Centre	**University College of Engineering of Vitoria-Gasteiz**
Name of subject	**26004 – Electronic Design and Simulation**
Qualification	**Degree in Industrial Electronic Engineering and Automatics**
Type	**Elective**
Credits	**6 ECTS**
Year	**4**
Term(s)	**2nd**
Department	**Electronic Technology**
Language	**Spanish**

Outcomes / Objectives

The intended outcome is for students to acquire a sound knowledge of the Spice simulation tool that will allow them to simulate any analogue or digital electronic circuit. The students will handle the tool using text files - CIR files – as well as from the schematic.

Syllabus

Electronic simulation. Introduction to analogue electronic simulation. Description of CIR files, structure, devices, independent and dependent sources. Characteristics of the analysis types supported by the simulation. Analogue simulation. Output files. Types of analysis, static, temporal, frequency, sensitivity, parameter, temperature, Fourier, Monte Carlo, sub-circuits. Simulation from text files and graphical presentation of results. Graphical presentation of results using Probe. Digital simulation with CIR files. Analogue simulation from the schematic. Work environment from the schematic. Analogue schemes. Work environment, component libraries. Connections, labels. Feeds and stimuli. Simulation from the schematic. Sub-circuit creation. Examples. Digital simulation from the schematic. Digital schemes. Libraries. Connections, buses. Digital stimuli, by attributes, editor. Digital simulations. Examples

Methodology

Teaching Method

Face-to-Face Teaching Hours

Lectures	Seminars	Classroom practice	Lab. practice	Computer sessions	Clinical practice	Workshops	Industrial workshops	Field practice
30			30					

Student Hours of Non Face-To-Face Activities

Lectures	Seminars	Classroom practice	Lab. practice	Computer sessions	Clinical practice	Workshops	Industrial workshops	Field practice
45			45					

Assessment System

General criteria

Clarification regarding assessment

Bibliography

Basic Bibliography

- Simulación Electrónica, J.A. Sainz, A.Mª Aledo 2002- Manuales DesignLab de MicroSim y Orcad-Documentación ON line

In-depth Bibliography

- Pspice. Simulación y análisis de circuitos analógicos asistidos por ordenador, García, Ibañez, Gil. Paraninfo, 1995
- Spice. A guide to circuit simulation and analysis using spice. Tuinenga. Prentice Hall, 1988