



Kadi Sarva Vishwavidyalaya
Faculty of Engineering & Technology
Second Year Bachelor of EC Engineering

Subject Code: EC407- N	Subject Title: ELECTRONICS SIMULATION LABORATORY
-------------------------------	---

Course Objective:

The educational objectives of this course provides introduction to Simulation and Designing software's for electronics. Using simulation software the students can design and analyze various analog and digital circuits. The students can design layout for the desired circuits using MULTISIM/ULTIBOARD/ORCAD/PCB design tools. Using LABView various electronics systems can be implemented.

Teaching Scheme (Credits and Hours)

Teaching scheme				Total Credit	Evaluation Scheme					Total Marks
L	T	P	Total		Theory		IE	CIA	Pract.	
Hrs	Hrs	Hrs	Hrs		Hrs	Marks	Marks	Marks	Marks	
00	00	02	02	01	00	00	00	40	60	100

Outline of the Course:

Sr. No	Title of the Unit	Hours
1.	Introduction to PSpice/Multisim software	16
2.	Introduction to LabView:	16
		32

Total hours (Practical): 32

Total hours: 32



Kadi Sarva Vishwavidyalaya
Faculty of Engineering & Technology
Second Year Bachelor of EC Engineering

Detailed Syllabus:

Unit No	Topics	Lectures (Hours)	Weight age
1.	Introduction to PSpice/Multisim software: General purpose circuit simulation using Schematic Editor, Introduction to netlist command-based SPICE simulation, basic netlist commands. Basic circuit analyses: OPamp, Diode, BJT, Frequency Response, Digital Circuit. Tools: Spice/Multisim/ NGspice/ LTSpice/Orcad/Proteus	16	50%
2.	Introduction to LabView: Front panel, control and indicator (Input/output) - Basic Controls Block Diagram, Nodes,Wires, LabVIEW small Projects designing, Toolbars Building Configuration of MYRIO Hardware with LABVIEW2014.MYRIO installation step. Introduction of RIO Hardware-Interfacing.	16	50%
Total		32	100%

Instructional Method and Pedagogy (Continuous Internal Assessment (CIA) Scheme)

- At the start of course, the course delivery pattern, prerequisite of the subject will be discussed
- Lecture may be conducted with the aid of multi-media projector, black board, OHP etc. & equal weight age should be given to all topics while teaching and conduction of all examinations.
- Attendance is compulsory in lectures and laboratory, which may carries five marks in overall evaluation.
- One/Two internal exams may be conducted and total/average/best of the same may be converted to equivalent of 30 marks as a part of internal theory evaluation.
- Assignment based on course content will be given to the student for each unit/topic and will be evaluated at regular interval. It may carry an importance of ten marks in the overall internal evaluation.
- Surprise tests/Quizzes/Seminar/Tutorial may be conducted and having share of five marks in the overall internal evaluation.

Learning Outcomes:

- Design the electronics circuits using software tools like NGspice/LTSpice/Multisim.
- Simulate various analog and digital circuits using NGspice/LTSpice/Multisim
- Implement a different circuit in LabView.
- Design a circuit in LabView and Implement in a real Hardware with MyRio.



Kadi Sarva Vishwavidyalaya
Faculty of Engineering & Technology
Second Year Bachelor of EC Engineering

TEXT BOOKS & REFERENCE BOOKS:

1. SPICE Second Edition by Gordon W. Roberts and Adel S.Sedra.

List of Software/learning website:

- NGspice, LTSpice, MULTISIM, Orcad, Proteus or other open source PCB design tools,
- <http://www.linear.com/>,
- <http://www.expresspcb.com/>
- <http://ngspice.sourceforge.net/>



Kadi Sarva Vishwavidyalaya
Faculty of Engineering & Technology
Second Year Bachelor of EC Engineering

LIST OF EXPERIMENTS

Sr. No.	Experiment Title
1.	Draw different types of rectifier circuit using CAD and take print out of : (a) Single phase half wave (b) Single phase full wave (c) Bridge rectifier (using Multisim/Pscpipe)
2.	Simulate three resistances in series circuit and find out voltage and current in each resistance. (using Multisim/ Pscpipe)
3.	Simulate R-L series circuit and observe voltage wave forms across each component (using Multisim/Pscpipe)
4.	Simulate Half wave Rectifier circuit and observe voltage wave forms across each component (using Multisim/Pscpipe)
5.	Simulate Full wave Rectifier circuit and observe voltage wave forms across each component (using Multisim/Pscpipe)
6.	Using simulation prove that NAND gate is universal gate. (using Labview)
7.	Using simulation prove that NOR gate is universal gate. (using Labview)
8.	Using simulation prove De Morgan's theorem. (using Labview)
9.	Using simulation prove half adder and full adder circuit. (using Labview)
10	PROJECT using any one software.