Course: Skill Enhancement Paper – HPHSE4021T & HPHSE4021P

Semester	IV
Paper Number	HPHSE4022T & HPHSE4022P
Paper Title	CIRCUIT DESIGN & SIMULATION
No. of Credits	02 (Theory – 1, Lab – 2)
Theory/ Composite	Composite
No. of periods assigned	Th:1 periods/week Pr: 2 periods/week
Name of Faculty member(s)	
Course description/ objective	This course aims at: 1) Understanding the basics of circuit simulation software. 2) Understanding concepts of electrical circuit design, interconnects, parasitic elements etc. through simulation software. 3) Understanding the analysis that is performed on a circuit such as AC, DC, Noise, Transient etc. 4) Studying the electrical behaviour of important circuits such as rectifiers, amplifiers, switches etc. 5) Understanding the importance of circuit simulation for Integrated Circuit design 6) Making the students aware of other software of similar nature present in the market
Syllabus	As enclosed
Textss	As enclosed
Reading/ Reference List	As enclosed
Evaluation	Theory - 40 (i)CIA – 10 (ii)Semester Examination (30) - Two 10 mark qs out of three qs; Two 5 mark qs out of four qs.

Syllabus:

HPHSE4022 T- CIRCUIT DESIGN & SIMULATION – Credits – 02 (Theory-1, Practical -2)

Introduction to PSpice software. **[1 lecture]**
Basic Analyses: Operating point, DC, AC, transient. **[2 lectures]**
Analog behavioural models. **[2 lectures]**
Advanced Analyses: Noise, Monte Carlo, worst case, Fourier, Sensitivity, parametric. **[5 lectures]**
Examples using semiconductor diodes, BJT, FET, OP Amp circuits. **[3 lectures]**

HPHSE4022P - Circuit Design &Simulation Lab (26 Periods)

Software: OrCADPSpice

1. Use of OP, DC, AC, TRAN, PROBE commands
2. Verifying the characteristics of a (a) Diode (B) BJT (c) FET
3. Advanced analysis: *(Optional) [depends on the software available]*
4. Design of a 555 timer and to show its astable operation.
5. Design of a XOR / ADDER/ MUX *(Optional) [depends on the software available]*
6. Project: *One experiment where the students develop and design an analog or a digital circuit of their choice and which has not been done above.*

Reference Books:

1. Introduction to PSpice using OrCAD for Circuits and Electronics by Mohammad H Rashid
2. SPICE : A guide to circuit simulation & analysis using PSpice by T W Tuinenga

<u>Practical – 60</u>

Two class examinations - 15 marks x 2 = 30 marks
(b) LNB – 20 marks
(c) Grand Viva – 8 marks
(d) Attendance –2 marks