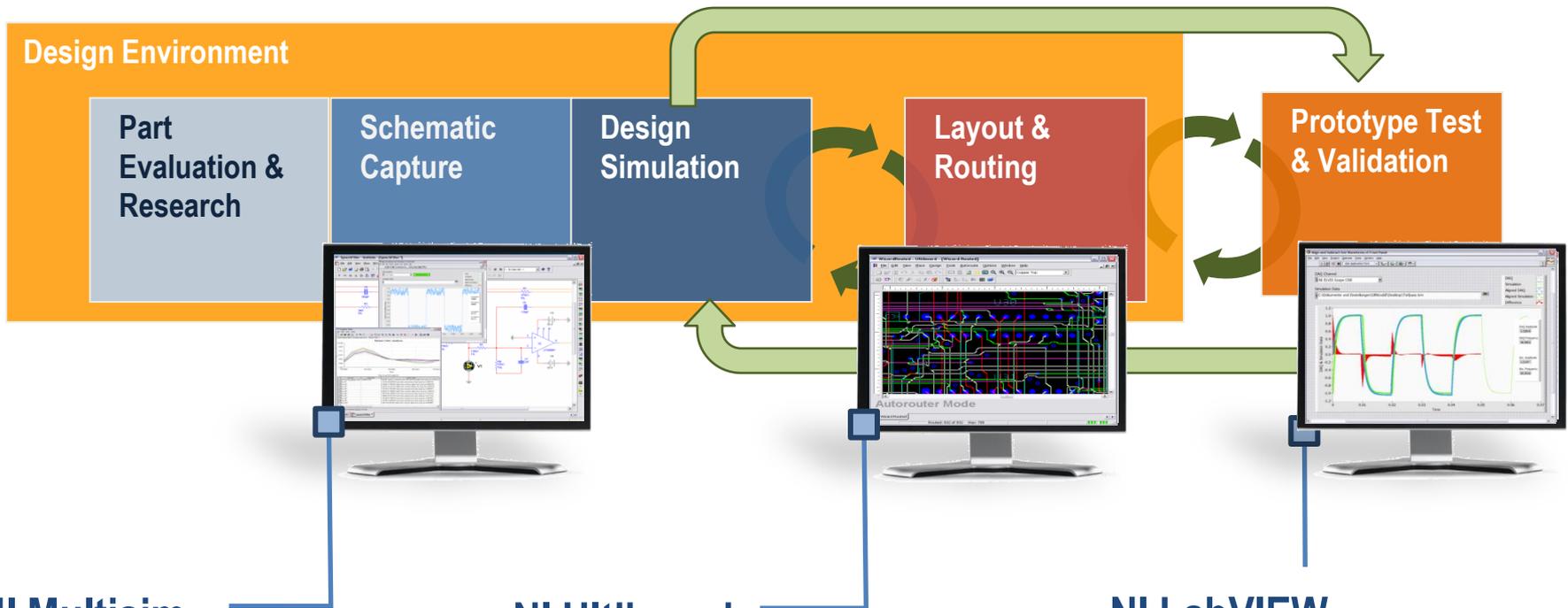


# NI Multisim - Electronic Design, Circuit Simulation and Prototyping

**Patrick Noonan**  
**Business Development Manager**  
**Electronics Workbench Tools**

# Integrated Design Flow | Simulation and Virtual Instrumentation



## NI Multisim

- Integrated capture and simulation environment
- Interactive mixed-mode simulation
- 20 SPICE Analyses
- 22 Measurement Instruments
- [ni.com/multisim](http://ni.com/multisim)

## NI Ultiboard

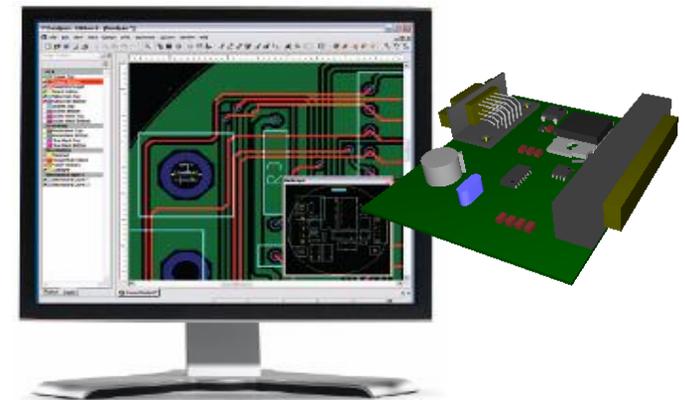
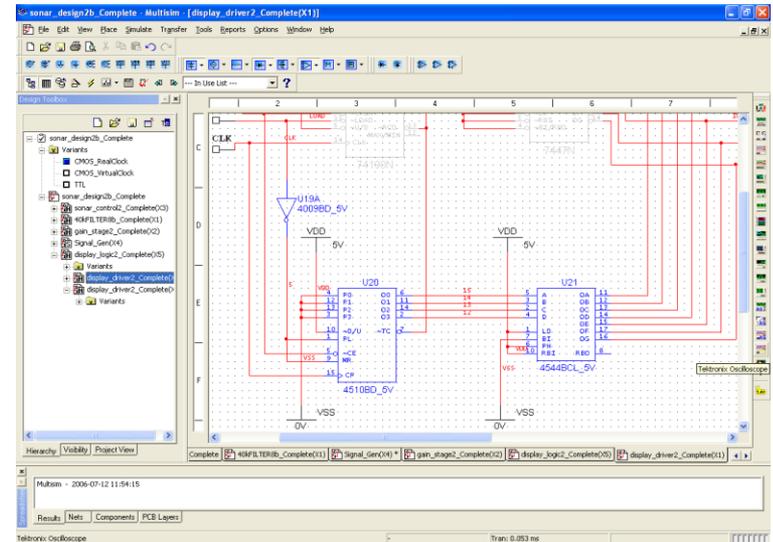
Flexible, easy-to-use PCB layout tool  
Design tools optimized for manual control or  
automated speed  
Integration with NI Multisim  
[ni.com/ultiboard](http://ni.com/ultiboard)

## NI LabVIEW

- Graphical development environment
- Tight integration of real-world I/O
- Easy measurement analysis, and data presentation
- [ni.com/labview](http://ni.com/labview)

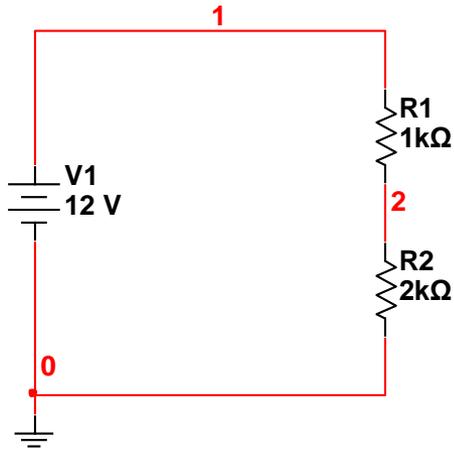
# NI Multisim | Schematics Simulation and Analysis

- **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

## Schematic Representation



## Equivalent SPICE Netlist

### 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
```

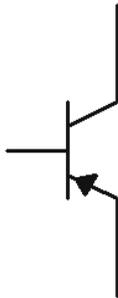
```
C1 n100 emitter 1.000E-9
```

```
D1 n100 emitter DX
```

```
e1 base n100 collector emitter 12.842917
```

```
R2 collector emitter 10
```

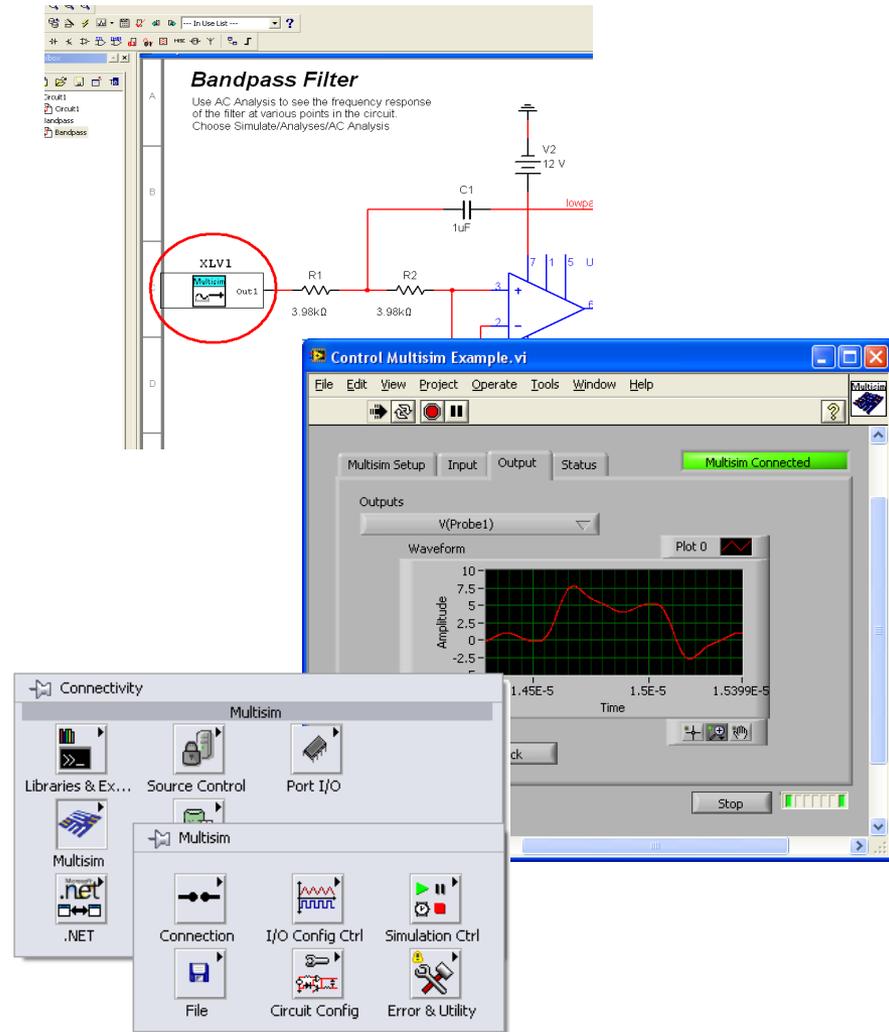
```
.MODEL DX D(IS=1e-15 RS=1)
```



# SPICE and Virtual Instrumentation

## Simulation, Measurements and Automation

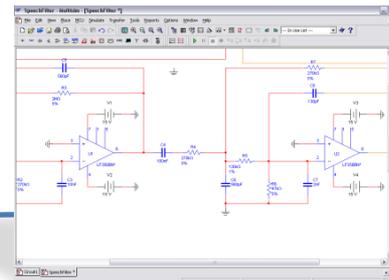
- **Bring Measurements inside of Multisim**
  - Readily available instruments emulate a test lab
  - Custom LabVIEW instruments for everything else
- **Compare Simulation and Measurements**
  - Improve Design Process
  - Troubleshoot and debug circuits
- **LabVIEW Multisim Connectivity Toolkit**
  - Multisim API Controls simulation
  - Create Virtual DUT and test with LabVIEW
  - Run, pause, stop simulation
  - **Change components**, view circuit
  - Set input, view output



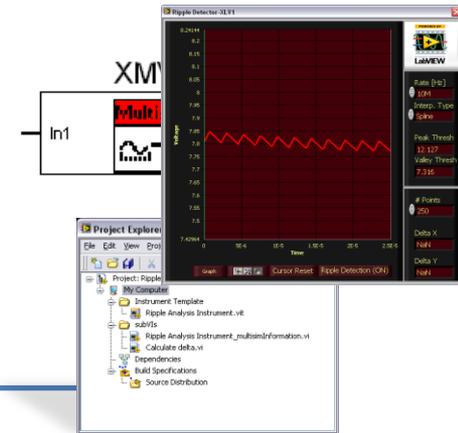
# Improving Analysis | Custom LabVIEW Instruments

- 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

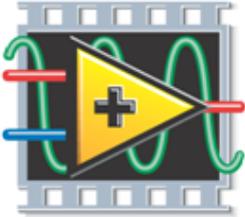


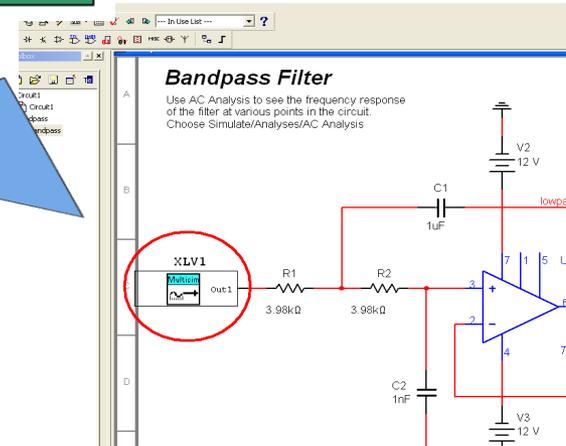
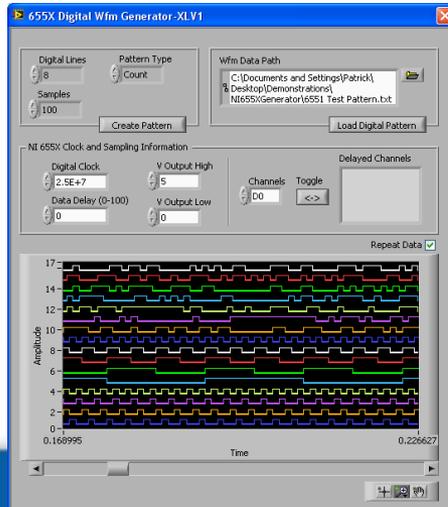
**Multisim** | Part Evaluation, Schematic Capture & Simulation

**LabVIEW** | Custom Analysis Instrument

**Ultiboard** | Layout & Routing

# Measurements in Multisim

<b>Advanced Control Design</b> (.system ID, Control Design, dynamic system simulation, etc)		<b>Order Analysis</b> (Order Tracking, Spectrum Selection, Tachometer Processing, Waterfall, Orbit / Polar Plots, Bode Plots, etc)
<b>Digital Filter Design</b> (FIR/ IIR Filter Design, Quantization, Fixed-point Modeling/Simulation, etc)		<b>Spectral Measurements</b> (Zoom FFT, Power-in-Band, Adjacent Channel Power, etc)
<b>Advanced Signal Processing</b> (Wavelets, Time-Series Analysis Time-Frequency Analysis, etc)	<b>Sound and Vibration</b> (Distortion, Octave Analysis, Swept Sine, Freq Measurements, Transient, S&V Level, Weighting, Waterfall Plot)	<b>Modulation</b> (Bit Error Rate, AWGN, Phase Noise, Constellation Plots, Eye Diagrams, etc)
<b>Signal Processing</b> (Signal Gen, Windows, Filters, Transforms, etc)	<b>Mathematics</b> (Numerics, Linear Algebra, Curve Fit, Prob/Stats, Optimization, Diff EQ, etc)	<b>Measurements</b> (Spectral, Tone Extraction, Pulse Params, Timing/Transition, Amp/Levels, etc)

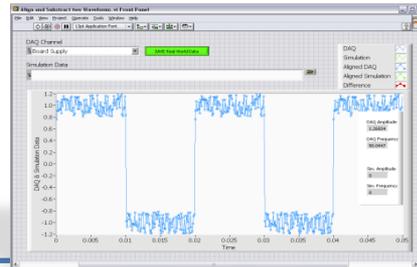


# Improving Simulation | Virtual Prototype

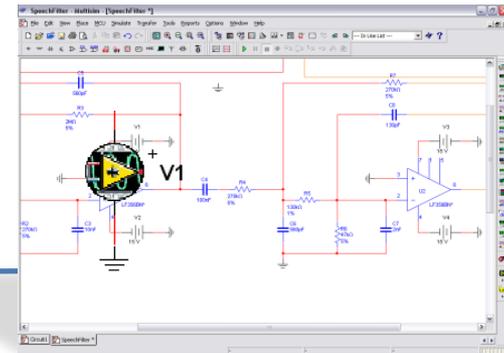
**Step 1** Create a real signal and connect to LabVIEW



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



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



LabVIEW | Measurement

Design Environment

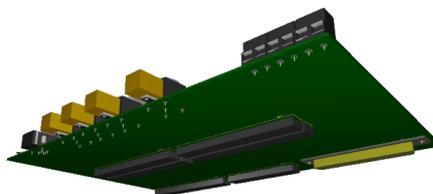
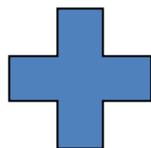
Multisim | Part Evaluation,  
Schematic Capture &  
Simulation

Ultiboard |  
Layout &  
Routing

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

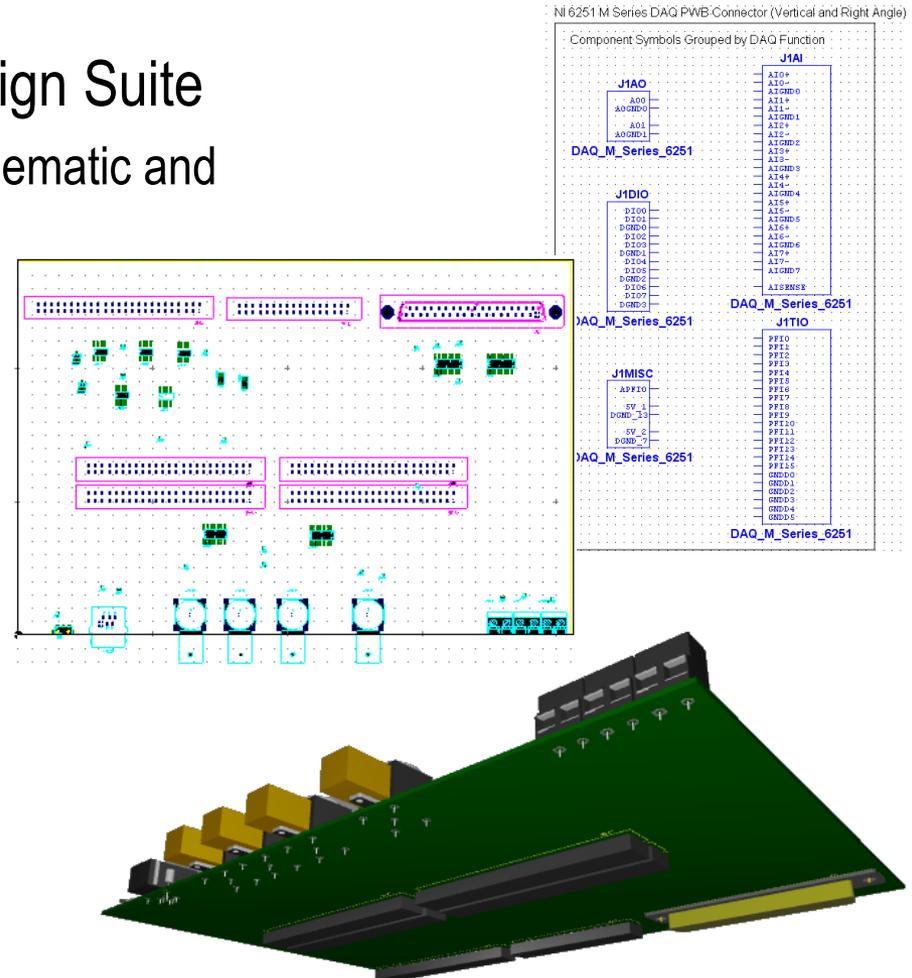
# NI for Custom Designs

- 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



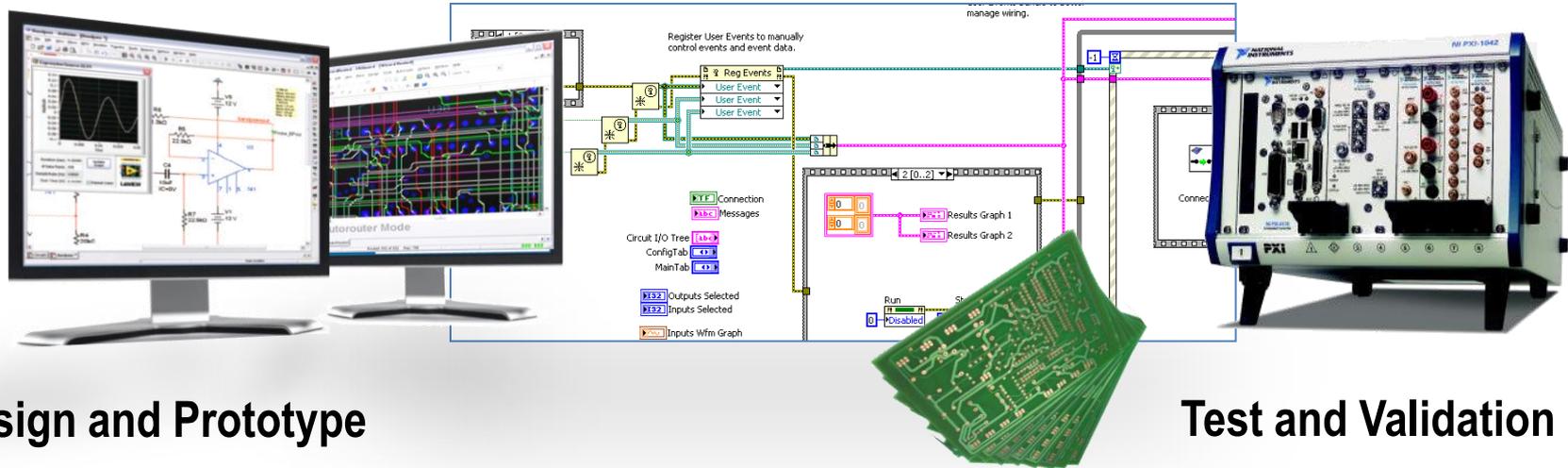
# Hardware Design References in Multisim

- 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

## Test and Validation

- Multisim 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](http://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



A screenshot of the NI Multisim website as viewed in Microsoft Internet Explorer. The browser's address bar shows "http://www.ni.com/multisim/". The website header includes the National Instruments logo, a "view cart" link, and a search bar. A navigation menu below the header lists "MyNI", "Contact NI", "Products &amp; Services" (highlighted), "Solutions", "Support", "NI Developer Zone", "Academic", "Events", and "Company". The main content area features a "NI Multisim" section with a sidebar menu containing links like "What Is NI Multisim?", "What's New in NI Multisim 10.0?", "Why Simulate Circuits?", "Technical Library" (highlighted), "Test-Drive Multisim", "NI Ultiboard - PCB Layout", and "Multisim in Education". The main content area includes a large image of a circuit simulation interface with a graph, and text highlighting "NI Multisim 10" with a "See what's new &gt;&gt;" link. Below this, there are three columns of product highlights: "Powerful SPICE Simulation", "New Over 1,200 Components Added", and "Improved Export to Mentor Graphics PAOS Layout". A paragraph of text describes the NI Electronics Workbench Group's capabilities. At the bottom, there is a section for "NI Multisim Development Systems" with three columns: "Power Pro", "Full", and "Base", each listing features and a "Compare | Buy" link. A "Highlights" section is partially visible at the very bottom.