Semester	4
Course	Skill I Physics Honors
Paper Code	S2PH230411P
Paper Title	Circuit Design & Simulation
No. of Credits	3
Theory / Practical / Composite	Practical
Minimum No. of preparatory hours per week a student has to devote	4
Number of Modules	01
Syllabus	Circuit Design & Simulation (Full Lab) <u>Software to be used: LTSpice</u> <u>List of Simulations/Experiments:</u> 1. Introduction to LTSPICE software and netlist writing 2. Implementation of DC, AC, Tran, OP analysis. 3. Simulation with devices in software: Diode, BJT. 4. Implementation of advanced analysis: Temp, noise, subcircuits, Fourier, Sensitivity, parametric. 5. Introduction to schematics. 6. Netlist import to schematics using simple circuits, OP AMP, 555 timer, Solar Cells etc. 7. Design of a XOR / ADDER/ MUX/COUNTER
Learning Outcomes	1. Understanding the working of the circuit simulator. 2. Learning to write SPICE netlist. 3. Understanding the working of both active and passive devices in a circuit from a practical standpoint. 4. Learning to add devices and ICs not in the default library of the software. 5. Understanding the use of both analog and digital devices and their comparisons from a practical standpoint. 6. Understanding the importance of electrical noise and distortions.
Reading/Reference Lists	Reference Books: 1. Essential Circuit Analysis using LTSpice by Farzin Asadi 2. Electronic Circuit Analysis using LTSpice XVII Simulator – a practical guide for beginners by Pooja Mahendru and Pankaj Mahendru
Evaluation	48 + 2