

### III Year B.Tech. EEE I-Sem

### POWER ELECTRONICS AND SIMULATION LAB

#### Course Outcomes:

<b>A16287</b>	<b>POWER ELECTRONICS &amp; SIMULATION LAB</b>
C328.1	Examine the characteristics of SCR, MOSFET, & IGBT, and analyze triggering circuits
C328.2	Examine input and output waveforms of AC-DC converters.
C328.3	Examine input and output waveforms of AC-AC
C328.4	Examine input and output waveforms of DC-DC Converters
C328.5	Design of converters and inverters using p-spice and verifying results.

#### LIST OF EXPERIMENTS

**Any ten of the following experiments are required to be conducted.**

1. Study of the characteristics of SCR, MOSFET & IGBT.
2. Gate Firing Circuits for SCRs (R- Triggering, RC Triggering & UJT Triggering).
3. Single Phase AC voltage Controller with R & RL Loads.
4. Single Phase fully Controlled Bridge Converter with R& RL Loads.
5. DC Jones Chopper with R& RL Loads.
6. Single Phase Parallel Inverter with R& RL Loads.
7. Single Phase Cyclo-Converter with R& RL Loads.
8. Single Phase Series Inverter with R& RL Loads.
9. Single Phase Half controlled converter with R Load.
10. PSPICE simulation of single-phase full converter using RLE loads and single-phase AC voltage controller using RLE loads.
11. PSPICE simulation of resonant pulse commutation circuit and Buck Chopper.
12. PSPICE simulation of single phase Inverter with PWM control.

#### REFERENCE BOOKS:

1. Simulation of Electrical and Electronics Circuits Using PSPICE- by M.H. Rashid, M/s PHI publications.
2. PSPICE A/D Users Manual-Microsim, USA
3. PSPICE reference Guide –Microsim, USA
4. MATLAB and Tool books user's manual and- MATH Works, USA