BMS INSTITUTE OF TECHNOLOGY AND MANAGEMENT

YELAHANKA – BANGALORE – 64

DEPARTMENT OF ELECTRICAL AND ELECTRONICS ENGINEERING

Open Course on Application of PSPICE in the Analysis of Electrical& Electronics circuits and systems

Dt: 25-07-2018

This course is planned to expose the students to develop and analyze electrical and electronic circuits. This course helps the students in analyzing challenging problems which are beyond the curriculum and also helps in solving complex circuits. This tool allows the student to compare the results which are obtained in classroom environment with the simple circuit models of devices, to those obtained by using complex SPICE models. After attending this course, student will be able to,

1. **Design** and **analyze** electrical, electronic circuits and systems.
2. **Analyze** transient response for a given system.
3. **Verify** experimental results with simulation results.

Schedule and Course Contents:

Sl.No.	Date	Time	Course Contents
1	2-08-2018	9:30AM–3:30PM (with one hour lunch break)	*PSpice basics*: Introduction, Circuit description, circuit format, understanding PSpice/D and simulator. Simple circuit simulation.
2	3-08-2018		*DC Circuit Analysis:* DC circuit simulation and verification of circuit laws, Network theorem verification, Transfer function model, Operating point.
3	4-08-2018	9:30AM–12:30PM	*AC Circuit Analysis:* Simulating 1 phase and 3 phase circuits, Bode plots AC systems. Simulation of Amplifier circuits and experimental verification.
4	6-08-2018	9:30AM-3:30PM	*Transient Analysis:* Different source models, Resistor capacitor and inductor model, Transient response of various circuits
5	7-08-2018	9:30AM-12:30PM	*Solid state devices and Circuits:* Simulation diode based rectifier circuits and experimental validation.
		1:30 PM to 4:30PM	A test for 50 Marks will be conducted

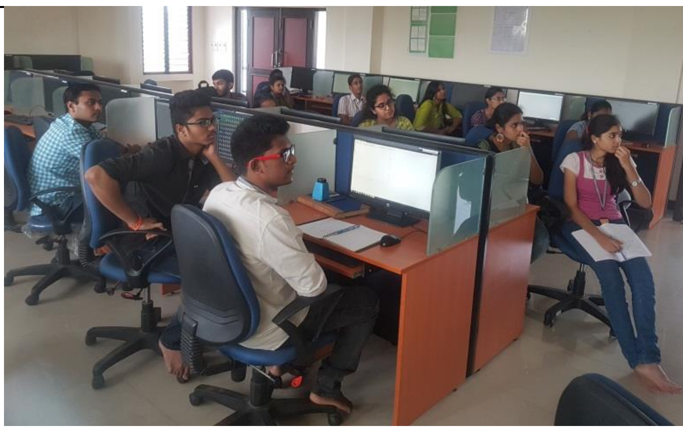
BMS INSTITUTE OF TECHNOLOGY AND MANAGEMENT

YELAHANKA – BANGALORE – 64

DEPARTMENT OF ELECTRICAL AND ELECTRONICS ENGINEERING

An Open course titled "Applications of PSPICE for the Analysis of Electric and Electronic circuits was conducted on 2nd August 2018(5day course). This course was attended by a total of 17 students.

This course was planned to expose the students to develop and analyse electrical and electronic circuits. This course helps the students in analysing challenging problems which are beyond the curriculum and also helps them in solving complex electrical circuits. This tool also allows the student to compare the results which are obtained in classroom environment with the simple circuit models of devices, to those obtained by using complex SPICE models.

 Session under progress	 Participants

Course Instructors:
Dr. Madhu Palati,
Ozwin Dsouza
Manjunatha Babu P