NI Multisim 13.0 Enhances Analog, Digital and Power Circuit Simulation for Education, Research, and Professional Design

This intuitive schematic capture environment improves student comprehension and reduces prototype iterations and development costs for design and test engineers.

PR Newswire

AUSTIN, Texas, Oct. 8, 2013

AUSTIN, Texas, Oct. 8, 2013 /PRNewswire/ -- National Instruments (Nasdaq: NATI) introduces Multisim 13.0, a best-in-class SPICE simulation environment used by educators, students and engineers worldwide to explore, design and prototype circuits.

New Multisim 13.0 benefits include:

- Circuit parameter and parameter sweep analysis
- Digital circuits education with NI myRIO and Digilent FPGA targets
- Power electronics analysis with IGBT and MOSFET thermal models
- Device library of 26,000+ components
- Design automation with the NI LabVIEW Multisim API Toolkit

Multisim 13.0 offers comprehensive circuit analysis tools for analog, digital and power electronics. The graphical, interactive environment helps educators to reinforce circuit theory and bridge the gap between the classroom and hands-on laboratory learning. The same advanced Multisim analysis capabilities are also used in various industries to explore design decisions and optimize circuit behavior with mixed-mode simulation.
To take students from basic electronics comprehension to complex final year design projects, Multisim includes courseware and laboratory hardware integration to NI myDAQ, NI ELVIS, NI myRIO and digital products from Digilent, for a complete solution to multiple courses. Multisim 13.0 also boasts ready-to-use daughterboard templates to speed design time for NI Single-Board RIO hardware and more.

"We chose Multisim for the breadth it provides us, allowing first year undergraduate students to understand fundamental analog and digital electronics, but also the depth so that final year masters students can use it in their projects," said Danielle George, Faculty Member of Engineering and Physical Sciences at the University of Manchester.

Using the latest device simulation models from leading semiconductor manufacturers in an interactive analysis environment, aerospace, energy and life science applications are evaluated, optimized and designed to meet specifications on time.

Additionally, the Multisim API Toolkit for LabVIEW system design software defines countless applications to correlate measurements, sweep domain specific conditions and analyze performance with a flexibility unavailable to conventional simulation environments.

**Additional Resources**

- Learn more about Multisim ([ni.com/multisim](http://ni.com/multisim/))
- Explore Multisim circuits teaching applications ([ni.com/multisim/courseware](http://ni.com/multisim/courseware))
- View Multisim circuit design applications ([ni.com/multisim/applications/pro/](http://ni.com/multisim/applications/pro/))

*LabVIEW, Multisim, National Instruments, NI, and ni.com are trademarks of National Instruments. Other product and company names listed are trademarks or trade names of their respective companies.*

CONTACT: Beth Williams, *[beth.williams@ni.com](mailto:beth.williams@ni.com)*, (512) 683-6394