

An analog integrated circuit design laboratory

Citation for published version (APA):

Mondragon-Torres, A. F., Mayhugh, J., Pineda de Gyvez, J., Silva-Martinez, J., & Sanchez-Sinencio, E. (2003). An analog integrated circuit design laboratory. In *Proceedings of the 2003 IEEE International Conference on Microelectronic Systems Education, 2003, 1-2 June 2003, Anaheim, California* (pp. 91-92). Institute of Electrical and Electronics Engineers. <https://doi.org/10.1109/MSE.2003.1205269>

DOI:

[10.1109/MSE.2003.1205269](https://doi.org/10.1109/MSE.2003.1205269)

Document status and date:

Published: 01/01/2003

Document Version:

Publisher's PDF, also known as Version of Record (includes final page, issue and volume numbers)

Please check the document version of this publication:

- A submitted manuscript is the version of the article upon submission and before peer-review. There can be important differences between the submitted version and the official published version of record. People interested in the research are advised to contact the author for the final version of the publication, or visit the DOI to the publisher's website.
- The final author version and the galley proof are versions of the publication after peer review.
- The final published version features the final layout of the paper including the volume, issue and page numbers.

[Link to publication](#)

General rights

Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- You may not further distribute the material or use it for any profit-making activity or commercial gain
- You may freely distribute the URL identifying the publication in the public portal.

If the publication is distributed under the terms of Article 25fa of the Dutch Copyright Act, indicated by the "Taverne" license above, please follow below link for the End User Agreement:

www.tue.nl/taverne

Take down policy

If you believe that this document breaches copyright please contact us at:

openaccess@tue.nl

providing details and we will investigate your claim.

AN ANALOG INTEGRATED CIRCUIT DESIGN LABORATORY

Antonio F. Mondragón-Torres, Terry Mayhugh Jr., José Pineda de Gyvez,
José Silva-Martinez and Edgar Sánchez-Sinencio

Department of Electrical Engineering, Texas A&M University, College Station, Texas, 77840
antoniofmondragon@ieee.org

ABSTRACT

We present the structure of an analog integrated circuit design laboratory to instruct at both, senior undergraduate and entry graduate levels. The teaching material includes: a laboratory manual with analog circuit design theory, pre-laboratory exercises and circuit design specifications; a reference web page with step by step instructions and examples; the use of mathematical tools for automation and analysis; and state of the art CAD design tools in use by industry. Upon completion of the course, the students have skills for an entry level analog designer position.

1. INTRODUCTION

There is a demand to prepare students in analog microelectronics, and to be familiar with industry grade IC design tools. The EE Department at Texas A&M University has developed a course in analog circuit design that covers the basic knowledge required for an entry level analog designer position.

This paper presents the organization of the laboratory section of the Analog VLSI Circuit Design (ELEN-474) class. The laboratory is a key part of the course that fills the gap between theory and practice and its objectives are:

- a) To understand the theory by designing circuits as required in the pre-laboratory exercises.
- b) To simulate and compare the design against the hand calculations.
- c) To produce a layout and compare it against the schematic.
- d) To run a parasitic extracted simulation in order to verify performance matches against specifications.
- e) To run statistical analysis in order to predict the behavior of the circuit under parameter variations.

The tools used in this laboratory are those available with the Cadence Design Systems: schematic capture (CAP), circuit simulation (SIM), circuit layout (LAY), design rule check (DRC), layout versus schematic check (LVS), circuit extraction (EXT), parasitic simulation (PSM) and statistical simulation (SSM). In addition, some mathematical tools are used as an aid to automate the design process, and also as a graphical user interface (GUI) for circuit analysis.

The innovation in the implementation of this lab consists of the creation, adaptation and integration of the different teaching materials available such as: Laboratory manual [1]; web page [2] with step by step instructions, examples and up to date information; use of industry grade design tools; and use of non-circuit related programs for automation and analysis.

2. COURSE DESCRIPTION

The laboratory is based on a bottom-up design methodology. Each session is based on simple configurations, characterized by circuit design, layout, simulation and measurement. By the end of the course, the student has the ability to integrate these building blocks to form complex systems such as continuous and discrete time filters. The laboratory meets once a week and is divided in two sessions: One for simulation and one for physical measurement (MEA).

2.1. Simulation

At the beginning of the simulation session the student turns in the pre-laboratory exercises. The design is captured into Composer using the hand calculated values. The simulations are performed using Analog Artist and Spectre. After the specifications are met, the circuit layout is designed using Virtuoso. The layout procedure is:

- a) Generate a floor plan by identifying which transistors should be matched and which would require special attention for proper layout.
- b) Execute the DRC to verify that the design complies with the design rules.
- c) Extract the layout to identify the devices generated and also the non-desired parasitic devices.
- d) Run LVS tool to verify the correspondence between the layout and the schematic.
- e) Generate an analog extracted cellview to perform parasitic extracted simulations.

2.2. Measurement

A set of ICs with equivalent circuits to those designed and simulated, are available for physical measurement. As part of their pre-laboratory exercises, the student proposes an experimental testbench setup to characterize and measure the circuit. After the students obtain their experimental results, it is verified that the results are coherent with simulations.

3. LABORATORY ORGANIZATION

1. Introduction: Review of the Cadence design system and of the UNIX. Basics of CAP and *dc* SIM.
2. Layout Design: LAY techniques: unit size transistors; common centroid and interdigitized structures; and dummy transistors.
3. MOS Device Characterization: KP, VT, λ and γ are extracted through SIM and MEA. The parametric (PAR) analysis is introduced.
4. Current Mirrors: Simple, cascode, feedback and low-voltage. Design constraints: Z_{in} , Z_{out} , compliance, accuracy and complexity. Analyses: *dc*, *ac* and PAR. Run LVS. *ac* analysis is introduced.
5. Inverting Amplifiers: Current Mirror Load, Digital CMOS, PMOS with self biased load and self biased CMOS. Design constraints: GBW, PM and A_{V0} . The effects of power supply, source resistance and load capacitance are investigated. Analyses: *dc* and *ac*. PSM is introduced.
6. Differential Amplifiers: Simple and cascode current mirrors. Design constraints: A_{DM} , A_{CM} , CMRR, CMR and SR. Analyses: *dc*, *ac* and transient (TRAN). TRAN analysis is introduced.
7. Operational Transconductance Amplifiers (OTA): Symmetrical OTA. Design constraints: G_M and SR. Analyses: *dc*, *ac* and TRAN. The static power (PW) dissipation is measured and reported.
8. Operational amplifiers (OP): Three stage OP. Design constraints: A_{DM} , A_{CM} , CMRR, PSRR, GBW, PM, Z_{out} , SR and PW. Monte-Carlo analysis is introduced, the *ac* response is used to explore the effect of random variations on parameters.
9. Analog System: Continuous low pass filter. Macromodelling is introduced.
10. Switched Capacitor Integrators: Use of transmission gates, capacitors and OP.

The grading policy applies 50% to the laboratory reports, 40% to the pre-laboratory exercises and 10% to random assigned five minutes quizzes.

4. TEACHING MATERIAL

4.1. Laboratory Manual

The laboratory manual is the foundation of this laboratory, it contains the guidelines and the required theory to perform each design. The structure of each laboratory session is as follows: Objectives, Introduction, Design Description, Simulation and Layout procedure, Test Procedure, Pre-Laboratory, Laboratory week 1, Laboratory week 2 and References.

4.2. Web Page

A WWW site has been created to complement the laboratory lectures [2]. This resource has proved to be a very useful tool. During the laboratory session, the student can follow the explanation given by the TA by

browsing on his/her own workstation as the TA explains the different steps required to complete the design. As the students work through their design, they can use the web site to verify that the procedure is correct.

4.3. Mathematical Tools

Two mathematical programming tools have been used to help automate the design procedure and to aid in the analysis of the design equations. These tools are Matlab and Maple. A very useful application of the mathematical tools is for the automatic generation of parameter files; e.g. the design equations are entered into Matlab, the output would be a text file which can be included into a Spice text simulation file. This procedure allows to run the mathematical tools and reflect the new design specifications in the simulation.

4.4. CAD Design Tools

Texas A&M University is part of the Cadence Design Systems North American University Software Program. The basic Cadence components used in this course are the following:

- a) Composer (schematic capture).
- b) Virtuoso (layout generation).
- c) Analog Artist (simulation front-end).
- d) Spectre HDL (circuit simulation).
- e) Diva tools (DRC, LVS, ERC, extraction).
- f) Hierarchy Editor (parasitic simulation)
- g) Artist statistics (Monte Carlo Simulation and statistical analysis tools)

Cadence Design Systems tools have become the de facto industry standard. Texas A&M University has formulated the course according to the industry requirements.

5. CONCLUSIONS

A laboratory structure for an analog integrated circuit design course has been presented. The different tools available, allow the students to concentrate more on the design issues rather than on software problems. The laboratory manual sets the guidelines, complements the theory, develops a design procedure and sets the specifications for each design. By using the web page, the student can go through the design steps and verify that the results are correct. The use of mathematical analysis tools helps the student to automate some tasks that are otherwise repetitive. Moreover, the use of a graphical interface allows the student to take design decisions. The best layouts are fabricated using the Mosis service. The fabricated circuits are to be measured by the students of the forthcoming semester.

REFERENCES

- [1] T. Mayhugh Jr and A. Mondragon-Torres, "Laboratory Manual for ELEN 474: VLSI Circuit Design", Dept. of EE, Texas A&M University, Version 4.0, Sept. 1998.
- [2] <http://amesp02.tamu.edu/~afmondra/elen474/>