

GUJARAT TECHNOLOGICAL UNIVERSITY

B.E Semester: 3

Electrical and Electronics Engineering

Subject Code 130801

Subject Name Simulation and Design Tools

Introduction to SPICE

Introduction to PSpice software, file types, netlist commands. Basic analyses: DC, AC, Transient. Analog behavioral models (ABM): equations setup, IF statement, voltage/current/ frequency dependent sources. Advanced analyses: noise, Monte-Carlo, worst-case. Spectral description of signals (FFT), measuring the total harmonic distortion (THD). Circuit optimization using PSpice Optimizer software.

Models of resistor, capacitor, inductor, energy sources (VCVS, CCVS, Sinusoidal source, pulse, etc), transformer, DIODE, BJT, FET, MOSFET, etc. sub circuits.

Laboratories should include

Simulation of following circuits using spice (Schematic entry of circuits using standard packages. Analysis- transient, AC, DC, etc.):

- a) Potential divider.
- b) Integrator & Differentiator (I/P PULSE) – Frequency response of RC circuits.
- c) Diode, BJT, FET, MOSFET Characteristics.
- d) Simulate and study half-wave, full-wave, and bridge-rectifier using PSPICE windows
- e) Simulate and study diode clipper and clamper circuits using PSPICE windows
- f) Voltage Regulators.
- g) Simulate and study emitter bias and fixed bias BJT and JFET circuits using PSPICE windows, and determine quiescent conditions.
- h) Simulate a common emitter amplifier using self biasing and study the effect of variation in emitter resistor on voltage gain , input and output impedance using PSPICE windows .
- i) Determine the frequency response of V_o/V_s for CE BJT amplifier using PSPICE windows. Study the effect of cascading of two stages on band width.
- j) Simulate and study Darlington pair amplifier circuit using PSPICE windows and determine dc bias and output ac voltage .
- k) Simulate RC Coupled amplifiers - Transient analysis and Frequency response.
- l) Simulate FET & MOSFET amplifiers.
- m) Simulate Multivibrators.
- n) Simulate Oscillators (RF & AF).
- o) Study an operational amplifier using PSPICE windows and find out: CMMR, gain band width product, slew rate, 3-db frequency, and input offset voltage.
- p) Simulate and study active low pass, high pass, and band pass filters using PSPICE windows.
- q) Simulate and study class A, B, C, and AB amplifier using PSPICE windows.
- r) Simulate logic expression..and determine its truth table.
- s) Simulate logic expression of adder and subtractor circuit and determine its truth table.

- t) Simulate a synchronous 4-bit counter and determine its count sequence.
- u) Simulate a master-slave flip-flop using NAND gates and study its operation.
Study the operation of asynchronous preset and clear .

NOTE : At least ten experiments have to be performed in the semester; out of which at least seven experiments should be performed from above list. Remaining three experiments may either be performed from the above list or designed and set by the concerned institution.

Introduction to MATLAB

Learning objectives

Introduction to Matlab, study of matlab functions. Writing simple programs using matlab, for handling arrays, files, plotting of functions etc. Writing M files for Creation of analog & discrete signals, plotting of signals etc. Filtering of analog & digital signals using convolution. Generation of noise signals (Gaussian, random, Poisson etc) Simulation using Simulink. Simulation study

Laboratories should include

- Design of analog low pass, bandpass, high pass and band elimination filters using Butterworth, Chebyshev etc.
- Anti-aliasing filters
- Bode plot
- Steady state and Transient analysis
- Z Transforms
- Fourier Analysis

Sr.No	Course content
1.	Exposure to different types of electrical accessories like types of switches, types of lamps, wires and cables
2.	Soldering Practice and fabrication of D.C Power supply circuits on General Purpose PCB/bread board.
3.	Importance of Neutral and structure Grounding and exposure to various earthing schemes.
4.	Assembling and disassembling of D. C. Machine, single phase motor and its meggering.
5.	Assembling and disassembling of single phase transformer and its meggering.
6.	Assembling and disassembling of three phase induction motor and its meggering.
7.	Calibration of Energy meter.
8.	Study of electric shocks and first aid treatments.

Note: Practical examination to be conducted covering entire syllabus given above.