ECE 521 **Fall 2016**

Final Project Proposal **Due 11/16/16**

You are required to submit a project proposal (maximum 2 pages including text, figures, and references). The following outline is to be used for the proposal.

- *Problem*: background and statement of the problem.
- *Objective*: what you plan to do.
- *Novelty* or *Significance*: what is the novel aspect of your work or why it would be interesting or useful.
- *Approach*: how you plan to solve the problem.

The project can be on modeling, simulation, analysis, or on a topic that you can justify being related to the course. I encourage you to choose a project that is of relevance to your research. Originality and creativity will be viewed positively and it is expected that you will read at least TWO journal papers and synthesize some new ideas for your project. You will be required to make a presentation in class the week of Nov. 28. **Final project reports are due no later than 12 noon of Dec. 8, 2016 (Thursday).** The final project report should include a ONE page *Executive Summary* that contains the following sections: 1) Project description 2) Approach 3) Accomplishments.

The final project will be graded on the following components. The total score is 30.

1.	Novelty/Initiative	5
2.	Analysis/Research	5
3.	Results/Accomplishments	6
4.	Presentation	5
5.	Exec Summary	3
6.	Final Report	6

Some Basic Project Suggestions (In case you are unable to define something)

1. Extend *myspice* to handle new analyses and/or models. These include PZ, sensitivity, distortion (Volterra series), Fourier integral method, or noise analyses. For models consider BSIM3/4, SOI, transmission lines, time-domain simulation of N-ports described in the frequency domain, or other elements.

2. Look at some different integration methods. (Exponential fitting, implicit Runge-Kutta methods, etc.)

3. Implement variable timestep/variable order integration methods with error control in *myspice.*

4. Implement a variable timestep integration method with error control for linear circuits which maximizes the use of uniform timesteps thereby minimizing the refactorization of the Jacobian matrix.

5. Implement a Homotopy method or damped Newton method for nonlinear equation solution in your simulator.

6. Study noise models and analyses.

7. Benchmark SPICE2/SPICE3/HSPICE/PSPICE on a set of circuits. Look at performance, memory, and convergence. PC-based environments are a good possibility.

8. Implement a parallel/multi-core version of your simulator to run on a cluster of workstations or a multi-core processor.

9. Study the feasibility of using the graphics processing unit (GPU) for circuit simulation.

10. Implement your own sparse matrix solver or interface the Univ. of Florida sparse matrix solvers in myspice and provide a performance comparison.

11. Implement iterative solvers in your circuit simulator.

12. Implement steady-state analyses methods in your simulator. These could be the shooting method or the harmonic balance method.

13. Oscillator analysis: robust simulation of periodic steady state, simulation of phase noise, jitter, or injection locking

14. Implement the Asymptotic waveform evaluation algorithm in myspice.

15. Implement a model-order reduction technique in myspice.