SPICE Simulation Fundamentals

Publish Date: Dec 23, 2013

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 simulation, SPICE modeling, and other concepts in circuit simulation.

The series is divided among a number of in-depth detailed articles that offer **HOW TO** 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).

NI **Multisim** is an example of an easy-to-use, powerful, and flexible SPICE simulation environment that allows educators to teach circuit theory, and engineers to quickly design topologies.

2. SPICE Simulation Overview

SPICE Simulation Overview
The basics of SPICE simulation. What is SPICE simulation? Learn about netlists, SPICE models, and speed versus accuracy in SPICE simulation.
Learn more >>

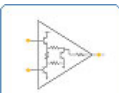### 3. SPICE Simulation History

SPICE Simulation History
SPICE simulation has been available for more than 30 years. Learn more about the history of SPICE in its earlier forms, and what it has evolved into today.
Learn more >>

4. SPICE Simulation Models

SPICE Simulation Models
One of the most fundamental facets of SPICE simulation is a set of accurate models of the components used in any given circuit. There are literally millions of different SPICE models available. Learn about the basics of a SPICE model, model makers, and where to look for vendor-specific models of components.
Learn more >>

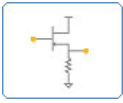### 5. Basic SPICE Simulation Model Parameters

Basic SPICE Simulation Model Parameters
Intrinsic to SPICE is the ability to model basic and more advanced single element real and ideal components. Read

this tutorial to learn the basics about the various models that are available in SPICE.
Learn more >>

---

6. Advanced SPICE Simulation Model Parameters

Advanced SPICE Simulation Model Parameters
The native SPICE models have a number of very specific parameters that control how the simulation model will behave under varying circumstances. For example, you can change the temperature coefficients for a resistor to make your simulation more closely match a vendor's component. This article lists the names and descriptions for the hundreds of parameters available in SPICE.
Learn more >>

---

7. SPICE Simulation Options

SPICE Simulation Options
Simulations can be accurate because of the quality of the SPICE models used and the simulation options. Parameters such as relative tolerance (RELTOL) and timestep can have a drastic affect on the results produced by the SPICE simulator. This article gives you detailed information on a number of the many options available and what effect changing them will produce.
Learn more >>

**8. SPICE Simulation Control Statements**

SPICE Simulation Control Statements
At the heart of every industry standard SPICE simulator is a text-based core. Power users of SPICE simulation take advantage of their knowledge to achieve the greatest control over exactly how simulations proceed. This article gives an overview of the control statements that engage the SPICE engine.
Learn more >>

**9. SPICE Simulation Source Type and Parameters**
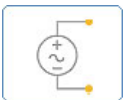
SPICE Simulation Source Type and Parameters
Proper SPICE simulation will always require some type of stimulus or source. Native to SPICE are a number of basic and more complex sources such as Pulse voltages, Exponential voltages, Sine waves, and Piece-wise linear sources. This article gives detailed information on the syntax of these source elements in SPICE.
Learn more >>

**10. SPICE Simulation Course on Connexions**
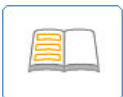
Introduction to Multisim Schematic Capture and SPICE Simulation
This course introduces Multisim through instructional modules and hands-on exercises. The entire course is available free online, and will walk you through the basics of capture and simulation using Multisim.
Learn more >>

---

11. SPICE Simulation User Guide
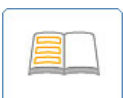
SPICE Simulation User Guide
The SPICE User Guide is an essential compendium of knowledge on the syntax and details of SPICE simulation. Use this link to access your copy of the SPICE User Guide.

Learn more >>

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