EENG 2920:
Project IV - Analog Circuit Design Project
Class 1: PSpice Introduction

Oluwayomi Adamo
Department of Electrical Engineering
College of Engineering, University of North Texas
Course Introduction

- This is a hands-on circuit design and analysis course with industry standard software tools and hardware equipments!
  - You will learn to use the software PSpice to do circuit design and analysis.
  - You will learn to use some basic EE laboratory test and measurement equipments.
  - You will also learn to write lab reports – for every lab that you do you will need to write a lab report.
Course Organization

- **Time Allocation**
  - About half of the classes will be on PSpice
  - Another half of the classes will be on laboratory experiments

- **Classroom Assignment**
  - Lectures and PSpice Sessions will be B207
  - Labs involving equipments will be in B207

- Look into syllabus for class topics
Labs and Assignments

- **Labs**
  - Before the lab you will need to read the lab manual and do the pre-lab assignments.
  - Turn in your solution for the pre-lab assignments when you come to the lab class. **No lab work will be allowed if the pre-lab assignment is not completed beforehand.**
  - The lab report will be due in one week.

- **Assignments**
  - Generally, assignments will be exercises on PSpice circuit simulation and analysis.
  - You will need to compile (put together) simulation/analysis results in a MS Word document (or similar word processing software). **Submit the print-out of the Word document.**
  - Assignments will be due in one week.
Format for Both PSpice Assignments and Lab Reports

- **Cover page (5%)**
  - Course name, title of the lab, your name, your email address, and the date of submission

- **Contents of Lab Report (85%)**
  - **Section 1**: Introduction: learning objectives and descriptions of the assignments or labs (10%)
  - **Section 2**: Theoretical background and analysis, lab procedures, … (20%)
  - **Section 3**: Simulation or lab results and observations (20%)
  - **Section 4**: Discussion and conclusions (20%)
  - **Section 5**: Solutions for the post-lab assignments (15%)
  - You may name the section titles differently in different reports.

- **Overall appearance of the report (10%)**
- Lab report must be prepared using MS Word or other similar word processing software.
- You need to turn in the print-out of the lab report document.

- Refer to the *Guidelines and Standards for Writing Assignments* from ECE Department, Rose-Hulman Institute of Technology.
Computer-aided Circuit Analysis using PSpice Software

- Computer-aided circuit analysis is essential in designing complex circuits
  - It can provide information about circuit performance that is almost impossible to obtain with laboratory prototype measurements.

- SPICE is a general circuit simulation program that can simulate electronic circuits.
  - SPICE contains models for common circuit elements, active as well as passive.
  - SPICE stands for Simulation Program with Integrated Circuit Emphasis
History of SPICE

- The development of SPICE spans about 30 years.
  - ECAP by IBM in mid 1960s
  - CANCER by UC Berkeley in late 1960s
  - SPICE by UC Berkeley in early 1970s
  - SPICE2 by UC Berkeley in mid 1970s; this has become an industry standard and it is commonly referred to as SPICE.
  - SPICE3 by UC Berkeley is specifically designed for research.
  - A wide variety of commercial SPICE software are available now, all of which use SPICE2 as core simulation engine.
    - Mainframe version: HSPICE, IG-SPICE, PSpice, AccuSim, …
    - PC version: AllSpice, Z-SPICE, PSpice, AIM-Spice, …
PSpice Platform from Cadence

- We will use PC version PSpice from Cadence
  - PSpice software has gone through several major company merge and acquisition, now under Cadence (OrCAD is a Cadence company)

- Different versions:
  - PSpice A/D or OrCAD PSpice A/D (ver 9.1 or above)
  - PSpice Schematics (ver 9.1 or below)
  - OrCAD Capture Lite (ver 9.2 or above)
    - This is the one that comes with your book. It is also called student version.
  - OrCAD Capture and OrCAD Capture CIS (Capture with CIS, ver 10.x) are installed in all computers in EE Labs, as part of Cadence design software suite.
    - CIS: component information system
OrCAD Capture and PSpice AD

- OrCAD Capture is similar to the PSpice Schematics, but the Capture has more user friendly features than the Schematics.
  - Capture is a platform where you can draw circuit diagram and run simulations.
  - After simulation is completed, the Capture automatically opens the PSpice AD platform for displaying and viewing the output results.
- OrCAD Capture Lite has some limitations
- Many SPICE/PSpice resources are available online.
OrCAD Capture Interface
OrCAD Capture Simulation Results
PSpice AD Interface
How SPICE works?

- SPICE is a general-purpose software that can be used to simulate and calculate the performance of circuits.
  - Input to the SPICE is a **circuit file**, which describes the circuit and circuit components.
  - Output of SPICE is **output file**, which contains simulation and analysis results.
  - Both circuit file and output file are text files.
Circuit Elements

- Circuit elements are identified by names. A name must start with a letter symbol corresponding to the element, e.g., R-resistor, C-capacitor, L-inductor, V-voltage source, I-current source, and etc. (see Table 2.3)
  - The component name can be changed to anything (letters, numbers, underscore) except the first letter of the name.

- In OrCAD Capture
  - R, L, C are from .\library\PSpice\analog.olb
  - V, I, and Ground (GND) are from .\library\PSpice\source.olb

- Element values
  - Element values are written in floating-point notation with optional scale (F, P, N, U, MIL, M, MEG, G, T) and units suffixes (V, A, HZ, OHM, H, F, DEG). (see page 16 and 17)
  - The scale suffix is always the first suffix, and the unit suffix follows the scale suffix. PSpice ignores any units suffix.
How to read the book?

- The book contains following materials
  - Text based SPICE statements and commands
  - PSpice Schematics (an earlier version of PSpice)
  - OrCAD Capture (newer version, better GUI)
  - Examples of circuit analysis and simulation

- The focus of this course:
  - Circuit analysis and simulation with OrCAD Capture and PSpice, which is GUI based software
  - You need to have some idea about the text SPICE statements and commands, but that’s not our focus.
  - Do not follow exactly the procedures in the examples. Instead, do the examples and exercises using OrCAD.
Circuit Drawing in OrCAD Capture

- Draw circuits following the Appendix B of the textbook, page 420.

Assignment 1 (due next week before class):
- Turn in the circuit drawn using the tutorial handout.
  - Copy the circuit to a MS Word document, compile a report, and print the document to turn in!
- Read the textbook Chapter 1 and 2
  - You have to do this reading assignments, otherwise you will not be able to do the in-class assignment next week!