II B.Tech I Sem

15AEE06-ELECTRIC CIRCUITS AND SIMULATION LAB

L	T	P	C
0	0	3	2

Objectives:

- To verify all theorems with practically.
- To know performance of RLC series and parallel circuits.
- To know the measurement of three phase power.
- To emphasis a brief introduction to PSPICE for analysis of DC Circuits.

List of Experiments

1) Verification of Thevenin's and Norton's Theorems
2) Verification of Superposition Theorem and Maximum Power Transfer Theorem
3) Verification of Compensation Theorem
4) Verification of Reciprocity , Millmann's Theorems
5) Locus Diagrams of RL and RC Series Circuits
6) Series and Parallel Resonance
7) Determination of Self, Mutual Inductances and Coefficient of Coupling
8) Z and Y Parameters
9) Transmission and Hybrid Parameters
10) Measurement of Active and Reactive Power for Star and Delta Connected Balanced Loads
11) Measurement of 3-Phase Power by Two Wattmeter Method for Unbalanced Loads

Any four simulation experiments are to be conducted

1) DC Transient Response
2) Mesh Analysis
3) Nodal Analysis
4) Frequency response of RLC Series Circuits
5) Analysis of RL and RC Series circuits for DC and AC Excitation
6) Analysis of Three Phase balanced and unbalanced systems
7) Verification of the maximum power dissipation (plot the power dissipated versus the load).

References:

1. Simulation of Power Electronics Circuit, M B Patil, V Ramanarayan and V T Ranganat, Alpha Science International Ltd., 2009.

U. Jeevan
BoS-Chairman

2. Public Domain Simulator: http:// www.ee.iitb.ac.in/~sequel
3. PSPICE A/D user's manual – Microsim, USA.
4. PSPICE reference guide – Microsim, USA.

Outcomes:

1. Understand and compare basic electric circuit theorems with actual working circuits.

2. Students can Design and understand RLC series and parallel circuits and its resonance condition.

3. They can able to measure power in three phase circuits in day to day life.

4. They can also be able to understand simulation programs for DC circuit analysis using PSPICE.