch 13.Zener as voltage regulator to determine Line / Load regulation
3	Day3 28/02/2024	Morning Session	Prof Lokesha A M	14.RC coupled Amplifier using multi-sim and virtual lab 15.BJT as RC phase shift oscillator
		Afternoon Session		16.RC coupled amplifier with feedback 17. Drain & Source characteristics of MOSFET

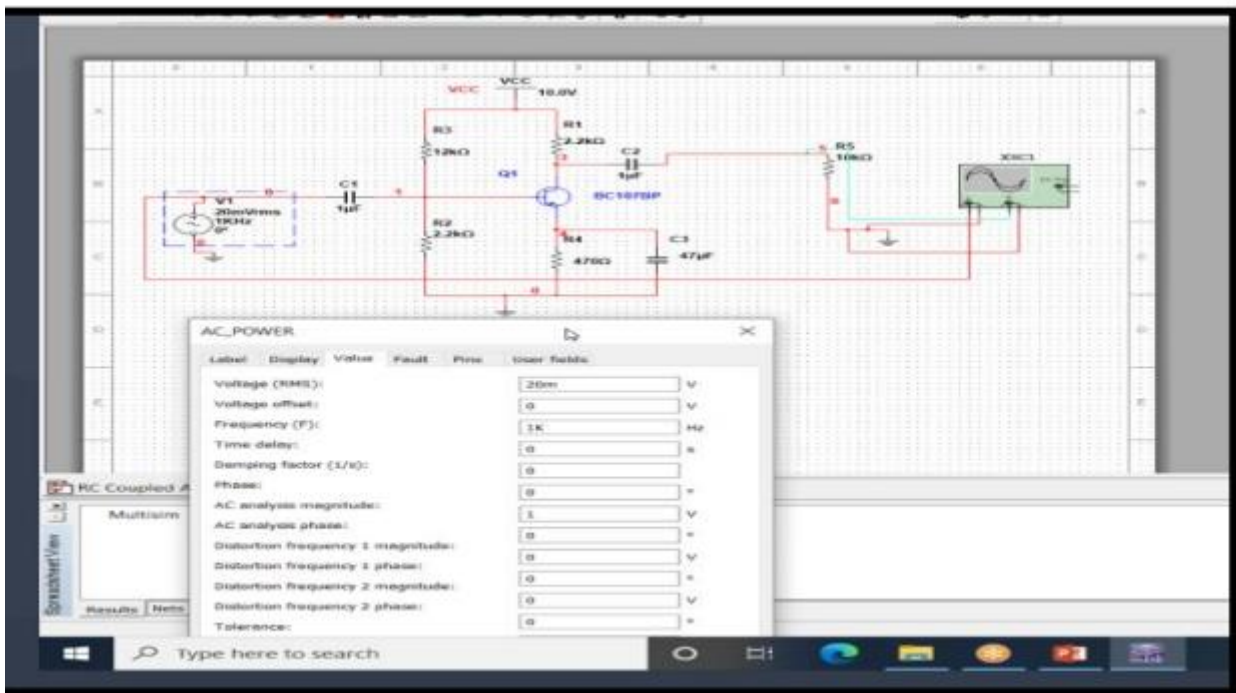
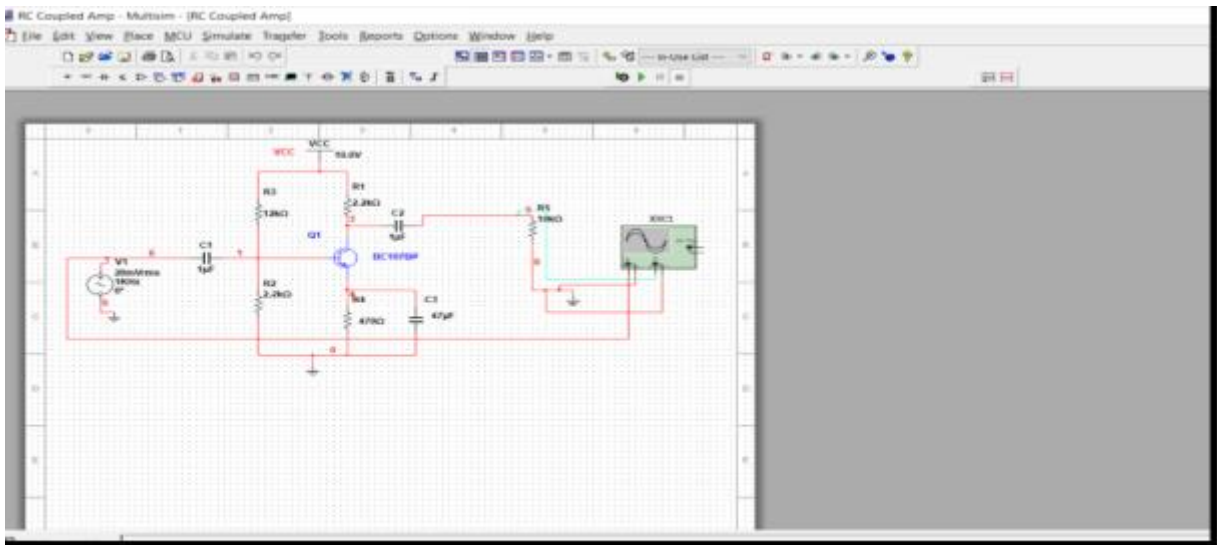
Screenshots of activities done in Workshop: Day1



Screenshots of activities done in Workshop: Day2



Screenshots of activities done in Workshop: Day3



Conclusion: -

The Multisim software workshop was a successful event that significantly contributed to the students' technical skills and understanding of electronic circuit simulation. It provided a solid **foundation** for their future endeavours in the field of automation and robotics

Description/ Report on Event

The Three days on workshop program began with inaugural function. Dr Nagendra Kumar M, HOD, Department of ECE, welcomed all the dignitaries including K V Srinivas Rao, Principal AIT, Dr Yathisha L Dean, AIT and Resource Persons Mr. Mahesh Deginal, Mr Harish N Karunadu technologies Private limited

Photos

