ECE 65 – Components and Circuits Laboratory – Fall 2010 – Dan Sievenpiper

Webpage: http://ece-classweb.ucsd.edu/fall10/ece65/index.html
Lectures: Monday, Wednesday, Friday, 10:00 – 10:50 am, Cognitive Science Building 002
Discussion: Wednesday, 1:00 – 1:50 pm, Peterson Hall 104

Instructor: Professor Dan Sievenpiper
Office: Jacobs Hall Room 3209
Phone: 858-822-6678
Email: sievenpiper @ ece.ucsd.edu
Office Hours: Monday, 11:00 am – 1:00 pm

Teaching/Lab Assistants:
Alex Bishop (arbishop @ ucsd.edu)
John Zhao (jzhao @ ucsd.edu)
Andy Leung (andytcleung @ hotmail.com)

Textbook:
None required
Professor Najmabadi's ECE65 notes (posted on class website) will be extremely helpful
Also Recommended:

- "Microelectronic Circuits," by Sedra & Smith (textbook for ECE102)
- "The Art of Electronics," by Horowitz & Hayes

Laboratory Sections: (please go to your enrolled lab section)
Location: EBU2, Room 339
Monday, 2:00 – 5:00 pm (Andy & Alex)
Tuesday, 12:30 – 3:30 pm (Alex & John)
Wedsday, 2:00 – 5:00 pm (Andy & John)

Pre-labs are due when you start the lab
Lab write-ups are due the following week in your lab section

Open lab times:
Thursday, 1:00 – 3:00 pm (Alex)
Friday, 11:00 am – 1:00 pm (Andy)
Friday, 2:00 – 4:00 pm (John)

Grades

Lab reports:	25%
Quizes	25%
Final:	50%

Course Description:
ECE 65 is an introduction to linear and nonlinear components and circuits. This course is designed to build upon material learned in ECE35 and ECE45, and introduces ECE students to practical circuits and solving real-world problems. The students will learn to model and design practical circuits using idealized circuit models, to account for the interactions among various parts of a circuit, the concept of feedback, and other issues encountered in designing a practical circuit.

Material Covered
- Diodes (Large and Small-signal models)
- BJT transistors (Large signal model, Digital Gates, Biasing, Small-signal models, Amplifiers)
- FET transistors (Large signal model, Digital Gates)
- Interaction of sub-circuits (e.g., Transfer function, Input & output resistance)
- Concept and utility of Feedback Circuit Design
- Using PSpice to simulate circuits (self study)

Course Mechanics
The course material is reviewed in the lecture. Lecture notes are posted on the class web site. The description of lab exercises for the following week is also posted on the class web site. The lab exercises are designed to enforce the material covered in the lecture and make students understand the differences between circuit models and practical circuits. We analyze many circuits by (1) simple analytical tools, (2) more detailed PSpice simulations, and (3) lab measurements. In this way, the students will understand when various models apply and how to account for various effects. Each Lab includes a design problem: you are asked to design a specific circuit, simulate its response with PSpice, built it in the lab and see if it performs according to design specifications.

PSpice
An essential part of the course work is PSpice simulation of various circuits. Lectures will not cover PSpice simulation syntax, and students should learn PSpice on their own. However, lab exercises are designed to introduce students to PSpice capabilities gradually. You can purchase commerical versions of PSpice, download free versions from the web, or use the PSpice lab (329 EBUII).
Sources of PSpice software:
- The PSpice Lab (329 EBUII) has OrCAD PSpice. Note that PSpice lab has the "full" version of PSpice (instead of smaller, less capable Demo version).
- OrCAD PSpice (http://www.cadence.com/products/orcad/pages/downloads.aspx) You can download a Demo version of this software for free. Click on "OrCAD Demo DVD (Capture & PSpice Only)". The download link is near the bottom of each section. You will need to register as a user of Cadenace (free).
- National Instruments Multisim (http://www.interactiv.com/) This is a commercial software (i.e., you have to buy it, but it is cheap for students).