SPICE Simulation Models

Publish Date: Apr 11, 2012

Table of Contents

1. Overview

The National Instruments **SPICE Simulation Fundamentals** series is your free resource on the internet for learning about circuit simulation. The series is a set of tutorials and information on SPICE simulation, OrCAD pSPICE compatibility, SPICE modeling, and other concepts in circuit simulation.

For more information, see the **SPICE Simulation Fundamentals** main page.

The series is divided among a number of in-depth detailed articles that will give you HOWTO information on the important concepts and details of SPICE simulation.

Circuit simulation is an important part of any design process. By simulating your circuits, you can detect errors early in the process, and avoid costly and time consuming prototype reworking. You can also easily swap components to evaluate designs with varying bills of materials (BOMs).

An important key to performing accurate and successful SPICE simulation is to use high quality SPICE models. While most circuit simulation packages such as **Multisim** come with thousands of components and SPICE simulation models, frequently designers need to use a part that does not exist in the available database. When these situations arise, the software tool will typically have a way of adding custom components and models to the database. **Multisim** for example has a detailed component creation wizard that will guide designers through the process of defining custom parts for simulation and PCB layout (See **Creating Custom Components in Multisim**).

2. What is a SPICE Simulation Model?

A SPICE model is a text-description of a circuit component used by the SPICE Simulator to mathematically predict the behavior of that part under varying conditions. SPICE models range from the simplest one line descriptions of a passive component such as a resistor, to extremely complex sub-circuits that can be hundreds of lines long.

SPICE models should not be confused with pSPICE models. pSPICE is a proprietary circuit simulator provided by OrCAD. While some pSPICE models are compatible with SPICE, there is no guarantee. SPICE is the most widely used circuit simulator, and is an open standard.

3. Model Makers

Some SPICE simulation programs such as **Multisim** include model makers to automatically generate SPICE models for various components. **Multisim** version 10.1 has 24 SPICE Model makers.

4. Where to look for SPICE Simulation models

The best place to look for SPICE models is to browse the vendor or manufacturer's website. Listed below are some of the most popular chip vendors that supply SPICE models on their website.

Vendor	Description
Analog Devices	Amplifiers and Comparators, Analog to Digital Converters, Digital to Analog Converters, Embedded Processing & DSP, MEMS and Sensors, RF/IF Components, Switches/Multiplexers, Analog Microcontrollers, Interface, Power and Thermal Management
Analog and RF Models	Analog and RF Models
Apex Microtechnology	Linear Amplifiers, PWM Amplifiers
Christophe Basso	Switch-mode power supplies
Coilcraft, Inc.	Power Magnetics, RF Inductors, EMI / RFI Filters, Broadband Magnetics
Directed Energy	Diodes, Switch-mode MOSFETs, HF / VHF Linear MOSFETs, MOSFET Driver ICs
Duncan Amps	Amplifiers, Vacuum tubes
Fairchild Semiconductors	Amplifiers & Comparators, Diodes & Rectifiers, Interfaces, Digital Logic Devices, Signal Conversion, Voltage to Frequency Converters, Microcontroller, Optoelectronics, Switches, Power Controllers, Power Drivers, Transistors, Filters, Voltage Regulators
Infineon Technologies AG	Fiber Optics, Microcontrollers, Power Semiconductors, Small Signal Discretes

International Rectifier	HEXFET Power MOSFETs, Diodes, Bridges, Thyristors, Relays, High Voltage ICs, Intelligent Power Modules, Intelligent Power Switch, HiRel Power MOSFETs, HiRel High Voltage Gate Drivers
Kemet Home Page	Surface-mount capacitors in aluminum, ceramic and tantalum and leaded capacitors in ceramic and tantalum
Linear Technology	Signal Conditioning, Data Conversion, Power Management, Interfacing, High Freuqency & Optical
Maxim	Amplifiers and Comparators, Analog Switches and Multiplexers, Clocks, Counters, Delay Lines, Oscillators, RTCs, Data Converters, Sample-and-Holds, Digital Potentiometers, Fiber and Communications, Filters (Analog), High-Frequency ASICs, Hot-Swap and Power Switching, Interface and Interconnect, Memories: Volatile, NV, Multi-Function, Thermal Management, Sensors, Sensor Conditioners, Voltage References, Wireless, RF, and Cable Amplifiers,Power Management, Temp Sensors, Interface, LVDS, Ethernet, USB Technologies, Micro SMD
National Semiconductor	Power Management, Amplifiers, Comparators, Analog Switches, Thyristors, Diodes, Rectifiers, Bipolar Transistors, FETs, Standard Logic, Differential Logic,
ON Semiconductor	Analog/Linear, Audio, Automotive, Connectivity, Data/Media/Video processing, Discretes, Displays, Interface and control, Logic, Microcontrollers, Power and power management, RF, Sensors
Philips	Polyfet transistors
Polyfet	Transient Voltage Suppression
Protek	Switch-mode power supply simulation
SMPS Power Supplies	Switch-mode power supply design
SMPS Technology	Mixed signal semiconductor, High-voltage interface products
Supertex	Amplifiers & Linear,Analog & Mixed Signal ICs, Diodes, EMI Filtering & Conditioning, Logic, Signal Switch, Memories, Microcontrollers, Power Management, Protection Devices, Sensors, Smartcard ICs, Thyristors & AC Switches, Transistors
STMicroelectronics	Buffers, Drivers and Transceivers, Flip-Flops, Latches and Registers, Gates, Counters, Decoders/Encoders/Multiplexers, Digital Comparators
Texas Instruments	Electromechanical components, passive components, power sources, RF & Microwave products
Tyco Electronics (formerly Amp)	Manufacturer of analog switches, capacitors, diodes, inductors, integrated modules, power ICs, LEDs, power MOSFETs, resistors and thermistors.
Vishay	DC-DC boost controllers, Voltage references, Current monitors, Motor control, Acoustar™ audio solutions, Linear regulators
Zetex	

Evaluate

Multisim for free for 30 days

Explore

Multisim with an Interactive Demo

Download

Free Courseware for Circuits

Learn

About the NI Electronics Education Platform