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

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: 2022 ECTS Credits: 2.0 Languages: Catalan, Spanish

LECTURER
Coordinating lecturer: Consultar aquí / See here:
https://telecos.upc.edu/ca/estudis/curs-actual/professorat-responsables-coordinadors/responsables-assignatura
Others: Consultar aquí / See here:
https://telecos.upc.edu/ca/estudis/curs-actual/professorat-responsables-coordinadors/professorat-assignat-idioma

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

<table>
<thead>
<tr>
<th>Type</th>
<th>Hours</th>
<th>Percentage</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hours small group</td>
<td>20,0</td>
<td>40.00</td>
</tr>
<tr>
<td>Self study</td>
<td>30,0</td>
<td>60.00</td>
</tr>
</tbody>
</table>

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 session 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:
\[ NF = \frac{(S2+S3+S4+S5+S6+2*S7)}{7} \]

BIBLIOGRAPHY

Complementary:

RESOURCES

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