2309 EIE440 Course Outline

Subject Code   : EIE440
Subject Title   : CAD FOR CIRCUITS
Course Type   : Elective
Level   : 3, 4
Credits   : 3
Teaching Activity   : Lecture   9 hours
           Lab   36 hours
Prior Knowledge*   : **Prerequisite:** Digital Circuits (EIE130)
           Co-requisite: Analog Circuits (EIE340)

Class Schedule

Class	Week	Time	Classroom	Date
D1	Wed	12:30-15:20	C402	4/9/2023-17/12/2023

Instructor   Dr. KinTak U
Contact Number   (853)8897-2249
E-mail Address   ktu@must.edu.mo
Office   C403
Office Hour   Tue. 13:30-15:30; Wed. 15:20-18:20
           Thur. 15:20-18:20; Fri. 13:30-15:30

Course Description

This subject aims to make the students familiar with modern methods and programs for computer-aided design of electronic circuits. Knowledge of circuit simulations allows the students to properly analyze the circuit, formulate the design task and to solve some possible problems when designing the circuits. Practical exercises will cover all the basic simulation methods in PSpice so that students can understand all the simulation procedures on how to analyze a circuit. Experiments should be finished by students to verify their practical abilities on analysis and designing some basic electronic circuits.

TEXT BOOK

Required Text Book:
No recommended textbook, but the learning materials will be provided to students during the classes.

Reference Book:
Donald Neamen. Microelectronic Circuit Analysis and Design. 4/E. 2006. McGraw-Hill.
ISBN: 9780071289474

INTENDED LEARNING OUTCOMES

Upon successful completion of this subject, students will be able to:
1. Construct the electronic circuit in Orcad/Capture correctly
2. Determine the simulation types and parameters carrying on the different circuits and perform the simulation analysis of the circuit in Orcad/PSpice.
3. Verify and analyze the circuit properties based on the simulation results.
4. Design the circuit to meet some given design requirement based on simulation

results.
5. Write the technical reports based on the experimental results.

Weekly Schedule

Week	Topic	Hours	Teaching Method
1	Introduction to OrCAD/Capture and OrCAD /Pspice, Principles of CAD, circuit drawing with Capture, circuit characteristic analysis with PSpice.	3	Lecture
Learning Capture and PSpice A/D by exercises:			Lab
2	1) Bias Point Detail Analysis 2) DC Sensitivity Analysis 3) DC Transfer Function Characteristic Analysis 4) Frequency Response Analysis of AC small signal (AC Sweep)	3	
3	5) Noise Analysis 6) DC Sweep Analysis 7) Transient Analysis 8) Fourier Analysis	3	
4	9) Temperature Analysis 10) Parametric Analysis 11) Performance Analysis 12) Worst-case Analysis	3	
5	Extra Exercise 1	3	
6	Extra Exercise 2	3	
7	Mid-term Review	3	Lecture
8	Mid-term Exam.	3	Lab
Experiments:			Lab
9	1. Common-emitter transistor amplifier	3	
10	2. The influence of Q-point on dynamic range	3	
11	3. The low frequency response of the amplifier	3	
12	4. Simulation and function analysis of the high frequency resonant amplifier	3	
13	5. Sine-wave generator and its Monte Carlo analysis	3	
14	Final Review	3	Lecture
15	Final Exam.	3	Lab

ASSESSMENT APPROACH

Assessment method	% weight
1.Exercise	10%
2.Experiment	20%
3.Midterm Exam.	20%
4.Final Exam	50%
Total	100 %

Guideline for Letter Grade:

Marks	Grade
93 - 100	A+
88 - 92	A
83 - 87	A-
78 - 82	B+
72 - 77	B
68 - 71	B-
63 - 67	C+
58 - 62	C
53 - 57	C-
48 - 52	D+
43 - 47	D
0 - 42	F

Notes:

Regular exercises with detailed steps are scheduled to let students learn different simulation methods. Two extra exercises with less instructions are given to students to test their understanding and skill in performing different simulation methods on the circuits. Five experiments with analysis tasks and design requirements are given to students to test their circuit analyzing and designing ability.

INDICATIVE READINGS

Journals:N/A

Trade and other Publications:N/A

Website:N/A