Course Announcement
EGN 187 – Engineering Tools: PSPICE
Spring 2016

When: Fridays, 11:00 AM – 12:40 PM, 1/19/16 – 5/13/16
Where: John Mitchell Center (Gorham Campus), Room 270
Instructor: Dr. Kenneth Soda

The Simulation Program with Integrated Circuit Emphasis (SPICE) is the world’s most widely used electrical and electronic design tool. Originally developed in the 1970's as a student project at UC Berkeley, it has been extensively refined not only to simulate integrated circuits, but to test, develop, qualify and document discrete circuits, sub-circuits as well as full electronic and electrical systems. In this course, you will be introduced to the latest version of PSPICE, a highly developed variant of the original program, optimized for personal computers. Methods for designing and simulating the static and dynamic operation of active and passive circuits will be demonstrated and practiced. A variety of simulation conditions and testing methodologies will be explored. These include the design of printed circuit boards directly from simulated designs. Skills developed in this course are essential for both electrical and mechanical engineering students and are applied in several other USM engineering courses. PSPICE finds wide use in industry for design and documentation of electrical and electronic systems.

Prerequisites: None
Delivery: Lecture, Guided Hands-On In-Class Practice, Free Student Software Available
Credits: 1

Syllabus Highlights
- Creating and Organizing Project Files
- Selecting and Placing Components
- Bias, DC, AC and Sweep Analysis Methods
- Displaying and Organizing Output
- Developing Robust Designs
- Creating and Managing Device Models
- Intro to Printed Circuit Design

Evaluation Methods: In-Class Quizzes, Project

Instructor Biography:
Dr. Soda is a former Associate Professor of Electrical Engineering at the US Air Force Academy, Colorado. He has over twenty years teaching experience and over ten years industry experience in electronics, electronics laboratory and integrated circuit design as well as extensive experience with the OrCAD PSPICE design tool.