Basics of Simulation Technology (SPICE), Virtual Instrumentation and Implications on Circuit and System Design

An Introduction

Patrick Noonan
Business Development Manager
National Instruments
Electronics Workbench Group

patrick.noonan@ni.com
cell. (207) 415-7754

Robert Berger
District Sales Manager – Long Island
National Instruments

robert.berger@ni.com
phone. (516) 507-7001

Presented at the IEEE –
Long Island Chapter on
10/25/2007

Agenda

- Introduction to SPICE
- What is Virtual Instrumentation?
- Using SPICE and Virtual Instrumentation Together
- Implications in Circuit and System Design (Demonstrations)
  - Circuit and Algorithm Development
  - Virtual Test
- Question and Answer

Introduction to SPICE

Circuit Simulation

- SPICE
- History
  - University of California at Berkeley- Larry Nagle
  - 1969 – CANCER (Computer Analysis of Nonlinear Circuits Excluding Radiation)
  - 1972 – SPICE 1
  - 1975 – SPICE 2
  - 19[illegible] 5 – SPICE 3
  - 1993 – SPICE 3F4
- Popular Commercial Versions
  - Orcad PSPICE
  - LTspice/SwitcherCAD III
  - Multisim
  - TINA by DesigSoft

SPICE Introduction

- SPICE
  - **S**imulation **P**rogram with **I**ntegrated **C**ircuit **E**mphasis
  - Developed at University of California at Berkeley
  - Three revisions, SPICE-3F5 is current

- Other simulation technologies
  - XSPICE – behavioral SPICE – combines SPICE with component behavior in C
  - VHDL – Programmable Logic Design
  - IBIS – Used to model transfer function of sophisticated components (A/Ds, etc…)
  - PSPICE, HSPICE – commercial variations of the Berkeley SPICE.

SPICE Primer

- SPICE Circuit
  - Built by creating a netlist of native SPICE primitive models.
  - Netlist is a text file that lists all connections and model information.
  - Schematic File
    - Vendor specific
    - May include package, footprint, and additional information
  - SPICE adds analysis commands on top of SPICE file allowing a SPICE simulation to extract information out of circuit (Transient, AC, Monte Carlo etc…)
- Variety of native SPICE components:
  - Resistors, Capacitors, Inductors, Sources, Transistors, etc…
- Subcircuit models
  - Can be derived to make higher order components out of these simple components

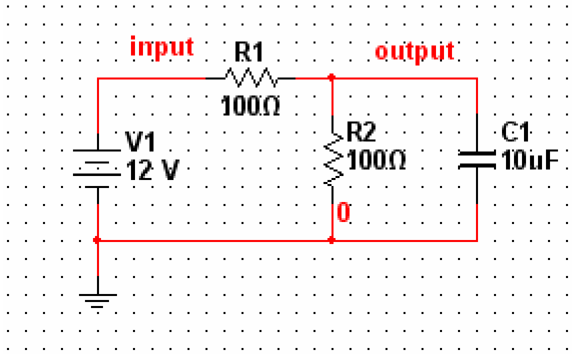
SPICE Examples

- Example SPICE netlist

  R1 input output 100

  R2 output 0 100

  C1 output 0 0.00001



- Subcircuit models
  - Command ".subckt" describes start of model
  - Command ".ends" encloses end of circuit
  - Example

    .subckt biplarjunctiontrans base collector emitter

      R1 base n100 200

      C1 n100 emitter 1.000E-9

      D1 n100 emitter DX

      e1 base n100 collector emitter 12.42917

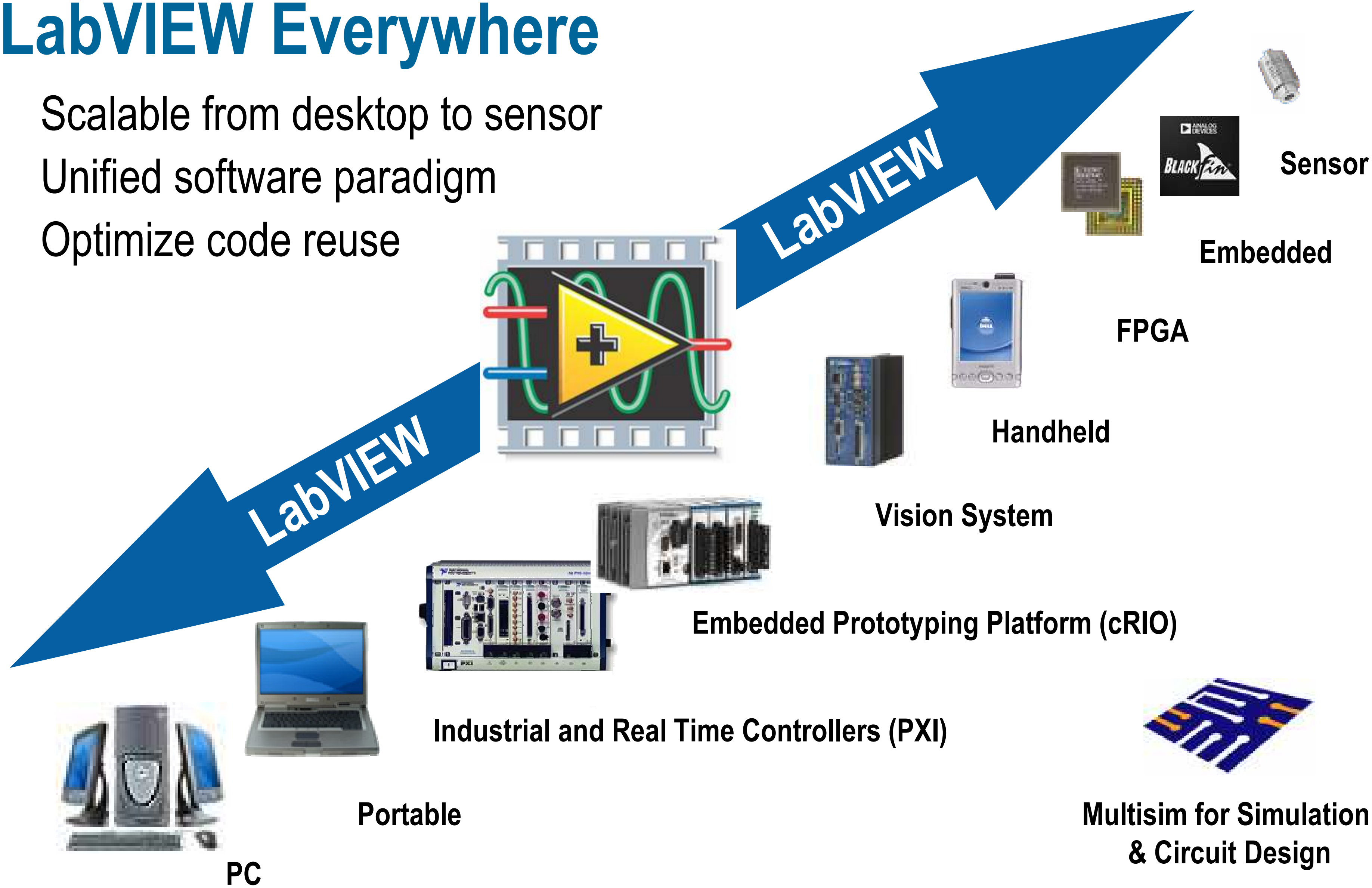
      R2 collector emitter 10

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

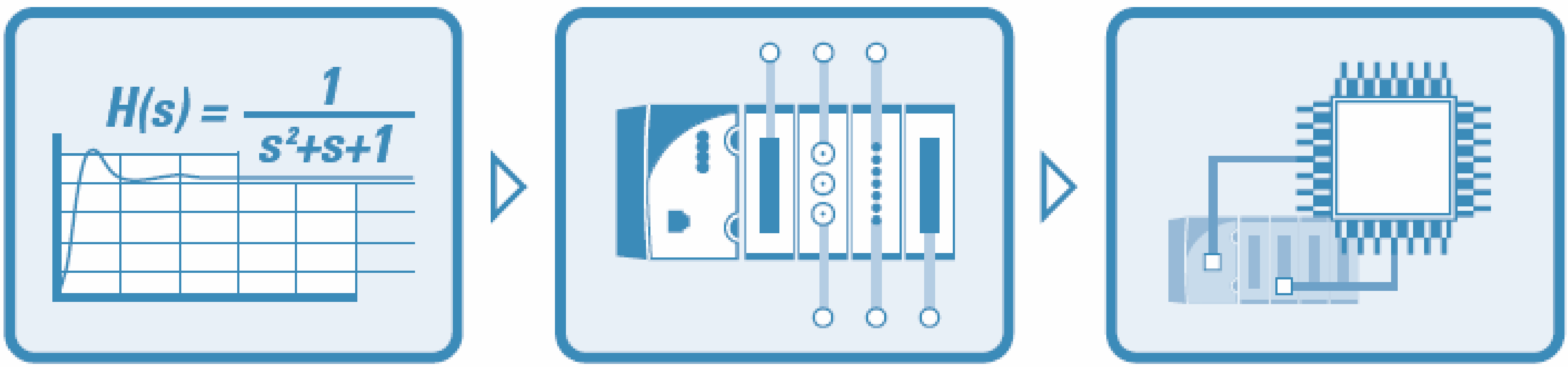
Introduction to Virtual Instrumentation

LabVIEW Everywhere

- Scalable from desktop to sensor
- Unified software paradigm
- Optimize code reuse



Graphical System Design



Design

Algorithm Design

- System Identification
- Control Design
- Dynamic System Modeling
- Digital Signal Processing

Prototype

Tight Integration with I/O

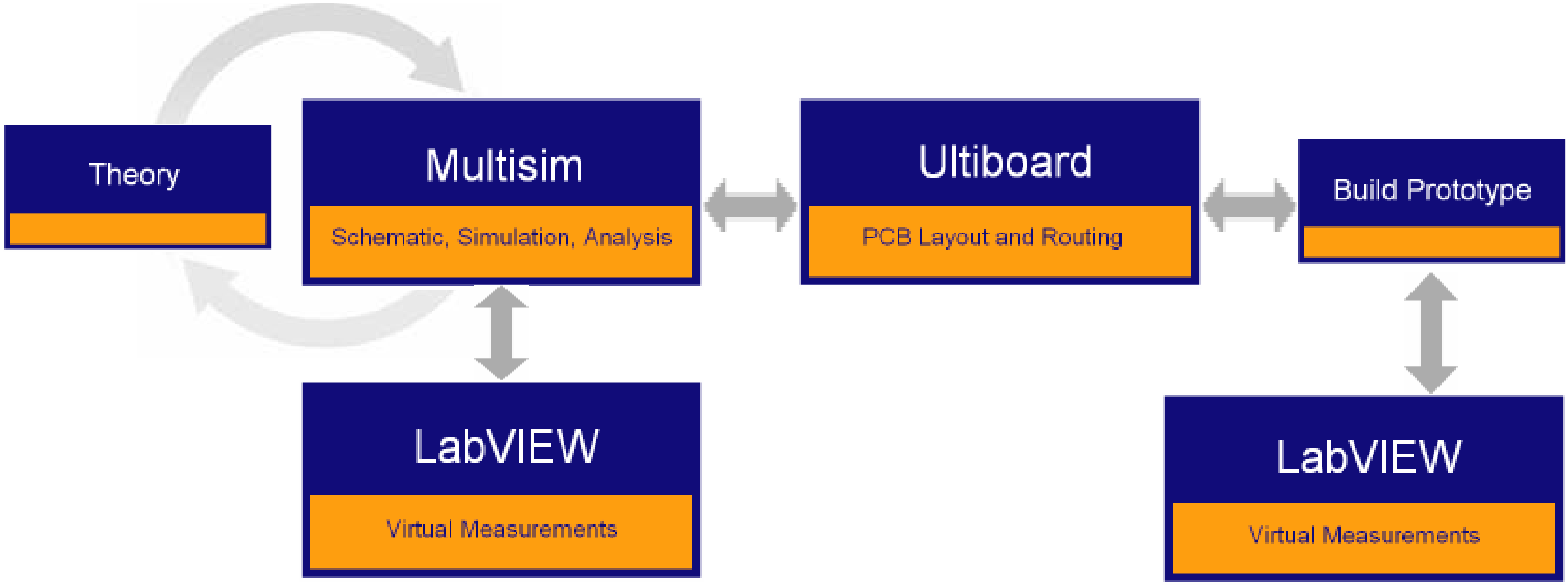
- Off-the-Shelf Device Drivers
- LabVIEW Real-Time
- LabVIEW FPGA
- LabVIEW Embedded

Deploy

Deployable Devices

- PXI
- CompactRIO
- Single Board Computers
- Custom devices

System Level Integrated Design Flow – Simulation and Virtual Instrumentation



1. Theory: Experience and Knowledge
2. Multisim and LabVIEW: Schematic, Simulation, Analysis, Real-World Input
3. Ultiboard: PCB Layout, Routing, Generation of Gerber Files
4. Prototype and LabVIEW: Virtual Measurement of Prototype

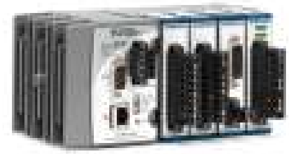
LabVIEW Toolkits

Advanced Control Design (,system ID, Control Design, dynamic system simulation, etc)		**Order Analysis** (Order Tracking, Spectrum Selection, Tachometer Processing, Waterfall, Orbit / Polar Plots, Bode Plots, etc)
Digital Filter Design (FIR / IIR Filter Design, Quantization, Fixed-point Modeling/Simulation, etc)		**Spectral Measurements** (Zoom FFT, Power-in-Band, Adjacent Channel Power, 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)	**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)

Available Graphical System Design Tools – Design, Prototype and Deploy

- Design
  - Electronics Workbench Multisim®
  - NI-ELVIS with data acquisition
  - LabVIEW and Design Toolkits
  - PXI with Modular Instrumentation
  - Signal Express

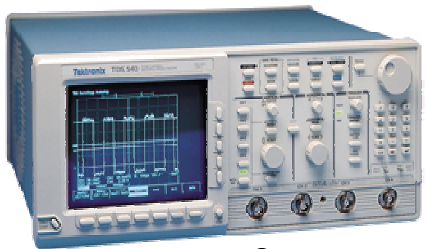
- Prototype
  - LabVIEW RT & FPGA
  - Compact RIO (cRIO)
  - Custom cRIO module kit
  - R Series DAQ



- Deploy
  - LabVIEW RT & FPGA
  - Compact RIO (cRIO)
  - LabVIEW Embedded
  - Electronics Workbench Ultiboard®

Test Tools for Design Engineers

Traditional fixed functionality bench-top Tools



Scope

Logic Analyzer

Power Supply

Function Generator

DMM

Engineer-defined computer based instrumentation

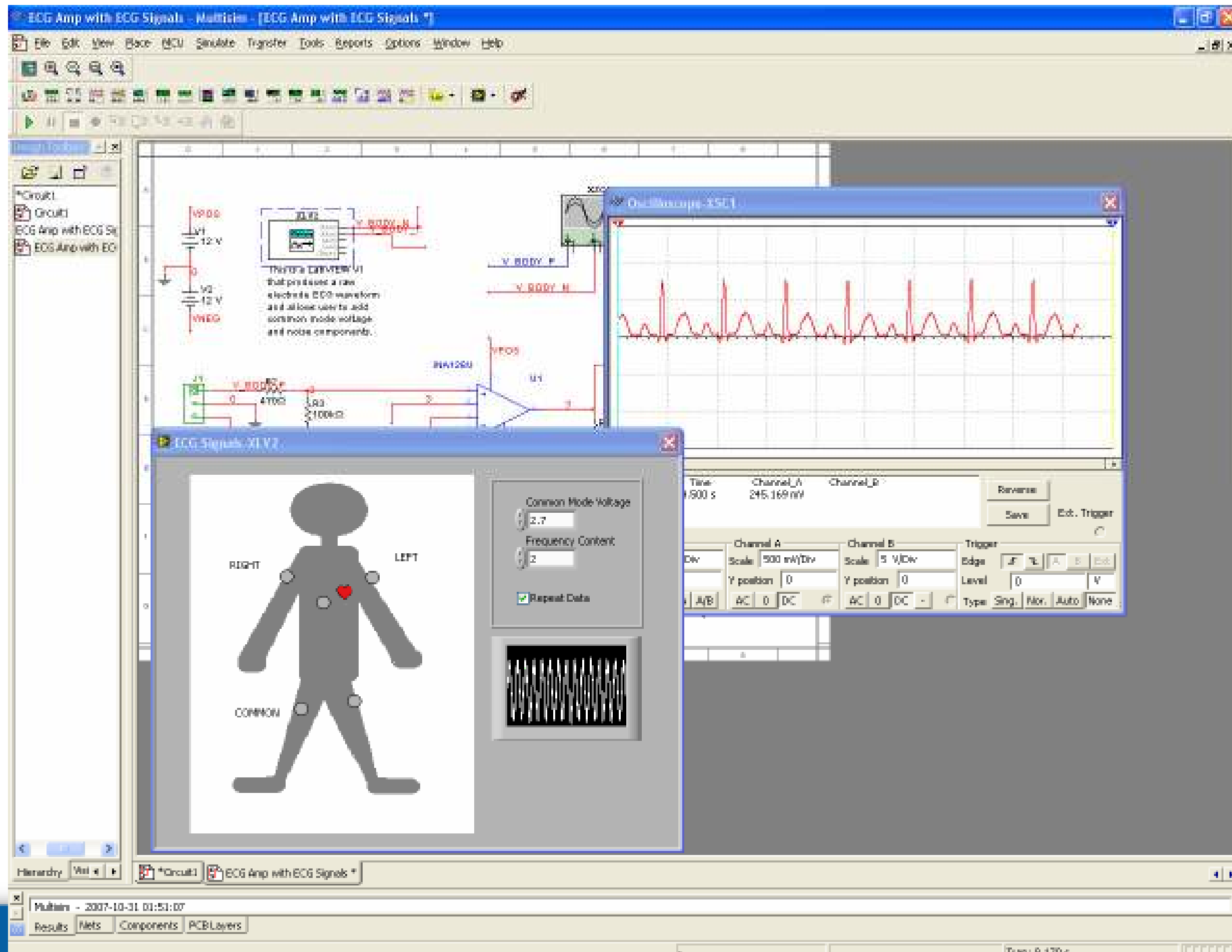
1. **Automation**
   (LabVIEW Signal Express)
2. **Flexibility (Custom Measurements)**
3. **Smaller Footprint**

Implications in Circuit and System Design

Design Examples

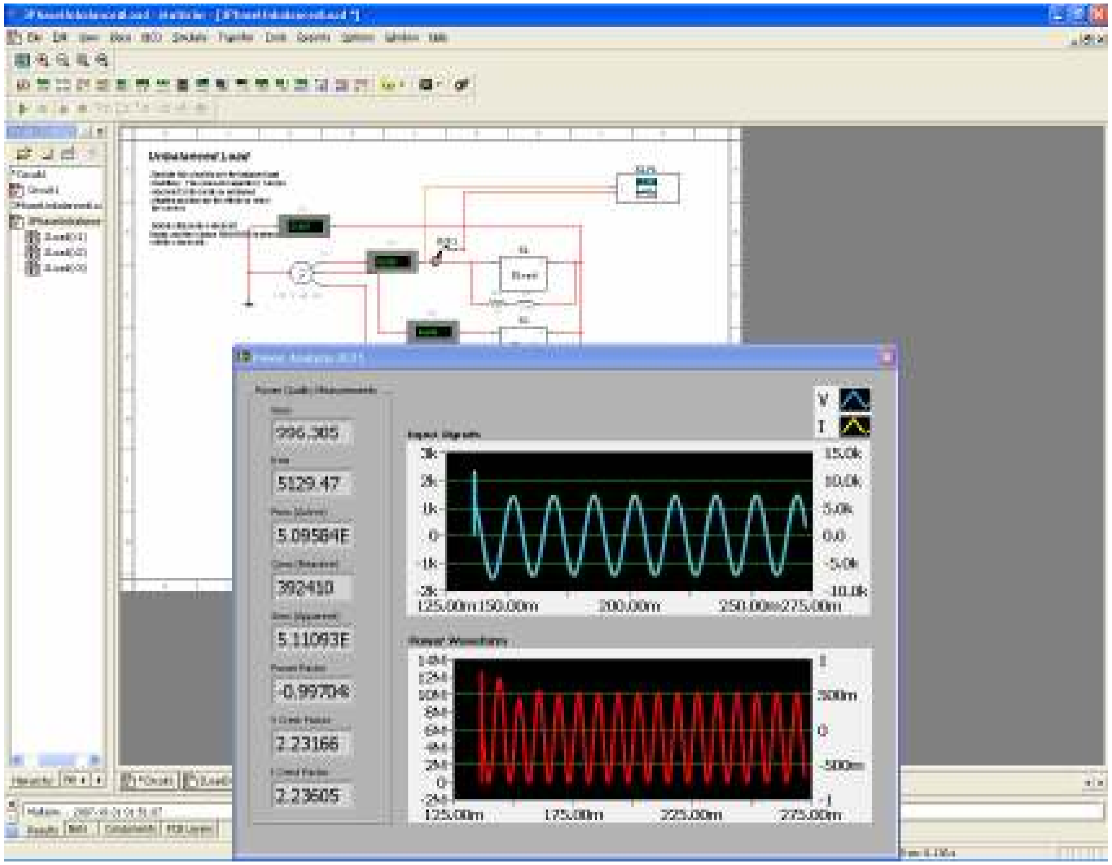
Example 1 – Using LabVIEW VI as a Signal Source

- Real World Signals: LabVIEW ECG Signal Generation + Impairments for Physiological Amplifier Development



Example 2 Using LabVIEW for Custom Measurements within SPICE

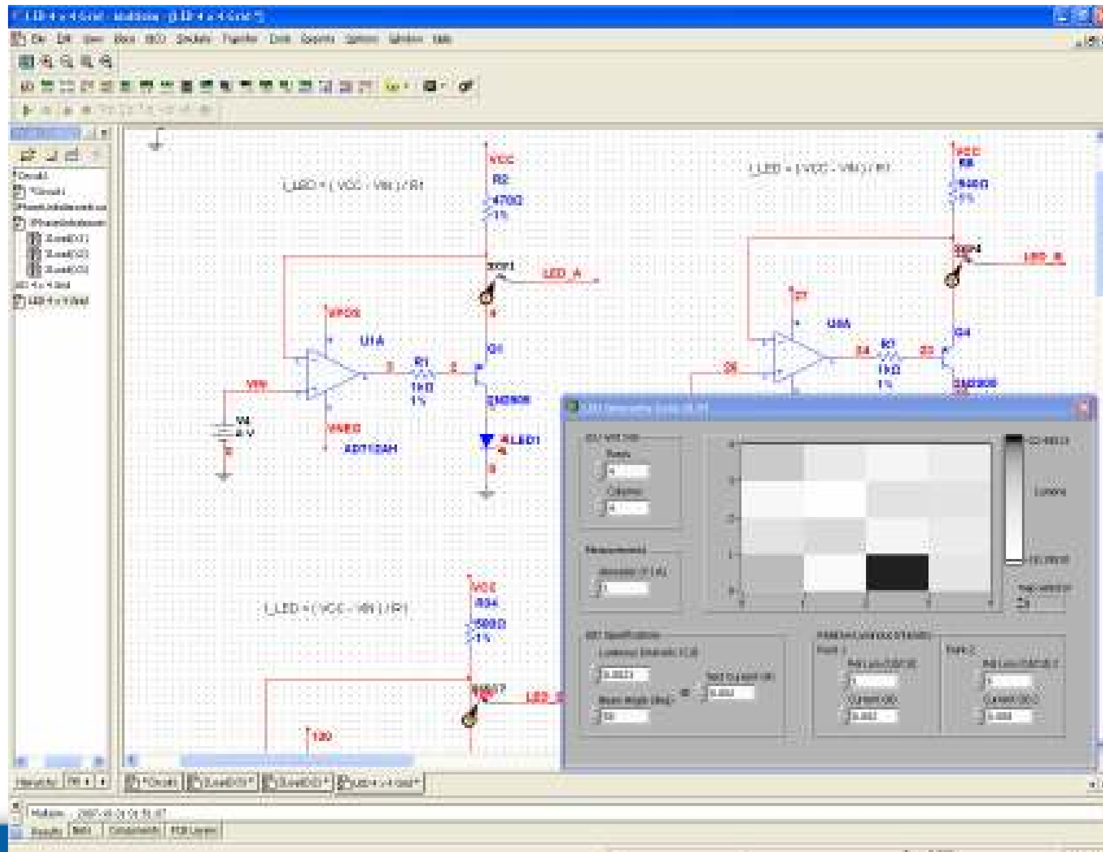
- Power Quality Analysis Measurements using LabVIEW inside of Multisim



NATIONAL INSTRUMENTS
ELECTRONICS WORKBENCH GROUP

Example 3 – Physical Measurements from SPICE Simulation

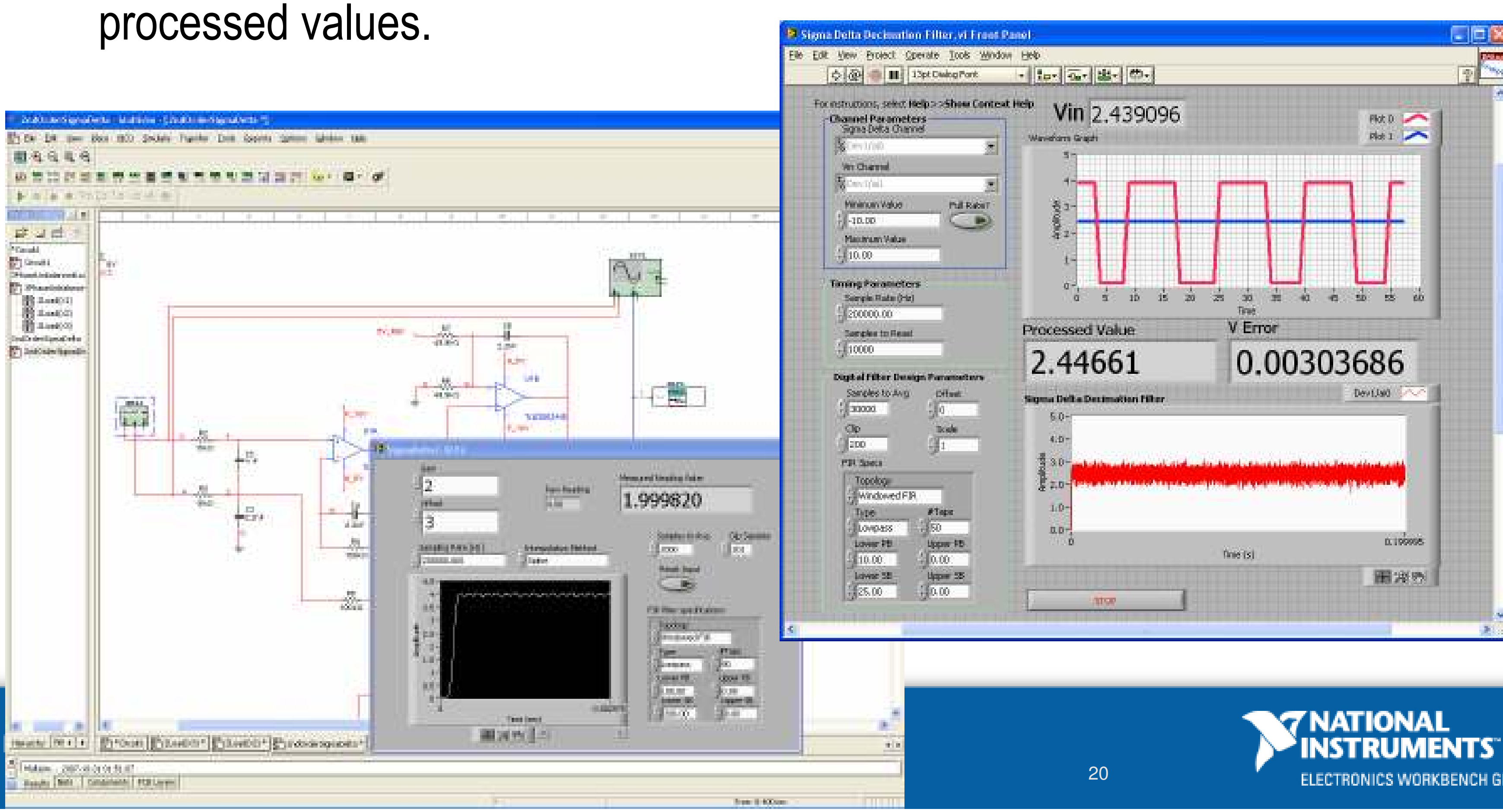
- Derived Physical Measurements from SPICE - Optical Uniformity Measurements (Lumens) on a 4x4 LED Array from SPICE Simulation



NATIONAL INSTRUMENTS ELECTRONICS WORKBENCH GROUP

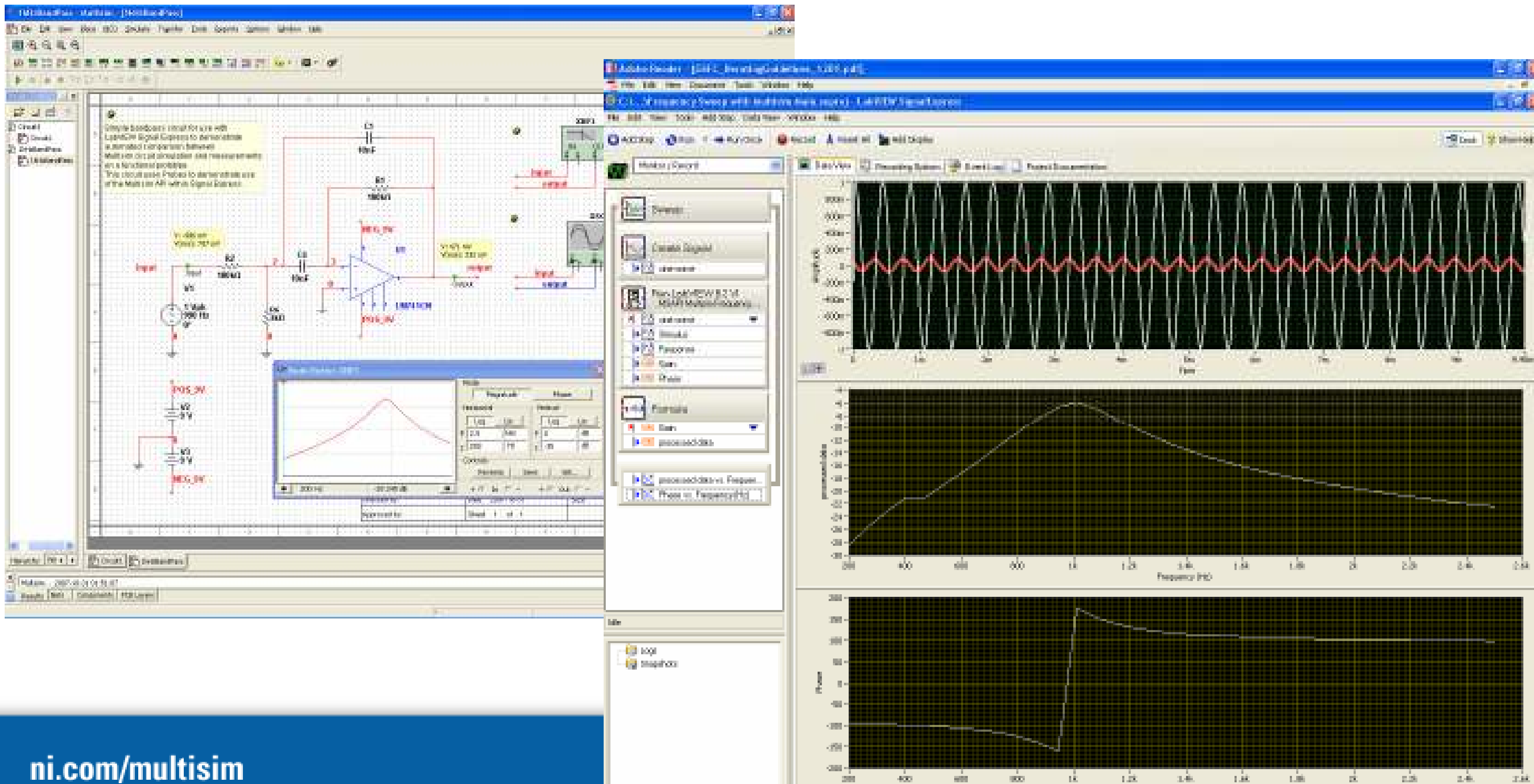
Example 4 – Using LabVIEW for DSP Filter Development within SPICE simulation

- Sigma Delta ADC – Circuit running in Multisim – LabVIEW used to design and implement DSP Filter.   Test VI on right showing implementation of ADC and good agreement between input and processed values.



Example 5 – Virtual Device Testing

- Signal Express Test Script – Running ‘Virtual Device’ simulation in Multisim to compare and correlate simulation with real test data. This example uses LabVIEW to control Multisim via ActiveX API



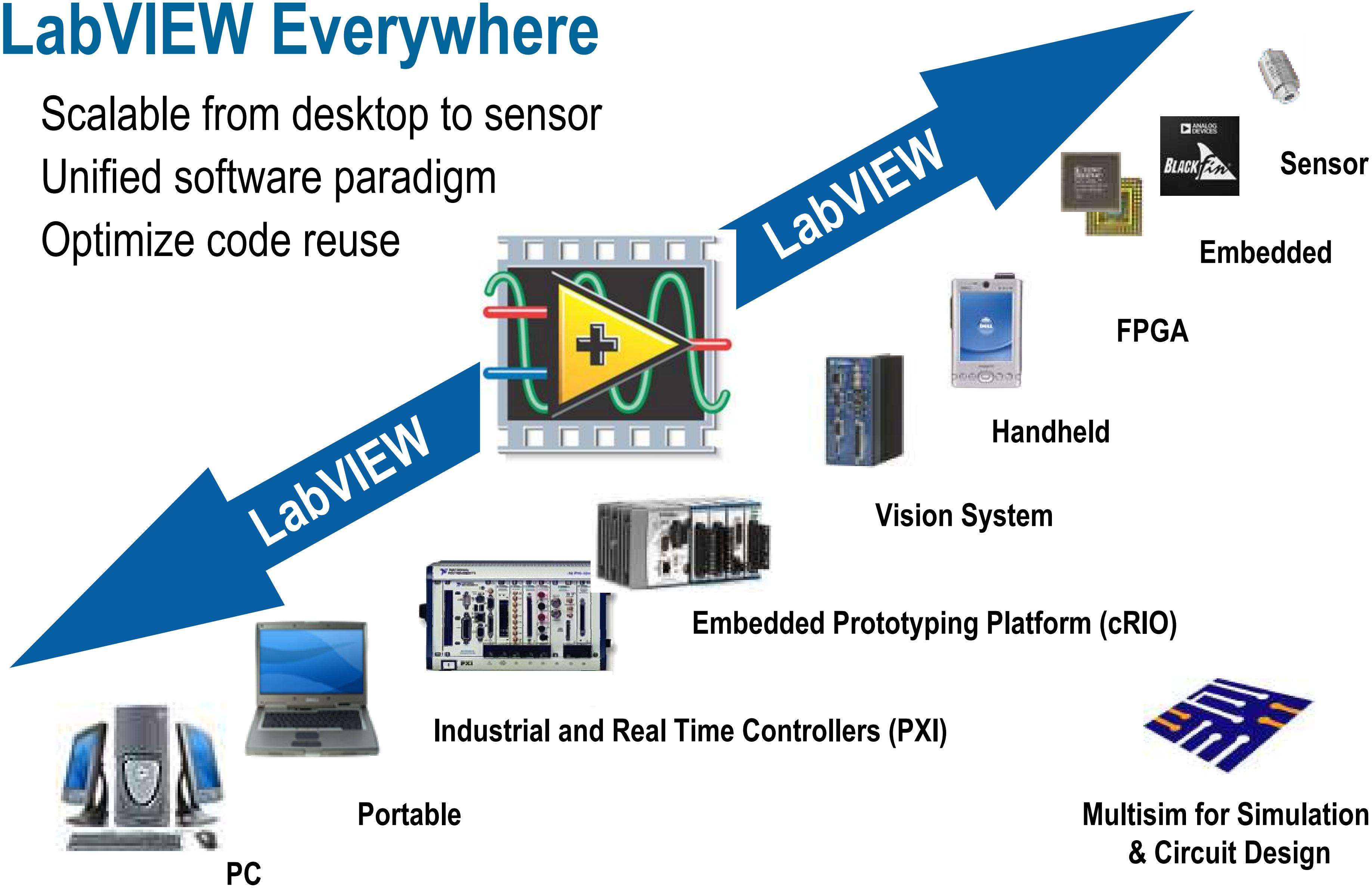
NI Multisim Tutorial

[Not Given in Presentation]

see ni.com/multisim

LabVIEW Everywhere

- Scalable from desktop to sensor
- Unified software paradigm
- Optimize code reuse

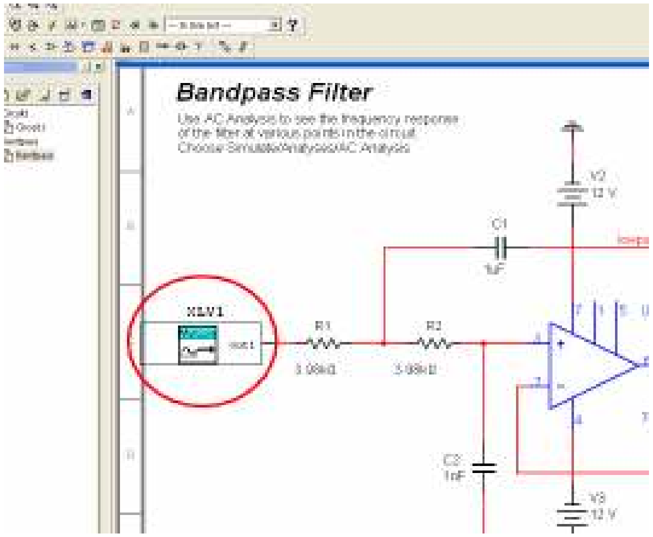


Multisim and LabVIEW Integration

- **Multisim**
  - Great for rapid designing of a circuit (schematic entry and simulation)
  - Placement and wiring technology speeds development
  - Once circuit is wired simulation is ready to run.
- **LabVIEW**
  - Great for rapid development of test, measurement and automation (Flowchart)
  - Over 4000 instrument drivers directly accessible for LabVIEW
  - Once control and functions objects are wired, program is ready to run.
- **Multisim and LabVIEW**
  - Real stimulus signals can be directed added to circuit simulation
  - Advanced LabVIEW measurements and algorithms can be tied into simulation
  - LabVIEW can be used to ‘drive’ the prototype and verify the design specs!

How Easy is to use LabVIEW Virtual Instruments in Multisim?

1. Utilize LabVIEW to measure and save real-world signals
2. LabVIEW VIs represented as part of Multisim simulation. Simply place the VI as you would a component
3. Input signals measured in in **LabVIEW** (step 1) into the VIs that are placed in **Multisim** (step 2)
4. Simulate!



OR – Create your own CUSTOM LabVIEW Instruments!

NI Multisim | Where to Learn More

- For product information: ni.com/multisim
- Professional Resources: ni.com/multisim/professional
- Academic Resources: ni.com/academic/circuits
- Circuit Design Technical Library
  - SPICE Simulation fundamentals
  - Example Circuits
  - Custom LabVIEW Virtual Instruments
  - User Guides and Manuals
  - Discussion Forum
  - Support Page
- Free Component Evaluation – Multisim - Analog Devices Edition

  ADI Edition: analog.com keyword search: multisim