--- University of Minnesota ---
Institute of Technology
Department of Electrical Engineering

EE5505: Analog Integrated Circuit Design — Fall 1998

Credits:	3	*Instructor:*	Ramesh Harjani
Time:	08:00 - 08:50 MWF	*Location:*	MechE 108
Office Hours:	09:00 - 10:00 MWF	*Location:*	4-165 EE/CSci, 625-4032
Call #:	968315		
Email:	harjani@ee.umn.edu		
Web:	**http://www-mount.ee.umn.edu/~harjani/courses/5505.html/**		

Course Description: Advanced techniques for the design of analog integrated circuits. Emphasis will be placed on the design of the fundamental circuits required for analog signal processing. Design issues associated with both MOS and BJT devices will be explored though the primary focus of this course will remain as CMOS design. Students will be expected to design and test several design problems. Topics will be selected from the following: modelling of basic IC components, operational amplifier design and comparator design. The objective of this course it to provide the basic design concepts and tradeoffs involved in analog integrated circuit design.

Course outline:		*Lecture Time*
i)	Integrated circuits devices and modeling	1 1/2 week
ii)	Introduction to processing & layout	1/2 week
iii)	Basic current mirrors & single stage amplifiers	2 week
iv)	Noise modeling	1/2 week
v)	Basic opamp design and compensation	2 week
v)	Advanced current mirrors and opamps	2 week
vii)	Misc topics	1/2 week
vii)	Revision & project reviews	1 week

Text: (Available in the bookstore)
Analog Integrated Circuit Design
David Johns & Ken Martin
John Wiley & Sons, Inc. 1997

References: CMOS Circuit Design, Layout, and Simulation
R. Jacob Baker, Harry W. Li and David E. Boyce
IEEE Press, 1998

Homework Assignments:
Some of the homework assignments will require circuit simulation on the computer. The rest of the homeworks will require hand calculations and drawing on graph paper. However, some of the hand calculations can get quite complex and so it is recommended that you get hold of some symbolic analysis program on a computer. Examples of such symbolic analysis programs include: Mathematica, Milo, MathCAD, REDUCE, MAPLE, Theorist, etc. I personally use MathCAD (available at the bookstore) the most. The purpose of the homework is to build upon your understanding of course concepts and to develop analog circuit design skills.

Projects:
All along the quarter students are expected to work on a project of significant size. You are expected to work on a single project. As these projects make up a significant proportion of your final grade, students are advised to start thinking of topics for their final project immediately. All students have to design an opamp. You will have to complete circuit design and layout, i.e., everything that is necessary to get your design fabricated at an IC foundry.

Computer use:
Students are expected to use HSPICE, or an equivalent circuit simulator, for design problems. On-campus students can access HSPICE on the SUNs and X workstations in the IT workstation lab (EE/CS 4-204). Other versions of SPICE are also acceptable (PSPICE, MSPICE, ACCUSIM, SPICE2G6). Students are expected to either already know SPICE or are expected to learn SPICE on their own. It will not be explicitly taught in class. So if you do not know SPICE start learning it at the earliest. As part of the final project students will also be required to use a layout program (Magic or equivalent). As mentioned earlier it is recommended that you learn a symbolic analysis program. Mathematica on the MACs on the IT workstation lab will be reserved for this class though other versions of symbolic analysis programs are acceptable.

Grading Policy:

Homework assignments	15%
Midterm I examination	25%
Midterm II examination	25%
Project	30%
Project Proposal	5%

Exam Timings:

Midterm I examination	Friday Oct 23
Midterm II examination	TBA