Course guide
230301 - PSPICE - Circuit Simulation and Analysis Using PSpice

Last modified: 24/05/2024

Unit in charge: Barcelona School of Telecommunications Engineering
Teaching unit: 710 - EEL - Department of Electronic Engineering.

Degree: BACHELOR'S DEGREE IN TELECOMMUNICATIONS TECHNOLOGIES AND SERVICES ENGINEERING (Syllabus 2015). (Optional subject).
BACHELOR'S DEGREE IN ELECTRONIC ENGINEERING AND TELECOMMUNICATION (Syllabus 2018). (Optional subject).

Academic year: 2024 **ECTS Credits:** 2.0 **Languages:** Catalan, Spanish

LECTURER

Coordinating lecturer: ANTONIO TURO PEROY

Others: Primer quadrimestre:
JUAN ANTONIO CHAVEZ DOMINGUEZ - 10
SANTIAGO SILVESTRE BERGES - 10
ANTONIO TURO PEROY - 10

PRIOR SKILLS

Basic electronic devices and circuits.
Circuit Analysis.

TEACHING METHODOLOGY

Laboratory classes.
Autonomous individual work.

LEARNING OBJECTIVES OF THE SUBJECT

To introduce the student in a guided way in the use of an electronic circuit simulator.
To deepen the knowledge of circuits and electronic devices.
To strengthen skills in the analysis of electronic circuits and linear systems.

STUDY LOAD

Type	Hours	Percentage
Self study	30,0	60.00
Hours small group	20,0	40.00

Total learning time: 50 h

CONTENTS

Session 1. Introduction

Description:
Introduction to the electrical simulators and the PSPICE of the Cadence Orcad.

Full-or-part-time: 3h
Laboratory classes: 2h
Self study : 1h

Session 2. Getting started with PSPICE of Cadence OrCad.

Description:
The goal of this session is to introduce students to get used in the use of this simulation software. This sesssion is organized as a tutorial to achieve it.

Full-or-part-time: 7h
Laboratory classes: 3h
Self study : 4h

Session 3. DC and TRANSIENT simulations.

Description:
In this session students will deepen in the use of the PSPICE simulator. Special attention will be paid to DC and TRANSIENT type analysis and some of the capabilities of the PROBE visualization tool.

Full-or-part-time: 7h
Laboratory classes: 3h
Self study : 4h

Session 4. AC simulation.

Description:
In this session, students will deepen into the use of the PSPICE electrical simulator in order to perform AC circuit analysis. The simulation of the circuits in the frequency domain will be learned.

Full-or-part-time: 7h
Laboratory classes: 3h
Self study : 4h

Sessions 5 & 6. Simulation by means of netlists and subcircuit and symbol creation.

Description:
In these sessions, students will deepen into the use of the PSPICE simulator by learning their capabilities in order to describe the circuits by means of schematic and/or netlist. Also, the definition of specific subcircuits, parts and symbols to use in our simulations will be introduced.

Full-or-part-time: 12h
Laboratory classes: 5h
Self study : 7h

Session 7. Applications.

Description:
In this session, the knowledge acquired throughout the seminar will be used in the simulation of an electronic system project that may be of interest.

Full-or-part-time: 14h
Laboratory classes: 8h
Self study : 6h

GRADING SYSTEM

Grading is based on the documents that must be delivered after each session with the results of the simulations and the severe justifications.
Final grade is calculated from the partial grades of each session as follows:
NF=(S2+S3+S4+S5+S6+2*S7)/7

BIBLIOGRAPHY

Complementary:
- Goody, Roy W. OrCAD PSpice para Windows, Volumen I. 3a ed. Madrid: Prentice Hall, 2003. ISBN 8420536492.
- Goody, Roy W. OrCAD PSpice para Windows, Volumen II. 3a ed. Madrid: Prentice Hall, 2003. ISBN 8420537047.
- Goody, Roy W. OrCAD PSpice para Windows, Volumen III. Madrid: Prentice Hall, 2003. ISBN 8420541729.

RESOURCES

Other resources:
PSpice® User's Guide:
https://resources.pcb.cadence.com/i/1180526-pspice-user-guide/