EE/CE/TE 3101
ELECTRICAL NETWORK ANALYSIS LAB
Fall 2016 Syllabus
Updated 8/20/2016

Instructor: Jeanne K. Pitz, Ph.D.
E-mail: jxp133430@utdallas.edu
Office: ECSN 4.312.

Lab sessions:
Sec 103; Room: ECSN 3.108-3.110; Time 1:00-3:45
TA: Arjona Angarita, Ricardo Javier email: rja150230@utdallas.edu

Sec 101; Room: ECSN 3.108-3.110; Time 4:00-6:45
TA: Gen Li email: gxl150630@utdallas.edu

Sec 102; Room: ECSN 3.112-3.114; Time 4:00-6:45
TA: Gen Li email: gxl150630@utdallas.edu

Sec 104; Room ECSN 3.118-3.120; Time 4:00-6:45
TA: Baruti, Kudra Hamidu (Kudra) email: khb160030@utdallas.edu

Course objective: This course is designed to provide students with lab skills needed for a future profession in electrical engineering. Students will design, assemble and test linear electrical networks and systems. Students will learn how to troubleshoot and debug circuits using lab equipment.. Students will simulate circuits with either Pspice or LTspice simulators. Lab classes are designed to accompany the co-requisite EE/TE3301 Electrical Network Analysis class.

Course learning objectives:
1. Ability to use spice program to analyze electrical circuits
2. Ability to measure and interpret basic electrical circuit parameters
3. Ability to use various techniques for electrical circuit analysis
4. Ability to analyze the behavior of an operational amplifier circuit.
5. Ability to measure and interpret response of first order RL and RC circuits
6. Ability to measure and interpret response of second order RLC circuits

Web site: eLearning site for the section.
Check the website each week before preparing for lab. It will be updated and will include hints, tips, etc. to help you prepare.

Lab Procedures: The Lab experiments are available at the course website described above. Acrobat reader is required for the lab manual files.
You will perform spice simulations of the circuits for the lab. Two popular spice packages are free to download and use.

PSpice from the cadence website.. Pspice is loaded on the lab computers.
http://www.orcad.com/buy/orcad-educational-program
OrCAD 16.6 Lite Software
Or
OrCAD 17.2 PSpice Designer Lite Software
Or you can download

LTSpice: You can download the free copy of **LTspice IV** from the Linear Technology Website. This is a very straight forward full spice version that is available at http://www.linear.com/designtools/software/

Both these tools are capable of performing the spice simulations for the circuits in this course. They each have pluses and minuses

Experiments:
Download the documents for each lab from the eLearning web site. A description of the lab equipment is on the eLearning site.

Lab preparation:
1. Review the lab manual prior to lab.
2. Students should read the labs carefully and complete the lab pre-work before coming to class and submit their pre-work reports to the eLearning site in the appropriate assignment entry.
3. Be prepared for the experimental procedures by understanding the relevant theory

.**Lab procedure:**
1. Arrive on time.
2. Ask for the component kit box (labeled with a number on the cover), then start the experiment. Use the same box for every experiment.
3. TAs will assist students to complete the experiment. Ask for help when students are in trouble with circuits and equipments.
4. When you have recorded all data, BOTH LAB partners should upload it to the DATAIN for the lab, the report can be written and data copied from this location.

5. Clean the workstation area and return all wires to their storage location. Turn off the power on any equipment used during the experiment.
6. Complete the lab report independently and upload the final report to the eLearning site before the expiration time. (will be posted)

Lab reports:

1. Read the descriptions of the lab pre-work and the lab report in the preface of the lab manual. You must follow the designated formats.
2. **Lab reports must be completed independently.** You can share only the collected data sets with your lab partner. Copying any part of the report from others is strictly prohibited and is against the school's scholastic integrity policy.
3. Lab reports are always due **the day prior** to the next lab. You will be notified of any changes. The last lab report is due the week after the last lab at a time to be announced. Submit lab reports to the eLearning site assignment.
4. **No late lab reports are allowed.**
5. Students should generate lab reports in a professional manner. Lab reports must be typed so they can be submitted electronically.
6. Lab pre-work must be submitted prior to the lab on the eLearning site.

Grading: Grades are based on the lab reports.

Lab Reports: 80%
Lab Pre-work: 20%

Safety rules: Read the safety rules that are presented in the preface of the lab manual and understand them for your safety.
Note: Students will work on experiments in two-person teams.

Scholastic Integrity: Scholastic dishonesty at The University of Texas at Dallas includes, but is not limited to, plagiarism and/or collusion. Scholastic dishonesty will not be tolerated. For details refer to the Scholastic dishonesty policy of University of Texas at Dallas.

Lab schedule: Lab schedule is subject to change.

Schedule	Tue	Topic
Week 1	23-Aug	**no class**
Week 2	30-Aug	Lab 0 Intro to spice and lab equipment
Week 3	6-Sep	Lab 1 lab equipment - basic circuits
Week 4	13-Sep	Lab 2 series and parallel circuits
Week 5	20-Sep	Lab 3 nodal and mesh analysis
Week 6	27-Sep	Lab 4 Spice only lab
Week 7	4-Oct	Lab 5 DC OP Amp lab
Week 8	11-Oct	
Week 9	18-Oct	Lab 6 first order response: RC and LC
Week 10	25-Oct	
Week 11	1-Nov	Lab 7 second order response: RLC
Week 12	8-Nov	
Week 13	15-Nov	Lab 8 sinusoidal response
Week 14	22-Nov	**holiday**
Week 15	29-Nov	
week 16	6-Dec	Makeup (instructors permission required)