PSPICE

<https://www.youtube.com/watch?v=8_pE68B6dqc>

**PSpice** is a program to simulate analog and digital logic circuits, where **Matlab** is a fully functional programming language designed to plot mathematical functions, implement various algorithms and solve complex mathematical problems.

Pspice is used for circuit simulation. The behavior of circuits under a particular set of conditions

**History**

SPICE was first developed at the University of California, Berkeley, in the early 1970s. Subsequently an improved version SPICE 2 was available in the mid-1970s especially to support computer aided design.

PSpice was released in January 1984, and was the first version of UC Berkeley SPICE available on an IBM Personal Computer. PSpice later included a waveform viewer and analyser program called Probe. Subsequent versions improved on performance and moved to [DEC/VAX minicomputers](https://en.wikipedia.org/wiki/VAX), Sun workstations, Apple Macintosh, and Microsoft Windows. Version 3.06 was released in 1988, and had a "Student Version" available which would allow a maximum of up to ten transistors to be inserted. PSpice (even the student version) increases the students' abilities to understand the behavior of electronic components and circuits.

Analysis

The type of simulation performed by PSpice depends on the source specifications and control statements. PSpice supports the following types of analyses:

* DC Analysis - for circuits with time–invariant sources (e.g. steady-state DC sources). It calculates all nodal voltages and branch currents over a range of values. Supported types include Linear sweep, Logarithmic sweep, and Sweep over List of values.
* Transient Analysis - for circuits with time variant sources (e.g., sinusoidal sources/switched DC sources). It calculates all nodal voltages and branch currents over a time interval and their instantaneous values are the outputs.
* AC Analysis - for small signal analysis of circuits with sources of varying frequencies. It calculates the magnitudes and phase angles of all nodal voltages and branch currents over a range of frequencies.

Introduction to PSPICE

PSPICE is a circuit analysis tool that allows the user to simulate a circuit and extract key voltages and currents. Information is entered into PSPICE via one of two methods; they are a typed ‘Net List’ or by designing a visual a schematic which is transformed into a netlist. However, to fully utilize the schematics we must first understand and become familiar with designing the net list.

Net List Format

The first line of the net list is the title line. This should contain pertinent information to the circuit and your name. The next lines are for circuit parameters – as many as needed. The next section is for output control statements. The file is closed with a statement. Below is the syntax for various elements. The [] and the <> are not actually typed – they are for visual purposes only.Parameter components must be separated by spaces.

Parameter Syntax

Resistor: R [+ node] [- node] [value]

Capacitor: C [+ node] [- node] [value] [IC = , optional]

Inductor L [+ node] [- node] [value] [IC = , optional]

#### Example

Name - Lab # – Circuit Number (Header Line)

R1 0 1 1k ; 1000 ohm resistor named R1 from ground to node 1

V1 1 0 DC 0 ; Zero volt DC source from node 1 to ground

.END ; Required formal end statement

**Multisim** is the preferred SPICE circuit simulator for use in EE-331.  The current version that is installed on the general purpose computers in the EE Department is 11.0.  Multisim was originally developed by Electronics Workbench in Canada, along with the companion printed circuit board (PCB) layout tool Ultiboard.  Electronics Workbench was bought by National Instruments in 2007, and the Multisim and Ultiboard products are now marketed and supported by National Instruments.  Multisim and Ultiboard also couple closely to LabVIEW when needed.

Several separate tasks comprise the overall process of circuit simulation.  These include:  (1) schematic capture, (2) device modeling, (3) setting up and running the simulation, and (4) output analysis.

FEATURES

A virtual electronics learning lab on the desktop.

Reduced need for expensive equipment.

Interactive analog and digital simulation.

Animated lectures with live simulation.

Wide range of components available.

Teaches real world troubleshooting.

Compliment any teaching style.

Use as pre-lab, co-lab or post-lab resource.

##### Who makes Multisim?

NI Multisim and Multisim Live are developed and owned by [National Instruments](http://www.ni.com/)

##### What is Multisim?

NI Multisim is an electronic schematic capture and simulation program which is part of a suite of circuit design programs, along with NI Ultiboard. Multisim is one of the few circuit design programs to employ the original Berkeley SPICE based software simulation.

**ETAP** extends to a Real-Time Intelligent Power Management System to monitor, control, automate, simulate, and optimize the operation of power systems.

<https://etap.com/student-edition>

<https://www.youtube.com/embed/8OY-lMHOsOo>

 ETAP [power engineering software](https://en.wikipedia.org/wiki/Power_engineering_software) utilizes an electrical digital twin in order for [electrical engineers](https://en.wikipedia.org/wiki/Electrical_engineering) and operators to perform following studies in offline or online mode:

* [Load flow](https://en.wikipedia.org/wiki/Power-flow_study) or power flow study
* Short circuit or fault analysis
* Protective device coordination, discrimination or selectivity[]](https://en.wikipedia.org/wiki/Electrical_Transient_Analyzer_Program#cite_note-8)
* Transient or dynamic stability.
* Substation design and analysis
* [Harmonic](https://en.wikipedia.org/wiki/Harmonic) or power quality analysis
* Reliability
* [Optimal power flow](https://en.wikipedia.org/wiki/Optimal_power_flow)
* Power system stabilizer tuning
* Optimal capacitor placement
* Motor starting and acceleration analysis
* Voltage stability analysis
* [Arc flash](https://en.wikipedia.org/wiki/Arc_flash) hazard assessment
* Ground loop impedance calculation
* Battery modeling and simulation

## Software applications

**ETAP**® is the most comprehensive electrical engineering **software** platform for the design, simulation, operation, and automation of generation, transmission, distribution, and industrial systems.

**ETAP stands for** Electrical Transient and Analysis Program.

ETAP is the most comprehensive analysis platform for the design, simulation, operation, and automation of generation, distribution, and industrial power systems. ETAP is developed under an established quality assurance program and is used worldwide as a high impact software. ETAP is completely localized in four languages with translated output reports in six languages

Advantages of etap software:

* Short circuit study. You can design rupturing capacity of HV, MV and LV boards (switchgear) using **ETAP**.
* Load flow study. ...
* Relay coordinations.
* Transformer sizing.
* Motor stability.
* Arc flash study.
* Cable scheduling.
* Solar system analysis and so many other studies too.

ETAP software applications include:

* Power system design for ANSI and IEC networks
* Electric supply substation simulation
* Monitoring and feeder analysis
* Simulation of distributed photovoltaic power
* Study of a DC network
* Open-phase fault analysis - Multiple events across the nuclear power industry have highlighted the need for greater understanding of what happens during an open phase fault. These open phase events have occurred on the high side of offsite power supply transformers and have involved loss of one or two phases.
* Diesel power plant analysis
* Combined cycle power plant analysis
* AC/DC hybrid system simulation
* Wind turbine design and analysis
* Harmonics in railway power systems
* Rural distribution system analysis
* Distributed generation protection
* Reliability assessment of renewable energy systems
* Wind and PV penetration studies