

## Course Online

1. *Course number and name*

**ENGR 290: Introduction to PSPICE**

2. *Credits and contact hours*

1 credit hours

3. *Instructor's or course coordinator's name*

Instructor: Hao Jiang, Associate Professor

Course coordinator: Hao Jiang, Associate Professor

4. *Text book, title, author, and year*

J. G. Tront, *PSPICE for Basic Microelectronics*, McGraw Hill, 2007

Web: [://www.linear.com/designtools/software/#LTspice](http://www.linear.com/designtools/software/#LTspice)

5. *Specific course information*

a. *brief description of the content of the course (catalog description)*

Introduce students to a simple computer-aided-design (CAD) circuit design tool, PSPICE or LTSPICE, to support electronic circuit analysis.

b. *prerequisites or co-requisites*

ENGR 205

c. *indicate whether a required, elective, or selected elective course in the program*

Elective for Electrical Engineering and Computer Engineering

6. *Specific goals for the course*

a. *Specific outcomes of instruction, ex. The student will be able to explain the significance of current research about a particular topic.*

- To do dc, transient domain, frequency domain, noise and Monte Carlo analysis of circuits with LC, diode, BJT and MOSFETs using a PSPICE or LTSPICE circuit simulator
- To enable students to conduct circuit analysis using a PSPICE or LTSPICE circuit simulator

b. *Explicitly indicate which of the student outcomes listed in Criterion 3 or any other outcomes are addressed by the course.*

- Student understands what is PSPICE or LTSPICE and its use in industrial applications
- Student knows how to simulate a circuit using a PSPICE or LTSPICE simulator.
- Student can demonstrate how to simulate an actual circuit using a PSPICE or LTSPICE in laboratory setting

7. *Brief list of topics to be covered*

- Dc analysis
- Time domain analysis

## Course Online

- Frequency domain analysis
- Analysis on Diode circuits
- Analysis on BJT circuits
- Analysis on MOSFET circuits