IV Year I Semester	L	T	P	C
17EE711	0	0	3	2

ELECTRICAL SIMULATION LAB

Learning objectives:

1. To simulate integrator circuit, differentiator circuit, Boost converter, Buck converter, full convertor.
2. To simulate transmission line by incorporating line, load and transformer models
3. To perform transient analysis of RLC circuit and single machine connected to infinite bus (SMIB).

Any 10 experiments of the following are to be conducted.

1. PSPICE simulation of series RLC circuits for step, pulse & sinusoidal input.
2. PSPICE simulation of D.C. network with sub circuit
3. PSPICE simulation of A.C circuits.
4. Transfer function analysis of i) time response for step input ii) frequency response for sinusoidal input.
5. Stability analysis using nyquist plots for the transfer functions of systems up to 5th order
6. Simulation of Boost converter.
7. Simulation of Buck converters.
8. Effect of source inductance on single phase fully controlled bride rectifier.
9. Dynamic characteristics simulation of SCR.
10. Integrator circuits using op–amp.
11. Differentiator circuits using op–amp.
12. Simulation of D.C separately excited motor using transfer function approach.
13. Modeling of transformer and simulation of loss transmission line.
14. Analysis of three phase circuit representing the generator transmission line and load. Plot three phase currents & neutral current.
15. Transient analysis of single machine connected to infinite bus (SMIB)

Learning outcomes:

1. Able to simulate integrator circuit, differentiator circuit, Boost converter, Buck converter, full converter.
2. Able to simulate transmission line by incorporating line, load and transformer models.
3. Able to perform transient analysis of RLC circuit and single machine connected to infinite bus (SMIB).

Text books:

1. "Simulation of Power Electronic Circuit", by M.B. Patil, V.Ramanarayan, V.T. Ranganathan. Narosha, 2009.
2. Pspice for circuits and electronics 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 user`s manual – Mathworks, USA.
6. MATLAB – control system tool box – Mathworks, USA.
7. SIMULINK user`s manual – Mathworks, USA.
8. EMTP User`s Manual.
9. SEQUEL– A public domain circuit simulator available at www.ee.iitb.ac.in/~sequel.