

# Advanced SPICE Simulation Model Parameters

Publish Date: May 16, 2008

## 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).

## 2. Advanced Model Parameters

The SPICE language can model many sophisticated real world effects such as the result of temperature variations on a component.

The attached document lists the detailed model parameters for all the native SPICE models.