NI Multisim - Electronic Design, Circuit Simulation and Prototyping

Patrick Noonan
Business Development Manager
Electronics Workbench Tools
NI Multisim
- Integrated capture and simulation environment
- Interactive mixed-mode simulation
- 20 SPICE Analyses
- 22 Measurement Instruments
  - [ni.com/multisim](ni.com/multisim)

NI Ultiboard
Flexible, easy-to-use PCB layout tool
Design tools optimized for manual control or automated speed
Integration with NI Multisim  
  - [ni.com/ultiboard](ni.com/ultiboard)

NI LabVIEW
- Graphical development environment
- Tight integration of real-world I/O
- Easy measurement analysis, and data presentation
  - [ni.com/labview](ni.com/labview)
NI Multisim | Schematics  Simulation and Analysis

• Graphical based schematic capture and integrated simulation
  ▪ Rapidly build and simulate circuits
  ▪ Analog and Digital co-simulation (SPICE/XSPICE)

• Thousands of components immediately ready for simulation
  ▪ Place components, wire and click run to start the simulation

• Integration with Measurements
  ▪ Simulation is an mathematical approximation
  ▪ Measurements are the REAL answer

• Virtual Instruments for immediate testing
• Advanced analyses for design validation
• Integration with NI Ultiboard for Full PCB Design
What is SPICE? | Examples

Example 1: Voltage divider netlist
* Voltage Divider - comment
vV1 1 0 12
rR1 1 2 1000
rR2 2 0 2000

Example 2: Subcircuit model
.subckt bipolarjunctiontrans base collector emitter
R1 base n100 200
C1 n100 emitter 1.000E-9
D1 n100 emitter DX
e1 base n100 collector emitter 12.842917
R2 collector emitter 10
.MODEL DX D(IS=1e-15 RS=1)
SPICE and Virtual Instrumentation
Simulation, Measurements and Automation

• Bring Measurements inside of Multisim
  ▪ Readily available instruments emulate a test lab
  ▪ Custom LabVIEW instruments for everything else

• Compare Simulation and Measurements
  ▪ Improve Design Process
  ▪ Troubleshoot and debug circuits

• LabVIEW Multisim Connectivity Toolkit
  ▪ Multisim API Controls simulation
  ▪ Create Virtual DUT and test with LabVIEW
  ▪ Run, pause, stop simulation
  ▪ Change components, view circuit
  ▪ Set input, view output
Improving Analysis | Custom LabVIEW Instruments

- Define custom Measurements and Multisim analyses – in the simulation stage.
  - Fully leverage simulation
  - Advanced characterization
  - Analyses outside the realm of traditional SPICE analyses

- Instruments can be defined as input and/or output

- Instruments can import and export real signals to simulation
  - Built-in virtual prototype

Step 1 Design a circuit, simulate and analyze in Multisim

Step 2 Create custom instruments in LabVIEW for simulation

ni.com
Measurements in Multisim

- Advanced Control Design
  - system ID, Control Design, dynamic system simulation, etc.

- Digital Filter Design
  - (FIR / IIR Filter Design, Quantization, Fixed-point Simulation, etc)

- Advanced Signal Processing
  - (Wavelets, Time-Series Analysis, Time-Frequency Analysis, etc)

- Sound and Vibration
  - (Distortion, Octave Analysis, Swept Sine, Freq Measurements, Transient, S&V Level, Weighting, Waterfall Plot)

- Order Analysis
  - (Order Tracking, Spectrum Selection, Tachometer Processing, Waterfall, Orbit/Polar Plots, Bode Plots, etc)

- Spectral Measurements
  - (Zoom FFT, Power-in-Band, Adjacent Channel Power, etc)

- Modulation
  - (Bit Error Rate, AWGN, Phase Noise, Constellation Plots, Eye Diagrams, etc)

- Signal Processing
  - (Signal Gen, Windows, Filters, Transforms, etc)

- Mathematics
  - (Numerics, Linear Algebra, Curve Fit, Prob/Stats, Optimization, Diff EQ, etc)

- Measurements
  - (Spectral, Tone Extraction, Pulse Params, Timing/Transition, Amp/Levels, etc)
Improving Simulation | Virtual Prototype

**Step 1** Create a real signal and connect to LabVIEW

**Step 2** Save as signal to the LVM format and transfer to Multisim

**Step 3** Simulate in Multisim with a real signal

**Step 4** Improve the design in Multisim **BEFORE** prototyping

**LabVIEW | Measurement**

**Multisim | Part Evaluation, Schematic Capture & Simulation**

**Ultiboard | Layout & Routing**
NI for Custom Designs

- Customizing the NI Platform Products
  - RIO Products Series, cRIO, sbRIO
  - DAQ, Instruments
  - Products Can be Customized  [Ex. Test Fixtures, Embedded Targets]

- Solution:
  - NI Multisim and NI Ultiboard provide an effective hardware design solution for customizing NI’s embedded platforms

- Examples:
  - NI Connector Database
  - sbRIO Custom daughter card reference designs
Hardware Design References in Multisim

- Multisim and Ultiboard = Circuit Design Suite
  - Complete low cost and easy to use schematic and PCB layout tool
- NI SW and HW Connectivity
  - Multisim interface toolkit for LabVIEW
  - NI Connector Component Library
- Multisim Design Examples
  - sbRIO Daughter Card Architecture
  - cRIO Module & cRIO Accessories
  - Custom NI connector boards
  - DAQ Accessories & Test Fixtures

Ex: sbRIO Daughter Card Reference Design
Multisim | Recap

Design and Prototype

- Multisim is integrates *simulation and validation* with LabVIEW
- Improve design models by using *real measurements* with simulation
- Troubleshoot design errors through correlated simulation and measurements
- Programmatically automate and control Multisim with LabVIEW (Virtual DUT)
- Easily create custom NI interfaces and hybrid embedded platforms
NI Multisim | Where to Learn More

- For product information: ni.com/multisim
- Circuit Design Technical Library
  - SPICE Simulation fundamentals
  - Example Circuits
  - Custom LabVIEW Virtual Instruments
  - User Guides and Manuals
  - Interactive Discussion Forum
  - Support Page
- For product information call: 1.800.263.5552