Analog Electronic Devices
(ESI038 / SE047)

Jan-May 2007
7-9 hrs Tuesdays and Fridays
Classroom D-301

Instructor Information

José Ernesto Rayas Sánchez, Ph.D.
Office: T-333
Office hours: 10-14, 17:00-21:00 hrs
Tel: 3669-3598, Ext. 3096
Email: erayas@iteso.mx
Website: http://iteso.mx/~erayas

General Description

This course will introduce students to the field of analog electronics analysis and design, by studying the most fundamental semiconductor devices (main types of diodes and transistors) as well as their basic applications. The physical structure and behavior of these devices is studied. Students will be enabled to analyze and design common diode and transistor circuits, with emphasis on fundamental linear amplification stages. The use of simulation software and practical implementations in the laboratory will be emphasized throughout the course.

Prerequisites

Circuit Analysis I and II

Objectives

By the end of the course the student will be able to:

A. Apply basic semiconductor theory to explain the junction diode operation (APPLICATION)

B. Explain the I-V characteristics of junction diodes (COMPREHENSION)

C. Analyze the most typical diode applications (ANALYSIS)

D. Describe the physical structure and modes of operation of the BJT (COMPREHENSION)

E. Describe the simplest DC and AC models of the BJT (COMPREHENSION)

F. Analize DC circuits with bipolar transistors (ANALYSIS)

G. Explain the operation of the BJT as an amplifier (COMPREHENSION)

H. Identify the fundamental BJT amplifier configurations (COMPREHENSION)
I. Design single-stage and multi-stage BJT amplifiers (SYNTHESIS)
J. Apply bipolar transistors as switches (APPLICATION)
K. Describe the physical structure and modes of operation of field-effect transistors (COMPREHENSION)
L. Describe the simplest DC and AC models of the FETs (COMPREHENSION)
M. Analize DC circuits with field-effect transistors (ANALYSIS)
N. Identify the fundamental FET amplifier configurations (COMPREHENSION)
O. Design single-stage and multi-stage FET amplifiers (SYNTHESIS)
P. Apply field-effect transistors as switches (APPLICATION)
Q. Analyze basic MOS and bipolar digital circuits (ANALYSIS)
R. Verify the behavior of simple electronic circuits using simulation software (APPLICATION)
S. Use measurement instruments to characterize simple electronic systems (APPLICATION)
T. Implement in the lab simple electronic systems previously analyzed, and contrast measurements with theoretical predictions (ANALYSIS)

General Contents

1. Diode physical operation and modeling
    1.1. Ideal Diode
    1.2. I-V characteristics
    1.3. Diode physical operation
    1.4. Types of diodes
    1.5. Large signal and DC models
    1.6. Small signal models (low and high frequency)
    1.7. Diode manufacturing data sheets
2. Diode applications
    2.1. Rectifiers
    2.2. Peak detectors
    2.3. Clippers, clampers and comparators
    2.4. Voltage regulators
    2.5. Voltage multipliers
    2.6. Samplers
    2.7. Other applications
3. Bipolar Junction Transistors (BJT)
    3.1. Physical structure and general operation of BJTs
    3.2. Detailed operation of npn BJTs
    3.3. The pnp BJT
    3.4. Static characteristics of BJTs

   - 3.5. BJT models for DC and large signal
   - 3.6. Two-Port Networks
   - 3.7. BJT models for small signal (low and high frequency)
   - 3.8. BJT manufacturing data sheets
   - 3.9. Basic biasing circuits for BJTs
   - 3.10. Common Emitter (including emitter resistor), Common Base and Common Collector configurations
4. FET physical operation and modeling
   - 4.1. Types of field effect transistors
   - 4.2. Physical structure and general operation of MOSFETS and JFETs
   - 4.3. Static characteristics of MOSFETs and JFETs
   - 4.4. MOSFET and JFET models for DC and large signal
   - 4.5. FET models for small signal (low and high frequency)
   - 4.6. FETs manufacturing data sheets
   - 4.7. Basic biasing circuits for MOSFETs and JFETs
   - 4.8. Common Souce (including source resistor), Common Gate and Common Drain configurations
5. Biasing Circuits and Multistage Amplifiers
   - 5.1. Biasing circuits (for discrete and integrated circuit implementations)
   - 5.2. Stability of biasing circuits
   - 5.3. Introduction to multi-stage amplifiers
6. Introduction to MOS and bipolar digital circuits
   - 6.1. CMOS technology for digital integrated circuits
   - 6.2. General features of logic circuits
   - 6.3. Basic CMOS logic gates

Relationship between Contents and Objectives

	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T
1	√	√																		
2			√															√	√	√
3				√	√	√	√											√	√	√
4											√	√	√					√	√	√
5								√	√					√	√			√	√	√
6										√						√	√	√		

Course Skeleton

Below it is shown the course skeleton. For the proposed course skeleton it is assumed: a group of 15 to 35 students; 2 sessions per week, 2 hours long each, during 16 weeks; laboratory available 8 hours a day, from Monday to Friday; simulation software available at ITESO and/or at home (WinSpice, OrCad –Cadence–, Electronic Workbench, or any other similar circuit simulator).

Week	Activity	Week	Activity
1		9	
2		10	Lab Report 3, Exam 3(Contents 4)
3	Lab Report 1	11	
4	Exam 1(Contents 1 and 2)	12	
5		13	
6	Lab Report 2	14	Lab Report 4, Exam 4 (Contents 5)
7	Exam 2 (Contents 3)	15	
8		16	Lab Report 5 (Contents 6)

Assessment

The overall grade in this course will be built from the following elements:

4 Exams	50%
5 Lab reports	40%
Participation	10%
Final exam	10% (optional)

To pass this course, the student must have an average in the 4 exams of at least 6/10.

The contents of the exams and lab reports are most of the time cascaded, so that the knowledge and skills developed in a given unit are usually needed in the next units.

The exams will be applied during class and will take 1 hour and 50 minutes each. Each exam typically consists of three sections: selected response assessment (multiple choice, true/false, etc.), brief analysis problems, and a more complex analysis problem or a design problem.

The lab reports might consist of simulations in the computer, implementations in the laboratory, or a combination of both. Lab projects and reports can be realized individually or in teams of 2-3 students.

Missed exams and lab reports deadlines can not be made up.

An optional final exam can be taken to earn extra credit. This is applied at the end of the course, and evaluates all the six contents units.

Assignments in the form of solved problems will be suggested for the student to test their comprehension of concepts and their ability to analyze and design common electronic circuits. It is strongly suggested to the students to solve these problems, since they are fundamental in the learning process.

The quality of the participation of the students during the lectures will be graded. This participation will be evaluated based on student’s attitude and performance during class: punctuality, willingness to ask relevant questions, respect to others, attention during class, ability to answer questions, ability to solve problems during the problem solving sessions, ability to solve assignments, etc.

It is expected that the student will be able to dedicate an average of 10 hours of work per week to this course, including attending classes (6 hours of extra-class work).

Teaching Methods

This course will use a variety of teaching methods including: lecturing, problem solving sessions, self-conducted laboratory work, computer simulations, lab report writing, assignments, and readings.

Important information related to the course will be posted in the instructor's web site through out the semester. Open and frequent communication with the instructor is encouraged. Collaboration between the students for the realization of the lab projects and assignments is also encouraged.

Out of class student's work is fundamental in the learning process. It is extremely important to read the corresponding materials before each lecture. It is also very important to solve the suggested assignments, either alone or in collaboration with other classmates.

Students will be asked to solve problems in most lectures. These problems are intended to re-inforce the learning process as well as to evaluate students' participation. Students should bring a calculator to each class.

The course will be conducted mainly in Spanish, but some of the lectures and discussions might be in English. Most of the written material for the course will be available in English. The lab reports can be submitted in either English or Spanish.

Textbook

Microelectronic Circuits, Fifth Edition
Adel S. Sedra and Kenneth C. Smith
Oxford University Press / McGraw-Hill
(621. 381 SED)
http://www.us.oup.com/us/companion.websites/0195142519/

Other Bibliography

Electronic Devices and Circuit Theory, Seventh Edition
Boylestad, Robert L. and Nashelsky, Louis
Prentice Hall
http://wps.prenhall.com/chet_boylestad_electronic_8

Microelectronic Circuit Design
Jaeger, Richard C.
McGraw-Hill
http://www.eng.auburn.edu/~jaeger/

Análisis y Diseño de Circuitos Electrónicos, Tomos I y II
Neamen, Donald A.
McGraw-Hill Interamericana
http://www.mhhe.com/engcs/electrical/neamen/

Electronic Principles

Albert P. Malvino
McGraw Hill
http://www.malvino.com/ep/index.html

Electronics, 2nd edition
Allan R. Hambley, Michigan Technological University
Prentice Hall

Edición y Simulación de Circuitos con OrCAD
José Luis Calvo-Rolle
Alfaomega Ra-Ma, 2004.

Spice, A Guide to Circuit Simulation & Analysis Using PSpice
Paul W. Tuinenga
2nd Ed., Prentice-Hall, 1992.

Other Resources

Analog Electronics Area at ITESO
http://www.desi.iteso.mx/analog/

Prof. Rayas' website:
http://iteso.mx/~erayas/electronics_i.htm

Sedra/Smith web site
http://www.oup-usa.org/sc/0195116631/sedrasmith.org

Britney's Guide to Semiconductor Physics
http://www.britneyspears.ac/lasers.htm

WinSpice circuit simulator
http://www.winspice.com/

PSpice (Cadence, Orcad)
http://www.cadence.com/orcad/

Multisim circuit simulator from Electronics Workbench's
http://www.electronicsworkbench.com

Modeling and simulation websites
http://www-device.eecs.berkeley.edu/
http://www.ee.washington.edu/circuit_archive/models/
http://www.cadence.com/products/si_pk_bd/downloads/pspice_models/index.aspx

NOTA: En caso de alguna dificultad o confusión respecto de este programa de estudios (por estar en idioma inglés), favor de consultar directamente con el profesor.