Simulation Methods for Electronic Circuits
(Graduate Course)

Jan-May 2020
19-22 hours, Wednesdays
Classroom: W-202

Instructor Information

José Ernesto Rayas Sánchez, Ph.D.
Office: T-369
Office hours: 10-14, 17:00-21:00 hrs
Tel: 3669-3598, Ext. 3096
Email: erayas@iteso.mx
Website: https://desi.iteso.mx/erayas/

General Description

This course will enable students to efficiently simulate electronic devices and circuits using commercially available CAD tools. Students will gain basic skills on fundamental CAD methodologies, as well as on some state-of-the-art CAD techniques. Statistical analysis and yield estimations for electronic circuits under manufacturing uncertainties will be considered. Although circuit-based low-frequency simulation will be emphasized, an introduction to high frequency distributed-circuit simulation and full-wave electromagnetic simulation will also be realized. The use of simulation software for hands-on experience will be emphasized throughout the course.

Prerequisites

No previous graduate course is required. It is expected that the students taking this course have a good understanding of basic electronics and circuit analysis. Experience with SPICE simulation would be very useful. The students should already have some familiarity with Matlab or a similar numerical tool.

Objectives

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

A. Identify the main milestones in CAD history, explaining its up-to-date evolution (APPLICATION)
B. Analyze linear circuits using the Tableau formulation (ANALYSIS)
C. Analyze linear and nonlinear circuits using the Modified Nodal Analysis (MNA) method (ANALYSIS)
D. Perform transient-domain analysis of linear circuits using an MNA formulation with stamps (ANALYSIS)
E. Analyze nonlinear circuits using the Nodal method and the MNA method (ANALYSIS)
F. Identify the main advantages of the Harmonic Balanced technique for the steady-state analysis of nonlinear circuits (COMPREHENSION)
G. Identify the main advantages of the Adjoint Network technique for the efficient calculation of sensitivities (COMPREHENSION)

H. Efficiently simulate electronic circuits using SPICE (APPLICATION)
I. Perform statistical analysis of circuits, Monte Carlo simulations, and yield estimations using conventional simulators and mathematical software (ANALYSIS)
J. Simulate high-frequency electronic circuits using distributed circuit models (APPLICATION)
K. Simulate high-frequency electronic circuits using full-wave electromagnetics-based models (APPLICATION)

General Contents

1. A historical perspective of CAD and some future trends
2. Systematic analysis of linear and non-linear circuits
   - 2.1. Oriented graphs, incidence matrix, and topology equations
   - 2.2. The Tableau method
   - 2.3. The Nodal formulation
   - 2.4. The MNA method for DC and frequency-domain analysis
   - 2.5. The MNA for nonlinear circuits
   - 2.6. Time-domain transient analysis of linear circuits
   - 2.7. An introduction to the Harmonic Balance method
   - 2.8. An introduction to the Adjoint Network technique
3. Circuit simulation with SPICE
   - 3.1. Structure of the SPICE language
   - 3.2. SPICE basic functions
   - 3.3. SPICE models for basic components
   - 3.4. SPICE advanced functions
   - 3.5. Modeling sensors and other physical systems with SPICE
   - 3.6. Driving WinSpice from Matlab
4. Statistical analysis and yield calculations
   - 4.1. Sources of design-performance uncertainty
   - 4.2. Design and development processes and design for manufacturability
   - 4.3. A general formulation to statistical analysis
   - 4.4. Tolerance and acceptability regions
   - 4.5. Yield definitions
   - 4.6. Probability distributions and tolerances
   - 4.7. Nominal optimization *vs* yield optimization
   - 4.8. Monte Carlo implementation and yield prediction
   - 4.9. Sensitivities, worst-case estimations, and first-order statistical analysis
5. High-frequency circuit simulation
   - 5.1. Basic concepts on high-frequency circuits
   - 5.2. Empirical rules for using transmission line theory
   - 5.3. An introduction to APLAC's circuit simulator (and/or AWR, ADS, or Qucs)
   - 5.4. Driving APLAC or ADS from Matlab
6. Full-wave EM simulation
   - 6.1. An overview on full-wave analysis
   - 6.2. EM simulation using Sonnet (and/or Momentum, EM-Pro, or COMSOL).

Periférico Sur 8585
45604 Tlaquepaque, Jal., México
Tel +52 33 3669 3598
www.iteso.mx

Relationship between Contents and Objectives

	1	2	3	4	5	6
A	✓					
B		✓				
C		✓				
D		✓				
E		✓				
F		✓				
G		✓				
H			✓			
I			✓	✓	✓	
J					✓	
K						✓

Course Skeleton

For the proposed course skeleton it is assumed: a group of 3 to 20 students; 3 hours per week of class meetings during 16 weeks; simulation software available at ITESO and/or at home.

It is also expected that the student will be able to dedicate an average of 8 hours of work per week to this course, including attending classes.

Week	Activity	Week	Activity
1	Contents 1, 2	9	Assignment 3, Contents 4
2	Assignment 1, Contents 2	10	Contents 4, 5
3	Contents 2	11	Contents 5
4	Contents 2	12	Contents 5
5	Contents 2	13	Assignment 4, Contents 6
6	Assignment 2, Contents 3	14	Contents 6
7	Contents 3	15	Assignment 5
8	Contents 3	16	Final project

Assessment

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

Assignments	65% (1:5%, 2:15%, 3:15%, 4:15%, 5:15%)
Project	30%
Participation	5%

Each student will realize a final project during the course. The topic chosen must be approved by the instructor. Students will make a technical presentation on their final project selected. The report and presentation of the final project must follow some guidelines that will be indicated in the course. Depending on the selected topic, the project and corresponding presentation can be done individually or in teams of up to 2 students. The report must be hand-in on the day of the presentation. The presentation will be evaluated not only by the instructor but also by the classmates. Further instructions about the technical presentations will be delivered later.

Periférico Sur 8585
45604 Tlaquepaque, Jal., México
Tel +52 33 3669 3598
www.iteso.mx

Quality of students' participation during the lectures will be graded. This participation will be evaluated based on student's attitude and performance during class: punctuality, willingness to ask relevant questions, respect to others, attention during class, ability to answer questions, etc.

Teaching Methods

This course will use a variety of teaching methods including: lecturing, seminars, computer simulations, assignments, readings, project presentation and report writing, and self-conducted research work.

Important information related to the course will be posted in the instructor's web site throughout the semester. Some communication will be maintained through e-mail. Open and frequent communication with the instructor is encouraged. Collaboration between the students is also encouraged.

The course will be conducted in Spanish. Most of the written material for the course will be available in English. The assignments and final project can be submitted in either English or Spanish, excepting the PhD students who must present their final project in English (using the internal research reports' template for the Doctoral Program in Engineering Sciences at ITESO).

References

Instructor's course website
https://desi.iteso.mx/erayas/cad.htm

CAECAS Research Group
https://desi.iteso.mx/caecas/

Books on systematic circuit analysis:

Computer methods for circuit analysis and design
Jirí Vlach and Kishore Singhal.
New York, NY: Van Nostrand Reinhold, 1983.

Electronic circuit and systems simulation methods
L. T. Pillage, R.A. Rohrer, and C. Visweswariah
McGraw Hill, 1995

Computer-aided design of microelectronic circuits and systems: fundamentals, methods and tools
A. F. Schwarz
Academic Press, 1987

Mixed - Mode Simulation
R. A. Saleh, A. R. Newton
Kluwer Academic Publishers, 1990

Books on SPICE simulation:

Introduction to PSpice using OrCAD for circuits and electronics,
Muhammad H. Rashid

Pearson Prentice Hall, 2004

SPICE for Power Electronics and Electric Power
Muhammad H. Rashid
CRC, 2006

Semiconductor device modeling with SPICE
Paolo Antognetti, editor, Giuseppe Massobrio, coeditor.
Imprint New York : McGraw-Hill, c1988.

SPICE: A guide to circuit simulation and analysis using Pspice
Paul W. Tuinenga
Prentice Hall, 1992

Books on statistical analysis:

Yield and reliability in microwave circuit and system design
Mihael D. Meehan and John Purviance
Artech House, 1993

Statistical modeling for computer-aided design of MOS VLSI circuits
Christopher Michael and Moahmmer Ismail
Kluwer Academic Publishers, 1993

Books on RF and Microwave Simulation:

Computer-aided analysis of nonlinear microwave circuits
Paulo J. C. Rodrigues
Artech, 1998

Introduction to computer methods for microwave circuit analysis and design.
J. A. Dobrowolski.
Artech, 1991

Steady-State Methods for Simulating Analog and Microwave Circuits
Kenneth S. Kundert, Jacob K. White, and Alberto L. Sangiovanni-Vincentelli
Kluwer Academic, 1990

Microwave Circuit Modeling Using Electromagnetic Field Simulation
Daniel G. Swanson and Wolfgang J. R. Hoefer
Artech, 2003

Introduction to RF Design Using EM Simulators
Hiroaki Kogure, Yoshie Kogure, and James Rautio
Artech House, 2010

Software Tools

Introduction to Matlab, Yossi Farjoun, MIT
https://ocw.mit.edu/resources/res-18-002-introduction-to-matlab-spring-2008/

Matlab
https://www.mathworks.com/

Octave
http://www.gnu.org/software/octave/

SciLab
https://www.scilab.org/

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

Cadence (Pspice Orcad)
https://www.orcad.com/resources/orcad-downloads

NI AWR Design Environment
https://www.awr.com/software/products

APLAC (NI AWR) high-frequency circuit simulator
https://www.awr.com/resource-library/microwave-office-aplac-datasheet

Qucs, high frequency circuit simulator
http://qucs.sourceforge.net/index.html

Keysight ADS and EM-Pro
https://www.keysight.com/en/pc-1297113/advanced-design-system-ads?nid=-34346.0&cc=MX&lc=eng

Sonnet, EM simulator
http://www.sonnetsoftware.com/

COMSOL Multiphysics
https://www.comsol.com/

Simulia CST Studio
https://www.3ds.com/products-services/simulia/products/cst-studio-suite/

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.

Periférico Sur 8585
45604 Tlaquepaque, Jal., México
Tel +52 33 3669 3598
www.iteso.mx