

# SPICE Simulation Options

Publish Date: May 16, 2008

## Table of Contents

1. [Overview](#)
2. [What are SPICE Simulation Options?](#)
3. [A Tradeoff Between Speed and Accuracy](#)
4. [Changing SPICE Simulation Options](#)
5. [A Listing of SPICE Simulation Options](#)
6. [SPICE2 Emulation Mode](#)

### 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. What are SPICE Simulation Options?

SPICE simulation can help to predict the behaviour of electronic circuits of almost any complexity. The SPICE simulator will execute a desired transient, DC, AC or other simulation based on the parameters of the simulation (e.g. length of time, start/stop frequencies, initial conditions, etc.) and based on the options chosen. The SPICE options set will affect simulations in various ways.

### 3. A Tradeoff Between Speed and Accuracy

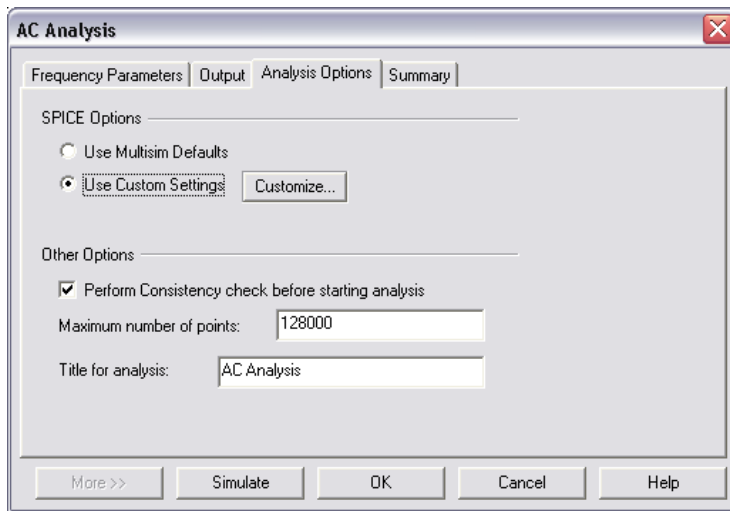
Although the SPICE models used in a SPICE simulation can greatly affect the accuracy of the results, simulation settings also contribute to varying degrees of accuracy. SPICE simulation options generally allow the user to gain more accuracy in the results at the cost of the speed of the simulation.

To understand the tradeoff between speed and accuracy in SPICE simulation one must consider a number of factors. SPICE simulation was created over 30 years ago and around that time a typical computer had less power than the average microwave oven did thirty years later. Computing power was very expensive. The simulation of a circuit to the highest degree of accuracy could have taken longer and cost more money than building the actual circuit to see the results. Also, consider that the broad purpose of circuit simulation is to augment basic hand calculations and predict general circuit behavior. With these considerations in mind, the designers of SPICE created a program that could produce reasonably accurate results in a cost-effective manner. They also included many options to allow engineers to customize the accuracy of a simulation.

As computing power has increased exponentially over the years, so have the complexity of circuit designs being simulated. Speed and accuracy are still important factors to consider when simulating circuits.

### 4. Changing SPICE Simulation Options

In [Multisim](#), you can change the SPICE options for interactive simulation by clicking Simulation >> Interactive Simulation Settings from the main menu. You can change settings for individual simulations by navigating to the Analysis Options tab for each desired simulation and clicking "Use Custom Settings".



## 5. A Listing of SPICE Simulation Options

Option	Effect
ABSTOL=x	Resets the absolute current error tolerance of the program. The default value is 1 picoamp.
BADMOS3	Use the older version of the MOS3 model with the "kappa" discontinuity.
CHGTOL=x	Resets the charge tolerance of the program. The default value is 1.0e-14.
DEFAD=x	Resets the value for MOS drain diffusion area; the default is 0.0.
DEFAS=x	Resets the value for MOS source diffusion area; the default is 0.0.
DEFL=x	Resets the value for MOS channel length; the default is 100.0 micrometer.
DEFW=x	Resets the value for MOS channel width; the default is 100.0 micrometer.
GMIN=x	Resets the value of GMIN, the minimum conductance allowed by the program. The default value is 1.0e-12.
ITL1=x	Resets the dc iteration limit. The default is 100.
ITL2=x	Resets the dc transfer curve iteration limit. The default is 50.
ITL3=x	Resets the lower transient analysis iteration limit. the default value is 4. (Note: not implemented in Spice3).
ITL4=x	Resets the transient analysis timepoint iteration limit. the default is 10.
ITL5=x	Resets the transient analysis total iteration limit. the default is 5000. Set ITL5=0 to omit this test. Note: not implemented in Spice3.
KEEPOPINFO	Retain the operating point information when either an AC, Distortion, or Pole-Zero analysis is run. This is particularly useful if the circuit is large and you do not want to run a (redundant) ".OP" analysis.
METHOD=name	Sets the numerical integration method used by SPICE. Possible names are "Gear" or "trapezoidal" (or just "trap"). The default is trapezoidal.
PIVREL=x	Resets the relative ratio between the largest column entry and an acceptable pivot value. The default value is 1.0e-3. In the numerical pivoting algorithm the allowed minimum pivot value is determined by $EPSREL = AMAX1(PIVREL * MAXVAL, PIVTOL)$ where MAXVAL is the maximum element in the column where a pivot is sought (partial pivoting).
PIVTOL=x	Sets the absolute minimum value for a matrix entry to be accepted as a pivot. The default value is 1.0e-13.
RELTOL=x	Sets the relative error tolerance of the program. The default value is 0.001 (0.1%).
TEMP=x	Sets the operating temperature of the circuit. The default value is 27 deg C (300 deg K). TEMP can be overridden by a temperature specification on any temperature dependent instance.
TNOM=x	Sets the nominal temperature at which device parameters are measured. The default value is 27 deg C (300 deg K). TNOM can be overridden by a specification on any temperature dependent device model.
TRTOL=x	Sets the transient error tolerance. The default value is 7.0. This parameter is an estimate of the factor by which SPICE overestimates the actual truncation error.
TRYTOCOMPACT	Applicable only to the LTRA model. When specified, the simulator tries to condense LTRA transmission lines' past history of input voltages and currents.
VNTOL=x	Sets the absolute voltage error tolerance of the program. The default value is 1 microvolt.

## 6. SPICE2 Emulation Mode

In addition, the following options have the listed effect when operating in SPICE2 emulation mode:

Option	Effect
ACCT	Causes accounting and run time statistics to be printed
LIST	Causes the summary listing of the input data to be printed
NOMOD	Suppresses the printout of the model parameters
NOPAGE	Suppresses page ejects
NODE	Causes the printing of the node table.
OPTS	Causes the option values to be printed.