



BOOKS TO BOARD



One Day Workshop on ELECTRONIC CIRCUIT DESIGN

Highlights

- Multisim Simulation.
- Hands on practice of power supply with filter and regulator.
- DSO Calibration and Tuning.
- Ripple Factor Calculations.
- PCB component mounting and testing of components.

Details

Participants: 2nd Year
Students Of I&C
Engineering
Department.

Date: April 12 , 2022.

Time: 11:00 AM Onwards.

Venue: Tagore Hall ,
IC Dept. , LDCE.

- Organised By:
- I&C Engineering Dept. Of
- LDCE

LD COLLEGE OF ENGINEERING

IC DEPARTMENT

REPORT

ONE DAY WORKSHOP ON ELECTRONIC CIRCUIT DESIGNING

The objective of this workshop is to enhance student's practical & technical skill and to make them compatible with industry. To fulfil this, the department of **Instrumentation & Control** has organized one day workshop on **Electronic circuit designing and simulation** for 2nd year BE IC students on 12th of April, 2022.

The workshop was aimed to provide knowledge about simulation of electrical circuit in Multisim software, implementation of electrical circuit on PCB, DSO calibration, Ripple factor calculation.

The workshop included the following topics:

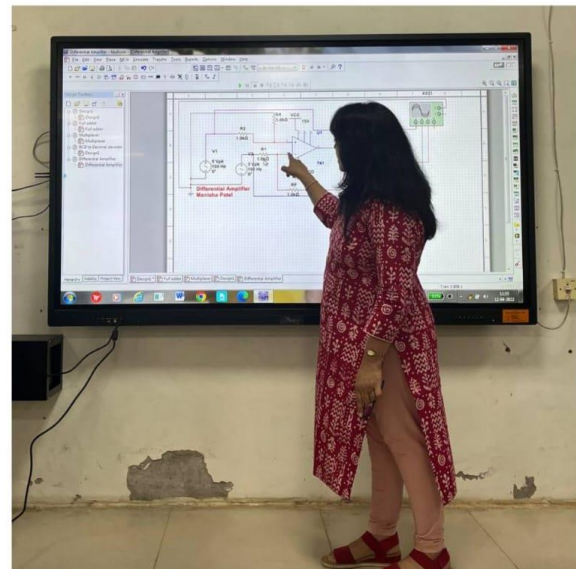
- Introduction to Multisim software and Digital circuit simulation
- Dual power circuit simulation on Multisim
- Work on Circuit Design
- Actual circuit implementation on PCB and final testing
- Feedback Session



Session 1: 11am to 11:45am

Introduction to Multisim software and Digital circuit simulation

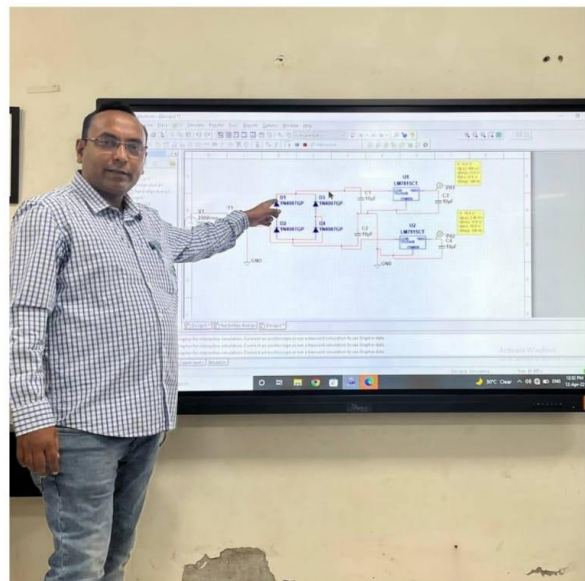
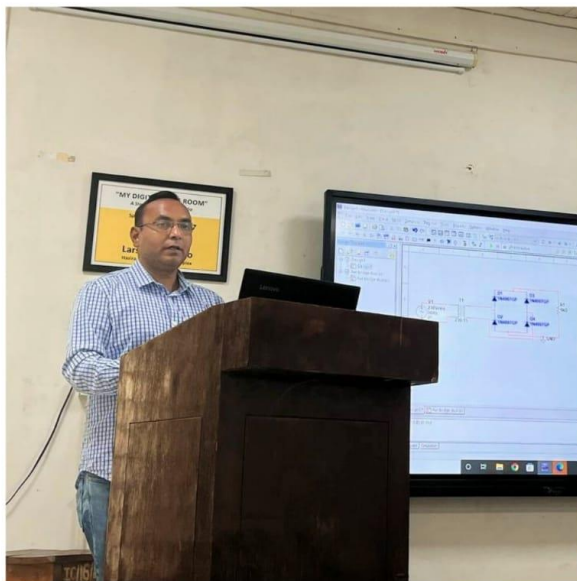
First session was conducted by **Prof. Manisha Patel** and **Prof. Nidhi Kanani**. They started with introduction of Multisim, benefits of the using Multisim and its applications. They explained that before directly implementing the circuit, it is necessary to simulate it on software, so when the circuit is being performed on bread-board, it becomes easier. They explained schematic design of circuit on software Multisim and pointers to make precise schematic design.



Session 2: 11:45am to 12:30am

Dual power circuit simulation on Multisim

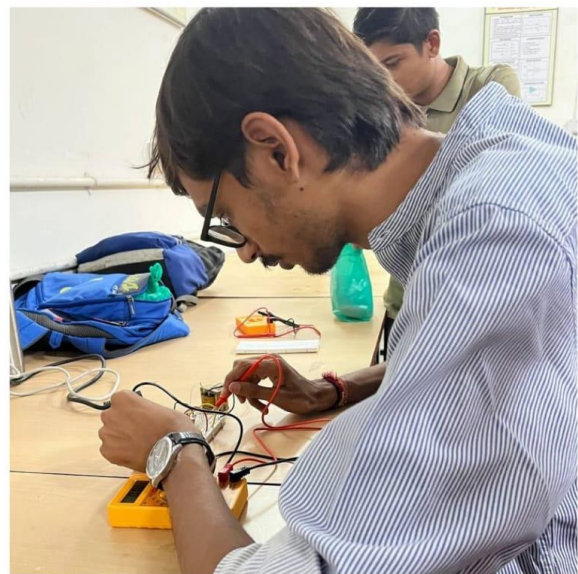
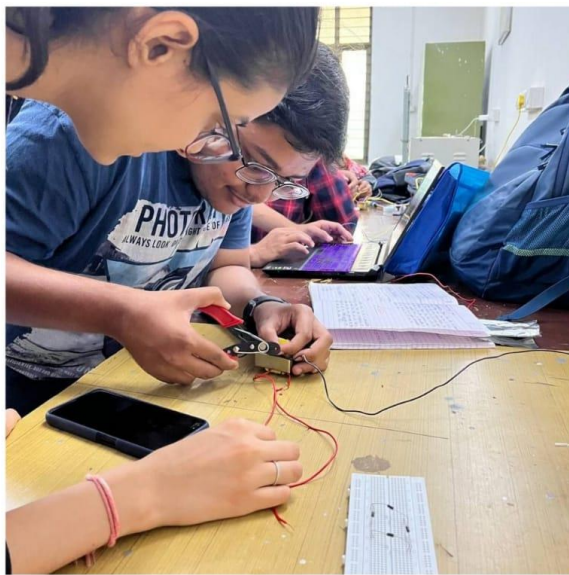
Second session was conducted by **Prof. Lalit Patel**. Two basic circuits were taken, first one is **Full wave bridge rectifier** and the other one is **Dual power supply**. He designed and explained the two given circuits on Multisim and simulated at the end.



Session 3: 1:00pm to 3:00pm

Work On Circuit Design

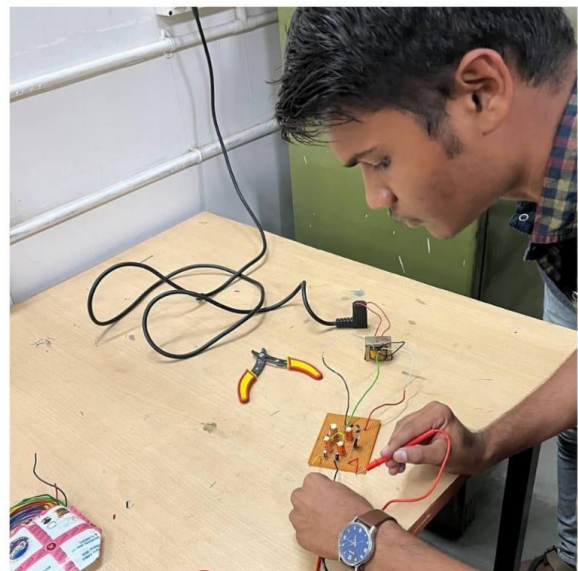
The afternoon session of Hands-on started at 1:00pm, where the students were provided with laboratory sessions on schematic circuit design on breadboard. Session 3 and Session 4 were divided into four batches B1, B2, B3, B4 accordingly. The students performed circuits on the breadboard which were explained in the previous session.



Session 4: 3:15pm to 4:30pm

Actual circuit implementation on PCB and final testing

The most important part of the workshop is the implementation of circuit on PCB through soldering and final testing. The circuits which were made on breadboard, has to be made on PCB through soldering the components on it.



Session 5: 4:30pm to 5:00pm

Feedback Session

At the end of the workshop, there was a feedback session, in which students gave their genuine feedback of the workshop. Students actively participated in this workshop and requested for more such workshops and industrial visit in the similar way to gain more practical skills.

