**B.Tech III Year II Semester**

## JNTUA COLLEGE OF ENGINEERING (AUTONOMOUS) PULIVENDULA

## 19AEE66- POWER ELECTRONICS LAB

| L | T | P | C |
|---|---|---|---|
| 0 | 0 | 3 | 1.5 |

**Course Objectives:**

- Understand and analyze various characteristics of power electronic devices with gate firing circuits and forced commutation techniques.
- Analyze the operation of single-phase half & fully-controlled converters and inverters with different types of loads
- Analyze the operation of DC-DC converters, single-phase AC Voltage controllers, cyclo converters with different loads
- Create and analyze various power electronic converters using PSPICE software.

**Any Eight of the following Experiments are to be conducted**

1. Study of Characteristics of SCR, Triac, MOSFET & IGBT
2. Gate firing circuits for SCR's: (a) R triggering (b) R-C triggering
3. Gate driver Circuits for MOSFET and IGBT
4. Single Phase AC Voltage Controller with R and RL Loads
5. Single Phase fully controlled bridge converter with R and RL loads
6. Forced Commutation circuits (Class A, Class B, Class C, Class D & Class E)
7. DC Jones chopper with R and RL Loads
8. Single Phase Parallel, inverter with R and RL loads
9. Single Phase Cycloconverter with R and RL loads
10. Single Phase Half controlled converter with R load
11. Three Phase half controlled bridge converter with R-load
12. Single Phase series inverter with R and RL loads
13. Single Phase Bridge converter with R and RL loads
14. Single Phase dual converter with RL loads.

**Any two simulation experiments with PSPICE /PSIM / MATLAB**

15. PSPICE / MATLAB / PSIM simulation of single-phase full converter using RLE loads and single-phase AC voltage controller using RLE loads
16. PSPICE/ MATLAB/ PSIM simulation of resonant pulse commutation circuit and Buck converters and chopper.
17. PSPICE / MATLAB/ PSIM simulation of single phase Inverter with PWM control.

**Reference Books:**


1. Power Electronics Laboratory: Theory, Practice and Organization (Narosa series in Power and Energy Systems) by O.P. Arora, Alpha Science International Ltd., 2007.
2. Simulation of Electric and Electronic circuits using PSPICE – by M.H.Rashid, M/s PHI Publications.
3. PSPICE A/D user's manual – Microsim, USA.
4. PSPICE reference guide – Microsim, USA.
5. MATLAB and its Tool Books user's manual and – Mathworks, USA.


**Course Outcomes:**

At the end of this Course the student will be able to

- Design a Commutation circuit of a thyristor, control a supply voltage using converters. **L1**
- Select a suitable power electronic device for different applications. **L2**
- Use PSPICE software for determining the performance of given power electronic Converters. **L3**