Analog Electronic Circuits
(ITE0341)

August-December 2006
9-11 hrs Tuesdays and Thursdays
Classroom A307

Instructor Information

Esteban Martínez Guerrero, Ph.D.
Office: T-334
Office hours: 9-14, 16:00-19:00 hrs
Tel: 3669-3598, Ext. 3177
Email: emguerrero@iteso.mx
Website: http://iteso.mx/~emguerrero

General Description

This course will enable students to analyze and design two of the most fundamental analog electronic systems: amplifiers and oscillators. The students will also gain a solid foundation for the analysis of integrated circuits. The use of simulation software and practical implementations in the laboratory will be emphasized through out the course.

Prerequisites

Analog Electronic Devices (ITE0342)

Objectives

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

A. Design differential and multistage amplifiers at mid frequencies, implemented with bipolar transistors as well as with field effect transistors (SYNTHESIS)

B. Explain the origin of the non-ideal effects in operational amplifiers (COMPREHENSION)

C. Analyze power amplifiers and output stages typically used in integrated circuits (ANALYSIS)

D. Apply the time constant method to analyze the frequency response of single and multistage amplifiers (APPLICATION)

E. Identify the effects of feedback on the amplifier's behavior (COMPREHENSION)

F. Efficiently analyze feedback amplifiers (ANALYSIS)

G. Analyze the stability of amplifiers and apply basic frequency compensation techniques (ANALYSIS)

H. Identify the fundamental principles for circuit oscillation (COMPREHENSION)

I. Design sinusoidal oscillators at low and mid frequencies (SYNTHESIS)

J. Verify the behaviour of amplifiers and oscillators using simulation software (APPLICATION)

K. Use measurement instruments to characterize simple electronic systems (APPLICATION)

L. Implement in the lab simple electronic systems previously analyzed, and contrast measurements with theoretical predictions (ANALYSIS)

General Contents

1. Differential and multistage amplifiers.
    - 1.1. The BJT differential pair
        - 1.1.1. Large-signal operation
        - 1.1.2. Small-signal operation
        - 1.1.3. Differential gain
        - 1.1.4. Common-Mode gain and Common-Mode Rejection Ratio
    - 1.2. IC bias circuits (current mirrors)
    - 1.3. Differential pair with active loads
    - 1.4. The FET differential pair
    - 1.5. Multistage amplifiers
        - 1.5.1. coupling types (capacitive, inductive, direct)
        - 1.5.2. Cascaded amplifiers
        - 1.5.3. Cascode amplifiers
2. Output stages and power amplifiers
    - 2.1. Classification of output stages
        - 2.1.1. Class A power amplifiers
        - 2.1.2. Class B and class AB power amplifiers
            - 2.1.2.1. Crossover distortion and reduction techniques
    - 2.2. Power transistors
    - 2.3. Thermal resistance and heat sinks calculations
    - 2.4. IC power amplifiers
3. Frequency response
    - 3.1. Transfer functions
        - 3.1.1. The complex frequency domain
        - 3.1.2. Bode plots of amplifiers
    - 3.2. Low frequency and high frequency models of transistors.
    - 3.3. Single time constants method to determine cutoff frequencies
        - 3.3.1. low frequency response
        - 3.3.2. high frequency response
        - 3.3.3. Miller effect
    - 3.4. Frequency response of differential pair and cascode amplifiers.
4. Negative feedback and stability
    - 4.1. Some properties of negative feedback
        - 4.1.1. Gain desensitivity

- 4.1.2. Bandwidth extension
- 4.1.3. Noise reduction
- 4.1.4. Reduction in non-linear distortion

4.2. The four basic feedback topologies
- 4.2.1. The Series-Shunt feedback amplifiers
- 4.2.2. The Series-Series feedback amplifiers
- 4.2.3. The Shunt-Series feedback amplifiers
- 4.2.4. The Shunt-Shunt feedback amplifiers

4.3. The stability problem of feedback amplifiers
- 4.3.1. Methods to stabilize feedback amplifiers
  - 4.3.1.1. Frequency compensation

5. Oscillators and signal generators

5.1. Basic principles of sinusoidal oscillators
- 5.1.1. The oscillation criterion

5.2. Non-linear amplitude control

5.3. RC oscillators
- 5.3.1. The Wien bridge oscillator
- 5.3.2. The Phase-Shift oscillator
- 5.3.3. The Quadrature oscillator

5.4. LC and crystal oscillators
- 5.4.1. LC tuned oscillator
- 5.4.2. Crystal oscillator

5.5. Bistable multivibrators

5.6. Non-linear waveform-shaping circuits

Relationship between Contents and Objectives

	1	2	3	4	5
A	✓				
B	✓				
C			✓		
D				✓	
E				✓	
F				✓	
G					✓
H					✓
I		✓			
J	✓		✓	✓	✓
K	✓				✓
L	✓				✓

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, Electronic Workbench, OrCad – Cadence –, or any other similar circuit simulator).

It is also expected that the student will be able to dedicate an average of 10 hours of work per week to this course, including attending classes.

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

Assessment

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

4 Exams	50%
4 Lab reports	20%
1 Project	15%
Participation	15%
Final exam	10% (optional)

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

Each exam and lab report focuses on one of the 5 general contents units. These contents are naturally cascaded, so that the knowledge and skills developed in a given unit are needed in the next units.

The exams will be applied during class, and will have a duration of 1 hour and 50 minutes each. Each exam 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.

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. These assignments will

not be graded. 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 mainly based on student's performance for homework and problem solution during class. Other factors will considered, such as attitude and performance during class: punctuality, willingness to ask relevant questions, respect to others, attention during class, ability to answer questions, etc.

An optional final exam can be taken to earn extra credit. This is applied at the end of the course, and evaluates all the five contents units. It has a duration of 2 hours.

Teaching Methods

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

A pre-test will be applied in the first day of class with the objective of assessing the background knowledge and skills relevant to the course. The results of this evaluation will be used by the instructor to suggest some remedial readings and exercises to individual students, as well as to redesign, if necessary, the approach to the course contents and objectives. This pretest will not be considered for grading.

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 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, either in the textbook or in the course handouts (lecture presentations, available in the website indicated below). It is also very important to solve the suggested assignments, either alone or collaborating with other classmates.

The students will be asked to solve a problem in most lectures. These problems are intended to re-inforce the learning process as well as to evaluate students' participation. The 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 held 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
ISBN: 970-10-5472-5

References and Other Resources

Lectures for the course available in the web site:
http://iteso.mx/~erayas

Complementary lectures and updated labs are available in the web site:
http://iteso.mx/~emguerrero

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

Electronic Devices and Circuit Theory, Seventh Edition
Boylestad, Robert L. and Nashelsky, Louis
Prentice Hall

Microelectronic Circuit Design
Jaeger, Richard C.
McGraw-Hill

Análisis y Diseño de Circuitos Electrónicos, Tomos I y II
Neamen, Donald A.
México, 1a. edición 1999
McGraw-Hill Interamericana

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

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

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

Cadence (Pspice Orcad), Spice simulator
http://www.cadencepcb.com/products/pspice/

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

MicroCap (Spectrum Software), Spice simulator
http://www.spectrum-soft.com/index.shtm

TopSpice (Penzar Development), Spice simulator
http://www.penzar.com/

T-Spice (Tanner Research Inc.)
http://www.tanner.com/EDA/

ICAP/4 (Intusoft), Spice simulator
http://www.intusoft.com/

Microelectronics Laboratory Using Electronics Workbench (EWB)
IEEE Self-Study Course
Muhammad Rashid, University of West Florida
2000, ISBN 0-7803-4804-4
IEEE Online Catalog: customer-service@ieee.org

On line internet seminars on electronics
http://www.netseminar.com/
http://www.techonline.com/

Notes on reading and self-study
http://iteso.mx/~gabi/tl/mat/algunas_estrategias_que_facilitan_la_comprension_de_la_lectura.htm

Learning English at ITESO
Centro de Lenguas, Office N-15
cenlenguas@iteso.mx

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.