Syllabus Fall 2019

EE60542 Analog Integrated Circuit Design

Instructor: Alan Seabaugh, Fitzpatrick Hall 230A, seabaugh.1@nd.edu

Time/place: MW 12:30-1:45 am, 242 DeBartolo

Netsite: sakai.nd.edu

Prerequisite: Graduate standing in the College of Engineering or Science, or by permission

Textbook: Design of Analog CMOS Integrated Circuits, 2nd edition, Behzad Razavi 2017 McGraw Hill

Description: The intent of this course is to set a firm understanding of circuit design and analysis to students across all disciplines of science and engineering. Students will learn to design and analyze transistor circuits in the context of breakthroughs in technology over the last century. Foundational circuits will be introduced, including voltage/current sources, mirrors, single-ended and differential amplifiers, operational amplifiers, oscillators, multipliers, etc. Clever circuits continue to be invented. Concepts of positive and negative feedback, frequency compensation, stability, and noise will be covered and related to current research and off-the-shelf hardware. The program SPICE (Simulation program with integrated circuit emphasis) will be used to get beyond hand analysis.

Class preparation: Readings for each class will be assigned. Students will provide questions for the lecture (due 9 pm the night before class). Classes will be a mixture of lecture and discussion to answer questions and reinforce concepts.

Homework: Homework will be due weekly on Wednesdays (5 pm). Open discussion of homework between students is permitted.

Design project: A design project will be completed in the second half of the course. Students can propose the circuit to work on.

Grading: Homework (15%), class preparation/quizzes (10%), midterm exam (25%), design project (20%), final exam (30%)

Office hours: By appointment. To schedule contact Heidi Deethardt, hdeethar@nd.edu, 631-0279, or stop by Cushing Hall, room 206.

For more reading:
- Carusone, Johns, Martin, Analog Integrated Circuit Design, John Wiley (2012)
- Horowitz & Hill, The Art of Electronics, Cambridge Univ. Press (1999)
- Gray et al. Analysis & Design of Analog Integrated Circuits, John Wiley (2009)
- Razavi, Fundamentals of Microelectronics, John Wiley (2014).

SPICE: Linear Technology **(**LT)-SPICE will be used for circuit simulation; download at https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html