

GUJARAT TECHNOLOGICAL UNIVERSITY

B.E Semester: 3

Power Electronics Engineering

Subject Code 132402

Subject Name Simulation and Design Tools

Objectives:

1. To develop computer skills for design and drafting electronics circuits through EDA Tool software.
2. To develop basic skills for understanding and using simulation for electronic circuits.
3. To develop ability in students for designing simple PCB and thereby make student understand terminology used with PCB.

Suggested Laboratory Work:

The laboratory work should be matching with the laboratory/theory work of the subjects 1. Network Analysis 2. Basic Electronics and 3. Digital Logic Design

Group 1

There should be **minimum 4 experiments** for drafting circuits through student /License version of any EDA tool software like OrCAD, Protel, Multisim or any other open source software. Suggested work is

1. To prepare schematic diagram for Diode characteristics.
2. To prepare schematic diagram for Transistor input and output characteristics.
3. To prepare schematic diagrams of Various Diode rectifiers with and without filter circuit.
4. To prepare schematic diagram of Diode application circuits like voltage clipper, clamper, zener regulator etc.
5. To prepare schematic diagram of Simple transistorized regulator
6. To prepare schematic diagram of Transistor biasing circuits.
7. To prepare schematic diagram of Transistor amplifier circuits
8. To prepare schematic diagram for implementation of all logic gates through NAND gate only.
9. To prepare schematic diagram for implementation of all logic gates through NOR gate only.
10. To prepare schematic diagram for implementation of various flip flops.
11. To prepare schematic diagram for implementation of various combinational and sequential circuits.

Group 2

There should be **minimum 4 experiments** of simulation of simple circuits through student/ Licence version of EDA tool software like OrCAD, Protel, Multisim or any other open source software. Suggested laboratory work is

1. Study of PSPICE description of various components.
2. Plotting diode characteristics
3. Plotting transistor input & output characteristics.
4. Study of diode rectifiers with and without filter circuits. Comparison of voltage and current waveforms at various places in the circuit through simulation
5. Simulation of zener voltage regulator
6. Simulation of transistorized regulator.
7. Simulation of transistor amplifier circuits.
8. Verification of truth table for various logic gates.
9. Verification of various flip flop circuits
10. Simulation of various combinational logic circuits.
11. Simulation of various sequential logic circuits.

Group 3

There should be 2 experiments for designing PCB for simple circuits covered in above group 1 and 2 lab work.