



REPORT

P SPICE

Date: 21st Aug, 2024.

The **IEEE Student Branch of Haldia Institute of Technology** successfully conducted a two-day workshop on **PSPICE**, a powerful circuit simulation tool widely used in the field of electronics and electrical engineering. Held on **August 21st and 22nd, 2024**, the workshop aimed to equip participants with foundational knowledge and hands-on experience in simulating and analyzing electronic circuits.

Day 1: Laying the Groundwork (August 21, 2024)

The first session began at **3:30 PM** in the **Control System Lab**, with **Dr. Partha Sarathi Das** and **Mr. Souvik Maulik** guiding participants through an introduction to PSPICE. The session offered a comprehensive overview of the software's core functions and its integration with SPICE simulation technology.

Attendees were introduced to essential circuit components and simulation techniques, including how to plot waveforms and identify probe points. The mentors explained the software's capability to calculate **node voltages** and **branch currents** across varying frequencies, laying the groundwork for more complex applications. The interactive session sparked interest among the participants, building anticipation for the hands-on activities planned for the next day.

Day 2: Practical Circuit Simulation (August 22, 2024)

The second day of the workshop took a more application-driven approach. Held in **Smart Room 6108**, the session was conducted by **Mr. Ayandeep Ganguly** and **Mrs. Alpana Barman**, who led participants through real-time PSPICE simulations.



IEEE HIT SB

ADVANCING TECHNOLOGY FOR HUMANITY

HALDIA INSTITUTE OF TECHNOLOGY

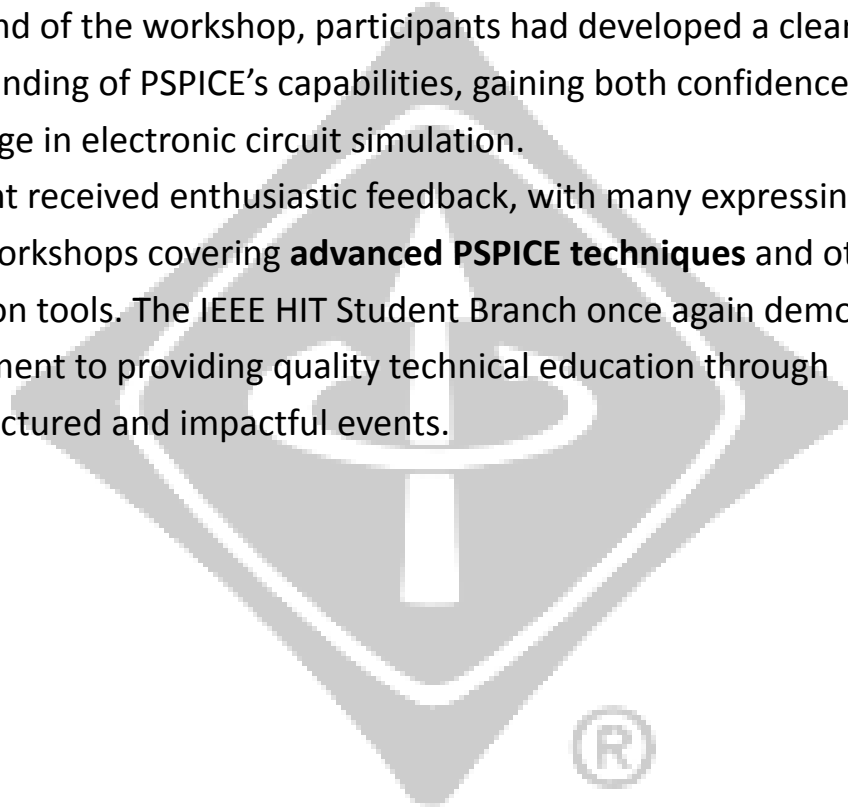
ICARE COMPLEX, HATIBERIA, HALDIA
DIST - PURBA MEDINIPUR, WEST BENGAL
PIN - 721657



Using a range of circuit examples, attendees learned to set up simulations, analyze output data, and apply theoretical concepts in a virtual environment. Key features such as **voltage and current analysis** and **waveform interpretation** were explored in depth, offering participants a chance to strengthen their skills through guided practice. Volunteers and mentors were actively involved in assisting participants, ensuring a smooth and engaging experience.

By the end of the workshop, participants had developed a clear understanding of PSPICE's capabilities, gaining both confidence and practical knowledge in electronic circuit simulation.

The event received enthusiastic feedback, with many expressing interest in future workshops covering **advanced PSPICE techniques** and other simulation tools. The IEEE HIT Student Branch once again demonstrated its commitment to providing quality technical education through well-structured and impactful events.



IEEE HIT SB