JSS Mahavidyapeetha

JSS Academy of Technical Education Noida

Department of Electrical Engineering

Report

on

Two-Days STTP

Introduction of Cadence Software (PSpice) and Its' Application

13th – 14th September, 2024

Mr. Nirmal Kumar Agarwal
Assistant Professor-JSSATEN
Program Coordinator-STTP

C-20/1, Sector-62, Noida-201301
Tel: 0120-2400114, 2400115, Fax: 0120-2400097, E-mail: principal@jssaten.ac.in

Two-Days STTP on "Introduction Of Cadence Software (Pspice) and Its Application"

INTRODUCTION

The two days STTP on "Introduction of Cadence Software (PSpice) and Its Application" was conducted from 13th – 14th September 2024, with the aim of providing lab instructors from the Departments of Electrical Engineering, Electronics and Communication Engineering, and Electrical and Electronics Engineering with the necessary skills to effectively utilize PSpice in their teaching and research activities. Total 10 participants across the college attended the STTP.

Training Content

The STTP covered a wide range of topics related to PSpice, including:

1. ***Basic Circuit Analysis***: Introduction to circuit elements, nodal analysis, mesh analysis, and Thevenin's and Norton's theorems.
2. ***DC Analysis***: Analysis of circuits with DC sources, including node voltage analysis, mesh current analysis, and superposition.
3. ***AC Analysis***: Analysis of circuits with AC sources, including phasor analysis, impedance, admittance, and frequency response.
4. ***Transient Analysis***: Simulation of circuits over time, including step response, impulse response, and sinusoidal steady-state analysis.
5. ***Small-Signal Analysis***: Analysis of circuits around a DC operating point, including small-signal models and gain calculations.
6. ***Non-Linear Analysis***: Simulation of circuits with non-linear elements, such as diodes and transistors.
7. ***Custom Component Creation***: Creating custom components using PSpice's behavioural models.

Hands-on exercises were conducted throughout the STTP to reinforce the theoretical concepts and provide participants with practical experience using PSpice.

Trainer Profiles

The STTP was co-ordinated and facilitated by Mr. Nirmal Kumar Agarwal, Assistant Professor, and co-coordinated by Mr. Rajesh Kumar, Assistant Professor and Mr. Abhishek Kumar Singh, Assistant professor in the Department of Electrical Engineering. The coordinator and co-coordinators also served as a trainer for the program. All the trainers have extensive experience in the field of electrical engineering and have been using PSpice for many years.

Participant Feedback

The STTP received positive feedback from the participants. They appreciated the comprehensive coverage of PSpice topics, the hands-on exercises, and the expertise of the trainers. Some of the key points raised in the feedback include:

1. The training content was relevant and up-to-date.
2. The trainers were knowledgeable and approachable.
3. The pace of the training was appropriate.
4. The hands-on exercises were helpful in understanding PSpice.
5. The overall organization of the STTP was well-planned.

CONCLUSION

The STTP on Introduction of Cadence Software (PSpice) and Its Application was a successful event that provided lab instructors with the necessary skills to effectively utilize PSpice in their teaching and research activities. The participants gained a deep understanding of PSpice's capabilities and are now equipped to incorporate it into their coursework and projects.

Future STTPs on related topics may be considered to further enhance the skills of faculty members and support the academic goals of the department.

TIMELINE

13-09-2024 (Day-1)

09:00 A.M.- Inaugural session has started with the addressing of HoD Dr. Sanjeev Kumar Sharma Sir. He motivated the participants from EE, EEE and ECE department followed by tea/snacks.

09:30-11:30 AM.- Introduction and simulation on PSpice started by Mr. Nirmal Kumar Agarwal (Assist. Prof-EED) on the topic:

Basic Circuit Analysis: Introduction to circuit elements, nodal analysis, mesh analysis, and Thevenin's and Norton's theorems.

DC Analysis: Analysis of circuits with DC sources, including node voltage analysis, mesh current analysis, and superposition.

13-09-2024(Day-1)

02:00 -04:00 P.M.- Setting up of Simulation is done by Mr. Abhishek Kumar Singh.

14-09-2024(Day-2)

09:30-11:30 AM	Designing of Lab exercise done by Mr. Rajesh Kumar
11:00 A.M.	During the inaugural session, the Principal sir, addressed and motivated the participants. Group Photograph Session followed by valedictory session.

KEY TAKEAWAYS

1. Working view of twin units with many functioning devices created a sense of inquisitiveness amongst the lab instructors.
2. Detailed discussion and doubt solving about the circuit simulation by PSpice.
3. Gained Knowledge about the dc ac analysis along with transient analysis.
4. Great Learning Experience for the lab instructors as they learnt the applications of PSpice software in the field of electrical and electronics engineering.

PHOTO GALLERY

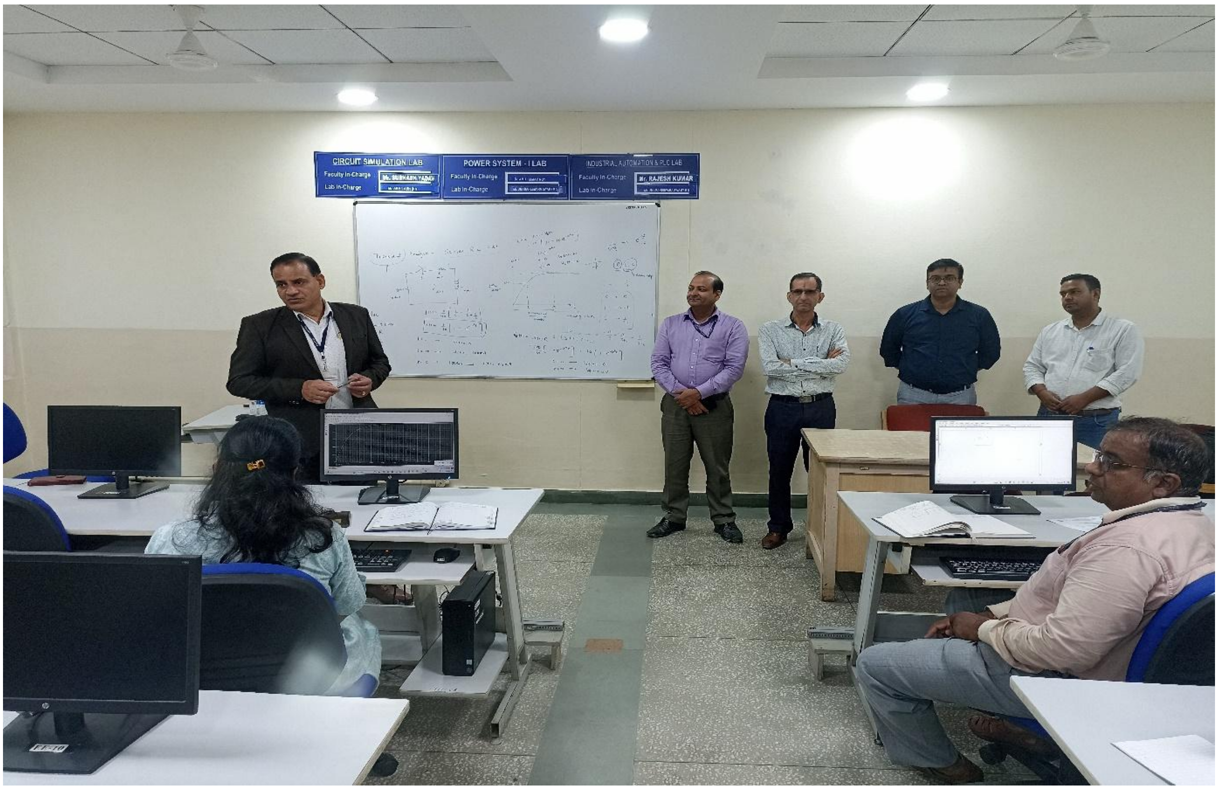

Fig.1 During the inaugural session, the Principal sir, addressed and motivated the participants

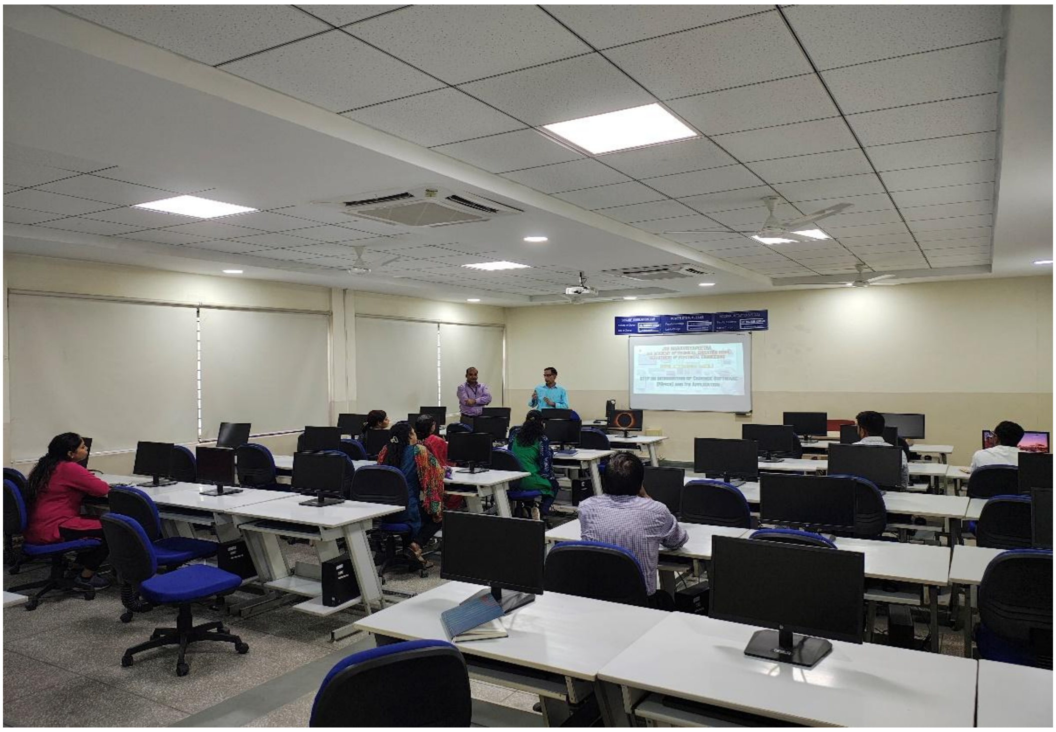

Fig.2 During the inaugral session, HoD Dr. Sanjeev Kumar Sharma sir, addresses and encouraged the participants.

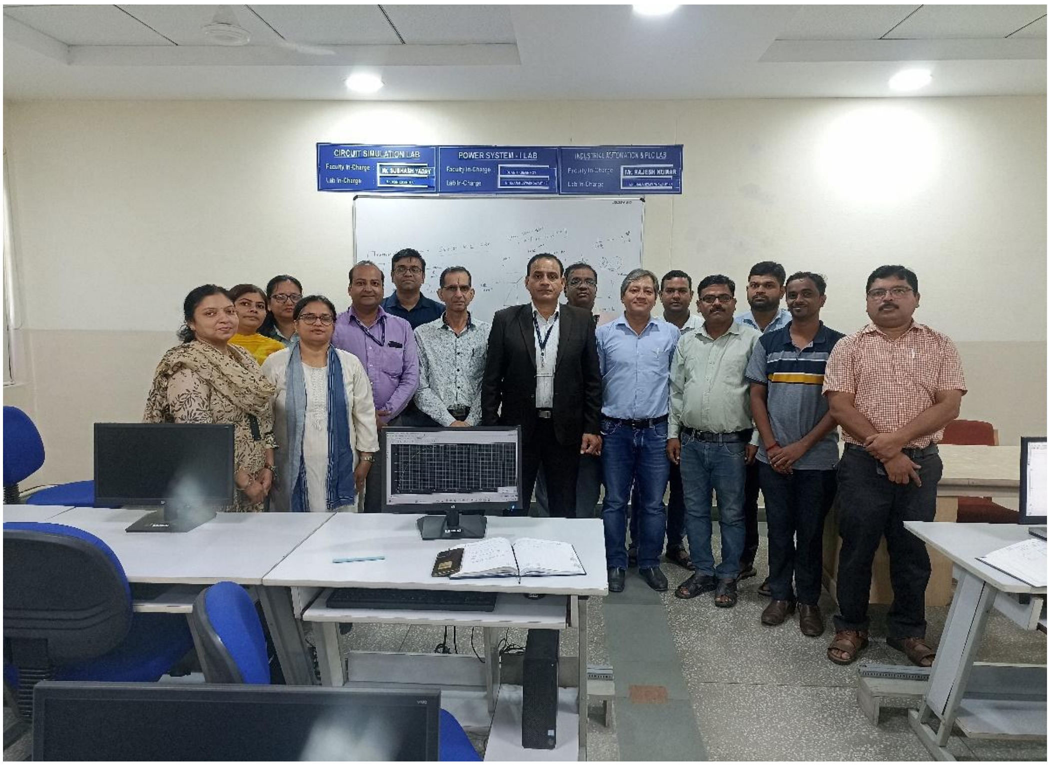

Fig.3 During the velidictory program the Principal Sir Dr. Amarjeet Singh addressed the participants.

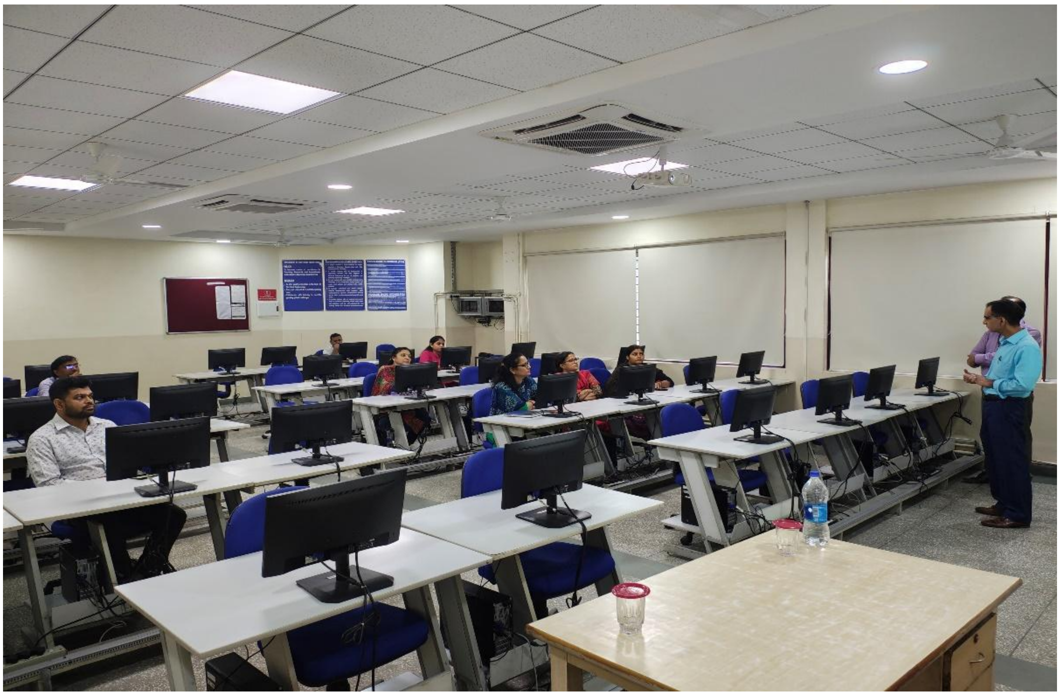

Fig.4 HoD Dr. Sanjeev Kumar Sharma Sir addressed the participants and presented a vote of gratitude during the velidictory program.