

Simulation Tool Allows Engineers To Design Larger, Complex Circuits

Analog Devices, Inc. (ADI) and National Instruments (NI) collaborated on a new release of NI's Multisim component evaluation tool with added features and functionality to provide engineers with an easy-to-use environment for the simulation of linear circuits using ADI components. The free component evaluation tool is available on ADI's website. Download a copy of the Multisim SPICE Simulation Program at <http://www.analog.com/multisim> [1].

Two important feature enhancements available in this edition of the component evaluation tool allow engineers to design larger, more complex circuits and easily import their own models into the tool. By matching more than 1,000 of ADI's amplifiers, switches and voltage references to over 550 of NI's simulation models, designers have free access to the industry's premier simulation environment allowing them to easily experiment with circuit designs and reduce system development time and cost.

SPICE Simulation and Component Evaluation Made Easy

"With this custom version of the NI Multisim simulation environment, engineers can take a graphical approach to evaluating many of ADI's industry-leading linear analog components. The result is the flexibility and creative freedom to quickly improve design decisions," said Bhavesh Mistry, general manager, National Instruments Electronics Workbench Group. "Engineers working to tight schedules and budgets can avoid costly and time-consuming re-design work due to component choice by enhancing productivity with this SPICE simulator."

- Download a free copy of NI Multisim Component Evaluator - Analog Devices Edition at <http://www.analog.com/nimultisimevaluator> [2].
- What's new in NI's Multisim Circuit Simulation Software: <http://zone.ni.com/devzone/cda/tut/p/id/12222> [3]
- Find Multisim technical support here: <http://www.ni.com/support/multisim supp> [4]
- Learn about SPICE simulation fundamentals: <http://zone.ni.com/devzone/cda/tut/p/id/5579> [5]

"Finding the right solution for a linear circuit often means sorting through many products and specifications," said Steve Sockolov, director, Precision Signal Conditioning Group, Analog Devices. "With ADI's extensive portfolio of amplifiers and other linear components, it's important that we offer innovative ways to help customers find what they need quickly. Multisim joins a suite of free resources, such as ADI's DiffAmp Calculator, developed with the engineer's product selection and evaluation needs in mind."

Simulation Tool Allows Engineers To Design Larger, Complex Circuits

Published on Electronic Component News (<http://www.ecnmag.com>)

NI Multisim Component Evaluator - Analog Devices Edition Features and Benefits:

- Build simulatable circuits to evaluate a library of Analog Devices' operational amplifiers, switches and voltage references.
- Simulate better with SPICE parser improvements, updated BSIM models, support for advanced parameters and enhanced digital simulation accuracy.
- Improve design communication with on-page connectors and a new WYSIWYG net naming system.

<http://www.analog.com> [6]

Source URL (retrieved on 04/25/2014 - 5:29am):

<http://www.ecnmag.com/product-releases/2011/01/simulation-tool-allows-engineers-design-larger-complex-circuits>

Links:

- [1] <http://www.analog.com/multisim>
- [2] <http://www.analog.com/nimultisimevaluator>
- [3] <http://zone.ni.com/devzone/cda/tut/p/id/12222>
- [4] <http://www.ni.com/support/multisim supp>
- [5] <http://zone.ni.com/devzone/cda/tut/p/id/5579>
- [6] <http://www.analog.com>