

## S.E. (ELECTRONICS) SEMESTER IV

### Electronics Workshop-I

Period per week	Lecture		
	Practical	4	
	Tutorial		
		Hours	Marks
Evaluation System	Theory Examination	---	---
	Practical	---	---
	Oral Examination	----	25
	Term Work	---	25
	Total	---	50

### Detailed Syllabus

	<p>This syllabus is designed to encourage students to design and implement innovative ideas. The syllabus will give them in depth practical knowledge from design to the final verification stage. Documentation of any project is an important part of the project and students are expected to document their work properly in standard IEEE format.</p> <p><b>Every group of students should select different projects. Number of students should not be less than TWO and not more than THREE in one group.</b></p> <p><b>1. Study of Soldering Techniques and PCB Design</b> Students are expected to select any experiment* that they have already performed in earlier semester. Soldering and testing is to be done for the selected experiment. Perform simulation of the same experiment by using CAD tools. Schematic as well as PCB design is to be carried out using CAD tools.</p> <p><b>2. Design, Simulation and Implementation of Analog Project</b> Students are expected to design any* analog application of their choice. Perform simulation using software tools. PCB design, fabrication of PCB, testing and implementation should be done. Documentation of the project is to be done in standard IEEE format using Latex/WinTex. Project report should include abstract in maximum 100 words, keywords, introduction, design, simulation, implementation, results, conclusion and references.</p> <p><b>3. Digital Project</b> Students are expected to design any* digital application of their choice. Perform simulation using software tools for hardware description. Design</p>
--	--

should be synthesized using FPGA/CPLD. Documentation of the project is to be done in standard IEEE format using Latex/WinTex. Project report should include abstract in maximum 100 words, keywords, introduction, design, simulation, implementation, results, conclusion and references.
--

**\* To be approved by the subject in-charge**

**References:-**

1. Roberts G. W. and Sedra A.S., “SPICE”, 2<sup>nd</sup> ed, USA
2. Rashid M.H. “SPICE for circuits and electronics using pSpice”, Prentice Hall
3. Tuinenger P.W., “SPICE: A Guide to circuit simulation and analysis using pSpice”, Prentice Hall
4. Bosshart, “Printed Circuit Boards: Design and Technology”, Tata McGraw Hill
5. Orcad/PCBII , “User’s Guide”.
6. [www.xilinx.com](http://www.xilinx.com)