EE605PC: POWER ELECTRONICS LAB

B.Tech. III Year II Sem.

L	T	P	C
0	0	3	2

Prerequisite: Power Electronics

Course Objectives:

- Apply the concepts of power electronic converters for efficient conversion/control of power from source to load.
- Design the power converter with suitable switches meeting a specific load requirement.

Course Outcomes: After completion of this course, the student is able to

- Understand the operating principles of various power electronic converters.
- Use power electronic simulation packages& hardware to develop the power converters.
- Analyze and choose the appropriate converters for various applications

Any eight experiments should 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 half controlled &fully controlled bridge converter with R and RL loads
5. Forced Commutation circuits (Class A, Class B, Class C, Class D & Class E)
6. Single Phase Cycloconverter with R and RL loads
7. Single Phase series& parallel inverter with R and RL loads
8. Single Phase Bridge inverter with R and RL loads

Any two experiments should be conducted

1. DC Jones chopper with R and RL Loads
2. Three Phase half controlled bridge converter with R-load
3. Single Phase dual converter with RL loads
4. (a)Simulation of single-phase Half wave converter using R and RL loads

   (b)Simulation of single-phase full converter using R, RL and RLE loads

   (c)Simulation of single-phase Semi converter using R, RL and RLE loads

5. (a)Simulation of Single-phase AC voltage controller using R and RL loads

   (b)Simulation of Single phase Cyclo-converter with R and RL-loads
6. Simulation of Buck chopper
7. Simulation of single phase Inverter with PWM control
8. Simulation of three phase fully controlled converter with R and RL loads, with and without freewheeling diode. Observation of waveforms for Continuous and Discontinuous modes of operation.
9. Study of PWM techniques

Reference Books:

1. M. H. Rashid, Simulation of Electric and Electronic circuits using PSPICE – by M/s PHI Publications.
2. User's manual of related softwares
3. Reference guides of related softwares
4. Rashid, Spice for power electronics and electric power, CRC Press