III B.Tech II Semester

15AEE36-POWER ELECTRONICS AND SIMULATION LAB

L T P C
0 03 2

Course Objectives:

- *To learn the characteristics of power electronics devices*
- *To understand the control methods of rectifiers and choppers*
- *To learn different gating circuits for thyristor turn on*
- *To learn the operation of ac voltage controllers and inverters*
- *To learn the simulation of power electronics circuits*
- *To understand the concepts of different loads.*

Any Eight of the following Experiments are to be conducted

1. Study of Characteristics of SCR, MOSFET & IGBT
2. Gate Firing Circuits for SCR's
3. Single Phase AC Voltage Controller with R and RL Loads
4. Single Phase Fully Controlled Bridge Converter with R and RL Loads
5. Forced Commutation Circuits (Class A, Class B, Class C, and Class D & Class E)
6. Buck converter with R and RL loads
7. Single Phase Parallel, Inverter with R And RL Loads
8. Single Phase Cycloconverter with R and RL Loads
9. Single Phase Half Controlled Converter with R Load
10. Three Phase Half Controlled Bridge Converter with R-Load
11. Single Phase Series Inverter with R and RL Loads
12. Single Phase Bridge Converter with R and RL Loads
13. Single Phase Dual Converter with RL Loads

Any Two Simulation Experiments are to be conducted

1. PSPICE Simulation of Single-Phase Full Converter Using RLE Loads and Single-Phase AC Voltage Controller Using RLE Loads
2. PSPICE Simulation of Resonant Pulse Commutation Circuit and Buck Chopper
3. PSPICE Simulation of Single Phase Inverter with PWM Control
4. Develop Simulation Model for a Four Quadrant Chopper and Observe the V-I Characteristics in All Quadrants.
5. Write a Program to Plot the Load Voltage and Current Waveforms of an Inverter for Different Delay Angles.

Course Outcomes:

- *Design a Commutation circuit of a thyristor, Control a supply voltage using converters.*
- *Select a suitable power electronic device for different applications.*
- *Use PSPICE software for determining the performance of given power electronic converters.*

REFERENCE BOOKS:

1. Simulation of Electric and Electronic circuits using PSPICE – by M.H.Rashid, PHI.
2. PSPICE reference guide – Microsim, USA.
3. MATLAB and its Tool Books user's manual and – Mathworks, USA.

V.[illegible]
BOS-chairman