

SNJB's  
KKHA Arts, SMGL Comm. & SPHJ Science College, Chandwad  
**DEPARTMENT OF ELECTRONIC SCIENCE**

Workshop Report

**Title:** Two -Day Workshop on “**Electronic Design Automation Tools**”

**Dates:** 27<sup>th</sup> March, 2021

**Organized by:** Department of Electronic Science

Faculty Coordinators: Dr. Tushar S. Salve

Student Coordinators: Ms. Priyanka Khairnar, Miss. Rutuja Chavan

**No. of Participants: 23**

Department of Electronic Science organized a Two-day workshop on ‘**Electronic Design Automation Tools**’ on **29<sup>th</sup> & 30<sup>th</sup> November, 2021**.

The workshop registration was free to all the UG and PG students of Electronics department.

Vice-Principal Mr. S. P. Khairnar inaugurated the function as a chief guest. In inaugural speech, Khairnar Sir motivated the students about various opportunities in the Electronic sector and future of AIML. Head of the department Dr. Tushar Salve put forward of the workshop. Dr. D. R. Patil, Asst. Prof. O. M. Pardeshi, Mr. B. R. Pagar, Miss. Prajakta Jadhav & Mr. S. B. Borade were present for the inaugural function.

**The main Objectives of the Workshop were:**

- 1: To Design the electronics circuits using EDA software tools
- 2: To Simulate various analog and digital circuits using EDA software tools
- 3: To Plot various waveforms.
- 4: To Simulate basic electronic system blocks.

The motive behind organizing this workshop was to encourage and make aware the Electronics students about different **Electronic Design Automation Tools**. It is already included in T. Y. B. Sc. syllabus but other Electronics students, PG students were not familiar with these tools. For future prospective, it is helpful for them to simulate various circuit designs without hardware. They can also perform the simulation at home.

In first session of day one, the resource person **Prof. S. A. Wankhede** Sir, from Pimpalgaon College discussed Definition of Simulation, Need of Simulation, Brief introduction of various

simulators, Description to simulator tool, Hands on practice on available library of components, wiring and schematic designing in **Proteus**.

On day two, Infirst session **Dr. D. R. Patil** explained in detail Introduction to Simulator: Brief History, New Versions, Representing Components, Understanding the simulation Environment, Using Model Editor, designing a Circuit and drawing a schematic, Preparation for Simulation: Preparing schematic for simulation, Understand the sources for simulation, Use of different markers. DC, AC, Transient and Fourier Analysis of circuit, Digital circuit Simulation using **LT-SPIICE**

In second session of day two, Mr. SachinBorade Sir focused on Environment: Design Process, Setting environment preferences, The Multisim /CircuitMode: Schematic capture of circuits, Placing components, Wiring components, simulation and result display in MultiSim.Device modeling: Design of Half-Wave rectifier, Bridge rectifier, clippers and clampers using diode, voltage regulator, AC voltage measurement, DC transfer curve analysis

The workshop was beneficial to the students in various aspects. They learnt and interacted to resource persons to know the basics of Arduino board and its programming logic. Students interacted through questions and answer sessions.

Valedictory function was anchored by Coordinator of the workshop Dr. Tushar Salve and put the vote of thanks.



**Prof. S. A. Wankhede discussing various simulator tools**