

GUJARAT TECHNOLOGICAL UNIVERSITY

B.E Semester: 3 Electronics Engineering

Subject Code 131001
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) Study the operation of 555 timer oscillator using PSPICE.
- s) Simulate logic expression..and determine its truth table.
- t) Simulate logic expression of full adder circuit and determine its truth table.

- u) Simulate a synchronous 4-bit counter and determine its count sequence.
- v) 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 PCB Design

Using OrCAD, Altium, Eagle, PowerPCB or others Package.

Learning objectives:

This module conveys knowledge on the design and specification of PCB's, fabrication basics, classes of packages as well as basic technologies of packaging.

Learning outcomes and competences:

The student himself will be able to design, carry out simulation and manufacture the printed circuit board.

Contents to be cover:

- Principles of circuit design, design tools, layout techniques.
- Characteristics and materials of printed boards, layout and simulation
- PCB Fabrication : Mechanical production, pick and place, conventional components and SMD, via and
- Plated through connections, soldering, surface coating, multilayer boards, thermal effects.
- HYBRID PCBs: Thick film and thin film technology, material properties and fabrication.
- PACKAGING: Packaging of ICs, PCBs and hybrids, cases and thermal effects.

Laboratories should include

- Schematic Entry
- Netlist Creation
- Working with component libraries
- Design of Boards
- Layout of Parts
- Optimizing Parts Placements
- Pads and Via
- Manual and Auto Routing
- Handling Multiple Layers

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

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