University of Southern California
Viterbi School of Engineering
Ming Hsieh Department of Electrical Engineering

EE 479 - Analog and Non-linear Integrated Circuit Design

Instructor: Ali Zadeh
Email: usc.ee479@yahoo.com
Lecture: Tuesday 6:30 – 9:10pm
Classroom: SSL 150
Term: Fall 2013
Office Hour: Tu. 6:10 – 6:30 & 9:10 – 10:00pm
Pre-requisite: EE 348L
Office location: Powel Hall 532

Pre-requisite:

- Linear-Time Invariant (LTI) systems, time domain (t-domain) vs. complex frequency domain (s-domain) analysis, RLC networks, Laplace transform, KCL, KVL, Diode, BJT, and MOS circuits, ac small-signal analysis of basic transistor circuits, basic feedback block diagram.
- Some material in *Frequency Domain vs. Time domain* is covered in the discussion section.

Possible Text Book:

- P. R. Gray, P. J. Hurst, S. H. Lewis, & R.G. Meyer, *Analysis and Design of Analog Integrated Circuits*, 5-th Edition, John Wiley & Sons, Inc., New York, 2009.

Reference Text Books:

- B. Razavi, *Design of Analog CMOS Integrated Circuits*, McGraw Hill, 2001.
- R. Jacob Baker, *CMOS Circuit Design, Layout, and Simulation*, 3rd Edition, IEEE Press, 2010.
- P. E. Allen & D. R. Holberg, *CMOS Analog Circuit Design*, 3rd Ed, Oxford University press, 2011.

Class Website:

- All lecture notes, assignments discussions, and announcements will be posted on DEN.
- Please check EE 479 Class Website on DEN once a day.

Course Goal:

- EE479 covers analysis and design of Analog and Non-Linear Integrated Circuits. The course consists of two sides of dealing with Analog Integrated circuits: (a) Analyzing a given Analog Integrated circuits through homework exercises and exams. (2) Designing Analog integrated circuits is accomplished by a major final project.

- The emphasis of the course material will be on CMOS analog circuits such as current sources, active load, bias current and voltages, differential pairs, frequency response, feedback amplifiers, output stages, noise behavior, Miller compensation, one-stage and two stage CMOS operational amplifier, folded-cascode, telescopic-cascode, operational transconductance amplifier (OTA), high-performance, low-voltage, and high-speed CMOS Opamp. We may cover band-gap circuits and voltage references. Finally, we will cover material on the Analog non-linear integrated circuits such as large signal analysis, distortion, and crystal oscillators.

Grade: Final course grade is based on the following formula:

Homework Assignments	=	25%
Midterm Exam	=	25%
Final Exam	=	25%
Class Project	=	25%
==========	=	========
Total	=	100%

Instructor's Background:

I am a part-time faculty in the Electrical Engineering department. I have worked in Semiconductor and Medical companies since 1982 and designed many Analog and Mixed-Signal Integrated circuits: Switched-Capacitor Filters, Analog-Digital Interface Integrated Circuits, Micro-power Biomedical Data-Converter Integrated Circuits and Systems, and CMOS Image sensors.

Exams

There will two Exams: a Midterm and a Final. The dates of exams will be announced in the class schedule. The Final Exam is NOT cumulative; it covers only the material after the Midterm exam.

Exams will be closed book. Students can bring one 8.5" x 11" page (both sides of the sheet) of notes for the Midterm Exam and two sheets of notes for Final Exam. Paper for exams will be supplied. Just bring a pencil, eraser, and a calculator. You must show how you have derived the answers fully in order to receive full credit.

Homework Assignments

There will be 6 homework assignments given during the semester. Homework assignments will be given through the class web-site and are due by the announcements. If you cannot make it to the lecture, have a friend to turn in your homework for you.

Homework Policies:

1. Your homework must be a professional quality work.
2. 10 points of your homework grade is based upon the following policies:
   - Use only standard 8.5 in x 11 in papers.
   - Use only one-side of each page.
   - Put staples on the upper-left corner of your homework.
   - Write nicely, large, neatly and legibly.
   - Put each schematic diagram or simulation results in a separate page.
   - Add a cover page with your name according to USC records: Last Name, First Name.
3. Each person does his/her homework individually.
4. Copying homework from each other is considered cheating.
5. Turn in your homework in the classroom at the beginning of the lecture.
6. Late homework and E-mail homework will NOT be accepted.

Homework Grading: Each homework problem has five points:

- 5 = Perfect Solution
- 4 = Minor Error
- 3 = Half-way
- 2 = Major Error
- 1 = Attempted
- 0 = No solution

LTspice Simulation:

- We will be using LTspice for simulations of our circuits, homework and project. In this class we will be using ***65nm CMOS process technology from IBM***.
- LTspice can be down-loaded free of charge from Linear Technology, Inc. Website. It has schematic capture, simulations engine, and waveform view. LTspice does NOT have any limitations on the number of components in your circuit schematic. I have tried ADC circuit with 10,000 transistors.
- We have a model files for ***65nm IBM CMOS technology***. This is Level 54 (BSIM4V3) parameters, IBM65nm.txt. This model file is from an actual processed wafer lot of IBM provided by MOSIS IC design Services.
- I will provide you with the model file on the DEN website. Copy the text file "IBM65nm.txt" into your PC and place it in your working folder (the folder in which you place your circuit schematics and symbols).
- You should become very familiar with the LTspice: Schematic Capture, Creating SPICE Net-list, Simulations, and Viewing the waveform by the first homework.

1. Download the free LTspice software from the following website:
   http://www.linear.com/designtools/software/
2. Watch five short videos LTspice Tutorials in this website:
   http://cmosedu.com/videos/ltspice/ltspice_videos.htm
3. There are several YouTube videos about LTspice:
   Go to YouTube website and search for "Circuit simulation in LTspice"
   http://www.youtube.com
4. To get all the information about LTspice.
   http://cmosedu.com/cmos1/ltspice/ltspice.htm
5. LTspice User Manual is in this website:
   http://ltspice.linear.com/software/scad3.pdf
6. LTspice Yahoo User Group: 10years and thousands of circuits and ask questions
   http://tech.groups.yahoo.com/group/LTspice

Class Project

One of the main purposes of the course is for student to have hands-on experience in designing a major CMOS Operational Amplifier integrated circuit using LTspice.

- Your project report must be professional quality work, done in the ***MS Power-Point***. Handwritten project will NOT be accepted.
- Use: Insert Object > Microsoft Equation 3.0, to make formula and equations in your report.
- Either one or two persons can work on the same project. Because of the amount of work, I do recommend two people.

The performance specification of the project operational amplifier is in Table 1. This opamp is an embedded operational amplifier and it is designed to be a part of a larger high-speed Mixed-Signal Integrated circuit or system.

The class project is graded based upon the following on:

Organization and Project Report:	10%
Analysis and Hand Calculations:	10%
Circuit Design Innovation:	10%
Circuits /Test Circuit Diagrams:	10%
Graphs and Waveforms:	10%
Circuit Performance:	50%
==========	======
Total	**100%**

Achieving all specifications simultaneously will be challenging. Several papers and patents will be provided at the end of the semester to give you some ideas how to push the performance of your circuit. First try to achieve the minimum possible performance. Then, improve the opamp performance if you have additional time.

EE479 Weekly Schedule, Summer Semester 2013

Week	First Half	Second Half	Readings
(1)	Introduction: • Syllabus • Analog vs. Digital	Frequency vs. Time Domain: • Linear-Time Invariant (LTI) system • First order & Second order networks	Chapter 1
(2)	PN Junction., MOS Device: • Ohmic (Triode) Region • Saturation (Active) Region	MOS Device in Saturation Region: • AC Small-Signal Equivalent • Transconductace, Output Resistance	Chapter 2
(3)	MOS Device 2nd order effects: • Channel Length Modulation (λ) • Body Effect (γ, χ)	MOS Device: • High-Frequency Model • Gate & Junction Capacitances	Chapter 3
(4)	Basic Amplifier Configurations: • Common Source (CS) • Common Gate (CG) • Source Follower (SF)	Multi-Transistor Amplifiers: • Cascade Configuration • Cascode Configuration • Gain enhancements	Chapter 3
(5)	Differential-Pair Amplifiers: • Differential Mode • Common Mode	Current Mirrors/Sources: • Cascode • Low-voltage Cascode	Chapter 4
(6)	Differential-Pair Amplifiers: • Active Load	References: • Current & Voltage References	Chapter 4
(7)	References: • Supply Independent Bias Current • PTAT Current Source	Opamp Specifications: • Single-Ended output • CMRR, PSRR+, PSRR-, ICMR	Chapter 6
(8)	Opamp Topologies: • Telescopic, Folded-Cascode, OTA	Opamp Topologies: • Telescopic, Folded-Cascode, OTA	Chapter 6
(9)	**Midterm Exam:** **Chapters: 1, 2, 3, 4**	**Midterm Exam: 7:00 pm – 9:00 pm** **Tuesday,**	----------
(10)	Amplifier Frequency Response: • CS, CG, SF Configurations	Amplifier Frequency Response: • CS, CG, SF Configurations	Chapter 7
(11)	Analysis of Feedback System • Return Ratio Analysis • Loop-Gain	Amplifier feedback: • First Order Model. • Gain-Bandwidth Trade-off	Chapter 8
(12)	Amplifier feedback: • Second Order Model • Frequency vs. Transient Response	Amplifier feedback: • Second Order Model • Frequency vs. Transient Response	Chapter 9
(13)	Two-Stage Operational Amplifier: • Miller Compensation	Two-Stage Operational Amplifier: • Frequency vs. Transient Response	Chapter 9
(14)	Non-Linear Analog Circuits: • MOS Distortion Analysis	Non-Linear Analog Circuits: • MOS Crystal's oscillator.	Notes
(15)	Final Exam Review Final Exam: Chapters: 6, 7, 8, 9	Project Review, Class Evaluation • Design of Two-Stage CMOS Opamp	Notes
(16)	----------	**Final Project: 7:00pm – 8:00pm** **Thursday,**	----------
(17)	**Final Exam:** **Chapters: 6, 7, 8, 9**	**Final Exam: 7:00pm – 9:00pm** **Tuesday,**	---------